Contents

What's New in 2017 Extension 1

1

Welcome to Autodesk Flame 2017 Extension 1. This release introduces many new creative features, performance and workflow enhancements, and other user-requested improvements. Make sure to check *http://www.autodesk.com/vxf* for the latest Flame documentation (including Installation Guides, Release Notes, and other documents). Also, many new feature videos (as well as other workflow videos) are available at *https://www.youtube.com/user/FlameHowTos*. This Learning Channel is updated frequently, so be sure to subscribe or bookmark the page.

NOTE For 2017 features, see What's New in 2017 (page 15).

Follow these links to navigate to the sections below:

What's New in Project Management

- After years of good and loyal service, the Flame Classic Engine is now retired. Every project opened in this release (newly created, converted, or restored from archive) now uses Flame Reactor.
- When opening a project created in a previous version, you are now offered only one option: Convert. The conversion creates a copy of the project and converts this copy to the new version of the Flame, duplicating the setups. The original project remains untouched and accessible to the application that created it.

What's New in Colour Management

This release introduces a more active approach to colour management. All media is now tagged to identify its colour space and this information is used to drive an automated new viewport system. In addition, Thumbnails, and the Sparks and Stabilizer viewports are now colour managed.

See Tagging Clips with a Colour Space (page 1756).

What is New in Supported File Formats

Import

Support for XF-AVC files from the Canon EOS C700:

- Supports both AVC-Intra and Long GOP
- Spanned clips
- SD/HD/UHDTV/4K in 8-, 10-, and 12-bits

Updated RED support to RED SDK 6.2.1:

- Support for RED HELIUM sensor (8K/S35)
- New REDWideGamutRGB.colorspace
- New Log3G10 Gamma Curve
- Lens Metadata in the Metadata tab of MediaHub

Updated ARRIRAW support to ARRIRAW SDK 5.3:

- Support for the ALEXA mini and AMIRA cameras
- Support for new colour spaces:
  - Video ITU2020
  - Video DCI D60
  - Video DCI D65
- Supports new processing version 5.0 (SW & HW)
- Supports anamorphic ratio 1.25

OpenEXR

- Tape Name is now read from the OpenEXR header. The default import option is now set to Tape Name from File Header, instead of Tape Name from Directory.
- Keycode data is now read from the OpenEXR header.
- Support for Data & Display windows. The new **Pixel Space Window** option allows you to import an OpenEXR using either the Data or the Display window. Available in **Format Specific Options ➤ OpenEXR ➤ Image**. See Pixel Space Window (page 213).

Support for Panasonic AVC-Intra LT formats from Varicam LT cameras:

- The AVC-Intra LT format is encoded to half-height resolution to reduce the data rate.
- A new import option, **Scale To Full Resolution**, debayers AVC-intra LT media to its full resolution. This option is enabled by default. Available in **Format Specific Options ➤ Panasonic ➤ Image** .

Open Clip

You can now create multi-version open clips where each version has a different number of channels.

NOTE To improve browsing speed, media files without proper filename extensions are no longer recognized by the MediaHub.

Enhanced Multi-channel Support

In previous versions, a multi-channel file would be imported as a Matte Container where each RGBA layer was itself wrapped in a Matte Container. The only way to import and keep the multi-channel structure of a clip was through Batch's Read File node. The Read File node allowed for keeping the multi-channel structure but at the cost of not being able to archive, cache, proxy, or wire the media.

This release enhances multi-channel clip support through the following changes:

- New import options in the MediaHub.
- Improvements to the Conform, Timeline FX, some Tools, and Batch FX worlkflows to improve support for multi-channel clips.
- New Import node in Batch and Batch FX, which behaves like any other media in the schematic. Read File node is now deprecated, and only included for backward compatibility.

See About Multi-channel Clips (page 82).

Export

Miscellaneous improvements

- Media encoding speed has been improved when source content is located on a network file system.
- Tape name and Keycode data is now included in exported OpenEXR files.
- Backburner Monitor now displays the jobs that are part of a Sequence Publish as a `sequence name/segment name` pair. This makes it easier to identify background jobs related to the sequence publish. Only applies to background export jobs.
- MXF XDCAM 422 exported by Flame are encoded based on Sony's XDCAM format, and all start with a full Group of Pictures (GOP). Some broadcasters use a different flavour of XDCAM 422, where the media start with a short GOP (starting on an I-frame). You can meet these encoding requirements by setting an environment variable. See Exporting Clips and Sequences (page 89).
- You can now use the command line tool `flame_export` to export contents from an opened project. See Exporting from the Command Line (page 254).

What's New in MediaHub

Open Clip new import options:

- You can now override the frame rate of an Open Clip with the new Rate box. **Format Specific Options ➤ Open Clip ➤ Rate box**.
- You can now use the new Align to Zero option to ignore the timecodes defined in the Open Clip, and align every track (and their versions) to timecode 00:00:00:00. **Format Specific Options ➤ Open Clip ➤ Rate box**

OpenEXR improvements:

- The Clip Name option is now set to Channel by default.
- In the Previewer, the Metadata tab now displays all the metadata available from OpenEXR files, in section **Clip Information ➤ Attributes**.

Previewer improvements:

- The Metadata tab now presents the available metadata in collapsible groups, improving legibility.
- P2 and Varicam clips now display additional metadata in the Previewer's Metadata tab, in section **Clip Information ➤ Panasonic XML**.

What's New in Conform

Miscellaneous Improvements

- Audio tracks of Sources Sequence are now correctly positioned when the sequences located in the Reel Group have different Record timecodes.
- Locators from a conformed AAF should no longer be missing or misplaced.
- The image in the Conform Previewer is now Fit to size by default. The Pan and Zoom values are now independent those of the timeline Player.
- When matching a source to an event in Conform, the handles amount displayed in the Event table is now displayed in red if the source does not have enough handles. You can still force a match by manually linking the source to the sequence's event.

Connected Conform Workflow

Miscellaneous Improvements

- There is a new preference: **Preferences ➤ Timeline FX / Batch /BFX ➤ Ignore in Publish/Conform**. When disabled, the Batch Group created with the Create Batch Group option starts at a frame matching the source frame number of the background element. When enabled, the Batch Group will start at frame 1. The preference is disabled by default.
- Head and Tail of clips are now unrolled if they do not visually modify the rendered range of a Batch Group created in a Connected Conform Workflow.

New Workflow: Sources Connections

It is now possible to create Sources Connections. The Segment Connection creation introduced in 2017 allowed you to enable Segment Connection on a segment, which could then be duplicated but remain connected to the original segment. The new Source Segment Connection allows you to set Connected Segment from a source to instances of this source within the same Reel Group.

See Connecting Sources (page 298).

What's New in Connected Colour Workflow

- In a two monitor configuration, where Flame is defined to use one monitor, the Tablet Monitor options are available and the tablet is set to the single monitor resolution. This provides the ability to use the full tablet size in both Lustre and Flame.
- When running Flame and Lustre at the same time, the Tangent Element panels are now automatically configured to work in the currently focused application.
- It is now possible to use Background Rector to render Batch FX. This is only available on Dual GPU configuration with a properly configured Flame Family product.

  NOTE The Background Reactor options are displayed even if the service is not available.

- The "Pointer Span" preference is no longer only available when Flame is configured in a two monitor setup. It is available the moment two monitors are used on by your workstation. This change was made to allow the Pointer Span to be temporarily locked to Flame when Connected Colour Workflow users have both Flame and Lustre running side-by-side.

What's New in Batch and Batch FX

The Write File Node **Clip Version** button is now an option box with 3 options:

- No Versioning: No versioning for the created Open Clip.
- Custom Version: Versioning in the Open Clip, with an version number set by the Version Field. This was the behavior when the Clip Version button was enabled in previous versions.
- Follow Iteration: This essentially new option displays the value of the current Batch Group's Iteration in the Version field. Follow Iteration ensures that the Version number always matches the value of the Iteration, so iterating the Batch Group automatically updates the Write File node's version.

The information displayed below a node in Batch, Batch FX and Modular Keyer is now more legible:

- The name is now always on top. The Indicators have been moved one row below the name.
- The name is now always displayed using the same colour.
- The Indicators and Resolution information are now shown in the same colour.

The Clip node now displays the Clip Info and Metadata sections, to the right of the Clip Channels User Interface. Extended View for Multi-Channel clips now displays the following information:

- **Type:** This column displays the Channel Type, as derived from the Channel Rules configuration file.
- **Name:** The name of the channel.
- **Colour Space:** The resolved colour space, based on the Channel Type.

There is a new preference: **Preferences ➤ Timeline ➤ FX ➤ Batch ➤ BFX ➤ Ignore in Publish/Conform**, disabled by default. When disabled, the Batch Group created from a Shot Sequence or Publish starts at a frame matching the source frame number of the background element. When enabled, the Batch Group will start at frame 1.

Expressions / Keyframes

Expression / Keyframes indicators are now displayed above a Batch, Batch FX and Modular Keyer node. The indicators appear when the node contains at least one Expression or one Keyframe. See: Keyframe Indicator Reference (page 1688).

Batch Schematic Cleanup

It is now possible to clean up a Batch, Batch FX and Modular Keyer Schematic using the Clean Up Branch Upstream and Clean Up Schematic functions:

- Clean Up Branch Upstream is available on all nodes and is accessible from the Contextual menu. All the nodes upstream are re-organized when this option is selected.
- Clean Up Schematic is available from the Contextual Menu when you click in the Schematic background. All nodes in the Schematic are re-organized.
- These operations are undoable.

Improved Batch or Batch FX Setup Portability

A Batch or Batch FX setup now contains the paths to the media files referenced by the clips in the schematic, making it easier to move Batch or Batch FX setups between Creative Finishing workstations. See Clips, Media, and Batch Setups (page 533).

Elbow Node

The Break Link feature has been replaced by the Elbow node. See: Creating Break Points in Node Connections (page 505).

Compass Node

The Compass node is a node that can be used to define and identify an area in the Batch, Batch FX and Modular Keyer schematic, by encompassing the nodes of your choice in your process tree. See: Using the Compass Node (page 510).

Batch Setup Start Frame

It is now possible to modify the Start Frame of a Batch setup, using the numeric field located above the Current Frame field. See: Batch Setup Start Frame (page 501).

Miscellaneous Batch or Batch FX Improvements

- Import Node in Batch or Batch FX: The Import Node allows you to import content from File System to the scene. The result creates clips in the Batch Group. These clips can then be cached or proxy generated (using the contextual menu from clips in the Batch Reels). Use the Import Node instead of the Read File node when you want your content to be part of an Archive.
- The Batch Read File Node Absolute Frame Offset preference also affects content imported using the Import node.

  NOTE This does not apply to content that was already in the application. It only applies to new content imported in Batch.

- The Import keyboard shortcut adds an Import node to the Scene and the imported media is placed at the cursor position in the Batch canvas.
- Nodes are now pasted under the cursor when the Paste shortcut is used while the cursor is over a Schematic view.
- Branch layouts are now preserved when you use the Explode FX, Convert to 3D Comp, Explode One, and Explode History functions. The nodes are also displayed where the exploded clip is now located rather than their original location.
- The following improvements have been made to Notes:
  - The Expand / Collapse status is now persistent and saved within the setup. A note's size is now persistent and saved in the setup.
  - Notes are now expanded by default when added from the Node bin.
- The current version number of a Write File node is now displayed within the Render List under the Version column.
- Disabled is displayed for a Write File node where the Clip Version button is disabled.
- The version number is displayed and can be edited when the Clip Version button is enabled in the Write File node.
- The Step rendering function has been moved from Batch's primary user interface to the Render List sub-menu. The setting is still global and is applied to all the Render and Write File nodes present in a Batch Group.
- Hidden Links connected to a MUX node are now visible when the upstream / downstream nodes are selected. The links are also visible when you connect a node to a MUX node that has the Hide Links function enabled.
- It is now possible to perform a `N-click` to only propagate a Node name to the Name layer in Burn-In Metadata.
- Starting Flame with Shotgun, using a Batch setup that includes a Write File node, now appropriately sets the correct version to render.
- The name of the tagged colour space is written under the clip when the Resolution option is selected in Batch or Batch FX Prefs.
- The Batch FX Schematic view now automatically fits the viewport when you enter Batch FX the first time.

What's New in Action

Camera and Viewport Improvements

- You can now create a camera on demand. Use the keyboard shortcut `Spacebar+C` you create a camera object that contains the viewing settings of the camera used in the current viewport. This allows you to navigate a scene using Working Cam or rendering camera and create new cameras as you look for creative angles.
- You can now easily frame the contents of the viewport or the Action schematic using keyboard shortcuts. See Framing Views (page 561).
- New keyboard shortcuts allows you to use a 3-button mouse or the tablet pen to orbit, dolly, or track the viewport. See Modifying the Camera (page 850).

Object Manipulations Improvements

Improved Selection

- Faster picking: Work has been done to improve picking in the viewport, making it more precise and faster than ever.
- Holding `Shift` as you click an object selects that object and every other object behind it (Z-depth picking).
- You can now multi-select objects using `Ctrl+click` directly in the viewport. You can also do the same thing in the Action schematic.
- The selection mechanism in Action's viewport and schematic now matches that of Batch or Batch FX. It now properly supports all Copy/Cut/Paste/Delete operation of a multi-selection scheme.
- You can enable multi-node editing with the `Spacebar+M` keyboard shortcut to display the multi-axes tab.
- You can now lock the selection in the viewport. See Locking Selection (page 580).
- See Selecting Objects in Action (page 575).

Improved Manipulations

- The new Axis widget provides an easy target for selection in 3D space, while preventing accidental edits by being clearly different from the new manipulators.
- New, 3D-inspired, manipulators for Move, Scale, Rotate, and Center (Pivot) replace the old axis widgets. See Action: Moving, Rotating, and Scaling Objects (page 613).
- You can now use the manipulators in new Object, World, or Camera coordinate spaces. See Camera Space, Object Space, World Space (page 617).
- A new preference sets the size of the Axis widgets. In **Action Preferences ➤ Display ➤ Axis Scale field**.
- The icon visibility setting contains a new option (Group Off), allowing you to remove the icons of objects contained inside of a group. This displays an All icon mode without the objects contained inside a group, improving the usability of the F4 scene view .

Miscellaneous Action Object Improvements

- Double-clicking a Surface, 3D Text, Light, or Camera node in the node bin now adds that object to the front of the currently selected object.
- A new keyboard shortcut, `Spacebar+H`, toggles between displaying or hiding texture-related axes. Use this keyboard shortcut to reduce the number of axes displayed in the viewport, making it easier to pick the correct axis to transform an object.
- Action widgets can now be affected by Z-depth occlusion. Use the keyboard shortcut `Spacebar+O` to toggle Z-depth occlusion on or off.
- Two new selection indicators are now available: Wireframe and Outline. The traditional Bounding Box is still available. Switch between indicators with the keyboard shortcut `Spacebar+I`.

- The Viewport bar displays three new preferences for quick access: Icons Visibility, Space Selection, and Z-depth Widget Occlusions.

See Action: Moving, Rotating, and Scaling Objects (page 613)

Image-Based Lighting (IBL) Enhancements

- IBL maps are now working properly with PBS Material, using the Shader node in Physically Based mode. The IBL now provides the exact diffuse reflection result based on the Roughness level of the material. When a material is not PBS, it behaves like before, so the same IBL can be used for both types of material.
- Ambient lights now work properly with a PBS material.
- Four new default IBL images are available.
- External textures are now supported for all Action map types. For IBL textures, the default location is `/presets/action/ibl`.

See Relighting: Image-Based Lighting IBL (page 690).

Physically Based Shading

- A new Action texture map node is available: PBS map. This node allows you to have a single map with all the map types that can be used in a PBS workflow: Anisotropic, AO, Base Color, Metallic, Opacity, Roughness, Subsurface scattering, Specular levels, and Specular tint. There is also a Custom box that gives you access to 10 map IDs to use in custom Lightboxes. Similar settings are available in the Shader Node when you select Physically Based as the Shader Type, but by using the PBS Map, you can affect the shader on a per pixel basis instead of as a global value for the whole object)

  NOTE On a Mac system, you are limited to Base Color, Roughness, Metallic, AO, and Opacity shader types.

  See PBS Mapping (page 813).
- Use the new Substance PBR (Physically Based Rendering) node to create physically based textures based on real world presets. Many Substance PBR preset assets are available. See Substance Textures and Substance PBR (page 762).

Action Geometry Import/Export Improvements

Importing FBX and Alembic files:

- You can now see the data structure of imported FBX and Alembic files. You can edit the content by connecting the soft-imported data structure to native Action objects using parenting, mimic link, lighting link, or GMask link, similar to the new Group node workflow.
- You can change some global geometry controls from the FBX Scene or Alembic Scene node UI. You can also locally cache the content by using the Cache button, which allows you to archive the data.
- New Maya hypershade assets have been added to the list of provided FBX files.

See Working with FBX and Alembic Scene Nodes (page 704).

Exporting FBX:

- You can now export FBX files from an Action setup. You have the choice of doing this from all supported objects in the scene, or only the selected objects. Supported objects are Surface, Geometry, Axis, Camera, Light, and Point Cloud. The textures are only exported for the current frame, as well as animated extended bicubics. See Exporting FBX Geometries (page 710).

Other Action Import Improvements:

- You can now access the Import browser through the right-click contextual menu in the Action schematic. The Import browser has also been updated to fix various issues.
- A new global preference is available to set the default import path (located by default in `~/presets/models`). A new python hook is also available to set this default path.

Action Miscellaneous Improvements

- Group nodes in Action have been improved. The Group node is now an object with a UI and has a schematic view (similar to the Batch Group) that can be edited. See Grouping Objects (page 581).
- You can now specify external media to be used by any Action texture map. If the media you want to use resides elsewhere on your filesystem, you can use the Read File tab of the map menu to load the texture, and then decide if you want the texture to be managed by Action, or not. See Working With Textures in Map Nodes (page 762).
- A new mode in the Action Software Anti-aliasing Setup menu Preferences section (AA N Samples) allows you to choose the number of samples in the corresponding Samples field. This allows you to have an arbitrary amount of Software AA values and animate them over time.
- Action Lens Flare nodes now allow you to save and reload presets containing animation channel expressions. Note that only animation channel expressions found inside the setup will reload properly (Lens Flare nodes and its children). Also keep in mind that node names need to be unique for expressions to resolve properly (if you load the same Lens Flare setup twice, you'll have an expression resolving clash).
- Multi-Render Target (MRT) allows Action outputs to be much more efficient when used by the Camera FX pipeline. Depending on the Rendering settings, the scene is rendered only once or twice instead of as many times as there were output passes in the past, leading to a significant speed increase in the case of Camera FX requiring many input connections. See Rendering Camera FX (page 825).

What's New in Viewing & Viewports

Colour Sampling

It is now possible to sample a pixel's colour inside a Viewport (Player or Tools) to get its RGB values. See: The Colour Picker (page 42).

Player

- A Fit button is now available next to the Home button in all Players and Viewers that support multi-views. Ctrl-Clicking on the Zoom % button no longer performs a Fit. It resets the zoom value to 100%.
- The Sequence Viewer is always displayed, even if the current sequence is located under the current Folder / Reel being shown in the Freeform view. The Sequence Viewer border is thicker so it can be differentiated from the other clips.
- Each Batch or Batch FX Group now has its own multi-view settings. The layout and its content are persistent between sessions and are saved with a setup. The layout is recalled when you load a setup.
- Modifying the Pan / Zoom of a Batch or Batch FX Schematic no longer persists when other Batch or Batch FX are loaded to the schematic. Improvements have been made to the Freeform view Sequence Viewer.

Action

- Some Action views now have their own Zoom / Pan settings that are not altered by the views that use the common Zoom / Pan settings. These views are:
  - Perspective view
  - Front view
  - Top view

- Side view
- Working view
- Source Working view
- Schematic view
- Group view
- Object view

What's New in Tangent Panel

- The Tangent Element panels are now supported on all Flame Family products (macOS and Linux).
- On Linux, the Tangent Hub (v1.3.1.2) is automatically installed with the application. The documentation and the Tangent mapper are located in the `/opt/Tangent/` folder.
- The Mac Tangent Hub (v1.3.1) must be downloaded from *here*.
- The update enables the following features:
  - Switching applications automatically updates the Hub. This is useful when running Flame and Lustre at the same time.
  - Support for Ripple panel (in Element TK emulation mode).
  - It is now possible to modify the user-based Element panels mapping for all products (macOS and Linux).
  - Support for Element-Vs, the iOS and Android application emulating the Element panels.

NOTE This supplied version of Tangent Hub is compatible with any third party application that supports the Tangent panel.

What's New in Wiretap SDK

You can now define or update a project's Colour Policy with the Wiretap API. If you have scripts that use Wiretap API to create projects, you must update your tools to create projects with a Colour Policy.

To define a Colour Policy, the new `syncolor` node is used. The `syncolor` node is found under the `root` node and under each project node.

`Project's syncolor node:`

- Only one child node: `policy`
- `getMetadata` can be called on a `syncolor` node. Displays the path to the shared transform directory.
- There is no `setMetadata` for `syncolor` or policy nodes under a project.
- A `syncolor` node can be used as source and destination by the `duplicateNode` to copy a policy from/to another project. For example, to copy the policy from one project to another using the command line tool:

```
wiretap_duplicate_node -s <src project node>/syncolor/policy
-n <dst project node>/syncolor
```

`Root's syncolor node:`

- Contains one child node: `policies` child node.
- The policies node has two children nodes, also known as policy category nodes:
  - Autodesk : Policies from the installed build.
  - Shared : Policies from the SynColor repository location.
- Each policy category node contains `policy` nodes where the node's name is the policy name.

- `getMetadata` can be called on `syncolor` node. It displays the path to the shared transform directory.
- `setMetadata` can be called on the `syncolor` root node to change the path of the repository
- A `syncolor` node can be the source node to set a policy on a project. Doing this using the the command line tool:

```
wiretap_duplicate_node -s "/syncolor/policies/Autodesk/ACES 1.0" -n
<dst project node>/syncolor
```

If you were to create a project without a Colour Policy, the Flame application will show an error message when starting the project. You will then need to do the following to assign a Colour Policy to the project:

1. Start the application with the project
2. When the error message about Colour Management Rules missing, press Cancel.
3. Enter the Project Management Edit menu.
4. Select a Colour Policy and save the project. Note: This is a known issue, the release will default to the Legacy policy.

Miscellaneous Features and Enhancements

InfiniBand

Remote connections and network transfers over InfiniBand have been revisited to take advantage of multi-threading, which should translate to faster data transfers. This should also improve playback over an InfiniBand network.

Backburner

On completion of a render job, the job can be left in the queue, it can be archived or it can be deleted a given number of days after job completion. Job handling can be set per application, per manager, or per job. See Backburner Monitor in a web browser.

Archiving

Located in the MediaHub's Archive Options, the new Silent Mode button allows the application to skip errors when archiving content. Error such as missing media when archiving with Cache On Archive option are skipped, allowing the archive to complete. Encountered errors are still printed to the application log files and to the shell, and a dialog box is displayed at the end of the archiving process if there were errors.

Path Translation

The Path Translation feature can now be used to translate not only media paths for conform sources, but also for clips in the Workspace, and for Batch setups' Clip and Write File nodes.

Media Panel

- Moving or deleting a shared library now locks the top-level Shared Libraries. This locks prevents other users on the network from editing the hierarchy of the Shared Libraries. Save the project to remove the lock. See Sharing Clips (page 238).
- Collapsed containers (libraries, reels, and folders) no longer expand automatically when you drop something on them.

Player

Use the new RAM Player option in the Player to allow you to cache a range of frames into memory in order to improve playback. This functionality is similar to the Play From Memory feature of pre-Anniversary Edition Flame. See Using the RAM Player to Cache Frames for Playback (page 400).

GMask Tracer

The GMask Tracer can now output colour values outside of the 0.0,1.0 range. There is a new Clamp Colours button in the GMask Tracer Setup menu's Rendering section.

What's New in Lustre

Installation

- When installing Lustre in a dual DVI monitor configuration, you no longer need to manually define the resolution of the Primary monitor in the init.conf file. The installation now handles that.
- Lustre Users can now create a Lustre-based Flame user profile and control Flame with similar shortcuts as Lustre for navigation, animation editor, etc.

Colour Management

- This release brings the new automated, rule-based, input colour conversion feature from Flame to Lustre.
- **CTF Plugin:** A new GPU plugin, enabling the application of a Colour Tranform or a 1D/3D LUT to a shot on output, is now available. Using this plugin, you can grade native media and view the effect through a CTT or a LUT. This plugin can be use to affect the entire image or parts of the image, using a secondary/shape or a key. You can see this plugin as a shot based output LUT. The location of the plugin in the Lustre grading pipeline is between the Input Primaries and the Output Primaries. This means that you can apply Output primaries color correction on top of the graded content onto which the CTF Plugin is applied. Then the Output LUT is applied, providing a lot of flexibility.

  NOTE Since the plugin references the CTF or LUT from its original location, make sure that location is available to remote render nodes or workstation that would use the plugin. If the CTF or LUT location is not available, rendering will still occur but without the plugin.

Multi-Channel Media

- It is now possible to update clips when external mattes are added from the file system. When you import a clip (locally or via Wiretap Gateway), and Matte media has been added in the Matte or Multi Matte structure, you can hold Shift when loading a Cut or a Grade and the clips and cuts are updated to reflect the newly added channels. This enables the following operations:
  - Converting regular clips to Multi-Channel clips.
  - Adding/Modifying Multi-Channel clips (addition, modification or removal of channels).

Flags System

- New CDL/SDL Import and Change flags.
- It is now possible to apply the current Flag ([ ]) to the event portion of an EDL. This functionality allows the use of events in an EDL to propagate a Flag to the matched event in the current cut. See: Flagging Shots.

Stereo Decision List (SDL)

- Aspect Ratio data is now correctly written in the DolbyVision XML file. The specification of Stereo Decision List (SDL) comments has been changed to reflect the StereoD SDL specification (V0.25, 2016). See: Flagging Shots.

ShotReactor

- It is possible to use multiple Shot Reactor servers. Shot Reactor servers can be local or remote. See: Using Multiple ShotReactor Servers.

What's New in 2017

2

Welcome to Autodesk Flame 2017. This release introduces many new creative features, performance and workflow enhancements, and other user-requested improvements. Make sure to check *http://www.autodesk.com/vxf* for the latest Flame documentation (including Installation Guides, Release Notes, and other documents). Also, many new feature videos (as well as other workflow videos) are available at *https://www.youtube.com/user/FlameHowTos*. This Learning Channel is updated frequently, so be sure to subscribe or bookmark the page. Links to specific 2017 videos are also provided in this What's New topic, denoted by this icon:

NOTE For 2017 Extension 1 features, see What's New in 2017 Extension 1 (page 1).

Follow these links to navigate to the sections below:

What's New in Project Management

New Compressed Intermediates

You can now use Avid DNxHR for compressed intermediates on Linux.

For more information, see Setting Cache and Renders Format (page 52).

Update to Project Compatibility

Flame 2017 (including Premium), Flare, and Flame Assist can now read contents created in Smoke Desktop Subscription (DTS), with the following limitations:

- Projects from Smoke DTS can be opened (and converted) by the Flame Family 2017 products.
- Projects can be seen and imported through MediaHub, using Browse for Projects.
- Archives created in Smoke DTS can be read in Flame Family 2017 products.

IMPORTANT Contents created in Flame Family 2017 products cannot be read or opened in Smoke DTS.

		Will import in...				
		Flame 2017	Flare 2017	Flame Assist 2017	Smoke DTS 2017	Training Edition 2017
Projects Created In...	Flame 2017	Yes	Yes	Yes	No	No
	Flare 2017	Yes	Yes	Yes	No	No
	Flame Assist 2017	Yes	Yes	Yes	No	No
	Smoke DTS 2017	Yes	Yes	Yes	Yes	No
	Training 2017	No	No	No	No	Yes
	Flame 2016	Yes	Yes	Yes	No	No
	Smoke DTS 2016	Yes	Yes	Yes	Yes	No
	Training 2016	No	No	No	No	Yes
	Smoke DTS 2015	Yes	Yes	Yes	No	No

Hardware Configuration

Flame can now use the full bit depth of a 10-bit precision graphics monitor.

On Linux You can toggle between 8- and 10-bit from the Flame Setup (and restarting the X server). The default it 8-bit. Note that other applications running on the same workstations will be impacted by this change (it edits the Xorg.conf),

and they will not run correctly if they do not also support 10-bit. There are no proscriptions against running 8-bit Flame on a 10-bit monitor.

On Mac (requires El Capitan 10.11.2 or later) The application should automatically configure itself to the correct bit depth. If the application still runs in 8-bit mode even when the attached monitor supports 10-bit, you can force the application to use the 10-bit mode.

To force the application to use the 10-bit mode:

- In a Terminal window, log in as root.
- Type: bash
- Type: export DL_XG_VISUAL_DEPTH=10
- cd to the application: cd /usr/discreet/<flame_version>/bin
- Start the application. Type: ./startApplication

Resource Management

A Resource Manager utility is now available from the Flame menu. The Resource manager indicates the amount of RAM and VRAM that are Available, Used by the system, Used by the software, and Free. It also lists the RAM and VRAM clients, and details their memory consumption.

For more information, see Resource Management (page 45).

Shotgun

Handles information is now available for export: `preExportAsset()` and `postExportAsset()` have now access to `handleIn` and `handleOut`. For more information, see exportHook.py.

Conform & Connected Conform

- The status of the "Hide Linked" button is now saved per user, and is now persistent between sessions.
- By enabling the new Keep Clip Name option, you can decide to derive the segment names from the clips used for relink, instead of using the names from the imported sequence (EDL, AAF, or XML). Consider using this options when conforming using content created in Batch.
- You can now filter out tracks from the Conform list by locking them. This means that sources matching segments on locked tracks are not display in the Conform list, nor in the Media Panel's Conform area.
- The settings found in the Conform Options in MediaHub are now saved per project.
- The support for imported AAF sequences has been improved:
  - The text and color of the original AAF Locators are now displayed in the imported sequence.
  - Time warps with negative keyframes are now correctly imported.
  - You can now keep RED source settings defined in Avid Media Composer when conforming an AAF in Flame: enable Keep Source Settings in MediaHub (XML & AAF Options tab).

    For more information, see Importing an AAF Sequence (page 137).

Connected Conform

The following are improvements to the Connected Conform workflow:

- *Shared Segment* has been renamed *Segment Connection* throughout the application.
- You can now create a Connected Segment: right-click a segment (including a Gap with BFX) and select Create Connected Segment. By copy-pasting this segment in other sequences within the same Reel Group, you now have access to the

same Sync Connected Segments operations you could previously only perform with a Shots Sequence, and propagate effects decision from one segment to every other related segments.

For more information, see Working with Connected Segments (page 448).

- You can now exclude one or more tracks from the Sources or Shots Sequence creation by locking it.
- Animated and reverse time warps are now properly created in the Shots Sequence.
- BFX applied to the segments of the source sequences are now carried over to the Shots Sequence when you create or update a Shots Sequence. Note that since gaps are not represented in the Shots Sequence, any BFX applied on gaps are not seen in the Shots Sequence. Also, BFX are now considered a criteria used to group segments from the different source sequences.
- You can now create a Shots Sequence from stereoscopic sequences (with left- and right-eye tracks), including any applicable Stereo BFX.
- You can now explode an existing BFX when creating Batch Groups from a sequence. The resulting segment from the Create Batch Group will be the rendered result of the BFX.

MediaHub

- Frame rate data is now displayed in the Preview window of the MediaHub when selecting an AAF.
- MediaHub options are now saved on a project-basis.
- When manually typing a path, if you make a mistake, the MediaHub will take you to the last valid location of your filepath.
- A confirm is now required when using the Revert All To Default button in the Format Specific Options tab.

For more details, see this video from the Flame Learning Channel: *MediaHub Updates*.

Media Import

The support for importing media in the following formats has been added, or improved:

- Canon XF-AVC:
  - Spanned clips are now seen as a single clip in MediaHub.
  - 444 RGB files from firmware-updated Canon cameras are now seen correctly as 10-bit instead of 12-bit.
- XDCAM-EX: The timecode of High Frame Rate (HFR - 50p, 59.94p, 60p) material is now accurately calculated at import.
- DNxHR:
  - 10- and 12-bit (QuickTime, MXF OP-1a and OP-Atom) generated by Avid applications are now supported.
  - DNxHR MXF (OP-1a only) media files from DaVinci Resolve are now supported.
- DNxHD: DNxHD 100 and 444 MXF (OP-1a) media files generated by Blackmagic Design Resolve are now supported
- ARRI: ARRIRAW SDK 5.3 is now supported.
- REDCODE: RED SDK 6.1 is now supported. It adds the following R3D Gamma Curves:
  - HDR2048
  - BT1886
  - LOG3G12
- SonyRAW: Support for ISO / Exposure Index MXF metadata.
- Audio files: The Frame Rate option is back in the Format Specific Options for Audio files.

NOTE Non-atomic Movie files (files where audio and video do not have the same durations) are now seen as valid sources. To ensure that audio and video tracks are of the same duration, silence or No Media is added at the end of the clip to make up for the difference.

For more information, see Supported File Formats (page 114).

Media Export

The following are new features or improvements to the Media Export:

Improvements to Sequence Publish

- When publishing a sequence, video content with Timeline FX can now have handles. In the Export's Sequence Options, enable both Use Media with FX and Include Handles.
- Publishing a single sub-clip from a sequence (such as when the Source and Record timecode are not identical) now results in Record Timecode being used to generate the media files.
- When you publish a sequence that contains multi-versions Open Clips, you can use the Source Version Name and Source Version ID tokens to access the version name and ID of the individual segments within the sequence.
- When publishing an EDL, Use Tape Name Extension and Use Long Tape Name are now mutually exclusive.
- The export process is now more efficient: media is no longer rendered when exporting without media an EDL, AAF, or FCP XML.

Improvements to available export codecs:

- You can now export content as Avid DNxHR compressions in QuickTime (.mov). The following compressions are available:
  - Avid DNxHR Low Bandwidth 8 bits (LB)
  - Avid DNxHR Standard Quality 8 bits (SQ)
  - Avid DNxHR High Quality 8 bits (HQ)
  - Avid DNxHR High Quality 12 bits (HQX)
  - Avid DNxHR 4:4:4 (10 or 12 bits)

- AAC audio encoding is now available to QuickTime containers (.mov). The following profiles are now available:
  - 56 kbps (mono only)
  - 96 kbps
  - 160 kbps
  - 224 kbps
  - 288 kbps
  - 320 kbps (stereo only)
  - 448 kbps (stereo only)
  - 576 kbps (stereo only)

  NOTE Every QuickTime movie preset that used IMA 4.1 now uses AAC.

Miscellaneous improvements

- Improved Avid Pro Tools compatibility. By improving the accuracy of the timecode information contained within Broadcast Wave files, an AAF generated by Flame for Pro Tools now automatically relinks to its media from within Pro Tools.

- RGBA Export. Exporting RGBA media files from a Matte Container that contains sections with No Media now generates Matte media as White if the Action on No Media found in the Export preferences is set to Use Black. Access the Export preferences tab by enabling the Show Advanced Options option in the Export window. This preference (like all export settings) is saved per project.
- There is no longer any consolidation of audio media when exporting.
- New tokens for Export are now available:
  - Source Name. Note that Source Name does not include the file extension or file index (as found in image sequences).
  - Background Name
  - Background Shot Name
  - Background Index

  NOTE Availability of the tokens depend on the context. For example, Source Name only makes sense in a Publish context, where each segment is exported individually and a source name is available.

Media Panel

- The following improvements have been made to the Media Panel sorting workflow:
  - You can now sort the Media panel hierarchy using the Tree: button at the top of the Media panel.
  - You can now sort the Media panel hierarchy by Creation Date or Name. Right-click the Tree: button to select either option.
  - The sorting options are now consistent between the Tiles and List view mode.
  - You can apply a localised sort to any object of the Media panel using the Localised Sort option in the right-click contextual menu.
- You can now search clips within your Workspace using more than just a name. You can still enter text in the field at the bottom of the Media panel to search using the clip name, but clicking the Magnifying Glass now opens the Advanced Search window, where you can search using many different criteria.
- When dual Media panels are displayed, you can now collapse/expand any container in one panel without collapsing/expanding it in the other panel.
- A Date Created column is now available in List View.
- After deleting an element from the Media Panel, the current Source clip is no longer selected. Instead, a sibling of the deleted element is selected. In the case where there is no sibling left, the parent container is selected.
- The Information Box shortcut (`Alt+click` in the Flame and Smoke Classic profiles or `/+click` in the Smoke FCP profile) can now be used on non-clip entries of the Media Panel, displaying its Name and Creation Date.
- If you add an item to a location where another identically named item already exists, you are now presented with options to either cancel, add, rename, or replace. You can also choose to apply your selection to all the selected items at once.

For more information, see Using the Media Panel (page 59).

For more details, see these videos from the Flame Learning Channel:

- *Media Panel Enhancements - Part 1*
- *Media Panel Enhancements - Part 2*
- *Media Panel Enhancements - Part 3*

MediaReactor

Drastic Technology MediaReactor build 169 provides the following enhancements:

- New Color Menu with an option to remove the DCI X'Y'Z' to REC709 conversion for JPEG2000 Digital Cinema Package (MXF). By default, imported DCP MXF is now provided in its native DCI X'Y'Z' color space. Disabling the option provides the REC709 conversion from previous releases.
- The DCI X'Y'Z' option has been renamed to 'Convert to Rec. 709'
- The “Enable Matrix” setting (in the Debayering menu) for RAW formats can now be disabled. When this option is disabled, along with "From File Header", the conversion to REC709 is disabled, providing the actual RAW color from the media files.
- Debayering Algorithm: New options to define the decoding method:
  - **Bilinear (CPU only):** Basic debayering.
  - **Adaptive (CPU only):** Looks for errors and alters the debayering based on those errors.
  - **Patterned (CPU only):** Detects patterns in each colour and selects the correct debayering algorithm based on the closest pattern.

    NOTE This version of MediaReactor is also compatible with previous releases. However the new features are only available in Flame Family 2017 applications.

Multi-Version Enhancements

Support for multi-version has been improved, making it easier to integrate multi-version clips into your workflow. Multi-version workflow requires the use of Open Clip files, and is an extension of the publish workflow previously available. The following enhancements are now available when you use multi-version clips.

For more information, see Working With Open Clip Files (page 207).

Some of the improvements:

- With multi-version clip in a Container or a Matte Container, you can switch between versions from the Timeline using the contextual menu.
- If you unlink a multi-version clip and relink it at a later time, it relinks to the original version.
- When an imported Open Clip used in a sequence is updated by an external application, its affected segments are highlighted in white.
- A new slate, No Media, is displayed whenever a selected version is shorter than the originally used clip. This prevents the structure of a timeline from changing when switching versions.
- Hard committing multi-version clips now preserves all available versions, but keep the link of handles that were located outside the range of the timeline segment at the moment on the Hard Commit operation.
- Proxies and Media Cache management:
  - Proxies and media cache are no longer flushed when you switch between versions of a clip.
  - A new option for proxies and media cache management is now available: Flush All But Current. This flushes all the media caches or the proxies attached to the versions of the clip, except for the currently displayed version. These two options are available from the contextual menu.
- When archiving multi-version clips, a new option allows you to cache every versions or only the ones currently used.

Changes to the structure of an Open Clip:

- Versions of a clip are now timecode-aligned.
- Versions of a clip can have different start time, end time, and duration. The actual duration of the clip is defined by the earliest start time of all versions, and the latest start time of all versions.

- An Open Clip now supports different media formats for different versions, but not different timings.
- The startOffset element is now correctly supported and computed when used in Open Clip.

Colour Management

- A new Auto mode in the MediaHub uses rules and format-specific colour space preferences to convert various types of media to a common scene-linear Working space.
- In Auto mode, the rules are resolved at import to assign a colour space to the clip. If you update your rules, you can go into Pre-processing options for already imported clips and reselect "From File or Rules" from the Input Colour Space menu in order to get the latest colour space from the updated rules.
- For ARRI, RED, and Sony camera native formats, Auto mode converts directly to the chosen Working colour space using the highest quality path (i.e. you do not have to worry about which camera-specific Gamma and Colour primaries to specify in each format's SDK options).
- In the Input Colour Space menu there is a "+ Add New" option to allow users to add their own colour spaces from LUTs or colour transforms. Doing this creates a ctf in the Shared directory specified in the Preferences.

  NOTE In a workgroup environment, these ctf files must be duplicated or the Shared Path must be configured in Colour Management Preferences to point to a common directory to make the colour space available to the other workstations. We hope to remove this limitation in a future release.

- A new table in Colour Management Preferences allows the rules to be defined which assign a default Input colour space based on filename.
- The MediaHub "Format Specific Options" tab provides a Colour Management setting for temporary adjustment of the Input colour space to assign when importing a specific format.
- A selection of Input and scene-linear Working colour spaces are offered as menu options. These options are currently the ones used in Maya and we are soliciting your input on the list you would like to have for the release. The ability to add user LUTs is being worked on for an upcoming beta.
- A new "Input Transform" tab in the clip "Pre-Processing" options allows adjusting the Input-to-Working space conversion for specific clips.
- A new "Input Transform" tab in the Clip, ReadFile, Render, Export and LUT Editor nodes also allows the Input-to-Working space conversion (and its inverse) to be applied in Batch.
- A source's assigned colour space is now displayed in the MediaHub clip info.
- New Input and Working colour space options for Media Hub.
- New colour transforms in the Autodesk collection for transcoded RED media.
- New default import rules for OpenEXR and DPX.
- The "LUT" preferences page has been renamed "Colour Management".

Tangent Panels

- The Tangent panels can now be used with every Linux product, without the need for a Flame Premium licence.
- When running Lustre and Flame at the same time, Lustre takes control of the panels. There is no way to switch the panels to Flame when Lustre controls the panels.

Player

- It is now possible to display and manipulate the different Timeline FX icons directly in the Player, enabling modifications at the Timeline level.
  - Edit Modes are available for GMask, GMask Tracer and Action, but only from the interface. Shortcuts cannot be used at the Timeline level.

- The Icons visibility can be enabled / disabled from the Player options menu, using the Show Icons / Hide Icons option.

Audio

The following are improvements to the audio capabilities of the application:

- A de-esser effect is now available from the Timeline. A de-esser allows you to remove the excessive prominence of the sibilant consonants ("s", "z", "sh"...) in human speech, often time experienced in ads with a heavily compressed audio dynamic range.
- You can now refresh a clip's audio waveforms. Right-click a clip, and select "Refresh Audio". This option deletes existing and pending audio waveforms from the selected clips, and then regenerates them. Note: The audio waveforms are also refreshed when you flush the Source Media cache from a timeline segment.

Desktop Reels

- A few enhancements have been to the display of clips, when scrolling through them, on the Desktop Reels:
  - While scrubbing a clip, the selected clip outline disappears, giving you a cleaner interface.
  - All clip information is hidden, except for the Current Frame / Timecode / Keycode and the File Format icon. This also applies to clips in the Media Panel (Tiles), the Freeform view and the MediaHub (Tiles and Preview Panel).
  - The scrollbar is now visible in all display modes (Collapsed, Frames and Storyboard).
- The Desktop Reels Previous and Next Transition buttons no longer place the Positioner on the very last cut of a clip. It now places the positioner on the last frame, eliminating the display of a black image in the clip when in Collapsed mode.
- Clips are now added to a reel at the location selected by the user. The clips are also always framed over the Reel controls after being added.
- You can load clips to a Reel using the Load button, available on each Reel. This button offers a workflow similar to the legacy "Load" button which allowed you to load clips directly from the Library to a Reel. The Load button opens a modal window displaying the Media Panel, from which you can select clips to be loaded to a Reel. It is not possible to do anything other than selecting a clip in this modal window. The advantages of this workflow are:
  - You can still load clips when the Media Panel is hidden.
  - The new Modal window has its own set of display settings (Tiles / List, Tiles Size, Expand / Collapse, Scrollbar, Filtering tab).

For more details, see this video from the Flame Learning Channel: *Loading into Reels*.

Batch / Batch FX

- The "Include History" and "Foreground/Background" buttons have been reintegrated into the Render node's user interface under the "Global Settings" section. In addition, "+H" is appended to the "Render / Burn / Reactor" button when "Include History" is enabled.
- It is now possible to easily deselect selected nodes by clicking in the Batch, Batch FX and Modular Keyer Schematic background using the Wacom tablet as well as the mouse.
- The Write File and Render nodes have been modified so the Render Range In and Out fields are in the same place, and display more digits.
- Hide the bottom Batch panel with a Ctrl-Swipe the bottom of the screen. The Viewport and navigation controls remain visible.

- The new Media Panel node allows you to load clips in Batch, Batch FX and the Modular Keyer. This node replicates the old Library node that allowed you to load clips from the Library directly to Batch. The Media Panel node opens a windowed Media Panel where you can select the clips to load. Especially useful when the Media Panel is hidden. Note that the windowed Media Panel display settings are independent from those of the regular Media Panel.
- The new Desk node allows you to load clips in Batch, Batch FX and the Modular Keyer. This node replicates the old Desk node that allowed you to load clips from the Desktop Library directly to Batch. But now you can toggle the visibility of the Desktop Reels view to display another Reel Group or Batch Group. This selection is not persistent and will not affect the visibility that was previously set.

It is now possible to copy a Batch FX setup on exit, enabling it to serve as a backup in the event the BFX setup is lost.

- This is done when the "Save Backup on Exit" setting is enabled in **Preferences ➤ TL FX / Batch / BFX ➤ BFX Setup**. It is enabled by default.
- The backup setup is saved in `/usr/discreet/project/<project name>/batch/bfx_backup`
- The backup setup is named after the BFX name.

NOTE Copying a BFX of the same on exit overwrites the previous backup.

For more details, see this video from the Flame Learning Channel: *Batch Workflow Updates*.

Batch Groups

- The Rename window now pops by default, when you click the Batch Iteration, Render node or Write File name field. This behaviour can be disabled using the Open Naming Window button in **Preferences ➤ TL FX / Batch / BFX ➤ Batch/BFX Iteration**.
- Shift+clicking the field inverts the status of the preference on-the-fly. The bottom Pen Button can also be used to invert the preference setting on-the-fly.
- It is now possible to clear the current Batch Setup using the "Clear Batch Setup" option, located in the "New" button's dropdown menu.
- You can choose whether a clip is copied or moved from / to a Batch Group using "Move From/To Batch" / "Copy From/To Batch" setting, located in Preferences / User Interface / Desktop.

  NOTE
  - "Move From/To Batch" is the default value.
  - The setting is saved within the User and is persistent.

- You can now set a custom value for an Iteration using the "Iterate As..." option located in the "Iterate" button's drop down list. This option enables you to:
  - Overwrite a previous Iteration.
  - Set a custom value so that the first iteration in your group can have a value other than 1.

- The file path to which a Batch Group was saved / loaded now persists in the Batch Group. This has been implemented to ensure that using the Save button will indeed save the updated setup in the right place, rather than the last visited directory in the File Browser.
- Using the "Save As" option updates the persisting file path.
- Using the "Load and Create" option updates the persisting file path.
- The "Load and Append" and "Load and Replace" options **do not** update the persisting file path.
- The <Iteration> token is no longer mandatory for Batch Iterations located outside of an "Iterations" folder.A new preference named "Append Original Name" enables you to decide whether you want to append the name of a Batch Iteration to another Batch Group Iteration name when you copy or move the first one into the second. This preference is disabled by default.

- The "Default Batch Iteration Name" value remains the same. The <Iteration> token is included, but can be removed.
- The <Iteration> token is automatically appended when you press the "Iterate" button.
- It is not possible to remove the <Iteration> token from an iteration located inside a Batch Group "Iterations" folder.
- The new "Default Batch Group Path" and "Default Batch Iteration Path" preferences, located in Preferences / TL FX / Batch / BFX enables you to choose where the setups should be saved or define a directory structure to be created when a Batch Setup is manually saved or when a setup is automatically saved when iterating. By default, the paths point to the same as directory as before. Python Hooks exist for both preference. They are located in the batchHook.py file.

  NOTE Removing the "~" from the path results in the setups located in the defined directory not being archived by Archive Setup or Archive Project.

- Batch Group creation order is now based on the order of the shots contained in the sequence used for the creation of the Batch Groups, rather than alphabetical order.
- Batch Groups are now collapsed by default in some situation:
  - When they are saved to Libraries or Shared Libraries.
  - When they are copied from one Library to another.
  - When a new Batch Group is created.
  - When a Batch Group is duplicated.
  - When a Batch Group Setup is loaded.

For more details, see this video from the Flame Learning Channel: *Batch Group Improvements*.

Burn Metadata

Burn Metadata is a new flexible tool created to replace and expand on the Burn-in Timecode tool. Available as a node in Batch and Batch FX, in the Tools tab, and as a Timeline FX.

NOTE The Burn In Timecode tool is no longer available in Batch, Batch FX, and the Tools tab. It can however be restored from an Archive or from a Wired project; Burn does not translate in a Burn Metadata. This is similar to what happens with the deprecated LogicOp node.

Burn Metadata includes the following features:

- Each metadata element is contained in its own layer. See Burn-in Metadata Menu Settings (page 1470) for the available metadata layers.
- Layers are managed inside a table and can be added, selected, muted, duplicated, deleted or simply converted to another type.
- A layer can be aligned to the Safe Action or Safe Title, or positioned using onscreen icons in the Viewport.
- A Layer can now be rotated in Z using the Rotation field located under the Alignment section.
- You can propagate some clip information to the Burn Metadata node in Batch or Batch FX using the T-Click keyboard shortcut.
- You can select to display the Frame information based on the Segment, Clip / Timeline, or Source Media from the Tools tab or the Timeline FX (this option is not available in Batch).
- Unlinked media and "No Media" frames are replaced by a black frame when a Burn Metadata is present.

For more information, see Burn-in Metadata (page 1469).

For more details, see this video from the Flame Learning Channel: *Burn-In Metadata*.

Action

Matchbox in Action (Action post-processing pipeline)

- You can now connect Matchbox shaders to the Action camera (Camera FX) and have the result of the shader applied to the result of the Action scene. Similar to using Texture Matchbox shaders, the priority editor is used to handle the pipeline order.
- The Action Post-Processing pipeline can connect the Matchbox shader automatically to the appropriate Action output. To support this, shader developers need to set the InputType tag properly in the shader XML (each Action output has its equivalent InputType tag). This includes RGBA shader rendering.
- Matchbox Camera FX also have access to the Acton output list in their patch panel, so you can manually set the proper connection if the Matchbox shader hasn't been tagged properly in the XML.
- Matchbox Camera FX can have 2D and 3D widgets visible in the Live Preview F4 view of Action. Only the widget(s) of the currently selected Matchbox are shown.

NOTE See the Matchbox / Lightbox (page 28) section below for other general Matchbox enhancements.

See Matchbox Shaders Connected to Cameras (Camera FX) (page 823).

For more details, see these videos from the Flame Learning Channel:

- *Action - CameraFX - Part 1*
- *Action - CameraFX - Part 2*

Action Live Preview Mode

Introducing Live Preview mode. You can now toggle between 3D and Live Preview in the F4 viewport of Action. This allows you to toggle between a 3D viewport, and a pixel-based viewport of the same scene. This solves a large number of issues, including allowing Action and GMask Tracer widget support in the different Context views for the selected output; including Timeline Primary/Secondary context, Batch context and MK Context. Live Preview mode also improves the interactive performance, since it is reducing the shading coverage of the scene, and makes better use of HWAA and properly supports proxy and resolution options. Hardware anti-aliasing or any post-processing effects (such as Camera FX, Lens Flares and Rays) are not available in the 3D view, and can only viewed in the Live Preview view mode. Adaptive Degradation has also been refactored, allowing for Live Preview specific options to create feature parity with the 3D view.

TIP Use the 3 keyboard shortcut to cycle between the Live Preview and 3D View modes.

See Live Preview and 3D View (page 561).

For more details, see this video from the Flame Learning Channel: *Action - The Live Preview Viewer*.

Action Shadow Cast Improvements

- The Shadow Cast node now supports up to 8K of resolution. Performance and quality have been improved for 3D Soft and 3D Hard Shadows.
- Shadow Cast now supports 32-bit shadow maps, fixing the shadow precision problem with large shadow maps.
- A new Shadow Casting option called Invisible Receiver allows you to select an object to be invisible in the scene while still receiving shadows. This option is available for both Surfaces and Geometries. All object properties are kept, so you can still use blend modes, transparency, or even texture the surface or geometry for more flexibility. The Shadow Only option has been renamed Invisible Caster for more coherence.

See Surface and Geometry Shadow Casters (page 672).

For more details, see this video from the Flame Learning Channel: *Action - Shadow Casting Updates*.

Action Lens Flare Improvements

- You can now quickly create and load Lens Flare presets. These presets are a different format than the ones found in the Presets node, and are easier and faster to use, as they do not load new nodes, but replace the Lens Flare node values, as well as all the nodes connected underneath it. This allows you to quickly replace Lens Flare presets, without having to perform schematic management (delete node, switch Light connection, etc.) The new Save and Load mechanism supports every node connected underneath a Lens Flare node, including Matchbox.

  NOTE All old Action Lens Flare presets have been converted to the new Lens Flare preset browser, so you do not have to use the Presets node for Lens Flare presets.

  See Using Lens Flare Presets (page 677)
- Action Lens Flare shader implementation allows you to re-order Action Lens Flare results anywhere in the post-processing pipeline. You need to deactivate the Lens Flare render in the Comp output and add the shader to the camera.
- Five new Matchbox shaders (LensBokeh, LensDirt, LensOptic, LensRefraction and LensSpatial) provide real lens graphical elements to be used with Action Lens Flares.

For more details, see these videos from the Flame Learning Channel:

- *Action - Working with Lens Flare Presets*
- *Action - Customising Lens Flares*

Action Outputs

- Introducing the concept of render layers in Action output, allowing you to set multiple render passes, based on a single camera and object selection. The Primary output selection allows you to quickly define a single render pass render layer of any output type, as well as the Primary output feeding the post-processing pipeline.
- More viewing flexibility in the new output system: F4 always shows the current output selection, even when the render pass is not set to be an active output socket of Action. When in an Output view, you can cycle through selected Render Passes associated with a Render Layer (press 1 to navigate down the list, or press 2 to navigate up the list).
- More output data passes are available from Action: Lens Flare/Rays, Albedo, GMask, Occluder, Background and Roughness. The Lens Flare/Rays result is still applied by default in the Comp output, but it can now be deactivated and put anywhere in the post-processing pipeline, using its own output pass via a Matchbox shader (by default Lens Flares are at the very end of the pipeline). The GMask and Occluder passes offer more flexibility in the post-processing pipeline, allowing them to be used as additional data, driven by GMask rendering. GMasks can be connected to a Camera to limit their effect to the post-processing pipeline exclusively. GMask output passes have an option to only show GMasks connected to the current camera, so you can set GMask to not interact with the scene, and only be created to be used with the post-processing pipeline.
- You can now create Motion Vector data passes out of Action (supports Axis, Camera, Surface vertices, Geometry animation, and limited support for 3D Shape and 3D Text). Motion Vector optimization can lead to better performance in generating motion data, but also provides better tracking performance, as this allows you to get access to transformation data without having to load the whole Action database. You can create 2D or 3D Motion vector render passes to represent vertex position changes in the scene.
- Colour coding for Action output render passes, Action output sockets, and Matchbox inputs has been implemented. New data types have colours associated with them (Normals, Position, ZDepth, Motion Vectors, and 3D Motion).

See About Rendering Outputs from Action (page 598).

For more details, see this video from the Flame Learning Channel: *Action - Updated Outputs*.

Action Miscellaneous

- The Toggle Icons Draw Mode shortcut has been renamed to Cycle Icons Draw Modes and it now cycles between the On, Off, and Selected Icons display modes.

  For more details, see this video from the Flame Learning Channel: *Action - Cycle Wireframe Icons*.
- The Action shader manager has been reworked to offer better stability and performance.
- The FBX SDK has been updated to version 2017.0.1.
- It is now possible to select all vertices of a GMask (in the GMask Tracer or Action) using the GMask: Select All Vertices shortcut:
  - **Ctrl+A** in the Flame profile.
  - **Keypad+/** in the Smoke Classic profile.
  - **Meta+A** (Linux) or **Command+A** (Mac) in the Smoke FCP profile.

Matchbox / Lightbox

- Matchbox now supports the usage of its previous result buffer with the uniform `adsk_accum_texture`. This is a regular sampler2D that automatically provides the previous frame result, and needs to be defined in both the XML and glsl file. You can use the uniform `adsk_accum_no_prev_frame` (which is a bool) to determine what happens if there are no previous results. There is an Accumulate example provided in the EXAMPLES folder. A non-recursive accumulation mode allows Matchbox developers to use this feature without the performance impact of the absolute recursive mode.
- You can now host a texture in a Matchbox shader, by bundling an image file as a texture grid, in the following formats: OpenEXR, DPX, JPEG, SGI, Targa, and TIFF. In the shader, you can define the content of the image file using the uniform sampler2D `adsk_texture_grid` (needs to be defined in both the glsl and xml files). There are three GridFetching examples in the EXAMPLES folder.

  For more details, see these videos from the Flame Learning Channel:

  - *Matchbox Texture Grids - Customizing Basics*
  - *Matchbox Texture Grids - HUD Graphics with a Custom Grid*

- A perspective camera is available inside Matchbox in relation to providing the Action camera API. This provides information about the current output camera, when the shader is used as a post-processing (Camera FX) effect in Action. GMask input has been implemented in most Matchbox shaders, so they work properly with Camera FX.
- Some existing Matchbox shaders have been updated to support the 3DSelective capabilities exclusive to Action Camera FX (showcased in the Action3DSelective Matchbox provided in the EXAMPLES folder). This functionality allows you to segment the render of Action using different 3D techniques, applying the given effects to a specific portion of the image. Effects that have been updated include: Blur, Glow, Exposure, Lightwrap, Colour Correct, Dots, Fog, Rays, Ripples, Twirl, and Chromatic Aberration.
- It is now possible to create presets in Matchbox and Lightbox, allowing shader developers to offer different sets of starting values for a given shader, without having to modify the glsl code. The preset file is a sidecar xml file in which you can declare as many sets of starting values as wanted, with a name to identify them. In the UI, a popup button appears underneath the Change Shader button, listing all the different declared presets. A button in the Node Prefs menu allows you to dump the current UI configuration to the shell, allowing for easy copy/pasting to create presets.
- In Action, you can now enable or disable Adaptive Degradation in shaders from the Shader tab. This indicates if you use the degraded result in the Interactive viewport for performance improvements.
- It is now possible to define a custom display name for a channel inside the Animation window, using the `ChannelName` token in the Matchbox XML.

- It is now possible set a fixed resolution for Matchbox intermediate passes that is different than the main output resolution.
- In Batch, Batch FX and the Modular Keyer, the Matchbox node's input tab colours are now based on the type of input (for example, yellow for composite, blue for matte).
- Matchbox packaging has been upgraded to offer better browsing performance.
- Matchbox menus can now use an unlimited amount of pages in either Batch or Action, by changing the tabs with a popup button when reaching more than five pages.
- Apart from the EXAMPLES shaders mentioned above, the following new Matchbox shaders have been added:
  - Stingray Motion Blur
  - Stingray Ambient Occlusion
  - Stingray Bloom
  - Stingray Reflection
  - Stingray Depth of Field
  - Stingray Lens Effects
  - Lightwrap
  - Five new Matchbox shaders (LensBokeh, LensDirt, LensOptic, LensRefraction and LensSpatial) provide real lens graphical elements to be used with Action Lens Flares.
  - Many existing Matchbox effects now have Action Camera FX 3D Selective functionality (these are new Matchbox shaders denoted with 3DSelective in their name, such as *Dots3DSelective*)
- The Lightbox API has been updated, allowing you to use the Physically Based Shader attributes to perform IBL rendering, for example (See PhysicallyBasedIBL in the EXAMPLES folder).

For more information on these new Matchbox and Lightbox features, as well as complete information of creating your own shaders, see the Shader Builder API Guide: *http://www.autodesk.com/shader_builder_api_guide*.

GMask Tracer

- Since the GMask Tracer is based on the Action ecosystem, some of the new Action features in this release are also present in the GMask Tracer interface. These include:
  - Live Preview mode
  - Render layer outputs
  - Matchbox Camera FX
  - Widgets in context view
- It is now possible to select all vertices of a GMask (in the GMask Tracer or Action) using the GMask: Select All Vertices shortcut:
  - **Ctrl+A** in the Flame profile.
  - **Keypad+/** in the Smoke Classic profile.
  - **Meta+A** (Linux) or **Command+A** (Mac) in the Smoke FCP profile.

General Workflow & User Interface

- The tools that include a Schematic view now automatically reveal the bottom portion of their user interface when you double-click on a node or when you use a shortcut to display the Node Bin while in Full screen mode.
- An enhanced and redesigned colour picker is now available. See The Colour Picker (page 42) for details.

For more details, see this video from the Flame Learning Channel: *The New Colour Picker*.

- The display of media's aspect ratio now includes 3 digits after the decimal, throughout the application.
- The "Name" field is now activated by default in all the Naming windows (Rename, Rename shot, Batch Name).

Shortcuts

- You can now use the "Enter" and "ESC" keys to confirm or cancel operations, respectively, in all Naming windows (Rename, Rename shot, Batch Name).
- It is now possible to use the "Rename Under Cursor" shortcut in the Timeline. The shortcut is assigned to "N" by default in the Flame Profile. It is not assigned to a keystroke in the Smoke Classic and Smoke FCP profiles, since these profiles use the Rename (selected) shortcut by default.
- A new "Rename Shot Under Cursor" shortcut is now available.
- A dialog window now appears on screen to indicate that a key is stuck on your keyboard, preventing you from using keyboard shortcuts. The message indicates which key must be released to fix the problem.
- The "Display Node Information" shortcut has been modified from Ctrl + I to Ctrl + Shift + I to avoid conflicts.
- The "Toggle Draw Icons" shortcut has been modified from I to Ctrl + I in the Smoke Classic and Smoke FCP profiles, to avoid conflicts and to make it consistent with other such shortcuts.

Preferences

- There is a new Preferences setting to define whether rendering takes into account the heads & tails for Timeline FX, BFX, Containers and Matte Containers. It is located in Preferences / Timeline / Rendering. Options are:
  - **Include Handles:** When enabled, rendering of Timeline FX takes the handles into account.
  - **Infinity Handles:** This value (expressed in frames) is used to limit the handles of virtual segments (handles are seen as * for freeze frames, etc.).

  NOTE
  - A new "Include Handles" option is now available in the Timeline's "Rendering Mode" dropdown menu. By default, it shows the same value as the Preference but can be changed at any moment. To limit the amount of handles to be generated, you should use the Consolidate functionality to the selected amount of handles. The "Commit FX" option also generates handles, based on the status of this option.
  - This option can also be used with Autodesk Burn and Background Reactor.
  - The "Cache On Playback" setting does not render handles, since the caching is performed on content displayed in the Player.
  - When performing interoperability with Lustre, use this option to generate more media for grading.

- A "Swipe Sensitivity" setting has been added to the Preferences. The setting is located in Preferences / User Interface / Gestural Workflow. The default value is 50% on Linux and 90% on Mac. This was added to solve the swiping issues on the Mac platform.
- A new "Duplicate Name Check" setting enables you to disable some or all of the messages related to media of the same name already existing in the Media Panel. The setting is located in Preferences / General / Media Panel. You can enable/disable the check for clips and containers. By default both clips and containers are checked.

Miscellaneous Features

Dual Monitor

- It is now possible to display an additional viewport on the secondary screen, while in Batch or Batch FX.
- You can set the Media Panel to Full Width, in order to have only the Media Panel displayed on the secondary screen.

- When the Media Panel is not set to Full Width, the secondary screen defaults to displaying an extra viewport. You can set any Batch view to that viewport.

Timeline

- The Select Current Segment shortcut is once again available. This shortcut can be used to explicitly select the segment under the positioner. The shortcut is assigned to the “Enter” key in the Smoke Classic and Smoke FCP profiles.

  NOTE The shortcut is not assigned by default in the Flame profile.

Editing

Smart Replace

- Replacing content is now easier using Smart Replace, a new replace option (enabled by default) that uses the Source clip attributes. Content duration is compared first, then Source Timecode. If these do not match, you are asked if you want to perform a positioner to positioner Replace operation. This option can be disabled from the Secondary Edit box pulldown menu. This option can be especially useful in the Connected Conform workflow where you need to re-insert a rendered clip in the Shots Sequence.

Timewarp

- It is now possible to slip the source Media underneath a Timeline FX Timewarp, without affecting your keyframes, using the editable Timing field.

Tokens

- The Record Frame token can be used to propagate the first record frame of a segment to its Name, Shot Name or Comment. It is also available in Sequence Publish.
- The Background Index token is now available in Sequence Publish.
- The Background Name and Background Shot Name tokens can now be used to propagate the name or shot name of the bottom-most layer to all the segments above. The token is available in Rename, Rename Shot, Comment and Sequence Publish.
- The Source Name token can be used to propagate the clip source name to its Name, Shot Name or Comment. It is also available in Sequence Publish.
- A <Depth> token is now available in Rename, Rename Shot and Export. Using this token adds the clip bit-depth to the segment / clip name.

What's New in Lustre

Connected Colour Workflow

Lustre 2017 introduces the Connected Colour Workflow. This collaborative workflow enables basic effects work during client-attended grading sessions, by harnessing the power of the Flame Family’s Batch procedural compositing environment (Flame or Flare, running concurrently on the same workstation).

For more details, see these videos from the Flame Learning Channel:

- *Connected Colour Workflow - Part 1*
- *Connected Colour Workflow - Part 2*
- *Connected Colour Workflow - Part 3*
- *Connected Colour Workflow - Part 4*

Dedicated What's New and User Guide

From the 2017 release onwards, Lustre will have a dedicated What's New and User Guide.

The Lustre 2017 What's New can be found here: *Lustre 2017 What's New*.

The Lustre 2017 User Guide can be found here: *Lustre 2017 User Guide*.

Flame Feature Differences by OS

3

Due to software and hardware differences, the following Flame features are not available on Mac systems:

- GPU debayering
- Background Reactor
- Real-time deliverables
- VTR archiving
- DNxHD Compressed Intermediates
- Floating license server

There are also some differences to note when working with certain effects on a Mac system:

- Hardware anti-aliasing (HWAA) is limited to 8x.
- You can add one Image-based lighting (IBL) map per Action object.
- When using the PBS Shader in Action on a Mac system, you are limited to Base Color, Roughness, Metallic, AO, and Opacity shader types.
- You are limited to 16 textures per Action object; therefore in large Action scenes, you may notice that some features are not available (shadow casting, for example).

Some preferences or settings are available only on Linux or Mac systems. These are noted where they occur within this User Guide.

Consult the online *System Requirements* for up-to-date information on supported configurations and components.

Flame Interface Overview

4

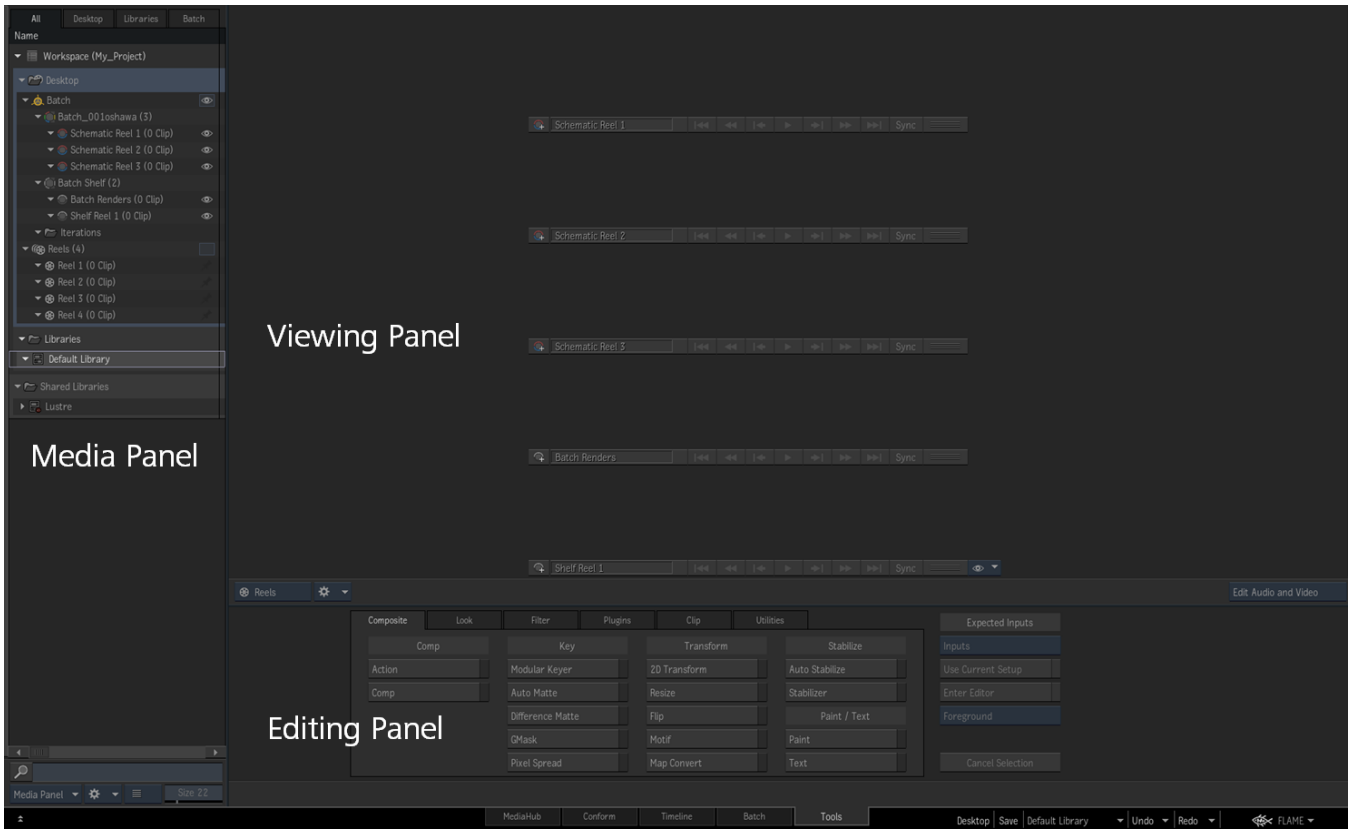
The Workspace is where you accomplish all of your project tasks. The Workspace consists of three panels:

- The Media panel
- The Viewing panel
- The Editing panel

Each panel has its own menu bar, where commands related to the panel can be performed.

At the bottom of the user interface is a series of tabs, which each give quick access to different views. Regardless of the current view, the working environment remains the same.

It is possible to hide the Editing panel area in the Conform, Timeline, and Tools tabs by performing a `Ctrl+Swipe` at the bottom of the screen. By doing so, you end up with a full height version of the Viewing panel (Player, Source-Sequence, Triptych, Freeform, Desktop Reels, or Conform list). Viewing panel areas other than the Conform list can also be displayed in a full screen manner if the Media panel is hidden.

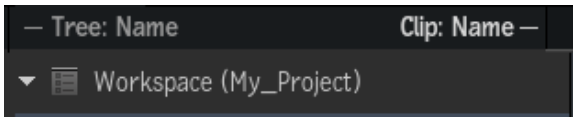
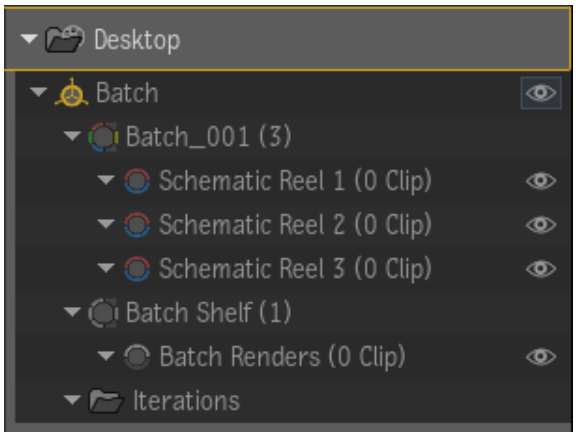
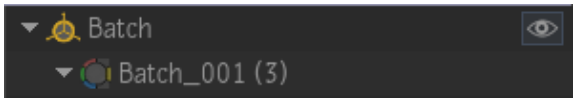
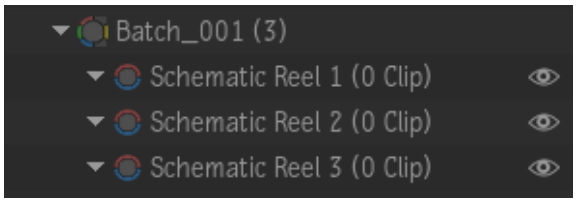
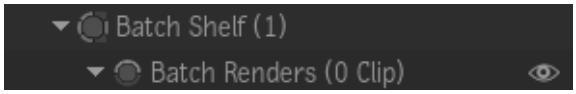


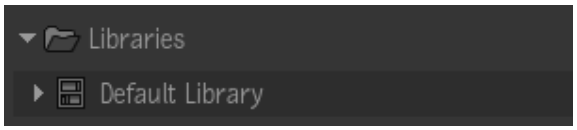
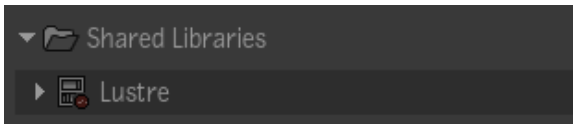
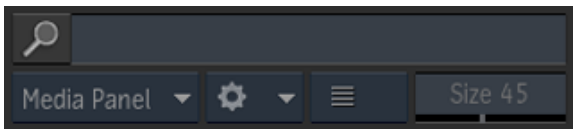
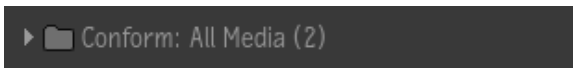
NOTE To display the traditional Mac OS menu bar, use the Setup utility, found in the Flame folder. In the Flame Setup utility, under the General tab, set Menu Bar to *yes*. Any changes will take effect the next time Flame is started.

About the Media Panel

The Media panel is where all Workspace media and creative finishing work is organized. To improve workflow efficiency, the Media panel offers several views of the current Workspace:

Section:	Description:
	**Media Panel tabs** Click All, Desktop, Libraries, or Batch to show most useful view of media and/or creative finishing work. When working in Conform, the Batch tab is replaced with the Conform tab, which lists all media available for conforming operations. When working in Batch FX, the Batch tab is replaced with the BFX tab.
	**Sorting Options** Sort the Media Panel Hierarchy using the Tree: option. Right-click the Tree: option to select whether to sort by name or creation date. Right-click the Clip: option to select clip sorting criteria, such as Name, Resolution, or Bit Depth.

Section:	Description:
	**Workspace Name (Project Name)** Displays the name of the current Workspace. The name of the project is shown in parenthesis.
	**Desktop** Organizes the Workspace into two distinct working areas: Batch groups and Reel groups.
	**Batch Group** The top level of the Batch compositing interface. The icon for the currently-active Batch group is highlighted in yellow. A project may contain multiple Batch groups, which can each be copied, duplicated, colour-coded, and renamed among other actions. Unless otherwise set in User Interface Preferences (page 1947), each new Batch group contains: the current Batch iteration containing the schematic reels where process trees are assembled; the Batch Shelf containing the Batch Renders reel and Batch Shelf reel; and the Iterations folder (if shown) where incremented saves of the parent Batch group are stored.
	**Batch Iteration** The second level of the Batch structure, containing the schematic reels. The name of the current iteration is displayed and the number of reels is shown in parenthesis. Schematic reels may be added or deleted, and renamed or colour-coded.
	**Batch Shelf** Organizes media on reels outside of Batch iterations. Includes the Batch Renders reel and the Shelf reel. The number of reels is shown in parenthesis. Shelf reels may be added or deleted, and renamed or colour-coded.
	**Iterations** Stores Batch iterations. This folder may be shown or hidden depending on the Workspace

Section:	Description:
	preferences. For more information, see Saving Work using Batch or Batch FX Iterations (page 534).
	**Reel Group** Organizes media on reels on the Desktop outside of Batch groups. The icon for the currently-active Reel group is highlighted in yellow. The number of reels is shown in parenthesis. When conforming sequences, also contains a Sequences reel, and usually a Sources reel to store saved sources.
	**Sequences Reel** Reel used to store imported sequences within a Reel group. The Sequences reel is also Shots sequences and Sources sequences are created during a Connected conform, allowing the sharing of segments across sequences stored within.
	**Libraries** Repositories for storing media outside of the Desktop. Each project contains a Default library, however additional libraries may be created as needed.
	**Shared Libraries** Allows content to be made available to other users and creative finishing applications running on the same network.
	**Search Field** Perform name based searches for media in the Media panel. Search results are displayed in alphabetical order. The search results list is always the last entry in the panel. If you need a more advanced Media panel search (page 62), click the magnifying glass icon to open the Find in Media Panel window. View Mode box changes the layout of the Media panel, the Gear menu displays options applicable to the selection, and the View menu switches the Media panel display between list and Tiles view.
	**Conform Media** Potential sources to conform are displayed in this section. This section is only displayed while in Conform view.

For a more detailed description of the Media panel, see Using the Media Panel (page 59).

Media Icons and Selection Reference

Media Thumbnail Icons

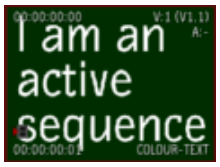
The Media panel displays icons next to the media so you can assess its contents at a glance.

Icon:	Media:
No Icon	Clip
	Sequence
	Effects Clip
	Opened Sequence (Timeline)
	Selected Clip
	Selected Sequence
	Selected Effects Clip

Media Selection Colour Coding

Clips on the Desktop or the Viewing panel are displayed with a coloured border based on the type of selection and the media type.

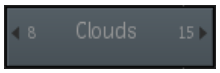
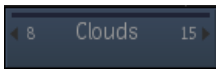
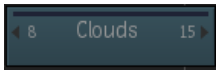
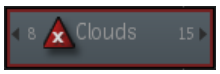
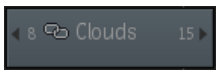
Border:	Selection:
	Unselected clip. No border.

Border:	Selection:
	Primary/secondary selection. A yellow border is displayed on the primary selection and a grey border on the secondary selection(s). The last selected clip becomes the primary selection.
	Active sequence. A red border is displayed when an active sequence is selected.
	Active source. A green border is displayed on the currently open source when an active sequence is selected.
	Last rendered clip. A blue border is displayed on the last rendered clip.
	Ganged clips. The clip information displayed on the thumbnails of ganged clips turns green when selected. Otherwise, the clip information on ganged clips is yellow.

NOTE When the selection is made from the Media panel, the yellow border appears in the Media panel selection and the clip on the Desktop or the Viewing panel is bordered in grey, regardless of the media type.

Timeline Colour Coding

On the timeline, different colours indicate different states of elements:

Element:	State:
	An explicitly selected video segment. If no timeline segment is explicitly selected, the segment closest to the timeline positioner's focus point is implicitly selected (and outlined with a white border).
	An unselected video segment.
	An audio segment.
	A container.
	A segment with a BFX applied.
	A segment with unlinked video.
	A segment shared across multiple timelines, due to the creation of Shots Sequence.

Media Cache Status for Clips

Clips provide you with a granular status of their media cache. There are four cache statuses, available from any list view from the *Cached* column, or from the icon on the actual clip or segment.

Media Cache Status Icon	Cache Status in List	Description
	Uncached	Not one frame of the clip is cached, and no caching operation has been requested for this clip.
	Pending	Backburner has been asked to cache the clip's media, but has not started the caching process yet.
	Caching	The clip's media is being cached, frame by frame.
	Cached	Every single frame of the clip is now cached. All of them.

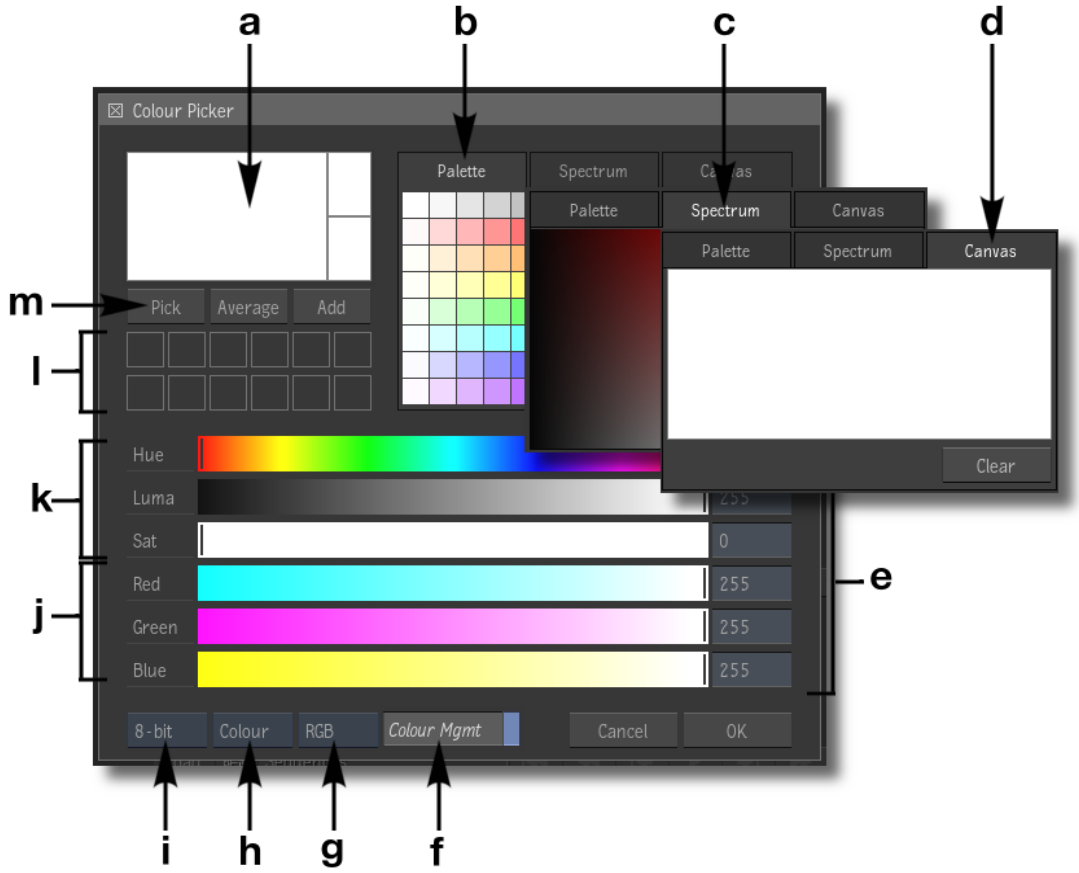
NOTE As a clip is being cached, Pending Cache is overlaid on frames not yet cached.

The Colour Picker

The Colour Picker appears as a floating window when you click a colour pot in the application. Using the Colour Picker, you can set the colour pot to the colour of your choice or you can pick colours by:

- Setting colour model channel values;
- Sampling pixels in a clip;
- Sampling a pixel's colour inside a Viewport (Player or Tools) to get its RGB values. The Colour Sampler can be accessed using the Toggle Colour Sampler shortcut while clicking on an image inside a Viewport. An unfiltered, zoomed-in part of the image enables you to precisely view the pixel being sampled.
- Selecting a colour from the Palette widget;
- Selecting a colour from the Spectrum widget;
- Mixing colours in the Canvas widget.

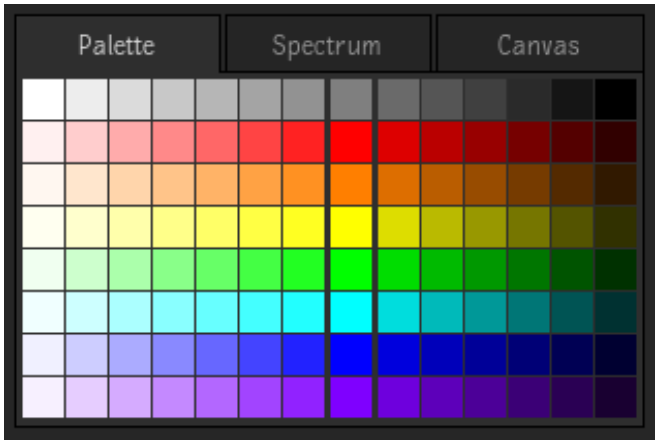
The Colour Sampler can be accessed using the Toggle Colour Sampler shortcut while clicking on an image inside a Viewport. The shortcut is assigned to Alt - Spacebar shortcut in all profiles.



a Colour Patches: The colour picker includes three colour patches:

- The large colour patch, on the left, displays the currently selected colour.
- The smaller colour patch, on the bottom right, displays the original colour that was selected.
- The smaller colour patch on the top right, displays the previous colour that was selected.

b The Palette Widget: Enables you to quickly select a default colour. Towards the middle of the palette, between the bold separators, you can select pure colours.



c The Spectrum Widget: Enables you to pick contrast and saturation variations of a same colour.

d The Canvas Widget: Enables you to mix and create custom colours.

e Colour Values: Displays the current value of the channel sliders. These can be either numeric or percentage values.

f Colour Management button: Enable viewport colour management on the colour swatches, based on the tagged colour space associated with the colour being picked. Additionally, in Preferences / User Interface / Colour Sampler menu, is the "Show Display CS Info" button which is used to add a second set of RGB values in the colour picker, showing the values after the viewport colour management has been applied. It is disabled by default.

g Colour Model box: Toggle to select between the RGB or YUV colour model.

h Mode box: Use to select how the numeric values represent each channel. Options are:

- **Colour:** Use a range based on bit depth (0-255 for 8-bit, 0-1023 for 10-bit, 0-4095 for 12-bit). Only available with Bit Depth set at 8-bit, 10-bit, or 12-bit.
- **Colour %:** Use a percentage value, relative to the entire range of the selected bit depth, ranging from 0-100%. Only available with Bit Depth set at 8-bit, 10-bit, or 12-bit.
- **Range:** Use the full range of colours in a 16-bit floating point colour space, entered as a floating-point number. The brackets enclose the 0-1 range. Only available with Bit Depth set at 16fp.
- **[0-1]:** Display the 0-1 range inside the full 16-bit floating point range, where 0 and 1 are enclosed by brackets. You can still use values outside the 0-1 range. Only available with Bit Depth set at 16fp.

i Bit Depth box: Use to select the media bit depth. Options are:

- 8-bit
- 10-bit
- 12-bit
- 16-bit fp

j RGB / YUV Channel sliders: Use to modify the values of the Red, Green and Blue channels or the YUV channels, as set by the Colour Model box.

k HLS Channel sliders: Use to modify the values of the Hue, Luma and Saturation channels.

l Custom Colour patches: 12 custom colours can be stored using the "Add" button.

m

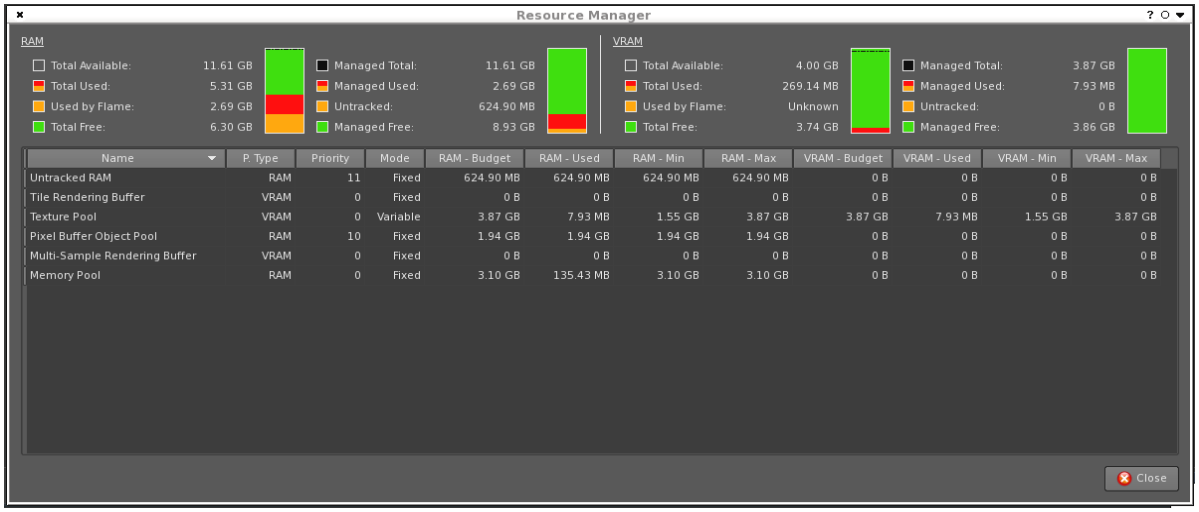
- **Pick button:** Click to sample the colour of the pixel under the cursor.

- **Average button:**
  - Click and drag to draw a box and sample the average colour of the pixels within that box.
  - Alt+Click and trace a path to sample an average colour from the pixels under the cursor, along the path you traced.

- **Add button:** Click to save a sampled colour to one of the 12 Custom Colour Patches.

Resource Management

A Resource Manager utility is available from the Flame menu. The Resource manager indicates the amount of RAM and VRAM that are Available, Used by the system, Used by the software, and Free. It also lists the RAM and VRAM clients, and details their memory consumption.



Memory Management Tokens

Two tokens located in the init.cfg file enable you to define the percentage of Memory and Graphics Memory to reserve for the application. These tokens deprecate both the `MemoryApplication` token and the `-M` startup switch. They are:

- `MemoryConsumptionTarget`
- `GraphicsMemoryConsumptionTarget`

See the init.cfg file for additional details, but in most cases, you do not need to worry about these. If you are using Connected Colour workflow, see Memory Settings for Connected Colour Workflow below.

NOTE The recommended ratio for system-memory (RAM) to graphics card memory (VRAM) is 3-1. If you are using Shot Reactor, add up the values of both graphics cards before calculating the right ratio.

Managing Projects and Users 5

Projects and users define your working environment.

Projects:

- Define the display environment
- Control values for resolution, aspect ratio, bit depth, scan mode, etc.
  You typically create a project for each job you work on.

For each person working on your system, you can also create a user.

A user is a profile that maintains the following preferences:
- User interface settings
- Pen and tablet preferences
- Keyboard shortcuts

Working with Projects

To create a project:

1. From the Startup screen, click New in the Project section.
2. In the Create New Project dialog box, in the Name field, enter a name for the new project, up to 120 characters long.
3. If you have multiple volumes on your system, you can select the volume you want to work with from the Volume field. If you only have one volume, the volume name is displayed. Projects are tied to the volume they were created on and cannot be accessed from a different volume. To access the content from a project on a different volume, wire the clips and sequences from that project to a project on the current volume using MediaHub > Browse for Projects.
4. Set the default resolution for the project. The default project configuration template is automatically loaded in Config Template or you can manually select a different project configuration template. This can be useful if your monitor does not natively support the resolution of your project. See About the Project Configuration Template (page 50).
5. Set the other Resolution options, if required. You should leave Graphics Rendering to the default option of 16-bit FP Graphics.
6. Choose a Colour Policy. The Colour Policy defines the colour management settings that will be used to configure the project. See Project-Based Colour Policies (page 1781).

7 Set the Cache and Renders settings.

8 Set the Proxy Settings.

9 Set the Lustre Settings.

10 When you are satisfied with project settings, click Create. Projects have an autosave feature. The project is automatically saved at the interval set in the preferences. You can manually save your project with the Alt+S keyboard shortcut or by selecting Save Project from the Flame (or Flare) menu.

To edit a project:

1 Do one of the following:
   - From the Startup screen, select a project.
   - In the middle of a session, from the Flame menu, select Project and User Settings. And then, from the Project and User Settings window, select the project.

2 Click Edit.

3 In the Edit Project dialog , modify the settings.

4 Click Done. The modifications are applied to your project.

To delete a project:

1 Do one of the following:
   - At start-up, select a project.
   - In the middle of a session, from the Flame (or Flare) menu, select Project and User Settings. From the Project and User Settings, select the project to delete. You cannot delete the current project.

2 Click Edit.

3 The Edit Project dialog click Project Edit and select Delete Project.

4 Click Delete and then Confirm. When you delete a project, all its associated clips and setups are deleted with the project.

You cannot open a project created in a previous version of Flame without first converting it.

To open a project created in a previous version:

1 From the Startup screen, in the **Project** section, set the **Versions** filter to **All Versions**.

2 Select the project to convert from the Project box. The version of Flame used to create it appears in brackets.

3 Click Start.

4 In the dialogue box that appears, click Convert to proceed with the conversion.

   The conversion creates a copy of the project and converts this copy to the new version of the application, duplicating the setups. The original project remains untouched and accessible to the application that created it.

If you cannot convert a project because of its age, use either of the following methods to open and access the contents of the project.

- Create an archive in the old version of Flame and restore the archive in the newer version.
- Open **MediaHub ➤ Browse for Projects** and browse to project to restore. You can then import those files in the new Flame.

Project Compatibility Between Releases

The Flame 2017 family of products (Flame Premium, Flare, Flame Assist) can now read contents created in Smoke Desktop Subscription (DTS), with the following limitations:

- Projects from Smoke DTS can be opened (and converted) by the Flame Family 2017 products.
- Projects can be seen and imported through MediaHub, using Browse for Projects.
- Archives created in Smoke DTS can be read in Flame Family 2017 products.

IMPORTANT Contents created in Flame Family 2017 products cannot be read or opened in Smoke DTS.

		Will import in...				
		Flame 2017	Flare 2017	Flame Assist 2017	Smoke DTS 2017	Training Edition 2017
Projects Created In...	Flame 2017	Yes	Yes	Yes	No	No
	Flare 2017	Yes	Yes	Yes	No	No
	Flame Assist 2017	Yes	Yes	Yes	No	No
	Smoke DTS 2017	Yes	Yes	Yes	Yes	No
	Training 2017	No	No	No	No	Yes
	Flame 2016	Yes	Yes	Yes	No	No
	Smoke DTS 2016	Yes	Yes	Yes	Yes	No
	Training 2016	No	No	No	No	Yes
	Smoke DTS 2015	Yes	Yes	Yes	No	No

About the Project Setup Directory

Setups are file-based resources such as effects settings and LUTs, and are managed in a set of subdirectories in the project's setup directory.

When you create a project, the project's setup directory appears in the Setup Directory box. The many setups and their directories are actually stored in */opt/Autodesk/project/<Project Name>/*, and are copied from the reference setups stored in */opt/Autodesk/<Version of the application>/*.

You are not restricted to these directories when you load or save setups: use the file browser to locate a different directory, or use project shortcuts to browse the setup directories of other projects on the same system.

To have the new project share the setups of an existing project:

1 From the Setup Directory box, select the project with the shared setups.

Any setups already saved to the existing project are made available to the new project.

To copy the setups from an existing project:

1. From the Setup Mode box, select Copy From.
2. From the Projects box that appears, select the project from which you want to copy the setups.

About the Graphics Rendering Bit Depth

Depending on your system, you can specify the bit depth for images rendered by the graphics card. The bit depth affects the quality of the resulting clip. The images can be rendered with 8-bit or 16-bit FP precision.

As a rule, if you are working with mixed resolutions and some clips are higher than 8-bit, you should select 16-bit FP graphics display. Even if you work only with 8-bit images, you can get better rendered results with 16-bit FP graphics rendering when transparencies, blending, and gradients are part of your effect. 16-bit FP graphics rendering produces better results but takes longer.

Also, if your output is ultimately an 8-bit format, having the best possible quality immediately prior to output produces the best results.

About the Project Configuration Template

Each project has a project configuration file. When you load a project, its project configuration file is read. Information in the project configuration file determines settings such as graphics monitor refresh rate, default time code, and default frame rate for playback.

A project's configuration file is created based on a project configuration file template. The template specifies project settings typically associated with the project's default resolution.

When you select a resolution, a project configuration template appears in the Configuration Template box. The project configuration file primarily defines the display environment, and in no way restricts you from working with clips of another resolution.

Setting Proxy Management Options

Proxies are low-resolution copies of high-resolution clips. Using proxies increases playback performance and responsiveness, especially when working with large files.

All projects are proxy capable, in that they support on demand proxies. At any point, for any media, you can generate proxies. Hence, there is no option to enable or disable proxy generation. You do, however, have proxy management options to specify how and when proxies are generated for clips used in the project.

For more information on On Demand Proxies, see: Working with On Demand Proxies (page 51).

Set proxy management options:

1. If you want proxy generation to be the default option in your project, enable the Generate Proxies as Default button.

   NOTE You can generate proxies on demand for any media in your project at any time, even if this button is not enabled. This simply makes it the default option at project creation.

2. Select Proxy 1/2, Proxy 1/4, or Proxy 1/8 to set the fraction of resolution to be displayed by proxies.

   NOTE Proxies maintain the same aspect ratio as their full-resolution sources.

3 Set the minimal width criteria for generating proxies by entering a value in the Conditional Proxy Width field. For example, enter 1280 to generate proxies only for clips with a width of 1280 pixels or greater.

4 From the Proxy Quality box, set the viewing quality of the proxy image.

Proxy quality affects rendering and proxy generation times. While lower qualities are faster to calculate, all proxies require the same amount of disk space for purposes of storage.

Select:	To specify:
Lanczos	Excellent quality, recommended for upscaling and downscaling. Slower to calculate.
Shannon	Excellent quality, sharper than Lanczos in small details. Recommended for upscaling and downscaling. Slower to calculate.
Gaussian	Medium quality and softer results.
Quadratic	Medium quality and softer results.
Bicubic	High quality, but not as sharp as Shannon. Use for both upscaling and downscaling.
Mitchell	High quality, but not as sharp as Shannon. Use for both upscaling and downscaling.
Triangle	Low quality, fast to calculate. Use for downscaling.
Impulse	Very low quality, fast to calculate. Use for downscaling.
Draft	Lowest possible quality, used when proxies are generated automatically following video I/O.

NOTE Once the Proxy options are set for a new project, they cannot be changed.

Working with On Demand Proxies

On Demand Proxies

Proxies can be enabled and disabled for individual clips, regardless of whether default proxy generation was enabled at project creation.

The Proxy Quality workflow allows you to reduce the amount of data being calculated in complex setups, without the need to have proxies stored on disk, by simply changing the quality.

However, as overall interactivity and performance is defined by the slowest link in the chain, if reading the source media is what is causing the performance loss for large setups, users can now optionally generate proxies on disk.

Creating Projects

Under the Proxy Settings tab, aside from the proxy size, quality setting and conditional setting, you can also enable the Generate Proxies By Default button. Because all projects are proxy-capable, what this button does is enable a sub-set of preferences, in relevant parts of the application, causing proxies to be generated on

disc, by default. This includes importing media from the MediaHub and rendering media from Batch or Batch FX, the Timeline, the Modules, etc.

Import Options

When you import media from the MediaHub, you have a Generate Proxies button. If you enabled the Generate Proxies By Default option when creating your project, the button will also be enabled by default in the MediaHub and proxy media will be generated on import. You are free to disable and re-enable it, at any point, to change the proxy generation behavior for specific media.

Rendering Options

Since proxy media is optional, there are preferences to define whether or not to write proxy media during rendering. These preferences affect the Timeline, Batch or Batch FX, Generators and the Modules.

In the General tab of the Preferences, under Default Rendering Options, there is now a Proxy Rendering drop-down menu, from which you can select one of three options:

- **Full Resolution and Proxy:** The application renders at both full resolution and proxy resolution.
- **Full Resolution:** The application renders only at full resolution.
- **Proxy:** This sets the Timeline Rendering options to "Proxy Only".

  NOTE The Proxy option only applies to the Timeline. When selected, Batch or Batch FX, Generators and the Modules are set to "Full Resolution and Proxy", as they do not support Proxy Only rendering. They instead have an "Include Proxies" option in the drop-down of their respective Rendering button, which is enabled or disabled, based on this preference setting.

Generating Proxies On Demand

On demand proxy generation is achieved through the contextual menu, under the Media header, which gives you two self-explanatory options, relative to proxy generation:

- Generate Proxies
- Flush Proxies

You can do this at any point, regardless of whether or not you enabled proxy generation by default when you created your project.

This can be done to an individual clip, to a Reel or a Library.

Visual Indication

In order to get a quick snapshot of which clips have proxy media, as well as their current status, you can:

- Enable List View in the Media Panel and look in the Proxy Resolution column, which will display a hyphen if no proxies exist on disc for that media, the resolution of the proxies if they exist on disc and "pending…" if proxies are in the midst of being generated.
- `Alt + Click` a clip in the Media Panel.

Setting Cache and Renders Format

Set the default media format for media cache and renders for your project under the Cache and Renders tab.

This setting is used whenever the application writes to its managed storage:

- When caching media. On demand (**contextual menu ➤ Media**) or on import (**MediaHub ➤ General ➤ Clip Options**).
- When rendering a Timeline FX, a clip using a Tools module, or when using a Render node in Batch or Batch FX.
- When creating virtual media such as coloured frames.
- When creating proxies. On demand (**contextual menu ➤ Media**) or on import (**MediaHub ➤ General ➤ Clip Options**).

To set the Cache and Renders format:

1. Click the Cache and Renders tab.
2. Select the media format from the Cache and Renders Preferred Format box.

NOTE Preferred formats are all available in RAW and non-RAW flavours. The difference between the two flavours is the presence or absence of OpenEXR as an alternate format: RAW flavours have only DPX and RAW, non-RAW ones have DPX, OpenEXR, and RAW. As a rule of thumb, use the RAW flavours, unless you are planning on exporting material as OpenEXR using linked publish, in which case you should the but the non-RAW flavours.

Notes regarding the use of DNxHD and DNxHR as an intermediate format (Linux only):

- Anything cached at HD resolution (1920x1080 or 1280x720), DNxHD is used. Everything else uses DNxHR.
- If you plan on using interlaced material in your project, avoid the DNxHD 36 and DNxHD 440 intermediates, or DNxHR. They do not support interlaced material, and any interlaced frame will be stored using one of the Alternate Formats, negating any space-saving advantage coming from using a compressed intermediate format. The other formats support mixing interlaced and progressive material.

Format Restriction:	Description:
Maximum Width	Displays the maximum width of a frame for it to use the Preferred Format. A frame wider than this is written using one of the Alternate Formats. *None* indicates that the preferred format is always used, unless the bit depth do not match the Depths field. See also the above note regarding DNxHD/DNxHR.
Maximum Height	Displays the maximum height of a frame for it to use the Preferred Format. Any frame bigger than this is written using one of the Alternate Formats. *None* indicates that the preferred format is always used, unless the bit depth do not match the Depths field. See also the above note regarding DNxHD/DNxHR.
Depths	Displays the bit depth required for a frame to use the Preferred Format. Frames with a bit depth higher than the ones listed is written using one of the Alternate Formats.
Alternate Formats	Displays the fallback formats used to cache or render frames that do not match the defined Bit Depth. The application follows a specific fallback strategy, based on the bit depth of the frame and the displayed formats: ■ DPX, OpenEXR, RAW: DPX for 8-, 10-, and 12-bit frames. OpenEXR for 16-bit floating point frames, RAW for 12-bit packed ones.

Format Restriction:	Description:
	■ DPX, RAW: DPX for 8-, 10-, and 12-bit frames. RAW for 12-bit packed, 16-bit floating point, and higher. Note that *12-bit packed* frames are always stored as RAW.

Setting Lustre Project Options

When creating a new Flame project, an associated Lustre library can be automatically generated by setting Lustre Project options. The new library is displayed in the Shared Libraries section of the media panel.

To create a shared Lustre library for a new project:

1. Click the Lustre Settings tab.
2. Enable Create Lustre Project.
3. From the Grade Mode box, select either Linear or Logarithmic to set the desired colour grading functionality.
4. Enable Stereoscopy to automatically create a stereoscopic Lustre project with library folders for both left and right eyes.

Working with Users

You can create, edit, or delete a user.

When you create a user, you have the option of copying preferences from an existing user. If the user whose preferences you want to copy was created on the same version of the application, you can copy all preferences. If the user was created on an older version of the application, you can only copy shortcut preferences.

NOTE You can create or open a project on a remote workstation using the Host Computer drop-down list.

To create a user:

1. On the Startup screen, click New in the User section.

   If you are past the Startup screen, you must restart Flame to access it, and proceed from there.
2. In the Name field, enter a name for the new user.
3. Optional: Add a nickname for the user. The nickname can be automatically inserted wherever tokens are available, such as batch iteration names, segment names or exported clip names.
4. Select your User Profile, which determines, among other things, the application startup behavior and the Keyboard Shortcuts scheme. Options are:
   - Smoke (FCP 7): Based on Final Cut Pro 7 keyboard shortcuts. When this profile is selected, the application starts in the Timeline tab.
   - Flame: When this profile is selected, the application starts in the Tools tab.
   - Smoke Classic: When this profile is selected, the application starts in the Timeline tab.
5. Set Directory to the location where the user preference files are stored. This can be local, or on the Remote Host selected from the Startup screen.
6. Enable Lustre User to create a matching user in Lustre.
7. Set the Creation Mode, either creating a new set of preferences, or copying them from another user.

8 Click Create to create the user.
Back on the Startup screen, the new

To edit a user:

1 Do one of the following:
- At start-up, select a user.
- During a session, from the Flame menu, select Project and User Settings. From the Project and User Settings, select a user.

2 Click Edit. The Edit User Profile dialogue box appears.
3 Modify the settings as required.
4 Click Done.

To delete a user:

1 Do one of the following:
- At start-up, select a user.
- During a session, from the Flame menu, select Project and User Settings. From the Project and User Settings, select a user.

2 Click Edit. The Edit User Profile dialogue box appears.
3 Click the User Edit box and select Delete User.
4 Click Delete.

Organizing Media in the Workspace 6

About The Workspace

Creative finishing projects are often complex endeavors. Multiple artists may perform many different tasks before sharing the results with other users for further development. In such heavily-collaborative environments, it is critical to keep these operations carefully separated in order to avoid the accidental overwriting of hours of creative effort.

To safeguard work, Flame is divided into the following components:

- The Workspace, a user-specific area where creative tasks are performed and media can be stored and organized locally.
- Shared Libraries, where source and finished media can be safely made available to other flame-family systems and applications connected to the same project.

Each user is assigned their own Workspace upon opening a project. In the Workspace, they can undertake any and all creative activities without the possibility of altering anyone else's work.

The Workspace is then further divided into:

- The Desktop, where creative finishing work is performed.
- The Libraries folder, for storing source media and safely saving work.

For complete information on all Flame components and their hierarchy, see About the Structure of Projects (page 58).

Managing Media in Desktops and Libraries

A standard practice is to first create an organized series of folders in the libraries and then import the required project footage directly into them. Clips are then copied to the Desktop as needed.

Clips can also be directly imported to any reel in the Desktop, with the exception of a reel on a saved Batch group iteration.

NOTE When importing a 3DS, FBX, PSD or Particles file, the application automatically creates a new schematic reel within the current Batch Group, placing all imported textures in the dedicated reel.

Through the Media panel, clips and even entire reels are easily moved by simply dragging and dropping them to a new location. Clips and reels can equally be cut, copy or pasted by right-clicking the target and selecting the desired operation.

NOTE Because libraries are used as storage for raw sources and finished material, dragging a clip or reel from a library to a Desktop location will copy the target rather than move it, in order to avoid any inadvertent loss of work.

Editing Clips in a Library

To protect raw and finished material from accidental alteration, all editing functions for clips in libraries are turned off by default. Should it become necessary to edit a clip directly in a library, disable **Preferences ➤ General tab ➤ Protect from Editing**.

Once this option disabled, the following operations can be freely performed on clips in all libraries:

- Editing of a source clip (Trim, Slip, Cut, etc.)
- Timeline FX operations (Add, Delete, Modify, etc.)
- Batch FX operations.
- Opening a clip as a sequence.

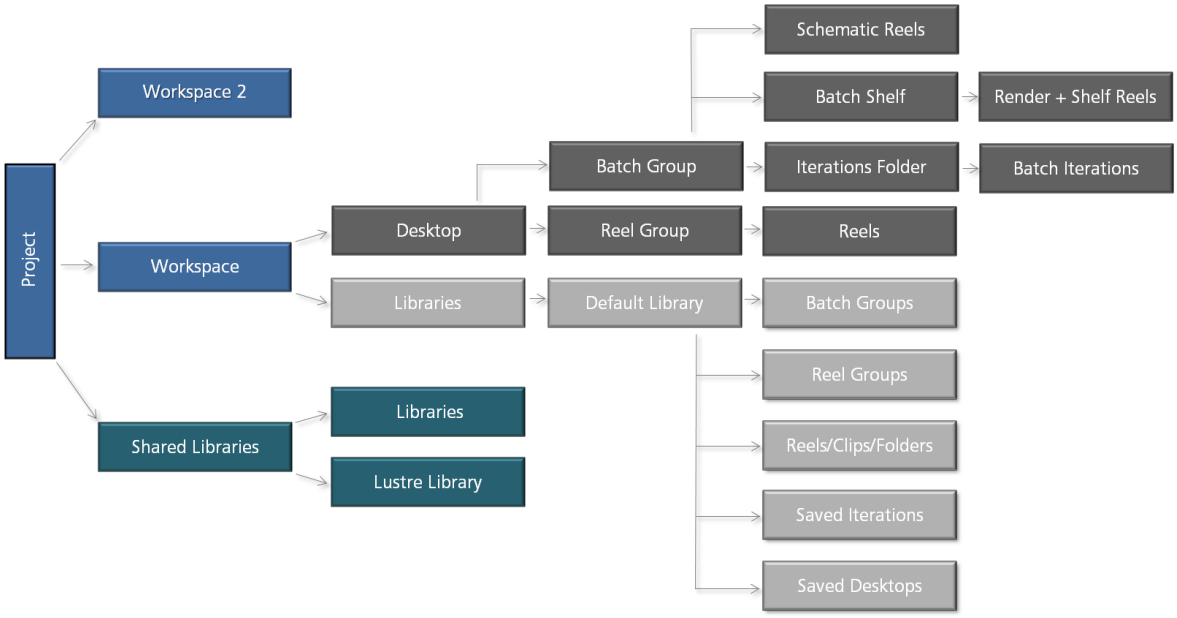
You can, however, copy a segment out of a sequence in the following situations:

- The clip is located in a Library and Protect From Editing is enabled.
- The clip is locked.
- The clip is located in a Shared Library and you do not have the exclusive write access.

About the Structure of Projects

Below is a diagram showing the hierarchy of components in the Flame project structure. The example below is shown without any changes to the User Interface preferences and with a Lustre project automatically created. For the purposes of illustration, a second Workspace, additional shared libraries, and sample contents of the Default Library have been added.

Each project could contain an unlimited number of Workspaces, which each comprise a protected hierarchy of components accessible to a single user.



Project Structure

Each Flame project is organized as follows:

Project The entire creative finishing project containing all Workspaces and shared libraries.

Workspace A new Workspace is automatically generated upon creating a new project. If the project is already created and in use and another artist connects to that project from another system, another Workspace will be generated to avoid conflicting saves and operations. Workspaces can never be explicitly created by a user, but existing Workspaces can be selected between at the start-up screen. The Workspace itself is divided into two components: the Desktop and the Libraries folder. The Desktop and Libraries folder is specific to the Workspace. To make material available to another Workspace, use the Shared Libraries folder described below.

Shared Libraries The Shared Libraries folder is common to all Workspaces in the project. All files in a shared library can be viewed and accessed by all other users in the same project. Though a shared library can be written to and read from, clips cannot be edited there: all clips must be first moved to the Desktop or a local library in order to perform any editing operations. A shared library can contain clips, sequences, folders, reels, Batch groups and iterations, and even complete saved Desktops. The Lustre library is special shared library for colour-grading operations and is only available if Create Lustre Project was enabled in the Lustre settings during the creation of the project.

Desktop The Desktop is the creative work bench of the program, containing:

- A separate Batch group for all compositions, each of which contain a Batch Shelf and an Iterations folder (if shown). The Batch Shelf contains the Batch Renders reel and all Shelf reels. The Iterations folder contains all saved iterations and their associated reels.
- All Reel groups (if created), which contain reels for storing media on the Desktop outside of Batch.

A Desktop can contain multiple Batch Groups and Reel Groups. In each workspace, there can be only one Desktop open at a time. A Desktop can be saved to a library and later restored, or copied to a shared library to be shared across Workspaces.

Libraries The storage area of the Workspace containing the Default library and any other individual user-created libraries. Each library can contain any combination of the following:

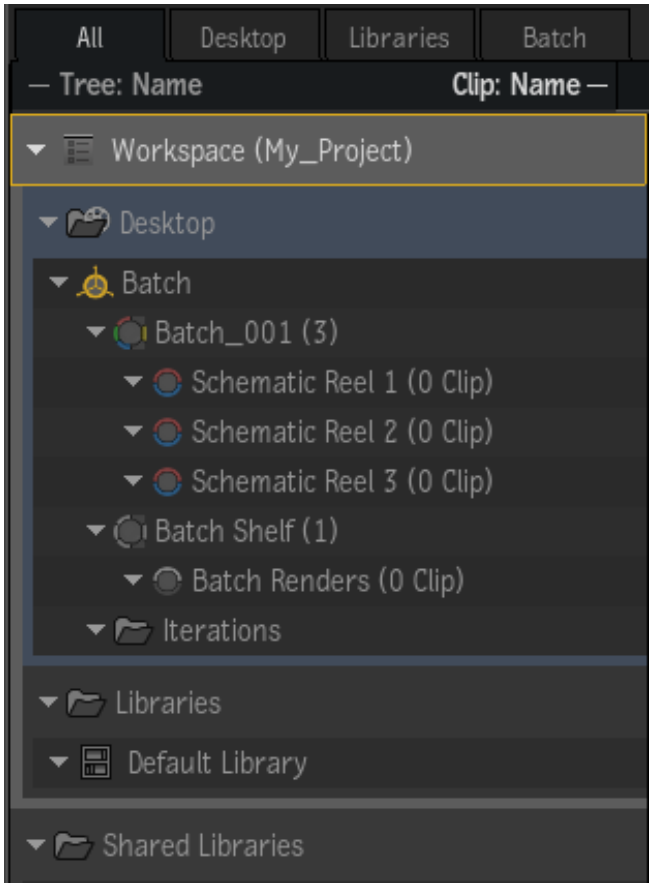
- Clips
- Folders
- Reels
- Batch groups
- Batch iterations
- Saved Desktops

There is no limit to the number of libraries that can be created.

Using the Media Panel

Displaying and Hiding the Media Panel

The Media panel is available for display at all times unless viewing a clip in Full Screen mode. The panel can be displayed on the left or right of the screen depending on the setting in General Preferences (page 1936).



NOTE If the Media panel is currently hidden, press `Shift+Esc` to redisplay it. In the Media panel View Mode box, you can select to display full height or width, or hide the Media panel.

Tiles and List View

The Media panel can be set to display clips in one of two modes using the View Mode box:

List View The List view provides detailed information about each clip in the Media panel, including file location and creation date. Widen the Media panel to show more information or extend to full-width by pressing `Shift + ~`.

Tiles View The Tiles view makes it easier to visually identify clips when working in Batch, identifying clips in Conform, or loading the right clip from a library to a reel. The thumbnails can be resized using the Size field. Clips can be freely dragged and sorted in Tiles view.

Tips:

- Use `Ctrl + Space + Up` to enlarge the thumbnail size.
- Use `Ctrl + Space + Down` to reduce the thumbnail size.

TIP `Alt + click` a clip to display extended clip information.

Tab Navigation

The top of the Media panel displays a number of tabs. Click a tab to focus on a specific area of interest in the Media panel.

All Displays all areas of the Media panel: Desktop, Libraries, and Batch.

Desktop Displays each instance of Batch, along with their schematic reels, shelves, and iterations.

Libraries Displays only the Libraries and Shared Libraries folders.

Batch Displays only the currently-active Batch, along with its schematic reels, Batch shelf, and iterations.

Conform Only available while working in Conform, replacing the Batch tab described above. Displays only media available for the conform, filtered by match criteria.

BFX Only available while working in Batch FX, replacing the Batch tab described above. Displays only media used in Batch FX.

Dual Panel View

Use Dual Panel View to manage clips, folders, reels, libraries. Dual Panel View allows the display of different areas of the same Workspace, making it easy to move material around.

To display the Media panel in dual view:

1. Set the Media Panel button to Full Width (`Shift+~`).
2. Click Dual Panel View to display two instances of the Media panel.

Dragging Clips into the Media Panel

Drag and drop clips from libraries into different areas of the Media panel to perform operations on working copies while leaving the originals intact:

- Drag clips to reels and edit directly in the Viewing panel using a mouse or tablet. For more information, see Gestural Editing on the Desktop (page 457).
- Drag clips to the schematic reels in Batch groups to apply procedural effects. For more information, see Procedural Compositing with Batch and ConnectFX (page 495).
- Drag clips to shared libraries to make them available to other applications and workstations on the network.

Sorting Clips and Sequences in the Media Panel

Whether in Tiles or List view, use the Tree: (name and creation date) and Clip: (many clip sorting criteria) sorting options at the top of the Media panel to sort the complete hierarchy. Right-click either option to select sorting criteria.

You can also apply a localised sort to any object of the Media panel using the Localised Sort option in the right-click contextual menu.

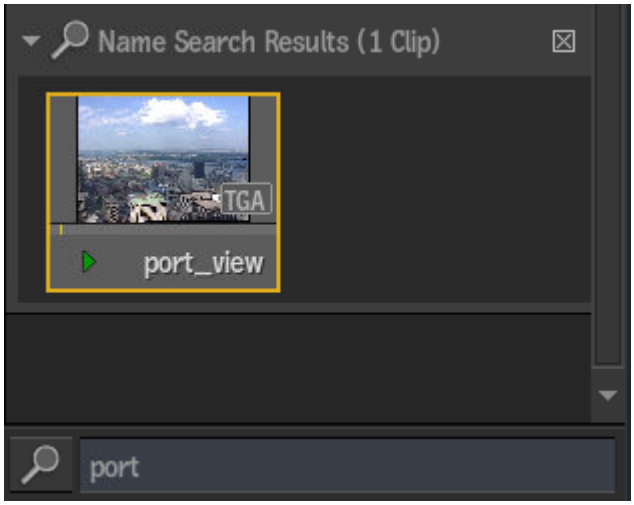
- The Media panel sorting options available at the top of the Media panel applies to all entries that are not under a localised sort.
- In the Media panel, an icon appears next to the container to indicate that a localised sort is being applied. You can click on the icon to cycle through Ascending, Descending, or Not Sorted. Alt-click the icon to clear the localised sort.

Assigning Colours

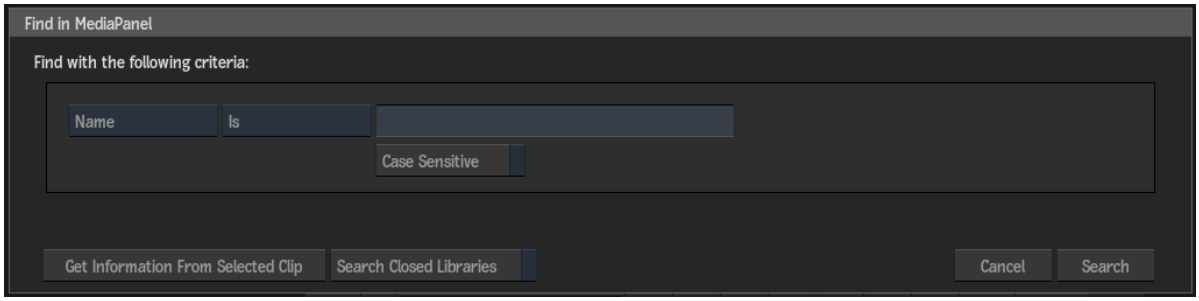
Assign custom colours to Batches, reels, folders and libraries to further organize material and enhance workflow in the Media panel. To do this, right-click a target, select Colour, and choose from the provided options.

Searching in the Media Panel

To perform a simple name search, enter text in the field at the bottom of the Media panel and press Enter. A Name Search Results area appears near the bottom of the Media panel to display the search results.



If you need a more advanced Media panel search, click the magnifying glass icon to open the Find in Media Panel window.



Here are some tips for advanced searches:

- Many options are available in the first criteria box, such as Name, Resolution, Tape, and Comment. Other criteria is displayed to further refine your search, depending on the selection in this box.
- Click Get Information From Selected Clip to fill the current search field with the information from the currently selected clip.
- Click Search Closed Libraries to also search the contents of closed Libraries and Batch Groups.
  - A Library or Batch Group remains closed if the Advanced Search cannot find any match.
  - A Library or Batch Group is opened if the Advanced Search finds a match.
- The contents of a Batch Group's Iterations folders are searched even if they are hidden in the Media panel.

Objects With the Same Name

If you add a clip or container to a location where another identically named item already exists, you are presented with options to either cancel, add, rename, or replace. You can also choose to apply your selection to all the selected items at once.

Select:	To:
Cancel	Cancel the operation (not available for clips).
Add	Add the entry without changing its name. You end up with more than one entry with the same name.
Rename	Rename the newly added item.
Replace	Overwrite the existing item in the destination.

TIP There is a Duplicate Name Check preference in **Preferences ➤ General ➤ Media Panel** where you can decide if you want to get this message for clip, containers, both, or neither. By default, it is set to both.

Tips for Working with the Media Panel

- Dragging a clip from a library to Batch Group entry within the Media panel creates a new reel with the clip name.
- Drag a library or a library folder to a reel to copy all of its contents onto the reel.
- Drag a reel from the Desktop area to a library to make a copy.
- After deleting an element from the Media Panel, the current Source clip is no longer selected. Instead, a sibling of the deleted element is selected. In the case where there is no sibling left, the parent container is selected.

Saving and Restoring Desktops

Save the Desktop at any time to back up work and have the option of restoring a copy later.

Saving the Desktop stores its contents and structure in a selected library. This an efficient method of packaging a shot, sequence, or work iteration, because when restoring a saved Desktop all settings and associated media are automatically included.

Saving the Desktop

To save the Desktop, do one of the following:

- Select a library or create a new one using the Target Library box, located in the bottom-right corner of the screen. Enter a new name in the Desktop Name field if desired, then click the Save Desktop button.

  The Desktop is saved in the target library.
- Drag and drop the Desktop onto a library folder.

  The Desktop is saved to the target library.

TIP The destination for saving the Desktop can also be set directly in the Media panel. To do so, right-click the desired library or folder, and from the contextual menu select Set As Save Destination.

If the Desktop is saved to a library containing a Desktop with the same name, the Saving to library dialog box appears:

- Click Add to create another Desktop save using the same name.
- Click Replace to overwrite the previously saved Desktop with the current Desktop.
- Click Rename to save the current Desktop with a new name.

Restoring a Desktop

To restore a Desktop, do one of the following in the Media panel:

- Right-click a saved desktop, and from the contextual menu select Replace Desktop.
- Drag and drop the Desktop onto the active Desktop.

The current Desktop is cleared and replaced without a confirmation message.

About Desktop Saves

Saving a Desktop backs up the following:

- All Batch groups and their complete contents
- All Reel groups and their associated media

Desktops can be saved as works-in-progress to be restored later or as backups of finished compositions. All clips, reels, and sequences are available as well as any Batch groups and iterations. Saved Desktops are designed for sharing through a Shared Libraries folder or the MediaHub as a single package containing all elements required to rebuild a shot, a sequence, or a work iteration.

The contents of a saved Desktop in a library are easily accessed. For example:

- Drag and drop a Batch group or iteration from a saved Desktop to Batch schematic to open it.
- Edit and add clips to the reels of a saved Desktop.
- Drag and drop a saved Desktop to a folder in Shared Libraries to make it accessible to other workstations.

NOTE When opening a Batch group created on a Flame in Flame Assist BFX, any nodes unavailable in Flame Assist are greyed out. The nodes will still be processed, however their settings cannot be edited.

Organizing the Desktop Reels

About Desktop Reels

Media to be used in creative finishing and editing operations on the Desktop is organized into reels for easy access. Individuals reels may be copied, moved, and deleted as well as have their visibility turned on and off. Only a single group of reels can be displayed in Desktop Reels view at any one time.

There are three types of Desktop reels available to help organize media:

- Schematic reels containing sources used in Batch process trees. Located under a parent Batch group.

  Additional schematic reels may be created if needed by right-clicking the Batch group or reel and selecting New Schematic Reel.

- Shelf reels, where media can be organized and accessed in Batch without cluttering up schematics and their process trees. The Batch Renders reel is a shelf reel provided as the default destination for all clips rendered via the Render node; however further shelves may be added or deleted as desired.

  Additional shelf reels may be created if needed by right-clicking the reel and selecting New Shelf Reel.

- Reel Group reels, where media can be organized outside of Batch but still on the Desktop. A Reel Group and the Sequences reel are central to the Connected Conform (page 294) workflow.

  To automatically include a Reel group on the Desktop in new projects, enable Create Reel Group in the User Interface Preferences (page 1947).

Viewing Reels

The contents of reels are best displayed using Desktop Reels view in the Viewing panel. To enable Desktop Reels view, do one of the following:

- From the View Mode box, select Desktop Reels.

- On the Desktop, double-click a Batch group or Reel group. If the desired reels are in a Batch group, that specific group must be active.

  The reels are displayed in the Viewing panel.

In the Media panel, click the eye icon beside each reel to toggle its visibility.

If the number of visible reels exceeds the maximum displayable in the Viewing panel, additional reels are hidden automatically. In the Media panel, the eye icon will turn black to show that visibility has been turned off by the system. Clicking a hidden reel when there is no space to display it will also turn the eye icon black. To view automatically hidden reels, first turn off the visibility of a corresponding number of reels in the Viewing panel.

NOTE The maximum number of reels that can be displayed in the Viewing panel depends upon screen size and resolution.

The currently-displayed group of reels can be quickly switched from directly in the Viewing panel. To do this:

- Click the eye icon displayed on the right side of the transport menu for the reel located at the bottom of the Viewing panel, or immediately above any pinned reels (for more information on pinning reels, see the following section).

- Select the desired Batch or Reel group from the drop-down menu.

The selected group is shown in the Viewing panel.

To change the order of the reels in Reels view:

1. Click and hold the Move button at the far right of the transport menu for the reel to be moved.
2. Drag and drop the reel to its new position on the Desktop.

NOTE Pinned reels cannot be moved.

Pinning Reels

Reels from any type of group can be pinned to the bottom of the Viewing panel. A pinned reel remains in place even if the current Batch group or Reel group is changed. This can be useful, for example, for keeping a particular reel in view while browsing through other reels for material to assemble. All pinned reels have a small white pin icon displayed to their right.

NOTE Reels from the currently-selected group cannot be pinned. To pin these reels, first select another group.

To pin a reel:

- In the Media panel, make sure the reel to be pinned is shown and not part of an already-selected group.
- Click the grayed-out pin icon to the right of the reel name and content details.

The reel is pinned to the bottom of the Viewing panel. If there are other pinned reels showing, the reel will be pinned above.

When pinned, a reel displays a drop-down menu marked by the pin icon. Use this menu to swap the pinned reel for another within the same group.

The order in which the pinned reels are shown in the Viewing panel reflects their order in the Media panel. To change the order in which pinned reels are shown, simply change their order in the Media panel by dragging and dropping them.

Priority is given to pinned reels. If more reels are displayed than the maximum number of reels allowed, other reels are hidden accordingly. As a displayed reel gets hidden, the eye icon to its right turns black.

Renaming Reels

Reels can be renamed for better organization. To rename a reel, do one of the following:

- In the Media panel, right-click the reel and select Rename.
- In the Desktop Reels view, select the name of the reel, the enter a new one.
- With the cursor over the item to rename, press `N`.

NOTE Reel names do not have to be unique: multiple reels can use the same name.

Adding Reels

New reels can be added directly in the Viewing panel while in Desktop view.

To add a new reel:

- Click the reel icon at the left of the transport menu to add a reel immediately below the selected reel.

- `Ctrl`-click the reel icon to add a reel immediately above the selected reel.

NOTE Adding more reels than the screen can display will automatically turn off the visibility of surplus reels.

Adding Clips to a Reel

To add clips to a Reel, do one of the following:

- Drag and drop a clip from the Media panel to a Desktop Reel.
- Press the "Load" button, to the left of the Reel. The Load button opens a modal window displaying the Media Panel, from which you can select clips to be loaded to a Reel. It is not possible to do anything other than selecting a clip in this modal window.

  The clip is added to the reel, framed over the Reel controls.

  NOTE
  - You can still load clips when the Media Panel is hidden.
  - The new Modal window has its own set of display settings (Tiles / List, Tiles Size, Expand / Collapse, Scrollbar, Filtering tab).

NOTE Adding more reels than the screen can display will automatically turn off the visibility of surplus reels.

Clearing and Deleting Reels

Reels can be cleared and deleted directly in the Viewing panel when in Desktop Reels view. Individual clips can also be deleted.

To clear the entire contents of a reel:

- With the cursor over the reel, press `Shift+D`.

To delete a reel or specific contents:

- With the cursor over a clip, press `D` to delete that clip only.
- With the cursor over a blank area in the reel, press `D` to delete the entire reel.

TIP Both clips and reels can also be deleted by dragging them to the bottom of the screen.

Reels and Reel Groups in Libraries

When in a library, a reel behaves like a folder, but can only contain clips. A Reel group in a library is like a Reel group on the Desktop, and contains multiple reels.

To copy a reel or group from a library, do one of the following:

1. Drag and drop the reel or reel group to the desired location on the Desktop. If a reel group, only the reels themselves are copied.
2. Drag and drop the reel on top of another reel to add the contents of the dropped reel.

NOTE In all cases, the original reel or reel group remains in its original location in the library.

Reel Groups

Reel groups are essential to the Connected conform workflow.

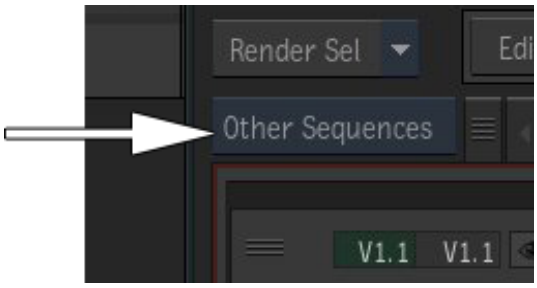
- Only a Reel group can have a Sequences reel. This reel is used to store sequences imported for conforming AAF, EDL, or XML sequences.

  NOTE By default, every new Reel group has a Sequences reel. To have Sequences reel only appear on demand, disable the preference *Create Sequences Reel* , found in **Preferences ➤ User Interface ➤ Desktop**.

- Create additional Sequences reel within a Reel group using the option **New ➤ Sequences Reel** from the contextual menu.
- It is possible to work with multiple Reel groups. In this case, you can define the active Reel group by clicking the Reel group's icon, or use *Set As Current Reel Group* in the Reel group Contextual Menu. The *current* Reel group icon is yellow.

  NOTE A newly created Reel group is automatically set as the Current Reel group.

- Use the Active Sequence Reel button to navigate between the reels of the currently active Reel group; it only displays reels with opened sequences within that Reel group. When you select reel, the timeline displays every opened sequence within that reel.



The Other Sequences option displays all the open sequences that are not in a Reel group.

- Reel groups can be created in Media Library and on the Desktop. If no Reel group exists when importing an AAF, EDL, or XML from Conform, one is automatically created in the Desktop area of the Media Panel.

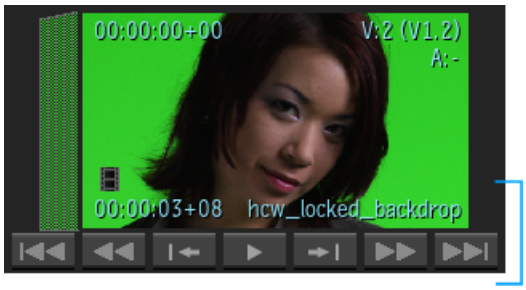
  NOTE Reel group can be copied in a Shared Library but cannot be used for any conform operation from this location.

- When you import a sequence with the option Save Sources Separately enabled, the sources imported are stored in a new reel, the Source Reel. The sequence is always imported in the Sequences Reel.
- Whenever a new Sequences reel is created, it is added as the last reel of the selected Reel group.
- In Conform view, if you enable the Save Sources option in the Conform tab, or the Save Sources Separately option in the Import options, the sources are saved in a Sources reel within the Reel group.

Scrubbing Clips, Sequences and Reels

Scrubbing a Clip or Sequence

You can scrub a clip or sequence by using the transport section under the reel or by dragging left or right on the lower third of a frame.



When scrubbing a clip, the selected clip outline disappears, giving you a cleaner interface. All clip information is hidden, except for the Current Frame / Timecode / Keycode and the File Format icon. This also applies to clips in the Media Panel (Tiles), the Freeform view and the MediaHub (Tiles and Preview Panel).

Scrubbing a Reel

Click:	To:
	Go to the previous clip or sequence on the reel.
	Go to the first frame of the clip or sequence currently over the reel's playback controls.
	Scrub the reel backward.
	Load the clip currently above the playback controls into the Player for full-resolution playback.
	Scrub the reel forward.

Click:	To:
	Go to the last frame of the clip or sequence currently over the reel's playback controls.
	Go to the next clip or sequence on the reel.

To scrub a reel using the cursor:

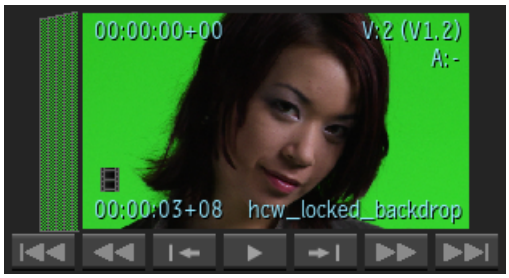
1. Click the grey area on top of the reel to be scrubbed.
2. Place the cursor in the top third of the reel.
   The cursor displays two white arrows on each side.
3. While holding the left mouse button, drag the cursor left or right.

NOTE The closer you are to the edges of the screen, the faster the scrubbing speed. The closer you are to the centre of the screen, the slower the scrubbing speed.

Displaying Collapsed, Frames and Storyboard Views

There are three ways to view clips on Desktop reels: collapsed view, frames view, and storyboard view.

In collapsed view, clips or sequences are displayed as a stack of frames.



In frames view, clips or sequences are displayed as a strip of contiguous frames.

In storyboard view, each segment within your sequence is represented as a frame. Any transitions or cuts between segments are represented as dotted green lines and full yellow lines respectively.

To toggle the view for one clip or sequence:

1. Move the cursor over any frame of an expanded clip or sequence.
2. Do one of the following:

- To toggle between collapsed view and frames view, press `C` repeatedly to cycle the two views.
- To toggle between collapsed view and storyboard view, press `Space` + `C` repeatedly to cycle the two views.

To toggle the view for all clips or sequences on a reel:

1. Position the cursor over a grey area between clips or sequences on a reel.
2. Do one of the following:

- To toggle between collapsed view and frames view, press `C` repeatedly to cycle the two views.
- To toggle between collapsed view and storyboard view, press `Space` + `C` repeatedly to cycle the two views.

To toggle the view for all clips or sequences on the Desktop:

1. Position the cursor outside the Desktop.
2. Do one of the following:

- To toggle between collapsed view and frames view, press `C` repeatedly to cycle the two views.
- To toggle between collapsed view and storyboard view, press `Space` + `C` repeatedly to cycle the two views.

Finding a Clip in a Reel

When there are many clips on a reel, locating a specific clip can be time-consuming. To quickly find a clip, perform a Find in Reels operation. Find in Reels automatically centers the selected clip on its reel, eliminating the need to scroll.

To find a clip in a reel:

1. Select the clip you want to locate from the Desktop section of the Media panel.
2. From the contextual menu, select Find in Reels.

The selected clip is centered on its reel.

Alternatively, select a clip and use the keyboard shortcut `F`.

Syncing Reels

You can sync clips and sequences on different reels together on the Desktop so that when you jog one, they are all jogged. This is useful for multicam setups. When you play a clip that is synced, all other synced clips are played.

To sync reels:

1. Align the frames of the clips you want to sync.
2. Click the Sync button on each reel with a clip to sync.

The selected reels are synced.

Changing the View of the Flame Workspace

The Viewing panel offers different visual representations of the clips in the Media panel.

From the View mode box, select from the following views:

- **Desktop Reels**: Displays reels where clips and sequences are placed. Clips and sequences can be edited and played back from the reels.
- **Freeform**: Displays the clips within the selected Media panel folder or reel as thumbnails.
- **Player**: Displays and plays back the selected clip in a Player.
- **Source - Sequence**: Displays and plays back the selected source clip and sequence in two side-by-side players.
- **Triptych Player**: Displays and plays back one or multiple clips in three side-by-side players. The triptych player can be useful for such operations as colour matching, for example.
- **Trim View**: Displays the last (outgoing) and first (incoming) frame above the Timeline from the two clips be trimmed.

TIP Double-clicking a thumbnail in the Media panel displays the clip in the Player.

Generating Clips

The following generic video and audio clips can be generated on a reel:

- Colour Source
- Colour Bars
- Noise
- Gradient
- Audio Tone

To generate a clip:

1. From the Media panel or Viewing panel, right-click a reel and select New.
2. Select the type of clip to generate.
3. Set the options in the dialog box that appears.
4. Click Create.

Once created, a clip is displayed as a source in the timeline, the Media panel and the Viewing panel.

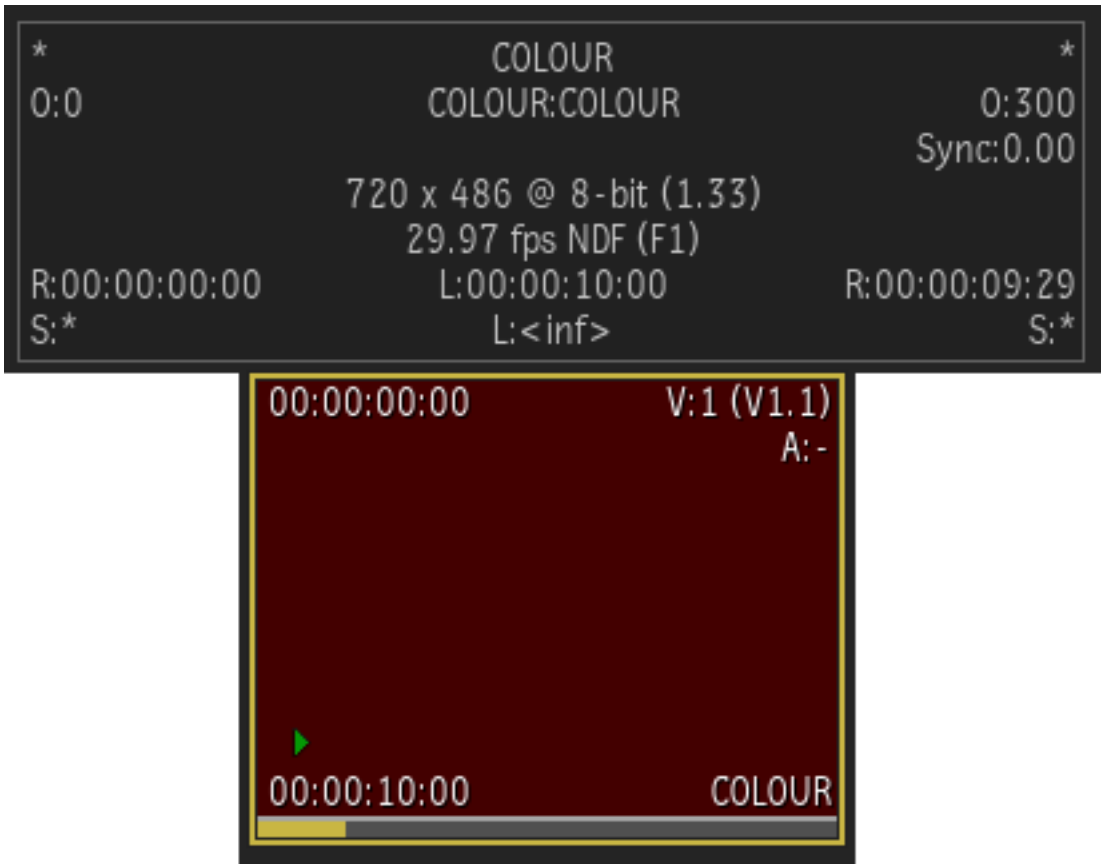
Displaying Media Metadata

By default, basic metadata such as the current timecode, video and audio tracks, duration and clip or sequence name is displayed on each corner of a thumbnail. More detailed metadata for each clip or sequence can be viewed directly on the thumbnail clip in the Viewing and Media panels.

To quickly view detailed clip information:

- In the Viewing or Media panels, position the cursor over a thumbnail.
- While holding the `Alt` key, click the thumbnail.

  Detailed clip information appears in a pop-up until the `Alt` key is released.



Clip information for multiple clips is also be displayed in a table in the Media panel for more thorough perusal. To view the complete table:

1. Click the Timeline tab.
2. From the View mode box, select List.
3. Expand the reels containing the clips of interest.
4. From the Media Panel View mode box, select Full Width.

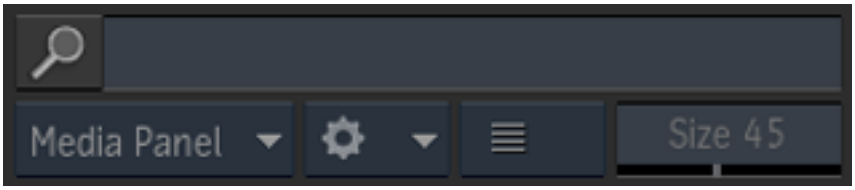
Metadata for each clip is displayed in a table.

Managing Sequences

A sequence is an edited clip that can contain multiple sources. Whenever a clip is edited, it becomes a sequence. When a new project is created, an empty sequence is automatically created within the project. New sequences can also be created from the Media panel.

Creating an Empty Sequence

1 From the Media Panel Gear menu, select **New ➤ Sequence**.



The New Sequence dialog appears.

2 Specify the settings for the sequence.

3 Press Create.

An empty sequence is created. It is displayed in the Timeline, the Media panel, and the Viewing panel.

Opening a Source as a Sequence

1 Select a source from the Viewing panel or the Media panel.

2 From the contextual menu, select Open as Sequence.

A new sequence is created from the selected source. It is displayed in the Timeline, the Media panel, and the Viewing panel.

NOTE When opening a source as a sequence, the sequence is created using the original media. When a source is added to an existing sequence, a copy is added, leaving the original source intact.

Opening Multiple Sequences Simultaneously

1 Select reels, folders or libraries in the Viewing panel or the Media panel.

2 From the contextual menu, select Open All as Sequences.

All the clips and sequences within the reel, folder or library are opened as individual sequences.

TIP This can also be achieved by dragging the reel, folder or library to the timeline.

Opening Multiple Clips as One Sequence

1 Create an empty sequence.

2 Select the clips to load and join as one sequence from the Media panel or the Viewing panel.

3 Drag the selected clips to the empty sequence at the desired timecode.

The selected clips are inserted in the sequence, in the order in which they were selected.

Closing all Sequences Simultaneously

1 Mouse over an open sequence tab in the Editing panel.

2 From the contextual menu, select Close All Sequences.

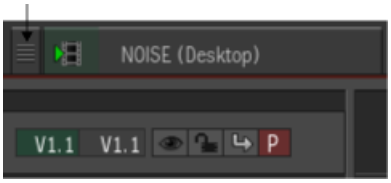
TIP This can also be achieved with the `Meta + C` keyboard shortcut.

Moving all Open Sequences to a New Reel or a New Folder

1. Do one of the following:
   - To move all open sequences to a new reel, in the Media panel, select the reel **above** which the new reel is to be created.
   - To move all open sequences to a new folder, in the Media panel, select the Library or folder **under** which the new folder is to be created. If a folder is selected, a sub-folder is created.
2. Mouse over an open sequence tab in the Editing panel.
3. From the contextual menu, select Move All Sequences.

   The open sequences are moved from their original location to the new reel or folder.

   TIP This can also be achieved by dragging the sequence handle to the left of the left-most tab in the timeline and dropping it on a reel on the Desktop or in a library. A prompt asks whether to move or copy the sequences to the new location.



Copying all Open Sequences to a New Reel or a New Folder

1. Do one of the following:
   - To copy all open sequences to a new reel, in the Media panel, select the reel **above** which the new reel is to be created.
   - To copy all open sequences to a new folder, in the Media panel, select the library or folder **under** which the new folder is to be created. If a folder is selected, a sub-folder is created.
2. Mouse over an open sequence tab in the Editing panel.
3. From the contextual menu, select Copy All Sequences.

   The open sequences are copied to the new reel or folder.

   TIP This can also be achieved by dragging the sequence handle to the left of the left-most tab in the timeline and dropping it on a reel on the Desktop or in a library. A prompt asks whether to move or copy the sequences to the new location.

Working in Freeform View

Freeform view displays the clips in the selected folder or reel as thumbnails.

Arranging the Thumbnails

Freeform view features a number of display options. To arrange the thumbnails, do one of the following:

From the contextual menu, select:	To:
Arrange ➤ Clean Up All	Tile the thumbnails across the Viewing panel.

From the contextual menu, select:	To:
Arrange ➤ Fit All	Tile the thumbnails across the Viewing panel at the highest possible resolution while fitting all the thumbnails within the Viewing panel.
Arrange ➤ Cascade Selection	Arrange the selected thumbnails as cascading stacks.

Displaying a Snap Grid

Use the grid to snap thumbnails into position when moved.

To display a snap grid in the Freeform view:

1. Make sure the Viewing panel is set to Freeform view.
2. In **Preferences ➤ User Interface ➤ Freeform View**, enable Snap To Grid.
3. Set the desired grid size.
4. Set the desired thumbnail height.
5. Click Close to close the Preferences dialog box.
   The Viewing panel displays a grid of the specified size.

Ganging Clips in the Viewing Panel

You can gang clips and sequences together in the Viewing panel so that when you jog one, they are all jogged. This is useful for multicam setups. When you play a clip that is part of a gang, the current frame of all other ganged clips is updated once playback has stopped.

To gang clips in the Viewing panel:

1. From the View Mode box, select Thumbnail view.
2. In the Viewing panel, locate all the clips or sequences you want to gang, and move their positioners to the timecode you want to lock.

   TIP Each clip or sequence can have its own timecode offset, but if you are working with a multicam setup, make sure all positioners are parked at the same location.

3. Hold the `Ctrl` key and select the clips or sequences you want to gang.
4. While the cursor is over one of the clips to be ganged, from the contextual menu, select Gang.
   All selected clips are ganged and the clip information turns green.

   NOTE The clip information of ganged clips turns yellow when a non-ganged clip is selected.

5. Jog the positioner of one of the ganged clips.
   All ganged clips or sequences are jogged.

To select all clips in a gang:

1. Select any clip or sequence that is part of a gang.
2. From the contextual menu, select **Gang ➤ Select Gang**.
   All ganged clips or sequences are selected.

To remove a clip from a gang:

1. Select the clip(s) or sequence(s) you want to remove from the group.
2. From the contextual menu, select **Gang ➤ Ungang**.
   All selected clips or sequences are unganged.

Undo and Redo

To the right of the tabs are the Undo and Redo boxes.

- **Undo box**: Click to undo the last operation. Alternatively, click the white arrow to list the last 10 operations performed. Selecting an operation from the list undoes all operations performed after the selected operation (including the selected operation).
- **Redo box**: Click to redo the last operation. Alternatively, click the white arrow to list the last 10 undone operations. Selecting one reapplies all undone operations performed before the selected operation.

The MediaHub, Conform, Timeline, Batch, and Tools tabs all share the same undo/redo stack, and are automatically switched according to the context of the last undo/redo operation. Commands such as Replace Desktop, Clear Batch, and Clear Desktop may also be undone.

The Undo levels may be set anywhere between 2 and 100 under the General tab in the Preferences. The default is 50.

NOTE While in Batch FX, all undo levels are accessible. Upon exiting Batch FX and returning to the timeline, all operations performed in Batch FX are considered one undo level. This means that even if seven operations were performed in Batch FX, the undo affects all Batch FX operations at once.

Importing and Exporting Media 7

Importing File-Based Media

To import media using the MediaHub:

1. Click the MediaHub tab.
2. Using the MediaHub file browser, navigate to the file to import.
3. Drag and drop the file to the Media Panel.
   The file is now imported.

To import media directly from the Desktop (Linux desktop or Finder on Mac OS X):

1. From Flame, switch to the Desktop.
2. Navigate to the file to import.
3. Drag the file from the Desktop and drop the file in Flame Media panel.
   MediaHub does not need to be displayed for this to work.

NOTE In .mov files where both audio and video tracks are present, it can happen that one track is shorter than the other one. To preserve the integrity of the source, if the audio track is shorter than the video one, silence is added to fill the gap. If the video track is shorter, No Media slates are added instead.

Modifying the Options After Importing a Clip

You can access format and import settings of an imported clip directly on the timeline. In the Editing panel, these options are represented in the Timeline FX pipeline, prior to the application of any Timeline FX. These options can be edited for the selected clip, and copied and pasted to other segments that use the same clip format.

1. Click the Timeline tab.
2. Select the clip to display it in the timeline to display its Timeline FX pipeline. You can also select a segment in a sequence.

   You do not need to open the clip as a sequence, but it cannot be locked. **Right-click ➤ Unlock** unlocks a locked clip. The same thing applies to clips in a Library with **Preferences ➤ General ➤ Protect from Editing** enabled: you need to open the clip from a reel.

3 In the Timeline FX pipeline, do one of the following:

NOTE Format and import options are accessible in the same editor. You can open the editor once, then switch between menus in the editor to edit different option types.

- Select the Format Options button in the Timeline FX pipeline, and click Editor in the quick menu. The Basic menu opens. Format options will be saved to this menu.
- Select the Pre-Processing button in the Timeline FX pipeline, and click Editor in the quick menu. The Resize menu opens. Import options will be saved to this menu and the RGB LUT menu.

4 Edit the settings as required, and then click Exit to return to the Timeline.

multi-channels import

Multi-channel OpenEXR files with multiple alpha channels / matte are imported as Matte Containers with all available mattes.

Importing File-Based Media Tips

- Drag and drop a folder in the Media panel from the MediaHub. Flame imports all the media files and folders contained therein. Note that only supported media files and the folder structure are imported: other files are ignored.
- Drag and drop multiple files in one operation: `Ctrl`-click or `Shift`-click to select multiple files to import before dragging them over to the Media panel.
- Double-click a clip to display it in the Preview panel. Use the Preview panel to display the clip information and additional metadata.
- For large media, use the Preview panel to set In and Out points and import only a subclip.
- If the media file to import is located on a network drive, and if you plan on using referenced media instead of cached sources, make sure that the network connection is at least 1 GB ethernet to have decent playback.
- MediaHub does not recognize media files without file extensions, as is it sometimes used with mov, dpx, or cin files.
- Flame is frame-based, which means that any audio-clip that does not end on a frame is padded with silence until the next frame. When this happens, the audio-only clip appears as Mixed in the Cached column of the Media Panel's list view, even if the clip was never cached.
- If the media file to import is located on a removable media such as a USB drive, and you plan to remove the drive before the end of your project, import with Cache Source Media enabled. This way Flame creates natively managed media out of the original, removing the need for the connected drive.
- From the Media panel, right-click > Import... to import media to that location.
- To work in a manner similar to offline editing suites, enable **MediaHub ➤ General tab ➤ Cache Source Media**. This creates local, transcoded, and managed versions of your media. To work online, disable Cache Source Media: the clips remain linked to the original media, and are not transcoded.
- The first time you browse a folder containing Long GOP based codecs (.mts- and .m2ts-structures for AVCHD), Flame creates invisible index files in that folder. These index files will speed up browsing the next time you open that folder.

  NOTE Technically, the index files are created with the suffix *.index* in the folder being browsed and can be removed if needed: this will only impact AVCHD browsing performances in that folder, not reading nor writing performances. If that folder is write-protected, Flame creates the index files in the local */var/tmp/*.

Long GOP Optimization

Flame optimizes the decoding of Long GOP (Group Of Pictures) codecs to facilitate playback, jogging and shuttling of clips the following codecs:

- Sony XDCAM
- Sony XAVC Long GOP
- Panasonic Long GOP
- Canon MPEG 2
- Canon XF-AVC
- AVCHD
- QuickTime Long GOP

The required optimizations are only available when importing from the Local Devices list from the MediaHub. Importing media from the Autodesk Network list of volumes can result in sub-par Long GOP decoding performances.

NOTE Lustre does not have access to this kind of Long GOP-read optimization.

About the Tape Name

Some media file formats have provisions on how to determine the tape name. For those that do not, MediaHub derives the Tape name from the file name. Here are the various formats rules. You can override these rules by specifying your own method in MediaHub, **Format Specific Options ➤ Metadata ➤ Tape Name box**.

Audio Files:

- Wave, AIFF, MP3, etc.: Uses the filename.

File Sequence:

- DPX: Uses the information stored in the file header (the *Input Device* data is used); falls back to filename if the header contains no tape name.
- OpenEXR: Uses the name of the directory where the OpenEXR is stored.
- ARRIRAW: Uses the information stored in the file header; falls back to filename if the header contains no tape name.
- Image Sequence (TIFF, TGA, JPEG, HDR, PNG, etc.): Uses the Tape Name entered in **Format Specific Options ➤ Metadata ➤ Tape Name field**.
- PSD: Uses the filename.

Movie Files:

- MTS (AVCHD): Uses the filename.
- REDCode (R3D): Uses the information stored in the file header.
- Canon (MXF): Uses the filename.
- Panasonic (MXF): Uses the name of the essence.
- Sony XDCAM EX (MP4): Uses the name of the essence.

  In case of spanned clips, set **Format Specific Options ➤ Metadata ➤ Tape Name box** to Tape Name from File Name. Leaving it to the default setting of Tape Name from Essence will actually use the name of the span segment for tape name.
- Sony RAW/SStP/XAVC (MXF): Uses the name of the essence.

The tape name is derived from the MXF Source Package that is presented as <TAPE>; falls back to the filename if empty.

- MP4/MXF/QuickTime: Uses information stored in the file heade; falls back to filename if the header contains no tape name.

About Multi-channel Clips

For Flame, a multi-channel file is any media file that contains more than just an RGB and an alpha channel. Flame supports the following multi-channel file formats: OpenEXR (.exr), Photoshop (.psd) and Open Clip (.clip). Each one of these file formats manages multiple channels of media in a single file.

In previous versions, a multi-channel file would be imported as a Matte Container where each channel was itself wrapped in a Matte Container. The only way to import and keep the multi-channel structure of a clip was through Batch's Read File node. The Read File node kept the multi-channel structure but at the cost of not being able to archive, cache, proxy, or wire the media.

NOTE Multi-Channel clips from Flame are not seen as multi-channel in Lustre: only the primary RGB channel is available through Wiretap. Use the Matte Container for any Flame to Lustre workflow involving Multi-Channel clips.

Importing Multi-Channel Clips Options

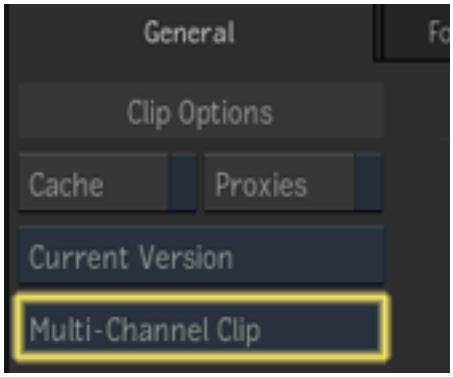
Multi-channel media is displayed in the MediaHub with the **Multi** icon.

Double-click the **Multi** icon to expand the channels that make up the clip. You can then decide to import the individual channels. A channel can be an audio track, an alpha channel, an alternate matte channels, or any sort of CGI render pass.

The MediaHub offers the following options to import multi-channel clips in the Multi Channel Processing box:

Matte Container (Full): This option lets you create a Matte Container out of a multi-channel file. This Matte Container includes every available channel, instead of only the Alpha Channel (or Matte) channel. This option is useful when importing content generated by 3D rendered or Desktop Compositing applications which contains RGBA matte or alpha. Use this option to conform in Flame material that will then be used in Lustre for Color Grading. In this case, the sequence provided to Lustre will contains all alternative channels so colorist can use these channels for secondary color correction.

Multi-Channel Clip: This option imports a multi-channel file (such as OpenEXR) as a multi-channel clip. This one clip contains every available channel from the file. This is the option to access the different channels in Batch or Batch FX. Overall, a multi-channel behaves like any other with a few exceptions related to TimelineFX and Tools. These limitations are also described below.



The other import options (Ignore Alpha Channel, Create Matte Container, Matte Container with Com, Include Alpha Clip) are always available to multi-channel clips, but splits the multiple channels into individual clips.

Working with Multi-channel Clips

Where you are in the Flame defines what you can do with a multi-channel clip. As in MediaHub, the **Multi** icon identifies multi-channel clips, although you cannot expand the channels.

In Conform Multi-Channel clips, even when containing more than just the classic RGBA channels, are seen in Flame as source clips that you can use for Conform. Make sure that the clips to be used as sources are imported with the **Multi-Channel Processing** box set to **Multi-Channel Clip**.

In the Timeline: With Timeline FX When you work with Multi-Channel clips in Timeline, only the main RGB and Alpha channels are available to the Timeline FX. Other channels are not accessible.

In the Timeline: With Batch FX If you add a Batch FX using Add Adjustment Segment option, you lose access to the channels of a multi-channel clip: the clip is converted to a back clip. If you add a Batch FX using any other option (including without any option selected), the multiple channels of that clip are available in Batch FX. But adding a TimelineFX on a multi-channel clip from the Timeline tab within BFX hides the alternate channels. Use Batch FX nodes to add effects if you want to have access to every channel.

In Tools Using a multi-channel clip in most Tools produces single channel clips in which neither the alpha nor the other channels are available in the Tools' results), with the exceptions of the following:

- Stereo 3D clip
- Create Matte Container
- Audio Mixdown
- Convert Rate
- Consolidate
- Change Keycode
- Change Timecode
- Copy/Duplicate
- Move/Delete
- Reformat

Import Node in Batch or Batch FX

NOTE In versions prior to 2017 Extension 1, you would import multi-channel clips in Batch or Batch FX using the Read File node. This node is now retired, but remains available for compatible with old setups.

In Batch or Batch FX, the Import node fully supports multi-channel clips.

- The Import node creates clips in the current Batch Group Schematic reel. These clips can then be archived, or transferred using the Projects tab in MediaHub like any other clips.
- Both Media Caching and Proxy Generation are available to the Import node, whether at import or through the contextual menu on the clip (in the Schematic Reels or the Desktop Reels.)
- An import node is like any other clip node in the Batch or Batch FXschematic. You can replace the media of a selected Import node with a **Shift-Double-click** (to select a clip from the Desktop) or with a **Shift-Alt-double-click** (to open MediaHub to import again). Existing connections are preserved for the channels named identically in the replacing file.

NOTE The Import node is one method to add media to Batch or Batch FX. Using the Desk and Media Panel nodes, as well as dragging a clip from the Media Panel are different methods to add a clip to the process tree. And all these methods result in the same clip node.

Channel Rules for Colour Management

The different rendering softwares available on today's market do not use standardized names to name the channels that they output. Even if they did, facilities often have in place media management systems that rename channels to fit internal standards.

CG renderers deliver render passes named according to their naming convention. But for maximum flexibility, renderers also allow renaming the passes to whatever is required by the user.

In Flame, colour management of render passes is resolved using the names of these render passes. Flame recognizes the default naming convention for the following renders:

- Arnold
- Mental Ray
- V Ray
- RenderMan

But if you receive multi-channel files with customized render pass names, you can edit a configuration file that allows Flame to translate these names into standardized ones. This translation ensures the renders are properly colour managed.

The translation file is named `channelrules.cfg`, and is found at `/opt/Autodesk/cfg/channelrules.cfg`. In it you can specify equivalents for the following channels.

- 3D Motion
- AO
- Beauty
- Matte
- Motion
- Normals
- Position
- Roughness
- Shadows
- UV
- Z-Depth

Non listed passes are seen as unknown in Flame.

These channel types are displayed in Batch, when you open the Extended menu for a clip node. They can all be assigned a colour space according to the defined colour management rules.

TIP In a collaborative environment, it is a good idea to share the `channelrules.cfg` file with other workstations. Consider creating a symbolic link (alias) to the file stored in a shared location.

Exporting Multi-channel Clips

Multi-Channel can be exported as RGBA media. Alternate channels are discarded.

Creating Multi-channel Clips

It is not possible to create Multi-Channel clips from within Flame.

But there is no obligation to use OpenEXR or Photoshop media files for Multi-Channel work. Any format can be used but since not all format allows embedding multi-channel content in single media file, one could create Open Clip to reference content that can be used as multi-channel clips in Flame Family. A good use case is when generating CGI assets. Single channel media files can be rendered out from CG applications as individual sequences or movie file and using the Open Clip XML specification, one could build an Open Clip that would reference these single pass sequences to be seen and used as Multi-Channel clips. The advantage of this workflow are the following:

- If a pass as to be regenerated, not need to recreate all channels
- Any file format can be used

Open Clip's versioning capability can be leverage to ease revision and content update. See the Open Clip samples and specifications for additional details.

Using WiretapCentral

About WiretapCentral

WiretapCentral is a flash-based application for remotely browsing, reading, and writing media using a standard web browser. WiretapCentral is installed by default on Autodesk workstations.

Use WiretapCentral to:

- Browse projects and material located on a creative finishing workstation or other system, and manage clip libraries, reels and directories.
- Encode media into a number of file formats including H264, Mpeg1, Mpeg2, Mpeg4, QuickTime, DV PAL, DV NTSC, and FLV at various resolutions.

Main Interface

The WiretapCentral main interface is comprised of three areas:

- The Network Tree area for navigating to connected systems and storage.
- The Main View area for viewing folders and their contents.
- The Details area, which shows information about the currently-selected node or clip.

Use these areas to navigate to media, and to select and apply actions to clips.

Menu Bar

The Menu bar offers the following options:

Click:	To:
Selection	Perform any of the following options on clips: Refresh All, Refresh Selected, Expand Children, and Select All.
Layout	Change the interface layout. The options are Normal, Extended, and Full Screen.
View	Change the current view. The options are Thumbnails, List, and Player.
Bookmarks	Create, manage, and navigate bookmarked media locations.
Tools	Open Backburner monitor, create new reels, libraries, and directories, as well as delete selected items.
Export	Export selected media or the entire list, or view packages or presets.
Help	Open application help.

Player

Use the player to view clips and play them back with more options. The player shows all clips at a width of 720 pixels, normally at 100% JPEG quality. In addition, the player offers controls for stepping through the action, looping, and playing backward.

Double-clicking a clip opens the player. WiretapCentral automatically generates and downloads the high-resolution version from the clip before streaming a playable version. For long clips, this may take a few seconds depending on the network interface and processing power of the WiretapCentral server. If the clip has an NTSC or PAL proxy on the volume, the process is often close to real-time. As with thumbnails, the clip begins playing as soon as it begins downloading.

Clips can also be panned and zoomed in WiretapCentral even while playing.

To pan and zoom:

- To pan the clip, drag it using the mouse.
- To zoom in and out, click the [+] and [-] buttons or use the wheel on the mouse.
- To reset the player's default pan and zoom settings, click Home.
- To fit the clip to the available player canvas, click Fit.

Export

After selecting media and choosing Export from the Menu bar, use the Export dialog to specify input and output settings. Settings may be saved as presets and reloaded using the Presets button.

If any media has not been rendered or cannot play back in realtime, black frames and empty audio will be generated. When Wiretap Central detects this before starting an export, a warning will appear, asking the user to cancel or continue.

In addition, if any source or rendered clips are moved, modified or altered, the export will fail.

Exporting Clips and Sequences to Files

To export a clip or a sequence:

1. Do one of the following:
   - Right-click the clip to export and select Export. And then, using the Media Export window, navigate to the clip's destination.
   - From the MediaHub, drag-and-drop the file from the Media panel to the location displayed in the browser.
2. Select an Export type and a Format Preset.
3. Click Export.

   Flame prepares the export job. Once that preparation is done, the rest of export happens in the background and frees up Flame for your use.

To export an RGBA movie file or file sequence, create a Matte Container prior to exporting.

- Use the Create Matte Container option in the MediaHub. The resulting clip will be a Matte Container and the Comp effect will be disabled.
- Use the Create Matte Container Tools and select the Contain or the Contain & Add Comp option.
- Use the Add Matte option in the Timeline FX ribbon to create a Matte Container.

  To be able to export an RGBA media file, make sure the Comp effect is muted before exporting.

Tips

- Muted tracks are not exported.
- You do not have to render your whole sequence before exporting it: Flame automatically renders only the media required for the export.
- Minimize the space taken on your storage by using **Export in Foreground**.

  For File Sequence and Movie, foreground export lets you export media without requiring you to re-render. This options make so Flame does not need to allocate frames on the framestore because content is streamed directly to the exported file, reducing the time to export.

  For Sequence Publish exports, when the media needs to be packaged, Flame renders before exporting for the segments that require to be pre-rendered. For example, using the Flattened tracks option with Timeline FX requires rendering prior exporting, as do segments with a BFX or Resize.
- Use `Ctrl-click` to select multiple clips for export in one operation.
- Select a folder, a reel, or a library from the Media panel to export all of its contents. If that folder contains a folder structure, that structure also gets exported.
- When exporting with **Export Between Marks**, you can have pre-roll and post-roll black by setting the in and out marks to include some empty track. Each frame of included empty track creates a frame of black on the export.
- When exporting a single frame as a file sequence, consider setting the Frame Padding field to zero to export that frame without any file numbering. And if you export a multi-frame clip as a file sequence by mistake without Frame padding, Flame detects this and automatically assigns a frame padding of 8 digits to correctly export the clip.
- Apple Final Cut Pro X XML Export

  When you open the XML in FCP X, the media will automatically relink if exported to a file location available on the FCP X workstation. You might have to use the FCP X Relink tool to link the XML sequence back to the QuickTime if the media file path is different.

- When working with CODEX files, you cannot use the Link Original Media option in Media Export to create file system links to the original media files because Flame cannot generate 12-bit packed DPX, the format used by CODEX recorders. Flame will instead create new 12-bit unpacked DPX files.
- OpenEXR exports include Tape name and Keycode data in the exported files' header.
- DPX metadata
  - The frame rate is written to the DPX file header, in both the TV rate and Movie rate fields.
  - The Drop Frame mode of 29.97 fps files is written to the DPX file header. Limitations from the DPX SMPTE-specifications prevent the drop frame mode from being specified in files using a timecode higher than 30 fps.
- In ProRes exports, Flame corrects the aspect ratio to match the Apple-defined standard. For PAL 16x9, the standard pixel aspect ratio is 118:81. For a frame of 720x576 this gives a an aspect ratio of ~1.82 = (720 / 576) x (118 / 81).
- MXF DVPro: a PAL export is always F2; a NTSC export is always F1. Fields are translated if required.
- When exporting, use Add Token to build a dynamic file name. Here are some of the available tokens.

Select:	To insert the token:	Definition:
Height	<height>	The clip's height, after resize if applicable.
Width	<width>	The clip's width, after resize if applicable.
Date	<date>	The current date (YYYY_MM_DD)
Event Number	<event number>	A number usually defined in a EDL, but that can be manually set in Conform.
Project	<project>	The name of the project, as displayed in **Flame ➤ Project and User Settings**.
Project Nickname	<project nickname>	The name of the project, as displayed in **Flame ➤ Project and User Settings ➤ Project Edit**.
Segment Index		A sequential index (from 1) indicating the relative position of each segment on the timeline; gaps are not indexed unless they have a Timeline FX applied.
Segment Name	<segment name>	User-defined name of the timeline segment.
Time	<time>	The time, formatted HH:MM:SS.
User	<user>	The user name, as displayed in **Flame ➤ Project and User Settings**.
User Nickname	<user nickname>	The user name, as displayed in **Flame ➤ Project and User Settings ➤ User Edit**.
Version ID	<version>	Number of the current version, as defined in the Clip Options tab.

Select:	To insert the token:	Definition:
Version Name	<version name>	Version Name from the Clip Options tab.
Workstation	<workstation>	The name of the workstation, as displayed in the Host Computer field in **Flame ➤ Project and User Settings**.

NOTE When exporting an Open Clip with Clip Version enabled, make sure the frame rate remains the same between versions: having differing frame rates within one Open Clip is not supported by the format.

Support for MXF XDCAM 422 Short GOP

MXF XDCAM 422 exported by Flame are encoded based on Sony's XDCAM format, and all start with a full Group of Pictures (GOP). Some broadcasters use a different flavour of XDCAM 422, where the media start with a short GOP (starting on an I-frame). You can meet these encoding requirements by setting an environment variable. You can configure either or both Background and Foreground exports to use this alternate encoding.

For Foreground Export, edit the application's `.cshrc`.

1. Close Flame.
2. In a shell, type: `cd /opt/Autodesk/<application>`
3. With a text editor, edit the file `.cshrc`
4. Add the following ligne to the .chsrc:

```
setenv DL_MXF_MPEG2_I_FRAME_FIRST 1
```

5. Save the file and restart Flame.

   Encoded MXF XDCAM 422 will now be encoded starting on a I frame. To return to the previous setting, delete the line `setenv DL_MXF_MPEG2_I_FRAME_FIRST 1` from the .cshrc.

For Background Export, you need to configure the Backburner Server process.

- **On Mac OS:**
  - Open the file `/Library/LaunchDaemons/com.autodesk.backburner_server.plist`
  - Add the following lines following the first `<dict>` line:

```
<key>EnvironmentVariables</key>
<dict>
<key>DL_MXF_MPEG2_I_FRAME_FIRST</key>
<string>1</string>
</dict>
```

  - Restart the Backburner Server service using the Service Monitor.

- **On Linux:**
  - Open the file `/etc/init.d/backburner_server`
  - Add the following line after the initial comments:

```
export DL_MXF_MPEG2_I_FRAME_FIRST=1
```

  - Restart the Backburner Server service using the Service Monitor.

Publishing an EDL

To publish a sequence as an EDL, and export its media:

1. Right-click the clip to export and select Export.

   In the MediaHub, you can also drag-and-drop the file from the Media panel to a location displayed in the MediaHub browser.
2. Navigate to the clip's destination, using the Media Export window.
3. Set Export Type to Sequence Publish.
4. Set Format Preset to one of the EDL Publish presets.

   NOTE Because of the limitations of the format used for the EDL (CMX3600), the published sequence is flattened to 1 video track and 8 audio tracks.
5. Click Export.

   Flame prepares the export job. Once that preparation is done, Flame performs the export in the background. You are free to use the application as the export happens.

The standard EDL published by Flame uses the CMX3600-DLEDL format, using DLEDL extensions if required. But if you require a format other than CMX3600, or need to change how events are combined, use the EDL editor, customizing the EDL as required.

NOTE A customized EDL is always published without its media.

To publish a customized EDL:

1. Right-click the sequence to export and select Export.
2. Set Export Type to Sequence Publish.
3. Set Format Preset to one of the EDL presets.
4. Click Show Advanced Options.
5. Click **Sequence Options ➤ Custom EDL Export**.

   The EDL editor appears.
6. Set the EDL Save options as required.
7. Click Generate, and then click Save Generated EDL.
8. In the file browser that appears, browse to the location where you want to export the EDL.
9. Click Save. This saves the EDL and returns you to the EDL Editor.
10. Exit the EDL Editor.

EDL Publishing Remarks

- Animated timewarps are converted into constant timewarps when written to an EDL.

  The Source In and Source Out written to the outgoing EDL match the span of the media used in the original animated timewarp. The constant timewarp speed in the outgoing EDL is calculated so that the media fits the segment's duration.
- When you export a sequence as an EDL, tape names with illegal characters are written to the EDL using the Tape Name Extension.

  The Tape Name Extension stores both the original name and a sanitized version, allowing third party applications to import and correctly conform the EDL, displaying the sanitized name. And when re-importing in an Autodesk application, the EDL displays the original tape names.

How the tape name is sanitized:

- Spaces are replaced with underscores; TAPE 1 becomes TAPE_1
- Illegal Unix characters are replaced with underscores; TAPE$1 becomes TAPE_1
- Lowercase characters are replaced with uppercase characters; tape1 becomes TAPE1

To publish an EDL with long tape names without relying on DLEDL extensions:

1. Follow the steps to publish a customized EDL.
2. Set the Save as menu to CMX 3600.
3. Enable Use Long Tape Name.

   NOTE Tape names are still sanitized according to the rules listed above.

Custom EDL Export Window Reference

Generate button Generates the new EDL and activates the Save Generated EDL button.

Save Generated EDL button Opens a file browser for you to select a location where to save the EDL.

Specify a file name and path for the saved EDL in the file browser that appears. The filename cannot contain any of the following characters: ` # ~ @ $ % ^ & * ( ) [ ] { } < > \ | / ! ? , ; : ' "

EDL Event Combination box Indicates how events with the same source timecodes, record timecodes, and tape ID are combined when the EDL is generated.

Select:	To use:
Combine All Events	A single entry for all video and audio events.
Combine Audio Events	One entry for audio events and a separate entry for video events.
Never Combine Events	A separate entry for each video and audio event.

EDL Format box Select the format of the generated EDL.

You can save EDLs in any of the following formats:

- CMX 340
- CMX 3600
- CMX OMNI
- GVG 4
- GVG 4 Plus (GVG v4.1 or higher)
- SONY 900
- SONY 910
- SONY 5000
- SONY 9000
- SONY 9000 Plus (v2.21or higher)
- SONY 9100

Segment Comments button Enable to allow comments added to the timeline to be included in the generated EDL.

Clip Name Comments button Enable to allow clip name comments to be included in the generated EDL.

2:3 Insertion Mode button Enable to convert the frame rate of a 24p clip from 23.97 fps to 29.97 fps, and maintain 2:3 pulldown information for all in and out points (including cuts, wipes, dissolves, and timewarps). 2:3 pulldown data is important when master tapes are sent out for hardware-based tape-to-tape colour correction.

Frame Code Mode box Select the drop frame mode for the output material: DF (drop frame) or NDF (non-drop frame).

The EDL file will include explicit notification of hybrid splices as punctuation marks in the record in and out data.

A:	Indicates a:
period (.)	Regular splice record-in point for 29.97 fps non-drop frame timecode tapes.
comma (,)	Regular splice record-in point for 29.97 fps drop frame timecode tapes.
colon (:)	Hybrid splice record-in point for 29.97 fps non-drop frame timecode tapes.
semi-colon (;)	Hybrid splice record-in point for 29.97 fps drop frame timecode tapes.

This button is enabled by default when a 24p template is selected at project creation.

Use Delayed Dissolves button Enable to include delayed dissolves in the generated EDL.

Default Tape field Enter a tape name if you want to override the default tape ID when saving an EDL.

Source clips are assigned tape IDs when loaded using the Clip VTR Input or through the Conform tab. For example, an edit that uses a clip created with the Colour Corrector does not have a tape ID. When the EDL is generated, the clip is given the tape ID in the Default Tape field.

Audio Patch Comments button Enable to allow clip audio patching comments to be included in the generated EDL.

Use Tape Name Extension button Enable to support long tape names (more than 8 characters) through the use of DLEDL extensions. This option and Use Long Tape Name are mutually exclusive.

This adds a list to the end of the EDL that shows the relationship between a abbreviated tape name (8 characters, maximum) used in the EDL and the actual tape name (52 characters, maximum).

Use Long Tape Name button Enable to support long tape names (more than 8 characters) in-place, without truncation or DLEDL-style name extensions. This option and Use Tape Name Extension are mutually exclusive.

Publishing an AAF Sequence

Use AAF export when you want to share a sequence with third-party applications. The AAF Export generates a simplified sequence that can be used in third-party application for creative editorial, colour correction, media management, etc. You can have a metadata-only or a metadata and media files publish.

Exporting an AAF allows video editorial interchange between Autodesk products and third-party applications supporting the AAF protocol.

The exported AAF file can be imported into the following applications:

- Avid Media Composer (version 8 and later)
- Avid ProTools 12.2
- DaVinci Resolve (version 11 and later)
- Filmlight Baselight
- Flame Family applications
- Autodesk Smoke

Not all the information part of the sequence can be published in an AAF. Look at the Limitations section to generate an AAF compatible with what you want to accomplish.

To publish a sequence as an AAF and export its media:

1. Do one of the following:
   - Right-click the sequence to export and select Export. And then, using the Media Export window, navigate to the clip's destination.
   - From the MediaHub, drag-and-drop the file from the Media panel to the location displayed in the browser.
2. Set Export Type to Sequence Publish.
3. Set Format Preset to one of the AAF presets.
4. Click Export.

   Flame prepares the export job. Once that preparation is done, Flame performs the export in the background. You are free to use the application as the export happens.

Customizing AAF Publication

1. In Media Export, set Export to Sequence Publish.
2. Enable Show Advanced Options to access the detailed settings.
3. In the Sequence Options, set Format to AAF.

From the Sequence Options tab, you can export an AAF file that references the original media files or generate an XML file that will reference new media files.

To export only an AAF, without media:

1. Enable Include Video and Include Audio, as required.
2. Disable both Export Video Media and Export Audio Media.
3. Click Export.

   This creates an AAF file where the video and audio tracks reference the original media files. Also note that the Tracks and Transitions option is implicitly set to Keep All Tracks, and cannot be changed.

   NOTE This process is especially fast, since no media need to be rendered or written to disk.

To export an AAF and its media:

1. Enable Include Video and Include Audio, as required.
2. Enable Export Video Media and Export Audio Media, as required.
3. Select a Media option, either Use Original Media or Use Media with FX.

4 Select an option for Tracks and Transitions. This option defines the exported structure of the sequence:
   - Flatten Tracks: The sequence is converted to a single segment.
   - Flatten Tracks with Transitions: A multi-track sequence is converted to a single track, where the dissolve transitions are preserved. Other transitions are flattened.
   - Keep All Tracks: The sequence structure is preserved but non-dissolve transitions are converted to centered dissolve transitions.

AAF Presets

You can use one of the following Sequence Publish presets to export an AAF for a specific purpose.

AAF for Avid Media Composer (Sequence only) Export an AAF file referencing the original media files. The exported sequence shows all video and audio tracks. No media files are generated. See the Limitations section to ensure that the exported sequence is compatible with this preset.

AAF for Avid Media Composer (DNxHD 36 and 16-bit WAVE) This preset generates a multi-tracks video and audio AAF file along with 8-bit 1080p QuickTime (DNxHD36/45) referencing the media with effects. 16-bit audio WAVE files are also generated.

AAF for Avid Media Composer (DNxHD 175X and 24-bit WAVE) This preset generates a video and audio multi-tracks AAF file along with 10-bit 1080p QuickTime (DNxHD175X/185X/220X) referencing the media with effects. 24-bit audio WAVE files are also generated.

AAF for DaVinci Resolve DNxHD 175X and 24-bit WAVE) This preset generates a video and audio multi-tracks AAF file with 10-bit 1080p QuickTime (DNxHD175X/185X/220X) referencing the media with effects. 24-bit audio WAVE files are also generated. Audio tracks are flattened.

AAF for DaVinci Resolve for Source Grading (DNxHD 175X and 24-bit WAVE) This preset generates a video and audio multi-tracks AAF file with 10-bit 1080p QuickTime (DNxHD175X/185X/220X) using the original media. 24-bit audio WAVE files are also generated, but the audio tracks are flattened.

AAF for DaVinci Resolve for Source Grading (Sequence only) This preset generates a video-only multi-track AAF file referencing the source media with effects. No media files are generated.

AAF for Avid Pro Tools (Audio only 24-bit WAVE) Creates an AAF with source audio files with crossfades.

AAF for Avid Pro Tools (AV DNxHD 36 1080p 24-bit WAVE) Creates an AAF file with a committed QuickTime file and source audio files with crossfades.

Limitations

The following guidelines will help you create AAF that should conform properly in third party applications.

Metadata-only AAF export:

The idea with metadata only AAF export is to provide to third party applications a sequence that reflects the editorial intention. You should prepare the sequence prior to exporting it to minimize problems in the third party application.

- Only use Dissolve transitions. Wipe, Action, and Matchbox transitions are not exported to the AAF, and should all be converted to Dissolves prior to export to avoid timing issues: these transitions are ignored and the timing of the sequence might be impacted.
- Avoid using generated sources. Since by definition a metadata-only AAF export does not export any media, the exported AAF would reference sources that are not available outside of Flame.
  - Timewarps
  - Containers and Matte Containers
  - Hard Committed segments
  - BFX

- Content generated internally (Tools, Batch, etc)
- Colour sources, Bars, Noise, Gradient
- Fade to / from colour

- Not all third-party application support all media formats and codecs that can be used in Flame. You should verify that the media files referenced by the sequence can be read by the third party application.

Media *and* metadata AAF export:

In this export type, Flame create AAF files referencing the newly generated media files. This is ideal when the destination application does not manage different file formats.

- Only use Dissolve transitions. Wipe, Action, and Matchbox transitions are not exported to the AAF, and should all be converted to Dissolves prior to export to avoid timing issues: these transitions are ignored and the timing of the sequence might be impacted.
- Using the Use Original Media will generate media files based on the source clips used with Timeline FX (except Timewarps). Segments with BFX, Container/Matte Container, content generated internally (Tools, Batch, etc), colour sources, bars, noise, gradient, fade from/to color, etc. all generate media files based on the Media with FX.
- To ensure maximum compatibility with third party applications, make sure to export the following media file types:
  - Exported video media should be Movie files, either QuickTime or MXF files.
  - Exported audio media should be Audio files (AIFF or WAVE). For workflows where the timecode is important, use the WAVE format.

Application-Specific Remarks

Avid Media Composer

- When importing a sequence that contains dissolve transitions, MC displays a warning about a sequence update for advanced keyframe effect. You can safely confirm the dialog, and even enable the *Do not ask again loading a sequence* option.
- When importing an AAF file in Avid Media Composer, the sequence appears as Offline (media files will not be linked to the sequence). Media Composer requires media files to be indexed prior linking. You need to manually import the media files (through either Media Import or AMA Import), and then relink the sequence to the master clips. If you generated MXF files from Flame, make sure to select the MXF file format in the AMA Import dialog; not doing so will produce an error.
- Also, avoid generating QuickTime files with Avid DNxHD codec if you plan on relinking the sequence using the Import dialog. QuickTime DNxHD can only be read through AMA.

Avid Pro Tools

- If the media files do not automatically link when importing the AAF file in Pro Tools, select the following options in the Missing Files dialog:
  - Select Manually Find & Relink.
  - In the Linking Options, use Find By Name.
  - Disable Match Format, Duration, and Modification Date.
  - Select Manual Link and disable Duration.

DaVinci Resolve:

- Avoid publishing a sequence that has audio crossfades and fades transitions. DaVinci Resolve will report errors and will remove the audio transitions.

Publishing an FCP X XML Sequence

Use FCP X XML Export when you want to share a sequence with third party applications. The XML Export generates a simplified sequence that can be used in third party applications for creative editorial, color correction, media management, etc.

You can also use XML export to share a sequence and media files with Flame, but the amount of exported metadata is limited by the FCP X XML format:

- Non-dissolve transitions (Wipe, Action, Matchbox) are converted to dissolve.
- Since Flame uses an effects and transitions technology different from FCP X, and because the timeline is not magnetic like in Final Cut pro X, it is not possible to completely represent a Flame sequence in FCP X XML format. For the same reason FCP X does not support FCP 7 XML.

The exported XML file can be imported in:

- Apple Final Cut Pro X (version 10.1 and later)
- DaVinci Resolve (version 11 and later)
- Autodesk Flame (version 2015 Extension 1 and later)
- Autodesk Smoke (version 2015 Extension 1 and later)

To publish a sequence as an FCP X XML and export its media:

1. Do one of the following:
   - Right-click the sequence to export and select Export. And then, using the Media Export window, navigate to the clip's destination.
   - From the MediaHub, drag-and-drop the file from the Media panel to the location displayed in the browser.
2. Set Export Type to Sequence Publish.
3. Set Format Preset to one of the FCP X XML presets.
4. Click Export.

   Flame prepares the export job. Once that preparation is done, Flame performs the export in the background. You are free to use the application as the export happens.

Customizing FCP X XML Publication

1. In Media Export, set Export to Sequence Publish.
2. Enable Show Advanced Options to access the detailed settings.
3. In the Sequence Options, set Format to Final Cut Pro X XML.

From the Sequence Options tab, you can export an AAF file that references the original media files or generate an XML file that will reference new media files.

To export only an FCP X XML, without media:

1. Enable Include Video and Include Audio, as required.
2. Disable both Export Video Media and Export Audio Media.

3 Click Export.

This creates an FCP X XML file where the video and audio tracks reference the original media files. Also note that the Tracks and Transitions option is implicitly set to Keep All Tracks, and cannot be changed.

NOTE This process is especially fast, since no media need to be rendered or written to disk.

To export an FCP X XML and its media:

1 Enable Include Video and Include Audio, as required.
2 Enable Export Video Media and Export Audio Media, as required.
3 Set Video Format to Movie, and Audio Format to Movie.
4 Select a Media option, either Use Original Media or Use Media with FX.
5 Select an option for Tracks and Transitions. This option defines the exported structure of the sequence:
- Flatten Tracks: The sequence is converted to a single segment.
- Flatten Tracks with Transitions: A multi-track sequence is converted to a single track sequence and dissolve transitions are preserved. Non dissolve transitions are flattened.
- Keep All Tracks: The sequence structure is not modified and non-dissolve transitions are converted to dissolve transitions. Also, non centered transitions are converted to centered dissolves.

FCP X XML Presets

You can use one of the following sets of Sequence Publish presets to export an FCP X XML for a specific purpose.

XML for Apple Final Cut Pro X (Flatten) Use these Export Presets to publish clips or sequences as a final asset to be used in a third party application that supports FCP X XML. Media with FX is used to generate media files.

XML for Apple Final Cut Pro X (Flatten With Transitions) Use these Export Presets to publish your sequence when you want editors, colorists, or visual effects artists to work on a simplified sequence version that allows to them to see shot boundaries. Media with FX is used to generate media files.

XML for Apple Final Cut Pro X (All Tracks) Use these Export Presets to publish your sequence when you want editors, colorists, or visual effects artists to work on a simplified multi tracks sequence version that allows them to see shot boundaries. The original media is used to generate media files for segments that have Timeline FX. Segments made from Tools, Batch, Container, Matte Container or Batch or Batch FX use media with FX.

NOTE The All Tracks Publish options uses the top most video tracks to generate content and not the current track focus. If the sequence you want to use for XML Export contains material on video track you do not want to export, remove these extra video tracks prior to exporting.

XML for DaVinci Resolve (ProRes 422 and 24-bit WAVE) This preset generates a video and audio multi-tracks FCP X XML file with 10-bit QuickTime ProRes 422 referencing the media with effects. 24-bit audio WAVE files are also generated. Audio tracks are flattened.

XML for DaVinci Resolve for Source Grading (ProRes 422 and 24-bit WAVE) This preset generates a video and audio multi-tracks FCP X XML file with 10-bit QuickTime ProRes 422 using the original media. 24-bit audio WAVE files are also generated, but the audio tracks are flattened.

XML for DaVinci Resolve for Source Grading (Sequence-only) This preset generates a video-only multi-track FCP X XML file referencing the source media with effects. No media files are generated.

Limitations

Note that when exporting a sequence as an FCP X XML file:

- The FCP X XML format does not support file sequences definition so exporting a metadata only XML referencing file sequence OR exporting file sequences will not work in FCP X and DaVinci Resolve. But the file will be loadable in Flame.
- The accompanying video media files exported by Flame must use either QuickTime or MXF containers.
- The accompanying audio media files exported by Flame must be either AIFF or WAVE.
- Audio sub-frame editing is not supported. Ensure audio edits are frame-based.

Metadata-only export limitations:

- When exporting a sequence as XML-only , the segments of virtual media generated in the application (Colour Sources, Gap FX, Tools or Batch, Container, Matte Container, etc) are seen as Unlinked Media.
- Avoid using transition *from*and *to* colour clips because these cannot be represented in third party applications when exporting only XML metadata. There is no problem for FCP X XML with exported media files.
- FCP X XML does not define the bit depth of the sequence. The resulting sequence is created at the project bit depth. Override the project bit depth using **MediaHub ➤ AAF & XML Import Option ➤ Select Resolution**.

Media file-related limitations:

- A sequence referencing AVC-HD media files exported as XML-only do import correctly in Apple Final Cut Pro and DaVinci Resolve. You should export the sequence and new media files.
- When working with mixed resolution content, it is possible to export an XML that references the Original Media and generate media files of the original source clip resolution.
- A sequence made of mixed frame rates is exported at the sequence rate, and the Original Media option generates files using the sequence rate. If clips were resized to accommodate the resolution difference, the generated media files are exported at the sequence resolution.
- Containers and Matte Container are exported as new clips and use the sequence's record timecode and rate. No alpha is exported for Matte Container.
- Transitions in FCP X XML are always centered and have even duration. Since Flame is capable or creating transitions of different orientation and duration, the transitions are modified to be compatible with FCP X.
- Avoid using illegal Unix characters in sequences and clip names because some third-party applications cannot read media files with such characters. Flame does not sanitize the file names when exporting XML with media files.
- XML files from DaVinci Resolve referencing file sequences cannot be imported in Flame since the media is flagged as Unlinked Media.

Publishing Shots

Publishing a Shot

Before distributing shots to others, verify that the structure of the sequence, name the shots, and then distribute them.

To create a shot:

1. Verify that the shot has a "layered cake" structure.

   This structure is the best layout when publishing a shot: Flame uses a bottom-most left-most segment part of the shot to create the shot's timing structure.

2. Identify segments that are part of the shot by assigning them an identical shot name. For Flame, a shot is a group of segments that have the same Shot Name.

   You can edit the shot name from the Conform view, or with the Rename Shot option from either the Timeline gear menu or contextual menu.

3. Repeat for each shot on your timeline. Make sure to assign each shot a unique name.

   TIP The Conform tab provides your with a great overview of your timeline, including assigned shot names.

Possible strategy for naming shots:

1. Select the bottom video track.
2. Select Rename Shot from the Timeline gear menu.
3. From the Add Token, select Segment Index. This assigns a unique shot name to each segment.
4. Now go from segment to segment, assigning each segment the segment id of the bottom-most segment. You can do this in Conform, to edit Shot names in bulk.

When you are ready to distribute the shots, publish them.

To publish the shots to be used by other applications:

1. Export the sequence.
2. In the Media Export window, navigate to the root folder.

   The root folder is the published material destination where the published shot structure is created. When you publish shots, you do not only export the media, you also export additional files that provide information crucial to the shot publish workflow, such as Open Clip XML files and Batch setups for each shot.
3. Set Export to Sequence Publish.
4. Set Format Preset to one of the Shot Publish existing presets.

   For this procedure, we will use *Shot Publish - Flame Root Directory (16-bit OpenEXR and No Audio)*.
5. Click Export.

   Once the export is completed, you can see, next your original sequence, the published timeline named as your sequence but appended with *_publish*.

When you open a shot published sequence, the versioned shot appears on a different Version layer of the sequence. The bottom Version contains the original sequence.

The versioned shot actually contains all of the media of the shot and effects, keeping the timing from the original sequence.

Create New Version

- Provides access to the open setup in a fashion similar to History. Like History, exiting automatically triggers a new render, however in this case it will create all of the required media and metadata in the appropriate paths in the job tree, and append a version to the timeline segment.

Open as Batch Group

- Load the currently selected version in Batch. This will include the Write File node needed to append versions to the shot, the job tree, and the segments in the timelines where it is used.

Promote to Batch FX

- Ingests the shot directly in the current timeline. This removes the connection to the job tree, as well as its versions.

Opening the Shot in Flare

1. In Flare's Batch view, click Load.
2. Navigate to the folder where the shot's Batch setups are stored.

   Batch setups for shots are placed in the path defined by **Clip Options tab ➤ Setup Pattern field** in the Media Export window. In the Format Preset *Shot Publish - Flame Root Directory (16-bit OpenEXR and No Audio)*, Batch setups are located in *flame/batch/*.
3. Double-click the Batch setup to open it.
4. Edit the setup.
5. When done, click Render.

   New media and version information are written to the paths set in the Write File node. The Open Clip XML is updated to include this new version information. And a new version of the Batch setup is automatically saved in the appropriate paths in the job tree.

Viewing a version in Flame:

1. Right-click the published sequence and select **Source Versions ➤ Update Source**. *_publish* identifies a published sequence.

   This reloads the associated Open Clip, making sure the latest versions are available for viewing. An updated segment is highlighted in white to indicate that is contains new versions.
2. Open the published sequence.

The original sequence is displayed on the bottom Version of the timeline. A new Version contains the track with the published shots and referenced media.

3. On the Timeline, locate the shot segment on one of the Published shots tracks.
4. On the Timeline FX ribbon, select a version from the Clip Versions box. Click Pre-Processing if at first you don't see it.

   You can now playback the new version, open it in Batch, promote it to BFX, output it, or simply revert back to a previous version.

Regarding Source Versions options:

- Update Source: Reloads the Open Clip, making sure all the versions are available for viewing.
- Select Current Version: Reloads the Open Clip, and then displays the version defined as current in the Open Clip by the `currentVersion` property of the `<versions>` element. Pipeline tools such as ShotGun can change the Current Version based on approval policies.
- Select Latest Version: Reloads the Open Clip, and displays the latest version.
- Mark All Updates as Viewed: Clears from all the selected clips the white halo that highlights segments that were updated and which help you track down updated segments.
- Version Name: If you select a clip, you can select a version directly from within the Source Versions menu.

NOTE You can perform these operations on a segment, a sequence, multi-selected sequences, and even a reel, Reels Group, a library, multiple libraries to updated all sequences within.

Naming With Tokens

You can make it so a segment's name, shot identifier, or comment uses dynamic tokens.

To rename a segment so it uses tokens:

1. Right-click the segment in the timeline.
2. Select Rename.
3. Enter the new name for the segment. Use tokens to use existing metadata.

   TIP Enable the Dynamic button to have a name that automatically updates its tokens.

To add the clip's height and width to the segment's name:

1. Right-click the segment in the timeline.
2. Select Rename.
3. Click the Pattern field, and then Esc to empty it.
4. From the Add Token box, select Clip Name.
   Pattern now displays: *<name>*.
5. From the Add Token box, select Width.
   Pattern field now displays: *<name><width>*.
6. From the Add Token box, select Height.
   Pattern field now displays: *<name><width><height>*.
7. Add characters to clear up the name: *<name>_<width>x<height>*.
8. Click Rename. The segment's name now contains its name and dimensions.

You can edit multiple segments in a single operation by first selecting them and then selecting, from the contextual menu, Rename, Comment, or Rename Shot. And what information you type in the dialog box overwrites whatever was already present in those segments.

To add information to multiple segments' shot name, comment, or segment name without overwriting what is already there:

1. Multi-select the segments.
2. From the contextual menu, select Rename, Comment, or Rename Shot, depending what you need to update.

   If you've never edited this information, both the Preview and Pattern are empty. If this is not the first time, they will contain some information.
3. Edit the Pattern field. Add the required text and tokens, but make sure that the relevant token is included. That token is a placeholder for the original information and makes sure. The tokens to include are:
   - For Rename: *<segment name>*
   - For Comment: *<comment>*
   - For Rename Shot: *<shot name>*
4. Click the required button to close the window. Now each segment contains the old information (thanks to the placeholder) and the new information that was added.

Example:

1. Two segments need to be renamed to display their width and height: *Segment_1* and *Segment_2*.
2. Select both segments, and from the contextual menu, select Rename.

   In the Rename dialog box, because this is the first time these segments are renamed, both Preview and Pattern fields are empty.
3. In the Pattern field, add the segments' name placeholder. The Pattern field is now:*<segment name>*
4. Add the width token, with a colon and space. The Pattern field is now:*<segment name>: <width>*
5. Add the height token, with an x and a space. The Pattern field is now:*<segment name>: <width> x <height>*
6. Click Rename.
7. The two segments are now named:
   - *Segment_1: 1920 x 1080*
   - *Segment_2: 1920 x 1080*

Regarding Batch Setups for Shots

Batch setups created with the Shot Publish workflow possess some interesting features that help minimize record keeping.

- The Write File node is already configured to use the correct render paths and the correct Format and Settings options.
- The version number automatically increases every time you render.
- The Batch timing view provides you with the required layer information.

Shot Publish Presets

Note that there is a limited amount of presets for Shot Publish. These presets are provided mainly as examples of naming patterns, examples you can customize to your specific needs. But as you customize those settings, make sure you enable:

- Copy Exported Clip in Media Library: This option creates in your Flame the special _publish sequence with the Version that contains all the individual shots.
- Create Clip Data: This option *is* the Shot Publish workflow by creating the Open Clip XML files, essential to this feature.
- Clip Versions: This option is required if you want to track the different versions for each shot.
- Create Shot Setup: This option exports, together with the clip, the batch setups required for the FX. Especially useful when working within a Flame-Flare setup.

But you can always edit the different patterns naming patterns to fit your facilities requirements.

Integration with Third Parties

You can integrate Flame in third-party asset management workflows, but it does require some additional work.

This additional work consists of configuring Nuke (or any third-party application) to update the Open Clip XML file. The Open Clip XML file stores version, media, and shot structure information. To leverage that information, the third-party app must be able to decode (and encode) Open Clip XML. You can find information about its format *here*.

You can integrate this info to a script in the Write node in Nuke, or to a post-render script in Maya. Rendering in such an application automatically runs the script to wrap the footage in the Open Clip XML format and automates this process. If any new renders are output from these applications, they automatically update in the Open Clip XML file.

If you additional monitoring or automation needs, you can use available Python hooks to integrate Flame into a shot management system.

And from the Flame, the process is identical to working with a Flare.

Creating New Export Presets

Flame comes with a number of export presets that should cover most of your needs. But you can also create your own presets.

Whenever you edit an existing preset, it gets appended *_custom*. That new custom preset remains until you change project, exit Flame, or click Reset.

To create a new export preset:

1. Select an Export Type.
2. Select a Format Preset.

   Select the preset that is the closest to what you want, to minimize the amount of required configuration.
3. Enable Show Advanced Options.
4. Edit the settings as required.

   Notice how the Format Preset box has appended the preset you selected with the *_custom* suffix.
5. Do one of the following:
   - To cancel your edits, click Reset.

- If this is essentially a one-off customization that you will not need later, click Export to export your sequence. You can always save that preset at a later time: a customized preset remains until you exit Flame or change project.
- To keep this new preset, click Save. You now the new preset in Project or Shared sections of the Format Preset drop-down menu, and the original preset back in the Autodesk section.

TIP Consider creating an Export preset with padding set to 0 to export single-frame clips without any file numbering. And to avoid overwriting the media files, Flame will automatically name file sequences of more than one frame with the default padding of 8 digits.

H.264 Codec Export Profiles

When exporting a clip as an H.264 in QuickTime, you can use one of the pre-configured H.264 codec profiles.

NOTE The H.264 codec profiles are xml files stored in */opt/Autodesk/mediaconverter/<release_version>/profiles/QuickTime/video/H264*.

The Suggested Clip Resolution guides you in the selection of a target resolution for the output. Using a different resolution can have unexpected results.

Profile	Description	Suggested Clip Resolution	Bit Rate
Baseline_1SEG_384Kbits	H264_CIF, Baseline profile	352x288 or 352x240	384 Kb/s
Baseline_3GP_256Kbits	H264_3GP 3GP, Baseline profile	352x288	256 Kb/s
Baseline_600Kbits	H264_BASELINE, Baseline profile	320x240	600 Kb/s
Baseline_Adobe_300Kbits	H264_FLASH_LOWRES, Baseline profile	320x240	300 Kb/s
Baseline_Apple_1_5Mbits	H264_iPOD Apple iPod, Baseline profile	320x240	1.5 Mb/s
Baseline_Apple_400Kbits	H264_iPOD Apple iPod, Baseline profile	320x240	400 Kb/s
Baseline_Apple_600Kbits	H264_iPOD Apple iPod, Baseline profile	320x240	600 Kb/s
Baseline_Apple_970Kbits	H264_iPOD Apple iPod, Baseline profile	320x240	970 Kb/s
Baseline_CIF_600Kbits	H264_CIF at, Baseline profile	352x288 or 352x240	600 Kb/s

Profile	Description	Suggested Clip Resolution	Bit Rate
Baseline_RIM_12Mbits	H264_BASELINE, Baseline profile	1920x1080	12 Mb/s
Baseline_RIM_20Mbits	H264_BASELINE, Baseline profile	1920x1080	20 Mb/s
Baseline_RIM_4Mbits	H264_BASELINE, Baseline profile	1920x1080	4 Mb/s
HDTV_1080i_10Mbits	H264_HDTV_1080i, High profile, interlaced	1920x1080	10 Mb/s
HDTV_720p_8Mbits	H264_HDTV_720p, High profile	1280x720	8 Mb/s
High_1080i_6Mbits	H264_HIGH, High profile, interlaced	1920x1080	6 Mb/s
High_AVC_HD_20Mbits	H264_AVCHD AVCHD, High profile, interlaced	1920x1080	20 Mb/s
High_AVC_Intra_111Mbits	H264_INTRA_CLASS_100 AVC Intra Class 100, High 10 profile, interlaced	1920x1080	111 Mb/s
High_AVC_Intra_54Mbits	H264_INTRA_CLASS_50 AVC Intra Class 50, High 10 profile, interlaced	1440x1080	54 Mb/s
High_Blu_Ray_20Mbits	H264_BD_HDMV Blu-ray HD, High profile, interlaced	1920x1080	20 Mb/s
High_Blu_Ray_8Mbits	H264_BD Blu-ray SD, High profile, interlaced	720x576 or 720x480	8 Mb/s
High_Divx_2Mbits	H264_DIVX DivX+, High profile	1920x1080	2 Mb/s
High_DVD_3Mbits	H264_DVD, High profile, interlaced	720x576 or 720x480	3 Mb/s
High_HD_DVD_20Mbits	H264_HD_DVD, High profile, interlaced	1920x1080	20 Mb/s

Profile	Description	Suggested Clip Resolution	Bit Rate
High_Microsoft_10Mbits	H264_SILVERLIGHT Microsoft Silverlight, High profile	1920x1080	10 Mb/s
High_Microsoft_500Kbits	H264_SILVERLIGHT Microsoft Silverlight, High profile	640x480	500 Kb/s
Main_3Mbits	H264_MAIN, Main profile	704x576 or 704x480	3 Mb/s
Main_Adobe_670Kbits	H264_FLASH_HIGHRES, Main profile	640x480	670 Kb/s
Main_Apple_1_8Mbits	H264_MAIN, Main profile	1024x576	1.8 Mb/s
Main_Apple_4_5Mbits	H264_MAIN, Main profile	1280x720	4.5 Mb/s
Main_D1_3Mbits	H264_D1, Main profile, interlaced	720x576 or 720x480	3 Mb/s
Main_Sony_2Mbits	H264_PSP_640x480 Sony PSP Level 3, Main profile	640x480	2 Mb/s
Main_Sony_700Kbits	H264_PSP Sony PSP, Main profile	320x240	700 Kb/s
Main_Sony_900Kbits	H264_PSP_480x270 Sony PSP Level 2, Main profile	480x272	900 Kb/s
Main_SVCD_1_15Mbits	H264_SVCD, Main profile, interlaced	480x576 or 480x480	1.15 Mb/s

Media Export Window Settings

Basic Options

Export Type box Select the type of export to use with the selected files.

Preset Selection box Select the export preset to apply to the exported files. Autodesk presets are built-in presets that you can still modify using the Advanced Options. An asterisk indicates that the preset's advanced options were modified.

Exported File Name field Displays the name given to the exported file, as defined in the Advanced Options. Editable when exporting a single clip.

Advanced Options button Enable to display the advanced options to customize existing presets, or create new ones.

Use Top Video Track button Enable to export the top video track of the current version. When disabled, exports the track currently set as the primary video track.

Export Between Marks button When enabled, sequences are exported between the In and Out marks: the marked frames are not exported. Disable to export the whole sequence, without considering the marks.

If there is only an Out mark, only frames from start to Out are exported. If there is only an In mark, only frames from In to end are exported. In both cases, the In and Out frames are not exported.

Export in Foreground button Enable to render sequences directly to destination files, without first writing frames on the framestore, saving space and time. When exporting as Sequence Publish, some elements might be rendered to fit the exported timeline structure.

For File Sequence and Movie, foreground export lets you export media without requiring you to re-render. You do not allocate frames on your framestore because content is streamed directly to the exported file, reducing the time to export.

For Sequence Publish export, when the media needs to be packaged, the application performs the required rendering before exporting. For example, using the Flattened tracks option with Timeline FX requires rendering prior exporting, as do segments with a BFX or Resize.

Sequence Options Tab

Sequence Format box Select the format of the sequence. Media Only exports the segments of the sequence as individual clips, but does not export the sequence itself.

Include Video button Enable to include in the published sequence the video tracks information. Required to export the video media.

Include Audio button Enable to include in the published sequence the audio tracks information. Required to export the audio media.

Sequence Filename field Displays the sequence filename based on the Pattern field. Each type of exported file has its own filename defined in the relevant tabs. Non-editable.

Sequence Filename Pattern field Displays how to name the exported media files. Build a dynamic naming scheme using Add Token, or characters normally allowed in a file name. Create folder structure using / . Add '#' to numerical tokens to define padding. The file extension is automatically appended. Editable.

Add Token box Inserts a token in the Pattern field to build a dynamic filename.

Export Video button Enable to export the segments of the sequence as files of the type specified in Video Format.

Video Format box Select the type of video file to create. For movie, select the wrapper and codec in the Movie Options tab. For file sequence, select the file type in the Video Options tab.

Media Source box Select Use Original Media to export the original source referred by the exported sequence, without any modifications. Select Use Media with FX to export the rendered media.

Video Tracks and Transitions box Select Keep All Tracks to export a clip for each segment of the sequence. Select Flatten Tracks to export commit every transition and flatten the sequence. Select Flatten with Transitions to flatten the sequence and commit every transition except dissolves, creating a single clip.

Include Video Handles button Enable to add head and tail frames to the exported video segments.

Video Handles field Displays the amount of head and tail frames. Editable.

Export Audio button Enable to export the audio tracks of the sequence.

Audio Source Selection box Select Use Original Media to export the original source referred by the exported sequence, without any modifications. Select Use Media with FX to export the rendered media.

Audio Track State box Select Flatten Tracks to commit all transitions and create one audio clip per track. Select Flatten Tracks with Transitions to create an audio clip per track but keep live transitions. Select Keep All Tracks to export one audio clip per audio segment.

Include Audio Handles button Enable to add head and tail frames the exported audio segments. Audio segments with Timeline FX applied are exported with 0 handles.

Audio Handles field Displays the amount of head and tail frames. Editable.

Movie Options tab

Movie Format box Select the container for the exported media.

Compression box Select the codec to apply to the exported movie file. The available codecs depend on the selected Movie Format.

Codec Profile box Select a pre-defined video compression codec profile when exporting QuickTime files using the H.264 or MPEG-4 codecs.

Filename field Displays the result of the pattern displayed in the Filename Pattern field. Non-editable.

Filename Pattern field Displays how to name the exported media files. Build a dynamic naming scheme using Add Token and characters normally allowed in a file name. Create folder structure using / . The file extension is automatically appended. Add '#' to numerical tokens to define padding. Editable.

Add Token box Inserts a token in the Pattern field to build a dynamic filename.

Include Audio button Enable to include audio tracks within the exported file. Available only if Export Type set to Movie.

LUT Activation button Enable to apply the LUT or colour transform displayed in the Applied LUT field to the clip.

Format box Select View Transform to apply viewport colour management to the export. Select Input Transform to apply import colour management (or its inverse) to the export. Select Colour Transform for CTF and most other formats except a legacy 1D or 3D LUT. Select Tag Only if you want to override the colour space written to an OpenClip XML file.

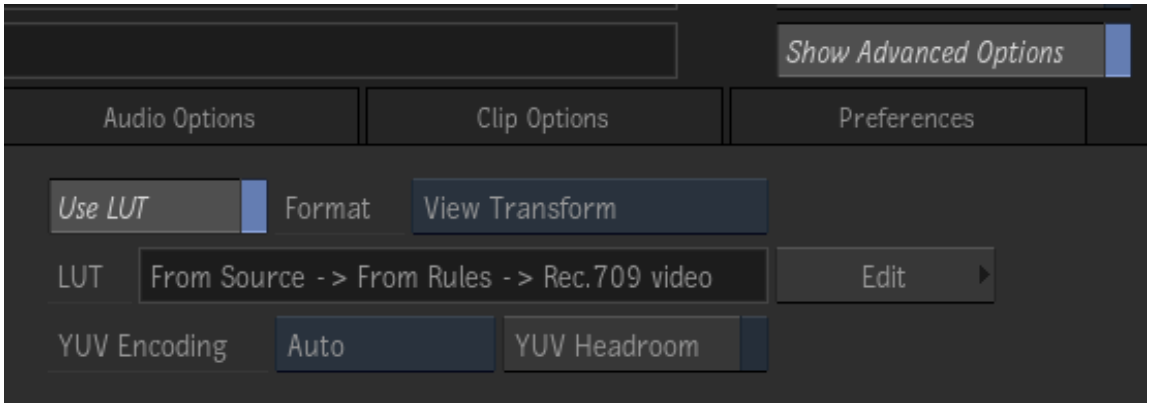
Applied LUT field Displays the type of colour management applied to the clip, either imported using Import, or edited using Edit.

Colour Management Access button Click to configure colour management settings.

Include YUV Headroom button Enable to include the YUV headroom in the exported media files, creating a full-range clip. Disable to convert your clip into a legal range/valid colour clip, with values between Reference Black and Reference White. Leave disabled when the deliverable is broadcast, and enabled when you want to provide wider range for post-production processes like grading and visual effects work. Automatically disabled when exporting using a QuickTime RGB codecs.

YUV Encoding box Select the clip's YCbCr colour encoding matrix. The application uses this to translate native RGB values into YUV ones. The Auto setting selects the colour encoder based on the resolution of the clip: Rec. 601 for clips with a resolution lower than 720 lines, Rec. 709 for everything else.

Flame works in an RGB colour space: it needs to translate RGB information back into YUV. This setting ensures that the right encoder is used for this. Specifying the wrong option will result in colour problems upon decoding. Per Recommendation BT. 2020, UHD media should use the Rec. 2020 colour space. But this is rarely the case at the moment: UHD clips still use Rec. 709. The Rec. 2020 option is there is case you need the new standard.



See also: Applying Colour Management to Clips (page 1760)

Video Options tab

Video Format field Select the format for the exported media.

Compression box Select the image compression to apply to the exported file sequence. The available compressions depend on the selected Video Format.

Link Original Media button Enable to hard link the published files to the original files if both the original and exported files are located on the same filesystem. If not, the application creates soft links back to the originals. Available for file sequences only.

This option saves disk space on export as files that remain unchanged by the export are not duplicated. Unchanged in this context means they were not rendered not resized in anyway. For example, a file sequence of 20 dpx are imported in Flame. Of those 20, 12 are modified in some fashion. With Link Original Media enabled, of the sequence of 20 dpx, only the modified 12 are actually created at export; the other 8 dpx are just linked to from the export folder.

This option can be used even when the colour space of the timeline is different from that of the frames, for example, when promoting 16-bit integer DPX files to a floating-point timeline (see Converting Images Between 12i or 16i and 16f Encodings (page 1804)). Any frames that have a colour transform assigned in the import settings, as well as the corresponding inverse colour transform assigned in the export settings, will be linked if those frames are otherwise unchanged. If you are creating your own .ctf files, note that the inverseOf attribute of the export transform must be properly set (see ProcessList (page 1813)). Any frames that do not have the inverse colour transforms applied on export are processed as usual.

NOTE You can have Flame copy files instead of creating symbolic links across filesystems: in the *stone+wire.cfg* configuration file, set the `SymlinkAcrossFilesystems` keyword to FALSE, in the `StandardFSMediaOptions` section.

DPX Transfer Characteristics box Select an option to identify the attributes associated with a particular film or video format, such as resolution, frame rate, or colour space. Available when Video Format is set to DPX.

When exporting DPX files, you can choose a DPX Transfer Characteristic. A DPX Transfer Characteristic is information that is stored in the DPX image file header. It identifies the attributes associated with a particular film or video format, such as resolution, frame rate.

Setting a Transfer Characteristic in no way changes the *image* information stored in the DPX file. The Transfer Characteristic simply indicates the attributes of the DPX file read by another device or application. Some devices or applications may take advantage of this information to improve workflow. For example, selecting

Logarithmic can allow a film recorder to adjust its parameters to print film-originated DPX files with the correct densities.

Select:	For:
Z depth homogeneous, Z depth linear, PAL, NTSC, CCIR 601 (525), CCIR 601 (625), CCIR 709-4, SMPTE 274M	Images that you want to identify as one of these types. Although the SMPTE 274M standard defines these DPX Transfer Characteristics, it does not provide usage specifications for them. As a result, these Transfer Characteristics are not generally used in the industry.
Unspecified	Images where the format is not specified.
Logarithmic	Negative film scanners recording status M densities.
Linear	Video images which have built-in gamma correction. This refers to images having a true linear quantization scheme (such as CG-originated material).
Printing Density	Negative film scans which use the SMPTE Printing Density settings. SMPTE Printing Densities use status M density measurements with a higher gain in the red component.
Academy Density Exchange (ADX)	Images at 10- and 16-bit film density encoding, in which colour information is encoded using a logarithmic scale. Usually used in the context of an Image Interchange Format workflow.

DPX Colorimetric Specification box Select an option to identify the colorimetric specifications used to encode the DPX files. Available when Video Format is set to DPX.

Setting a Colorimetric Specification in no way changes the *image* information stored in the DPX file. The Colorimetric Specification simply indicates the attributes of the DPX file read by another device or application. Some devices or applications may take advantage of this information to improve workflow. For example, selecting Academy Density Exchange (ADX) can allow a film recorder to adjust its parameters to print film-originated DPX files with the correct densities.

JPEG Quality field Specifies the degree of quality versus compression. A value of 0 gives the lowest quality (and highest compression), while a value of 100 gives the best quality (but applies no compression). Available when File Format is set to JPEG.

LUT Activation button Enable to apply the LUT or colour transform displayed in the Applied LUT field to the clip.

Format box Select View Transform to apply viewport colour management to the export. Select Input Transform to apply import colour management (or its inverse) to the export. Select Colour Transform for CTF and most other formats except a legacy 1D or 3D LUT. Select Tag Only if you want to override the colour space written to an OpenClip XML file.

Applied LUT field Displays the type of colour management applied to the clip, either imported using Import, or edited using Edit.

Import button Use to browse and select a colour transform.

Colour Management Access button Click to open the Colour Management editor.

Filename field Displays the result of the pattern displayed in the Filename Pattern field. Non-editable.

Filename Pattern field Displays how to name the exported media files. Build a dynamic naming scheme using Add Token and characters normally allowed in a file name. Create folder structure using / . The file

extension is automatically appended. Add '#' to numerical tokens to define padding. Frame identifiers are automatically added to image sequence files. Editable.

Add Token box Inserts a token in the Pattern field to build a dynamic filename.

Frame Padding field Define the padding of the frame identifiers appended to each file of an image sequence. Only used with image sequences.

For example, a frame pad of 6 indicates that each frames's file name has its frame identifier padded with a number of zeroes required to make it a 6-digit number: frame 1 is written as 000001, frame 22 as 000022, frame 55555 as 055555, and so on.

Use Timecode button Enable to offset the numbering of the file sequence by the start timecode, as a frame count. The start timecode must be positive or the export will fail.

Start Frame field Enter the start number to be used in the exported sequence of numbered image files. Disabled when Use Clip TC Names is enabled.

Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

Fit Method box Select a fit method option to be applied to the exported clip.

Select:	To:
Centre/Crop	Fit the source image, centred, over the destination frame. If the source is larger than the destination, it is cropped. If the source is smaller than the destination, it is surrounded by a black border.
Crop Edges	Fit one edge of the source into the destination frame without stretching or squashing the frame. Excess parts of the source frame after resizing are cropped. If the source—after the one edge is resized—is wider than the destination, its overhanging left and right edges are cropped. If the source is taller than the destination, the upper and lower edges are cropped.
Fill	Fit the source, width, and height, into the destination frame. If the source and destination frames do not have the same aspect ratio, the image can become distorted.
Letterbox	Fit the source to the destination frame without squashing or stretching it, and without cropping the source. If the source is wider than the destination, black bars fill the top and bottom of the destination frame. If the source is narrower than the destination, black bars fill the right and left sides of the frame. In all cases, the entire source frame is contained within the destination frame.

Resize Filter box Select the filter option to determine the quality of the interpolated resize result. The Resize Filter box is active only if Fit Method is set to Crop Edges, Fill, or Letterbox.

Select:	To get:
Impulse	Quick, low-quality results.
Triangle	Moderate results with little processing overhead.
Mitchell	Best results when resizing a clip to a higher resolution.
Bicubic	Very good results for resizing soft-looking images. Use to sharpen the image.

Select:	To get:
Quadratic	Good results for resizing simple images with straight edges. Similar to Gaussian but with more blurring. Use to soften the image.
Gaussian	Excellent results when resizing a clip with no patterns and numerous straight edges to a lower resolution. Useful for softening some detail.
Shannon	Excellent results when resizing a clip to a lower resolution. Very similar to Lanczos, but results are a little softer.
Lanczos	Best results when resizing a clip containing a variety of patterns and elements to a lower resolution. It is the most complex with the longest processing time.

Frame Width field Displays the frame width of the selected clip. By clicking it you activate the field, allowing you to enter the frame width value that you want to use on export.

Frame Height field Displays the frame height of the selected clip. By clicking it you activate the field, allowing you to enter the frame height value that you want to use on export.

Aspect Ratio Presets box Select a standard frame aspect ratio. Select the Set to w:h option to set the clip to use square pixels. Select Custom to define a custom frame aspect ratio in the Aspect Ratio field.

Aspect Ratio field Displays the aspect ratio defined by Aspect Ratio Presets. Editable.

Bit Depth box Select a bit depth to be used on export. Some file formats support multiple bit depths. Bit Depth is active only when Resize is enabled.

Scan Mode box Select an option to set the order in which the fields of interlaced material are scanned.

For interlaced material, you can specify whether the resize needs to be done from both fields or just from one of the two. In the latter case, the result is a progressive clip made from the same two fields.

Select:	To resize:
From Clip	Using the scan mode of the source clip.
Progressive	A frame-based clip to another frame-based clip.
Field 1	A clip by drawing Field 1 followed by Field 2.
Field 2	A clip by drawing Field 2 followed by Field 1.

Note that in the case of IMX exports, the Scan Mode box is ignored: NTSC clips are always exported as F2 dominant, while PAL clips are always exported as F1 dominant.

Audio Options Tab

Audio Format box Select the audio format of the exported audio tracks. If Export Type is Movie, the audio is embedded within the video file. Any other Export Type outputs a separate audio file.

Audio Bit Depth box Select the bit depth of the exported audio file.

Audio Compression field Select the compression for the exported audio file. The available options depends on the Audio Format.

Audio Sample Rate box Select the sample rate of the exported audio.

Audio Mixdown box Select a mixdown to apply to the audio tracks, if any are included with the source clip.

Select:	To mix down:
No Mixdown	Nothing
Mixdown As Is	With the current output strip assignments.
Mixdown To 4 tracks	To four tracks. The output strips are assigned sequentially in fours to the mixed-down channels (where M1 goes to 1, M2 to 2, M3 to 3, M4 to 4, M5 to 1, and so on).
Mixdown To Stereo	To one stereo track. The output strips are assigned sequentially in twos to the mixed-down channels (where M1 goes to 1, M2 to 2, M3 to 1, M4 to 2, and so on).
Mixdown To Mono	To one mono track.

Filename field Displays the result of the pattern displayed in the Filename Pattern field. Non-editable.

Filename Pattern field Displays how to name the exported media files. Build a dynamic naming scheme using Add Token and characters normally allowed in a file name. Create folder structure using / . The file extension is automatically appended. Add '#' to numerical tokens to define padding. Editable.

Add Token box Inserts a token in the Pattern field to build a dynamic filename.

Clip Options Tab

Copy Exported Clip in Media Library button Enable to automatically create a copy of the exported clip in the Media panel, linked to the exported media files. Available when performing a Sequence Publish with a File Sequence, with no resize or bit depth change. This is similar to importing back the exported media with Cache Source Media disabled.

Create Clip Data button Enable to create a .clip file for the exported sequence. A .clip contains extended metadata about the exported sequence, and enables versioning. The name of the .clip appears in the Filename field.

Clip Versions button Enable to version the exported sequence. This allows you to track revisions to the sequence.

Clip Filename field Name of the .clip created for the sequence. It displays the result from the Pattern field below.

Clip Filename Pattern field The pattern used to name the .clip created for the sequence. Use the Add Token box to use pre-defined tokens to build the name.

Create Shot Setup button Enable to export a batch setup and a .clip file for every segment with a shot name in the exported sequence. Only segments with non-empty Shot Name generate an associated .batch and .clip. For Shot Publish exports.

Shot Setup Filename field Example of the names given to the shot setup created each segment with a shot name. It displays the result from the Pattern field below.

Shot Setup Pattern field The pattern used to name each shot setup file. Use the Add Token box to use pre-defined tokens to build the name.

Shot .Clip Filename field Example of the names given to the shot .clip created each segment with a shot name. It displays the result from the Pattern field below.

Shot .Clip Pattern field The pattern used to name each shot .clip file. Use the Add Token box to use pre-defined tokens to build the name.

Output Path field Example of an output path and name used by the Write File node of a generated Batch setup. Displays the result of the Pattern field below.

Output Path Pattern field The pattern used to create the Output Path to ensure that the Write File node of each generated Batch setup uses a different path and name. Use the Add Token box to use pre-defined tokens to build the path.

Create Batch button Enable to create a Batch Group for each shot, back in the application at the location set in the Batch Group Path field.

Batch Group Path field Location in the Desktop where the Batch Group is created. Enabled by Create Batch.

Create Clip Nodes button Enable to create in the Batch Group reels matching clip nodes.

Supported File Formats

NOTE Flame cannot display 16-bit integer media (16i in the tables below). It automatically interprets it as 12-bit unpacked. Because of this, 16-bit integer media appears as 12-bit unpacked in the Preview panel. And when you import that media, Flame converts it to 12-bit unpacked. This process is perceptually negligible, as only the 4-least significant digits are discarded. But you can promote 16i media to 16f in order to maximize the precision available for compositing, by applying a colour transform. See Converting Images Between 12i or 16i and 16f Encodings (page 1804).

Also, if you use the Link Original Media video option during your export, image files that were not edited are not affected by this conversion.

Action-only Formats

Format	Extension	Import	Export	Note
3ds Max	.3ds	yes (page 699)	yes (page 846) (camera data only)	
Alembic	.abc	yes (page 699)	no	Supports version 1.5.4
Filmbox	.fbx	yes	yes (camera (page 846) or 3d camera (page 868) data only)	Supports up to FBX SDK 2017.0.1
Inventor	.iv	yes (page 699)	no	
Scalable Vector Graphics	.svg	yes (page 699)	no	
Substance Engine	Includes support for: ■ Substance engine compatible with Substance Designer 5.0 ■ Substance Materialize engine compatible with Substance Bitmap 2 Material (B2M) 2.2			

Format	Extension	Import	Export	Note
Wavefront	.obj	yes (page 699)	no	

Sequence Exchange Formats

Format	Extension	Import	Export	Note
Advanced Authoring Format	.aaf	yes (page 137)	yes	Supports importing Avid Media Composer AAF up to version 8.5. The exported AAF supports the following applications: ■ Avid Media Composer (version 8 and later) ■ Avid ProTools (version 12.2 and later) ■ Blackmagic Design DaVinci Resolve (version 11 and later) ■ Filmlight Baselight ■ Flame Family and Smoke applications AAF with embedded media are not supported.
Editorial Decision List	.edl	yes (page 133)	yes	■ Other supported formats: ■ ALE (Avid Log Exchange) —import only ■ CMX 340 ■ CMX 3600 —recommended format for import; default for export ■ CMX OMNI (ASCII version) ■ EDM (edit) —import only ■ GVG 4 ■ GVG 4 Plus (v4.1 or higher) ■ SONY 900 ■ SONY 910 ■ SONY 5000 ■ SONY 9000 ■ SONY 9000 Plus (v2.21 or higher) ■ SONY 9100 **NOTE** CMX 3600 is the default export format. To use another one, you must generate a customized EDL.
Final Cut Pro 7	.xml	yes (page 149)	no	Only supports Apple Final Cut Pro 7 xml files.

Format	Extension	Import	Export	Note
Final Cut Pro X	.fcpxml	yes (page 149)	yes	Supports up to version 10.2 of Final Cut Pro X.

Image Sequence

Format	Extension	Import	Export	Depth
Alias®	.als	yes (page 201)	yes	8 bits
ARRIRAW	.ari	yes (page 181)	no	12, 16fp bits
	NOTE Media is presented as a clip, never as a sequence of RAW images. Supports material shot on camera using ARRI SUP 11 or earlier, and the features from version 5.3 of the ARRIRAW SDK. Includes support for ALEXA Monochrome, ALEXA Open Gate, ALEXA mini, AMIRA, as well as the 3.2K format from the ALEXA SXT.			
Cineon®	.cin	yes (page 201)	yes	10 bits
DPX	.dpx	yes (page 196)	yes	8, 10, 12, 16i bits and ADX encoding.
	NOTE 8, 10, 12, and 16i bits support alpha, including 12-bit DPX (packed) from CODEX recorders.			
DPX - Single channel	.dpx	yes (page 196)	no	See Note.
	NOTE Monochromatic DPX files from the following film scanners have been validated: ■ FilmLight Northlight (10 & 16-bit) ■ DigitalFilmTechnology SCANITY™ (10 & 16-bit) ■ Imagica (8, 10 & 16-bit) ■ Laser Graphics (8, 10 & 16-bit) Lustre supports single channel DPX files only when imported through Wiretap Gateway.			
Open Clip	.clip	yes (page 206)	no	n/a
HDR	.hdr	yes (page 199)	no	32 bits
JPEG	.jpg	yes (page 201)	yes	8 bits
OpenEXR 2.2	.exr	yes (page 212)	yes	8, 10, 12u, 12, 16fp, or 32fp bits RGB, RGBA, and multi-channel

Format	Extension	Import	Export	Depth
	NOTE You can access RGB and RGBA channels from a multi-part file; each part appears as a track of a multi-track clip in the MediaHub file browser. Deep pixel channels are not supported and are not displayed. On import, 32-bit floating point is downconverted to 16-bit floating point. You can also access non-RGBA channels located at the root of the channels hierarchy. Flame reads frame rate and timecode information from the header of an OpenEXR file, when one of the following rates is used. It also supports writing that same information on export. ■ 23.97 fps ■ 24 fps ■ 25 fps ■ 29.97 fps ■ 30 fps 32-bit integer channels are seen as 12-bit clips in MediaHub.			
Photoshop	.psd	yes (page 216)	no	8 or 16i bits RGB and RGBA
	NOTE Supports both flattened and layered files. Rasterize any layer with shapes before saving the file to make sure those elements are available in Flame.			
Pict (Macintosh®)	.pict	yes (page 201)	yes	8 bits
	NOTE Supports alpha.			
Pixar	.pxr	yes (page 201)	yes	8 bits
	NOTE Supports alpha.			
Portable Network Graphics	.png	yes (page 217)	no	8 or 16i bits
	NOTE Supports alpha.			
SGI®	.sgi	yes (page 201)	yes	8 or 16i bits
	NOTE Supports alpha.			
Softimage®	.pic	yes (page 201)	yes	8 bits
	NOTE Supports alpha.			
TARGA®	.tga	yes (page 201)	yes	8 bits

Format	Extension	Import	Export	Depth
	NOTE Supports alpha.			
Tdi/Maya®	.iff	yes (page 201)	no	8 or 16i bits
	NOTE Supports alpha.			
Tiff	.tif	yes (page 201)	yes	8 or 16i bits
	NOTE Supports alpha.			
Wavefront®	.rla	yes (page 201)	yes	8 or 16i bits
	NOTE Supports alpha.			

QuickTime

Format	Extension	Import	Export	Depth
Uncompressed 8-bit RGB (RAW)	.mov	yes (page 218)	yes	
	NOTE Supports alpha both RGB and RGBA import and export.			
Uncompressed 8-bit VUY (2VUY)	.mov	yes (page 218)	yes	8-bit packed
	NOTE 4:2:2 codec.			
Uncompressed 8-bit YUV (YUV2)	.mov	yes (page 218)	yes	8-bit packed
	NOTE 4:2:2 codec.			
Uncompressed 10-bit RGB (R210)	.mov	yes (page 218)	yes	
Uncompressed 10-bit YUV (V210)	.mov	yes (page 218)	yes	10-bit packed
	NOTE 4:2:2 codec.			
Apple Animation	.mov	yes (page 218)	no	
	NOTE Supports alpha.			
Apple Graphics	.mov	yes (page 218)	no	

Format	Extension	Import	Export	Depth
Apple® Video	.mov	yes (page 218)	no	
Cinepak	.mov	yes (page 218)	no	
Component Y'Cb-Cr 8-bit 4:4:4	.mov	yes (page 218)	no	8-bit planar
Component Y'Cb-CrA 8-bit 4:4:4:4	.mov	yes (page 218)	no	8-bit planar
Component Y'Cb-Cr 10-bit 4:4:4	.mov	yes (page 218)	no	10-bit packed
Component Y'Cb-Cr 10-bit 4:2:2	.mov	yes (page 218)	no	10-bit packed
DV 25	.mov	yes (page 218)	yes	
	NOTE NTSC & PAL Although the specifications allow the DV format to be field 1 or 2, the industry standard is "bottom first". Thus, before exporting to Flame, ensure that the clip is Field 2. Reformat, if necessary.			
DVCPRO 50	.mov	yes (page 218)	yes	
	NOTE NTSC & PAL			
DVCPRO HD	.mov	yes (page 218)	yes	
DNxHD	.mov	yes (page 218)	yes	8 and 10 bits
	NOTE Includes support for the following varieties: 36, 60, 75, 80, 85, 90, 90x, 100, 110, 110x, 115, 145, 175, 175x, 185, 185x, 220, 220x, 444 (sometimes known as 350x).			
DNxHR	.mov	yes (page 218)	yes	8, 10, or 12 bits
	NOTE Supports the DNxHR family: 444, HQX, HQ, SQ, and LB. Supports import of Avid-created DNxHR files that are 10- or 12-bit.			
H.264	.mov	yes (page 218)	yes	
IMX	.mov	yes (page 218)	yes	
	NOTE Includes support for IMX 30, 40, and 50. When exporting, the Scan Mode box is ignored: NTSC clips are always exported as F2 dominant, while PAL clips are always exported as F1 dominant.			

Format	Extension	Import	Export	Depth
MJPEG	.mov	yes (page 218)	yes	
	NOTE JPEG compatible			
MPEG-1	.mov	yes (page 218)	no	
MPEG-4	.mov	yes (page 218)	yes	
MSMpeg 4v3 (Di-vX)	.mov	yes (page 218)	no	
PhotoJPEG	.mov	yes (page 218)	no	
	NOTE RT PhotoJPEG compatible			
PNG	.mov	yes (page 218)	yes	
	NOTE Without alpha channel support			
PNGA	.mov	yes (page 218)	yes	
	NOTE With alpha channel support			
ProRes 4444 (XQ)	.mov	yes (page 218)	yes	12-bit
ProRes 4444	.mov	yes (page 218)	yes	12-bit
	NOTE Includes support for ProRes files from Sony F5 and F55 cameras.			
ProRes 422 (HQ)	.mov	yes (page 218)	yes	10-bit
	NOTE Includes support for ProRes files from Sony F5 and F55 cameras.			
ProRes 422	.mov	yes (page 218)	yes	10-bit
	NOTE Includes support for ProRes files from Sony F5 and F55 cameras.			
ProRes 422 (LT)	.mov	yes (page 218)	yes	10-bit
ProRes 422 (Proxy)	.mov	yes (page 218)	yes	10-bit

Format	Extension	Import	Export	Depth
Quicktime Planar RGB	.mov	yes (page 218)	no	
TGA	.mov	yes (page 218)	no	
XDCAM	.mov	yes (page 218)	no	
XDCAM HD	.mov	yes (page 218)	no	
XDCAM EX	.mov	yes (page 218)	no	
	NOTE There is no essence-mode browsing in a QuickTime XDCAM EX.			

The Supported QuickTime Audio (page 125) table lists the audio codecs supported in QuickTime files.

IMPORTANT Movie files that contain multiple compressions are not supported and may display as checkerboard when scrubbed in MediaHub. Such movie files can be created when exporting a sequence made of different DNxHD / DNxHR media from Avid Media Composer, using the *Same as Source* export option.

Material Exchange Format

Format	Extension	Import	Export	Depth
AVC-Intra 50	.mxf	yes (page 214)	yes	
	NOTE Panasonic P2.			
AVC-Intra 100	.mxf	yes (page 214)	yes	
	NOTE Panasonic P2. Import includes support for 1080/60p. Import supports OP-Atom format.			
AVC-Intra 200	.mxf	yes (page 214)	no	
	NOTE Panasonic P2. Supports 1080/25p, 1080/30p, 720/50p, and 720/60p.			
AVC-Intra 4:2:2 HD	mxf	yes (page 214)	no	
AVC-Intra LT	mxf	yes (page 214)	no	
	NOTE AVC-Intra LT is encoded half-height to reduce data rate. Enable, in Format Specific Options for Panasonic, the option **Image ➤ Scale to Full Resolution** to import the media at its full resolution.			
AVC-Intra2K	.mxf	yes (page 214)	no	

Format	Extension	Import	Export	Depth
	NOTE Supports the VariCam 35 and HS, both 4:2:2 and 4:4:4 flavours.			
AVC-Intra4K	.mxf	yes (page 214)	no	
	NOTE Supports the VariCam 35 and HS, both 4:2:2 and 4:4:4 flavours.			
AVC-LongG	.mxf	yes (page 214)	no	
Canon XF-AVC	.mxf	yes (page 193)	no	
	■ Note: ■ Supports both intra and Long GOP flavours ■ Supports spanned clips.			
DNxHD	.mxf	yes (page 204)	yes	
	NOTE ■ Includes support for the following varieties: 36, 60, 75, 80, 85, 90, 90x, 100, 110, 110x, 115, 145, 175, 175x, 185, 185x, 220, 220x, 444 (sometimes known as 350x). ■ Supports DNxHD in .mxf files from ARRI ALEXA cameras. ■ Supports OP-1a media files from Blackmagic Design DaVinci Resolve (DNxHD 100 and 444). ■ Supports DNxHD 220x and 145 from Sony PMW-F5 and F55. ■ In MXF Op-Atom files generated by Avid Media Composer, audio tracks appear in the MediaHub as a single audio channel file (A1). But once imported, the tracks display the original channels.			
DNxHR	.mxf	yes (page 204)	no	8, 10, or 12 bits
	NOTE ■ Supports the DNxHR family: 444, HQX, HQ, SQ, and LB. ■ Supports import of Blackmagic Design DaVinci Resolve DNxHR files, but only if they are OP-1a (OP-Atom files from DaVinci Resolve are *not* supported). ■ Supports import of Avid-created DNxHR files that are 10- or 12-bit, using OP-1a or OP-Atom.			
DV 25	.mxf	yes (page 204)	no	
	NOTE Panasonic P2			
DVCPRO	.mxf	yes (page 204)	yes	
	NOTE Panasonic P2			

Format	Extension	Import	Export	Depth
DVCPRO 50	.mxf	yes (page 204)	yes	
	NOTE Panasonic P2 (PAL & NTSC)			
DVCPRO HD	.mxf	yes (page 204)	yes	
	NOTE Panasonic P2. Available in 1080p@25/50, 1080p@24/30/60, 720p@25/50, and 720p@24/30/60.			
SonyRAW	.mxf	yes (page 227)	no	
	NOTE Flame uses Sony's SDK to decode SonyRAW files created with Camera Firmware 7.0. It includes support for Sony F65, F55, and F5 camera outputs. Supports ISO / Exposure Index information.			
Sony SStP	.mxf	yes (page 230)	yes	
	NOTE Sony's implementation of MPEG-4 Simple Studio Profile, part of the SRMASTER format. ■ Supported Sony SStP formats: ■ SR-Lite (10-bit / YUV 422 / 220 Mbps) ■ SR-SQ (10-bit / YUV 422 / 440 Mbps) ■ SR-SQ (10-bit / YUV 444 / 440 Mbps) ■ SR-HQ (10-bit or 12-bit RGB 444 / 880 Mbps) These formats are supported in 1280x720 (50p, 59.94p), 1920x1080 (23.97p, 24p, 25p, 25i, 29.97p, 29,97i, 50p, 59,94p), 2048x1080 (23,97/24/25PsF) with up to 16 tracks of PCM audio.			
XAVC	.mxf	yes (page 225)	yes	
	NOTE Includes support for Sony F55 and F5 camera, recorded in HD and 4K. Includes support for Long GOP files (Sony XAVC-Long format), as recorded by cameras such as the PXW-Z100.			
XDCAM	.mxf	yes (page 225)	no	
	NOTE MPEG-2 IMX-30, IMX-40, and IMX-50			
XDCAM HD	.mxf	yes (page 225)	yes	
	NOTE MPEG-2 Long GOP. Import supports 4:2:0 and 4:2:2. Export is 4:2:2.			

NOTE Flame exports MXF as OP-1a files (including the timecode). The is audio encoded as PCM, 16-Bit or 24-Bit. Avid applications support OP-Atom MXF files: use the AMA MXF plug-in to bring the OP-1a files into an Avid application.

MP4

Format	Extension	Import	Export	Depth
H.264	.mp4	yes (page 202)	no	
XAVC-S	.mp4	yes (page 202)	no	
	NOTE Supports up to 4 tracks of 16-bit PCM audio.			
XDCAM EX	.mp4	yes (page 231)	no	
	NOTE MPEG-2 Long GOP. Includes supports for High Frame Rate (HFR) media—50p, 59.94p, 60p.—			

Other Streaming Codecs

Format	Extension	Import	Export	Depth
AVCHD	.mts or .m2ts	yes (page 192)	no	
	Only linear PCM audio is supported. Some cameras can record AC-3 audio, but this format is not supported: Flame can still access the video portion of the media.			
REDCODE	.r3d	yes (page 220)	no	12, 16fp bits
	Flame uses RED SDK 6.2.1, including support for the RED HELIUM sensor, REDWideGamutRGB color space, and for the RED Weapon (8K) and RED Raven (4K) cameras.			

Audio File Formats

Format	Extension	Import	Export	Depth
AIFF	.aiff	yes	yes	16 or 24 bits
AIFF-C	.aifc	yes	yes	16, 24, 32fp bits
Audio Visual Research	.avr	yes	yes	16 bits
Berkeley/IRCAM/CARL Sound (BISCF)	.bsf	yes	yes	16 bits
MP3	.mp3	yes	yes	16 bits
Nextsnd	.au	yes	yes	16, 24, 32fp bits
WAVE	.wav	yes		16, 24, 32fp bits
WAVE - Broadcast	.wav	yes	yes	16, 24, 32fp bits

Format	Extension	Import	Export	Depth
	NOTE Includes support for RF64 files (BWF-compatible format that supports files larger than 4 GB).			
WAVE - Extensible	.wav	yes	no	16, 24, 32fp bits
	NOTE Audio tracks are imported as regular audio tracks, without mapping the channels to spatial locations.			

NOTE Files of any sample rate can be imported, but they are all resampled to 48 kHz.

Supported QuickTime Audio

The table below lists the audio codecs supported in .mov files, as audio-video and audio-only files.

Audio CODEC	Import	Export
16-bit PCM	**yes**	**yes**
	NOTE Export supports both big and small endian.	
24-bit PCM	**no**	**yes**
	NOTE Export supports both big and small endian.	
32-bit floating point PCM	**no**	**yes**
	NOTE Export supports both big and small endian.	
32-bit float	**no**	**yes**
A-law 2:1	**yes**	**yes**
ADPCM ima WAV	**yes**	**no**
Advanced Audio Codec (AAC)	**yes**	**yes**

Audio CODEC	Import	Export
	NOTE Supported in mp4 and m4v files. ■ On export, the following data rates are supported: ■ 56 kbps (mono only) ■ 96 kbps ■ 160 kbps ■ 224 kbps ■ 288 kbps ■ 320 kbps (stereo only) ■ 448 kbps (stereo only) ■ 576 kbps (stereo only)	
Apple lossless	**yes**	**no**
IMA 4:1	**yes**	**yes**
Linear PCM (QT 7)	**yes**	**yes**
MPEG-2 Layer 2 Audio	**yes**	**yes**
MS ADPCM	**yes**	**no**
Ogg Vorbis (qt4l compatible)	**yes**	**no**
Ogg Vorbis (qtcomponents compatible)	**yes**	**no**
QDM2 Audio	**yes**	**no**
Raw 8-bit audio	**yes**	**no**
Sowt	**yes**	**yes**
	NOTE 16-bit PCM (Little Endian)	
Twos	**yes**	**yes**
	NOTE 16-bit PCM (Big Endian)	
Ulaw	**yes**	**yes**

Pixspan Support

Flame supports Pixspan 3 compression, when importing and exporting media.

Pixspan is a lossless compression algorithm that preserves the original media container and metadata while compressing the original media file.

Pixspan enables the following formats to be **decoded on import and encoded on export**:

- DPX
- Cineon
- EXR (Uncompressed only)
- TIFF (Uncompressed only)

Pixspan enables the following formats to be **decoded on import**:

- Pixspan encoded ARRIRAW media files

It is also possible to export content with the Link To Original Media option, to avoid duplicating media files on the destination file system. Compressed Intermediates with Pixspan compression is not supported.

Files encoded with Pixspan compression have a double extension: the original extension, plus ".pxz" (ex. dpx.pxz).

The Pixspan plugin is included with the application and supports encoding, out of the box. If you want to use Pixspan for decoding as well, you will have to properly license the plugin.

Files exported through Pixspan generate thumbnails when encoded using Autodesk products. These thumbnails can be seen under the Mac OS X Finder, making media browsing easier. This requires a Mac OS X QuickTime component available from Pixspan.

NOTE The Pixspan license server must run on Mac OS X or Red Hat Linux: it does not support CentOS.

For more information on Pixspan, visit: *http://pixspan.com*.

MediaReactor Support

Drastic Technology MediaReactor is an import plugin for Wiretap Gateway, that allows the ingest of multiple file formats not currently supported by Autodesk products. Among others, the following formats are available:

- Vision Research Phantom CineRAW (.cine)
- Cinema DNG (AJA CION, Blackmagic cameras, etc) (.dng)
- Canon C500 RAW (.rmf)
- Panasonic Varicam RAW (.vraw)
- Jpeg2000 (MXF)

The Format Specific options for MediaReactor are as follows:

- Metadata (Tape Name, Timecode and Frame Rate options).
- Clip Options (Clip name and Sequence Detection options).
- Debayering (RAW formats options) Various Colour settings for RAW media decoding. The default option is set to use the File Header information.

MediaReactor supports many more file formats than those listed above; including Cineform DPX, Flash Video, HDV, Matroska (MKV), Mpeg 1/2/4, Avid DV25, DV50, HDV, Uncompressed (MXF), Windows Media,

WebM and many more. However, only the formats listed above have been validated and are officially supported.

You must download and install the Drastic plugin, *available here*. Once installed, you can select MediaReactor from the File Format box in the MediaHub and browse to, and import the above file formats.

NOTE After installing the MediaReactor plugin, you must restart the Wiretap Gateway, Stone&Wire and Flame.

For more information on Drastic, visit: *www.drastic.tv*.

Linux installation

Installation on Linux

1. Exit Flame.
2. Download the file to the Linux workstation. /usr/tmp is a good spot.
3. Open a shell.
4. Change directory to /usr/tmp. Type: `cd /usr/tmp 5. Unzip the file`.Type: `unzip SetupMRWS_Linux-x86_64_2016_Beta.zip` and wait for the process to finish.
5. You now need to find the unzipped file. Type: `ls SetupMRWS_Linux`. Look for the file that does not end with .zip. Note that name. In the following steps, we will use SetupMRWS_Linux-x86_64_5_0_154 as an example: you will need to change the name to reflect your situation.
6. Change the file permissions. Type: `chmod 777 SetupMRWS_Linux-x86_64_5_0_154`.
7. Launch the installer. Type: `./SetupMRWS_Linux-x86_64_5_0_154 8`. Follow the on-screen instructions.
8. Close the shell once the installation is complete.

Once the MediaReactor component is installed, you need to restart Wiretap Gateway, Wiretap Server, and Stone & Wire.

1. In the shell, type: `ServiceMonitor`
2. In the ServiceMonitor window, restart Wiretap Server and Stone+Wire (this also restarts the Wiretap Gateway).

   NOTE For Lustre, only the Wiretap Gateway needs to be restarted. From the Editing / Browse menu, click Restart WTG.

Once all three services are green, you are good to go and relaunch Flame.

Uninstallation on Linux

1. Exit Flame.
2. Open a shell.
3. Change directory the MediaReactor install directory. Type: `cd /opt/mediareactor`
4. Launch the uninstaller. Type: `./uninstall`
5. Once MediaReactor is uninstalled, you must restart both Wiretap Gateway, Wiretap Server, and Stone+Wire. See the instructions above.
6. Relaunch Flame.

Mac Installation

Installation on Mac OSX

1. Exit Flame.
2. Download the MediaReactor setup file, and unzip it.
3. Launch the setup.
4. Restart Wiretap Gateway, Wiretap Server, and Stone+Wire using the Service Monitor. It is found in Applications / Autodesk / [your_application] / Utilities.
5. Restart Flame.

Uninstallation on Mac OSX

1. Exit Flame.
2. In the Finder, open Applications / MediaReactor.
3. Launch "uninstall".
4. Once the uninstall is done, you must Restart Wiretap Gateway, Wiretap Server, and Stone+Wire using the Service Monitor. It is found in Applications / Autodesk / [application] / Utilities.
5. Restart Flame.

NOTE You can install a new MediaReactor over an older version: you do not need to uninstall the older version beforehand.

About Supported XDCAM Formats

List of supported data rates and timings for Sony XDCAM HD, XDCAM EX, and XDCAM 422 codecs.

Import: Supported XDCAM Formats

XDCAM 422 (50 Mbps):

- 1080/50i
- 1080/60i
- 1080/24p
- 1080/25p
- 1080/30p
- 720/24p
- 720/25p
- 720/30p
- 720/50p
- 720/60p

XDCAM EX (35 Mbps):

- 1080/50i
- 1080/60i
- 1080/24p
- 1080/25p

- 1080/30p
- 720/24p
- 720/25p
- 720/30p
- 720/50p
- 720/60p

XDCAM HD (35 Mbps):

- 1080/50i
- 1080/60i
- 1080/24p
- 1080/25p
- 1080/30p

Export: Supported XDCAM Formats

XDCAM HD (MPEG-2 MP@HL LongGOP)

- 4:2:0 colour sampling
- 17 Mbps (Variable Bit Rate), 25 Mbps (Constant Bit Rate), 35 Mbps (Variable Bit Rate)
- MXF OP-1a
- Supported resolutions and associated rates:
  - 1280x720p (50 and 59.94)
  - 1440x1080i (25, 29.97)
  - 1440x1080p (23.97, 25, 29.97)
- Available movie export presets:
  - MXF (XDCAM HD 720p 8-bit)
  - MXF (XDCAM HD 1080i 8-bit)
  - MXF (XDCAM HD 1080p 8-bit)

XDCAM HD 422 (MPEG-2 422P@HL LongGOP)

- 4:2:2 colour sampling
- 50 Mbps (CBR)
- MXF OP-1a
- Supported resolutions and associated rates:
  - 720p (50, 59.94)
  - 1080i (25 and 29.97)
  - 1080p (23.97, 25 and 29,97)
- Available movie export presets:
  - MXF (XDCAM HD422 720p 8-bit)
  - MXF (XDCAM HD422 1080i 8-bit)
  - MXF (XDCAM HD422 1080p 8-bit)

XDCAM EX (MPEG-2 MP@HL LongGOP)

- 4:2:0 colour sampling
- 25 Mbps (CBR) & 35 Mbps (VBR)
- MXF OP-1a
- Supported resolutions and associated rates:
  - 720p (23.98, 25, 29.97, 50, 59.94)
  - 1080i (50i and 59.94i)
  - 1080p (23.97, 25 and 29,97)
  - 1080i Thin raster (50 and 59.94)
  - 1080p Thin raster (23.97)
- Available movie export presets:
  - MXF (XDCAM EX 720p 8-bit)
  - MXF (XDCAM EX 1080i 8-bit
  - MXF (XDCAM EX 1080p 8-bit)

NOTE The bit rate is defined by the selected codec and cannot be changed by users.

- XDCAM EX 1080i / 1080p / 720p: 35 Mbps
- XDCAM EX 1080 (Thin Raster): 25 Mbps

End-to-end 4K and UltraHD Support

SDI Preview - Linux

Flame and Lustre support 4K Preview timings and UltraHD 4K footage (UHDTV).On systems with an AJA Kona 3G card or an AJA Kona 4G card, the following timings are supported:

AJA Kona 3G:

- 3840x2160 (23,97p, 24p, 25p, 29.97p, 30p)
- 4096x2160 (23,97p, 24p, 25p, 29.97p, 30p)

AJA Kona 4G:

- 3840x2160 (23,97p, 24p, 25p, 29.9p, 30p, 50p, 59.94p, 60p)
- 4096x2160 (23,97p, 24p, 25p, 29.9p, 30p, 50p, 59.94p, 60p)

These timings can be selected at project creation and from the Preferences / Broadcast Monitor menu, if connected to a broadcast monitor.

NOTE

- In order to use the AJA Kona 3G card in Flame 2015, you must update the card's firmware to Firmware version "2014/03/19 07:22:10". A script is installed on the workstation when you install the application (in `/opt/Autodesk/flame_2015/scripts/cfgAjaUpdate` (Flame), that enables you to upgrade and downgrade the AJA Kona 3G's firmware. To run the script to upgrade the firmware for 2015, from a shell, type the following command: `cfgAjaUpdate 2015`. Once the card is upgraded, it is recognized as AJA Kona 3G **Quad**.
- Once you have upgraded the card to the AJA Kona 3G Quad firmware, it is no longer be useable with pre 2015 versions of Flame. If you are running a pre 2015 version of Flame alongside a 2015 version, you must run this script again, with the [pre2015] variable to downgrade the AJA Kona 3G' Quads firmware. To run the script to downgrade the firmware for pre 2015, from a shell, type the following command: `cfgAjaUpdate pre2015`.
- If you have downgraded the card's firmware and want to use it again in Flame 2015, you must re-update the card's firmware, by running the following command: `cfgAjaUpdate 2015`.
- After running the script, you must power down and reboot your workstation, to make sure the appropriate AJA board firmware is properly installed.
- Upgrading to Flame 2015 overwrites the init.cfg file to make the new timings available. The new timings must be uncommented in the init.cfg file to be available in the application. You must manually modify the init.cfg to reset the edited options or you can use the Flame Setup application.

SDI Preview - Mac OS X

- The following timings are available when using Blackmagic Design SDI I/O and Preview cards on Mac OS X. Availability based on hardware capabilities. See Blackmagic Design's website for more details:
  - Support for Single Link 4:4:4 RGB 3G for all HD timings;
  - Support for 2K DCI (2048x1080) 23.98, 24, 25, 29.97 & 30;
  - Support for UHD (3840x2160) 23.98, 24, 25, 29.97 & 30;
  - Support for 4k (4096x2160) 23.98, 24, 25, 29.97 & 30;
  - Support for HFR timings for AJA IoXT 4K on OS X - UHDTV/4K 50p, 59,94 and 60p.
- It is possible to monitor 8 audio tracks over HDMI, using AJA cards.
- Blackmagic Design HDMI audio monitoring is limited to two tracks.

GPU RAW Media Decoding

Flame and Lustre support GPU RAW Media Decoding for the ARRIRAW and RED file formats, providing much faster decoding speeds. GPU RAW Media Decoding is enabled by default, if the following conditions are met:

- GPU RAW Media Decoding requires the NVidia Quadro 6000, Quadro M6000 or Quadro K6000.
- You must update to the latest DKU.
- For ARRIRAW media, Debayering must be set to ADA-SW5. This is the default setting.

  NOTE In Lustre, the new ADA 5 SW Sharpening controls are located in Image / Transcode / Format menu. As always, you can copy these settings using the Selector.

- For RED, Colour Science must be set to version 3.x. This is the default setting.

About GPU Decoding and Wiretap Gateway Slaves

With GPU decoding, the concept of Wiretap Gateway local decoding slaves is being slowly phased out.

The slaves mechanism had a significant impact on performances for R3D CPU Decoding. No slaves will be used by default for R3D decoding when a GPU decoding capable system is detected (Q6000 and above) unless NumLocalSlaves is explicitly set to a non zero value (it is commented out by default). See the comments in the [Slaves] section of the file /opt/Autodesk/wiretapgateway/cfg/wiretapgateway.cfg.

Keep in mind that, as mentionned in the wiretapgateway.cfg file, R3D files using Color Science 2.x or HDRx cannot be GPU-decoded, in which case you will need to configure NumLocalSlaves.

Importing an EDL Sequence

To import an EDL from the MediaHub:

1. Open MediaHub, and set it to Browse for Files.
2. Review the EDL Import Options.

   An EDL has neither frame rate nor resolution. Unless you specify something in the EDL Options, Flame assigns to the EDL the project's resolution and frame rate.
3. Using the file browser, navigate to the EDL sequence to import.
4. Drag the file from the browser to the Media Library.

   Flame converts the EDL to its timeline format.

You can also import a sequence from Conform:

1. Open Conform.
2. Click in the Media Panel the destination for the imported sequence.
3. Right-click the Events list, and select Load New FCP XML/AAF/EDL...
4. Review the EDL Import Options.

   An EDL has neither frame rate nor resolution. Unless you specify something in the EDL Options, Flame assigns to the EDL the project's resolution and frame rate.
5. In the Media Import window, locate the sequence to import using the file browser, and click Import.

An EDL has no concept of track. To get around this limitation, one can export an EDL per track, and have Flame combine these related EDL into a single sequence. This process is called EDL multi-assemble.

To assemble multiple EDL into one sequence from the MediaHub:

1. Open MediaHub, and set it to Browse for Files.
2. Set the Clip Display box to List.
3. Review the EDL Import Options.

   An EDL has neither frame rate nor resolution. Unless you specify something in the EDL Options, Flame assigns to the EDL the project's resolution and frame rate.
4. Enable Multi-Assemble in the EDL Options.
5. Set the EDL Detection box to Automatic Detection.
6. Using the file browser, navigate to the EDL sequences to import.

   Flame groups the sequences logically to recreate the sequence (or sequences if it detects there is more than one).
7. Drag the aggregate sequence from the browser to the Media Panel.

Flame converts the EDL to its timeline format.

You can select multiple sequences to import, using `Shift-click` and `Ctrl-click`.

If a source is used multiple times in a sequence, or across multiple sequences, and Save Sources Separately is enabled, only one instance of the source is imported. And the sources are saved in a Sources folder.

If you plan on perform a Connected conform, you must import your sequences into a Reel group. Make sure to drag and drop the sequences on top of a Reel group, or to import from Conform. When you import your sequences through Conform, they are always added to the Sequences reel of the default Reel group, making sure you will be able to use the Sources and Shots sequences required for the Connected conform workflow.

If Multi-Assemble Automatic Detection does not work:

- Make sure that you are in List view within the file browser.
- Set the EDL Detection box to Selection Order, and then **Ctrl-click** to select multiple EDL sequences. The order in which you add the EDL to the selection defines the video tracks: the EDL selected first goes to the first video track, the EDL selected next goes to the second video track, and so on.
- Set the EDL Detection box to Custom Grouping. Once you have selected the EDL to import, click Import. In the windows that appears, set the sequence names, versions, and tracks for each of the EDL. Once set, click Import Sequences.

About EDL Files

An EDL (Edit Decision List) file is a digital list of commands used to describe a series of film or video edits. It consists of an ordered list of reel, keycode, and timecode data, which represents the original location (usually video tapes or sequences of images stored on disk) of each media clip used in the edit. These media clips can later be easily obtained in order to conform the final cut. EDLs are the primary method of transferring information about an edit project between offline and online editing systems.

The most primitive EDL formats are telecine log files. Telecine log files typically describe the most basic edit decisions, which contain only tape name, clip name, source timecode, keycode, and whether 2:3 pulldown was used.

Standard EDL formats, such as CMX3600, support basic functions, such as cuts, dissolves, SMPTE wipes, speed variations (including freeze frames), and direction changes (Forward/Reverse). EDL comments, clip names, and audio patching comments are also supported.

AAF and FCP XML file formats are more modern, enhanced formats of EDLs, supporting multiple layers of video and audio, as well as an expanded set of transitions and effects. Some may include embedded audio.

Collectively, all these formats are known as File Interchange® Formats, but they are still commonly referred to as EDLs.

Supported EDL Formats

The following Standard EDL formats are supported by Flame:

- CMX 340
- CMX 3600
- CMX OMNI (ASCII version)
- GVG 4
- GVG 4 Plus (v4.1 or higher)
- SONY 900
- SONY™ 910
- SONY 5000
- SONY 9000
- SONY 9000 Plus (v2.21 or higher)
- SONY 9100
- ALE (Avid Log Exchange)

■ EDM (edit)

NOTE It is recommended that you use EDLs in the CMX 3600 format.

About the DLEDL Format

DLEDL is an EDL file based on the CMX 3600 format supported by most digital nonlinear editing systems. DLEDLs are unique to Autodesk Flame applications and contain extra comments that they can interpret, such as media file locations. Using DLEDLs in your post-production workflow allows you to automate the import of media files (video and audio) residing in different locations. Instead of performing media import for each file separately, you can use a DLEDL to import all media files in a single operation.

DLEDL Structure

The following table lists the DLEDL flags used in creating an EDL file. It describes the comments used in creating DLEDLs and provides examples of basic EDLs that you can use in your projects.

Note that the order of flags always remains the same:

- Clip name (EDL comment)
- Start TC
- File path
- Video file name
- Audio 1 file name
- Audio 2 file name
- Audio 3 file name
- Audio 4 file name

DLEDL Flag Example	Description
DLEDL: START TC: 10;00;00;00	The start timecode of the clip that will be created from the media files imported into Flame. This timecode can be different from the source start timecode of the EDL event. Both drop-frame (00:00:00:00) and non-drop-frame (00;00;00;00) timecodes are supported.
DLEDL: PATH /CXFS1/PRODUCTION1/SHOW5	The location of the files on the filesystem. All the files for an event must be in the same directory.
DLEDL: EDIT:0 FILENAME: INTRO .(010@240).dpx or DLEDL: EDIT:0 FILENAME: (009@100).dpx	Name of the video file sequence to load. Numbers in parenthesis refer to frames in the sequence. In this example, the following files are referenced: *INTRO.010.dpx*, *INTRO.011.dpx*, ..., *INTRO.240.dpx* or *009.dpx*, *010.dpx*, ..., *100.dpx*
DLEDL: EDIT:0 FILENAME: TONE-8TRK-44KHZ.AIFF_CHANNEL_1.AIF	Audio file name to import. The audio files can contain one file per track (_1), one file per stereo pair (_12), or one file

DLEDL Flag Example	Description
	per multiple tracks (_1234). Supported formats are wave (.WAV) or AIFF (.AIF).
DLEDL: REEL: 12345678 123456781234556	The CMX EDL format limits the length of the tape name to eight characters. Some editing systems do not impose a tape name length limit. If the tape name exceeds eight characters, Flame generates a short name to be used in the EDL and stores the original name in the Source Table list at the end of the EDL file (one Source Table per EDL).

NOTE File names, tape names, and clip names cannot contain spaces or special characters. For example, TAPE 1 should be set to TAPE_1 (no space) and CLIP$,12/2 to CLIP_12_2 (no special characters).

Using DLEDLs Examples

The following examples are given to facilitate the creation of edit decision lists for the automatic import of multiple media files into a project. You can simply copy and paste the suitable example into a text editor, modify it accordingly, save the file with an *.edl* extension, and import the resulting file into your application.

Video-only EDL

```
TITLE: SIMPLE EDL VIDEO ONLY
FCM: DROP FRAME
001 TAPENAME V C 10:00:00:00 10:00:05:00 00:00:30:00 00:00:35:00
* FROM CLIP NAME: clip
DLEDL: START TC: 10;00;00;00
DLEDL: PATH: /CXFS1/PRODUCTION1/SHOW5
DLEDL: EDIT:0 FILENAME: INTRO.(010@240).dpx
DLEDL: REEL:TAPENAME TAPENAMEVERYLONG
```

Simple Video and Audio (2 Tracks) EDL

```
TITLE: SIMPLE EDL WITH VIDEO AND 2 AUDIO TRACKS (1 TRACK PER FILE)
FCM: DROP FRAME
001 TAPENAME AA/V C 10:00:00:00 10:00:05:00 00:00:30:00 00:00:35:00
* FROM CLIP NAME: TONE-8TRK-44KHZ.AIFF
DLEDL: START TC: 10;00;00;00
DLEDL: PATH: /CXFS1/PRODUCTION1/SHOW5
DLEDL: EDIT:0 FILENAME: INTRO.(010@240).dpx
DLEDL: EDIT:0 FILENAME: TONE-8TRK-44KHZ.AIFF_CHANNEL_1.AIF
DLEDL: EDIT:0 FILENAME: TONE-8TRK-44KHZ.AIFF_CHANNEL_1.AIF
DLEDL: REEL:TAPENAME TAPENAMEVERYLONG
```

Simple Video and Audio (2 Tracks - Stereo) EDL

```
TITLE: SIMPLE EDL WITH VIDEO AND 2 AUDIO TRACKS (2 TRACKS PER FILE)
FCM: DROP FRAME
001 TAPENAME AA/V C 10:00:00:00 10:00:05:00 00:00:30:00 00:00:35:00
* FROM CLIP NAME: TONE-8TRK-44KHZ.AIFF
DLEDL: START TC: 10;00;00;00
DLEDL: PATH: /CXFS1/PRODUCTION1/SHOW5
DLEDL: EDIT:0 FILENAME: INTRO.(010@240).dpx
```

```
DLEDL: EDIT:0 FILENAME: TONE_12.AIF
DLEDL: REEL:TAPENAME TAPENAMEVERYLONG
```

Simple Video and Audio (4 Tracks) EDL

```
TITLE: SIMPLE EDL WITH VIDEO AND 4 AUDIO TRACKS (1 TRACK PER FILE)
FCM: DROP FRAME
001 TAPENAME AA/V C 10:00:00:00 10:00:05:00 00:00:30:00 00:00:35:00
* FROM CLIP NAME: TONE-8TRK-44KHZ.AIFF
DLEDL: START TC: 10;00;00;00
DLEDL: PATH: /CXFS1/PRODUCTION1/SHOW5
DLEDL: EDIT:0 FILENAME: INTRO.(010@240).dpx
DLEDL: EDIT:0 FILENAME: TONE-8TRK-44KHZ.AIFF_CHANNEL_1.AIF
DLEDL: EDIT:0 FILENAME: TONE-8TRK-44KHZ.AIFF_CHANNEL_2.AIF
DLEDL: EDIT:0 FILENAME: TONE-8TRK-44KHZ.AIFF_CHANNEL_3.AIF
DLEDL: EDIT:0 FILENAME: TONE-8TRK-44KHZ.AIFF_CHANNEL_4.AIF
AUD 3 4
DLEDL: REEL:TAPENAME TAPENAMEVERYLONG
```

Simple Video and Audio (4 Tracks - Stereo) EDL

```
TITLE: SIMPLE EDL WITH VIDEO AND 4 AUDIO TRACKS (2 TRACKS PER FILE)
FCM: DROP FRAME
001 TAPENAME AA/V C 10:00:00:00 10:00:05:00 00:00:30:00 00:00:35:00
* FROM CLIP NAME: TONE-8TRK-44KHZ.AIFF
DLEDL: START TC: 10;00;00;00
DLEDL: PATH: /CXFS1/PRODUCTION1/SHOW5
DLEDL: EDIT:0 FILENAME: INTRO.(010@240).dpx
DLEDL: EDIT:0 FILENAME: TONE_12.AIF
DLEDL: EDIT:0 FILENAME: TONE_34.AIF
AUD 3 4
DLEDL: REEL:TAPENAME TAPENAMEVERYLONG
```

Importing an AAF Sequence

To simplify the conform process, create the AAF project on the workstation running Flame. And when exporting the AAF, save the AAF to the root of the media used in that timeline: the media should either be with the AAF, or within a folder alongside the AAF.

To simplify the conform process, when conforming on a Linux workstation, save the AAF to the root of the media used in that timeline: the media should either be with the AAF, or within a folder alongside the AAF. If you are conforming on a Mac, create the AAF project on that Mac. And when exporting the AAF, save the AAF to the root of the media used in that timeline: the media should either be with the AAF, or within a folder alongside the AAF.

To import a sequence using the MediaHub:

1. Open MediaHub, and set it to Browse for Files.
2. Review the AAF & XML Import Options. And because you are importing an AAF, pay attention to Preferred Media: if offline intermediates were used during the offline editing, decide now whether you wish to relink to the original media or to the offline intermediates.

   NOTE By default, the sequence is imported at the resolution specified in the FCP XML, not the project's. You can set a resolution with the Sequence Resolution box.

3 Using the file browser, navigate to the AAF sequence to import.

You can view the frame rate from the Preview Panel.

4 Drag the file from the browser to the Media Library.

Flame converts the AAF to its sequence format. According to the Media and Relative Search options, the sequence can be relinked or not, and the sources also imported, or not. The media itself is imported using the option file format options defined in the Format Specific Options tab.

You can also import a sequence from Conform:

1 Open Conform.
2 Click in the Media Panel the destination for the imported sequence.
3 Right-click the Events list, and select Load New FCP XML/AAF/EDL...
4 Set the AAF & XML Import Options.
5 In the Media Import window, locate the sequence to import using the file browser, and click Import.

You can select multiple sequences to import, using `Shift-click` and `Ctrl-click`.

If a source is used multiple times in a sequence, or across multiple sequences, and Save Sources Separately is enabled, only one instance of the source is imported. And the sources are saved in a sources folder.

If you plan on perform a Connected conform, you must import your sequences into a Reel group. Make sure to drag and drop the sequences on top of a Reel group, or to import from Conform. When you import your sequences through Conform, they are always added to the Sequences reel of the default Reel group, making sure you will be able to use the Sources and Shots sequences required for the Connected conform workflow.

NOTE Bit depth and scan mode are media properties and not sequence properties: set these options in the Resolution options if you plan on using them as conform criteria.

Tips to Simplify the Conform

- On a Mac: Create the AAF project on the workstation running Flame. And when exporting the AAF, save the AAF to the root of the media used in that timeline: the media should either be with the AAF, or within a folder alongside the AAF.
- On a Linux workstation: Save the AAF to the root of the media used in that timeline: the media should either be with the AAF, or within a folder alongside the AAF. Then you can move that folder to the Linux workstation to conform in Flame.

R3D Media Files and AAF

In Flame, when you conform an AAF using RED .r3d media files, you can also apply the RED Source Settings defined in Avid Media Composer. Source Settings are the color settings edited in either Media Composer.

To use the RED Color Settings defined in Media Composer:

1 From the Conform tab contextual menu, select Load New FCP XML/AAF/EDL...
2 In the Media Import window, enable **AAF & XML Import Options ➤ Keep Source Settings**.

This setting can also be set in the MediaHub.

3 Locate and **Import** the AAF with RED Color Settings.
4 Once the AAF is imported, link the segments back to their R3D sources by Conforming an Avid AAF (page 290).

The RED sources are displayed using the color settings defined in Avid Media Composer.

When using Keep Source Settings, never relink to clips already imported in Flame because they already have color settings that cannot be overridden by the relinking process. Instead, make sure to use one of the following approaches:

- Automatically link to media files: In the Media Import window, enable Link To Media Files so Flame links the media files using the paths from the AAF, (with or without Path Translation).
- Set a search location: Add sources from an external location (page 260).
- Manually relink to media files: Edit the file path and location directly in the event list.

Debayering resolution is not part of the information contained in the AAF Source Settings. You must therefore specify the debayering settings before locating the R3D files, or the media's resolution will not match that of the sequence. You should do this from the MediaHub before importing the AAF, or from the Media Import window as you locate the AAF.

Matching the resolution of an AAF to the fully debayered resolution of R3D files:

1. In the Media Import window, locate and select one of the R3D files.
2. In the **Format Specific Options** tab, enable **Show All Formats**.
3. Select **File Format ➤ R3D** from the box.
4. Select **Options Category ➤ Debayering**.
5. Set the Debayering Mode box to Full.
6. Open the AAF & XML Import Options tab.
7. Select *Select Resolution* from the Sequence Resolution box.
8. With the R3D file still selected, click Copy from Selected Clip.

   The resolution information is copied from the R3D file. Any AAF (or XML for that matter) will be imported to that resolution.

TIP Often times the AAF will be have been created at an HD resolution in Avid Media Composer. But by working at full debayering resolution on Linux workstation, you can benefit from the faster GPU debayering. Override the AAF resolution with the Sequence Resolution in the AAF & XML Import Options; use Copy from Selected Clip on one of the R3D files to match resolution. Make sure to also set the R3D debayering to Full.

To set the debayering settings before importing the AAF:

1. Open the MediaHub.
2. In the **Format Specific Options** tab, enable **Show All Formats**.
3. Select **File Format ➤ R3D** from the box.
4. Select **Options Category ➤ Debayering** .
5. Set the Debayering Mode box to the desired debayering resolution.

   You can then following the instructions above to import the AAF.

To set the debayering settings as you import the AAF:

1. From the Conform tab contextual menu, select Load New FCP XML/AAF/EDL...
2. In the **Format Specific Options** tab, enable **Show All Formats**.
3. Select **File Format ➤ R3D** from the box.
4. Select **Options Category ➤ Debayering** .
5. Set the Debayering Mode box to the desired debayering resolution.
6. In the Media Import window, enable **AAF & XML Import Options ➤ Keep Source Settings**.

   This setting can also be set in the MediaHub.

7 Locate and **Import** the AAF with RED Color Settings.

8 Once the AAF is imported, link the segments back to their R3D sources by Conforming an Avid AAF.

The RED sources are displayed using the color settings defined in Avid Media Composer.

NOTE Format Options you set from the Timeline in Flame are overwritten when you relink the media to the AAF: the media is imported using the MediaHub settings.

Link to Media Files and Search and Import Files

The options *Search and Import Files* and *Link to Media Files* are performed sequentially:

1 If Link to Media Files is enabled, Flame locates the media files based file paths and names found in the AAF.

2 For the media that cannot be found, and if Search and Import Files is enabled, Flame tries to locate the media files in the location defined by the Directory Up field, trying to match media to segments based on the defined Match Criteria.

About Intermediates Formats

Transcoded intermediates from Avid Media Composer supported in Flame:

- AVC-Intra 50
- AVC-Intra 100
- DNxHD
- XDCam EX
- XDCam HD

Transcoded intermediates from Avid Media Composer not supported in Flame:

- -J2K MXF
- 1:1 MXF
- 1:1p 10b MXF
- Apple ProRes in MXF
- Avid DNxHR (also known as DNxRI)

When conforming unsupported intermediates, use the **Preferred Media ➤ Original Sources** option from the Media Import window to link to the original media files. Or from Media Composer, transcode the sources to a supported format.

Sequences made using a unsupported frame rate, such as 48 fps, cannot be conformed in Flame.

NOTE In MXF Op-Atom files generated by Avid Media Composer, audio tracks appear in the MediaHub as a single audio channel file (A1). But once imported, the tracks display the original channels.

Working with Path Translation

Path Translation allows you to fix problems with media that was relocated after it was imported in Flame.

Use path translation for:

- Clips in the Workspace: If the file system location of the clip's source has been modified, add a translation path to fix the issue without having to use Conform or re-import the contents. Restore a broken connection between a clip and its media. This break can happen when the media referenced by the clip is moved

from its location. The original paths to the media are not changed, but translated by Flame. This means that if you open the workspace on the original workstation, the sources remain linked.

- Batch setups: Loading a Batch setup with Import, Read File, and Write File nodes uses the path translation to modify the location of media files, Open Clip files and Batch setup. The data from the setup files is not modified, to protect its integrity: a user on the original file system will not be impacted.
- Conform: When you import an AAF or an FCP XML file with Link to Media Files enabled, Flame tries to reconnect the segments to their sources, using the paths defined within the sequence file. Oftentimes in a collaborative environment, with varying OSes and volume mount points, or with dead drops, the paths in the imported sequence cannot be resolved by Flame.

  Usually, the AAF is produced on a Mac OS X or Windows, where external volumes are mounted under */Volumes/* or a drive name. On Linux, such volumes can be mounted on any point. Path translation allows you to convert a path from one format into another. Once path translation is set up, linking and relinking to sources should be seamless.

NOTE This mechanism is similar to Lustre's Wiretap path translation function.

You set up the path translation in a configuration file, where you define pairs of paths.

To set a path translation pair:

1. Open a shell window. You do not have to close Flame.
2. As root, open the following file in a text editor:

   /opt/Autodesk/cfg/pathTranslation.cfg
3. Define a Source path and a Destination.

   For example, we need Flame to search for media files seen from an offlining Mac at /Volumes/SAN, but seen from the conforming Linux workstation at /ProductionSAN:

   ```
   <PathTranslationTable>
   <PathTranslation src="/Volumes/SAN" dst="/ProductionSAN" />
   </PathTranslationTable>
   ```
4. Save and close the file.

   Flame can immediately start translating a source path found in pathTranslation.cfg to its destination path when you import a sequence or use Link To Media File in Conform.

Importing an AAF Sequence: Supported Transitions and Effects

Sections in this topic:

Video and Audio Effects

The following tables describe how effects are supported in Flame.

NOTE When you import an AAF, an effect that is present in the AAF and applied to a gap or a filler is not supported but discarded.

Locators

AAF Locators' text contents and color are translated into Segment Marks in the imported sequence.

Blend

Avid	Flame
Picture-in-picture	Supported
Superimpose	Translated to Action FX (transparency value is translated)

Film

Avid	Flame
3D Warp	Not supported; replaced by Cue mark
1.66 mask	Supported; bkg is black, horizontal position ignored
1.85 mask	Supported; bkg is black, horizontal position ignored
16:9 mask	Supported; bkg is black, horizontal position ignored
Anamorphic mask	Supported; bkg is black, horizontal position ignored
Blowup	Converted to Action FX
Mask	Supported; bkg is black, horizontal position ignored

Generator

Avid	Flame
SubCap	Not supported; replaced by Cue mark
Timecode Burn-in	Not supported; replaced by Cue mark

AVX Plugin

Avid	Flame
Illusion FX	Not supported; replaced by Cue mark
AVX Plugins	Not supported; replaced by Cue mark

Image

Avid	Flame
Avid Pan and Zoom	Not supported; replaced by Cue mark

Avid	Flame
Blur effect	Not supported; replaced by Cue mark
Colour Correction	Converted to Colour Correct FX (empty) + cue mark
Colour Effect	Converted to Colour Correct FX (empty) + cue mark
Flip	Converted to Action FX (Flip effect)
Flip-flop	Converted to Action FX (Flip-flop effect)
Flop	Converted to Action FX (Flop effect)
Freeze Frame	Converted to a TW FX set at 0% on the frozen frame
Mask	Supported; bkg is black, no mask, horizontal position off
Mosaic Effect	Not supported; replaced by Cue mark
Paint Effect	Not supported; replaced by Cue mark
Region Stabilize	Not supported; replaced by Cue mark
Resize	Supported; background is black, no left and right cropping
Safe Color Limiter	Not supported; replaced by Cue mark
Scratch removal	Not supported; replaced by Cue mark
Spot Color Effect	Not supported; replaced by Cue mark
Stabilize	Not supported; replaced by Cue mark
Submaster	Converted to Container

Reformat

Avid	Flame
14:9 Letterbox	Not supported; replaced by Action FX
16:9 Letterbox	Not supported; replaced by Action FX
4:3 Sidebar	Not supported; replaced by Action FX
Pan and Scan	Not supported; replaced by Action FX

Titles

Avid	Flame
Classic Title	Converted to Text FX; only text string is available (white Discreet font, size 50)
Marquee Text	Converted to Text FX; only text string is available (white Discreet font, size 50)

NOTE The BLUE Titler Pro effects are not supported. When applied on an empty video track, they are applied on a filler and filler are discarded when conforming AAF in Flame. If the BLUE Title effect is applied on a timeline segment, a cue mark with a *Not supported* note is displayed in the imported timeline.

Key

Avid	Flame
Animatte	Not supported; replaced by Cue mark
Luma key	Converted to Action FX (empty) + cue mark
Matte key	Converted to Action FX (empty) + cue mark
RGB keyer	Converted to Action FX (empty) + cue mark
Spectra Matte	Not supported; replaced by Cue mark

S3D

Flame can import Media Composer stereoscopic timelines; an Avid Stereo3D timeline is imported as a stereoscopic timeline with Left and Right tracks, with the following limits.

Avid Stereoscopic Sequence Limitations:

- Segments with multiple nested effects may not be translated as expected.
- The Avid stereoscopic sequence cannot contain S3D master clips with frame compatible contributors (clips with side-by-side or top-bottom stereoscopic media), it can only contain full resolution contributors.

Avid	Flame
S3D Floating Window Effect	Not supported; replaced by Action FX
S3D Spatial Alignment Effect	Not supported; replaced by Action FX
S3D Vergence Effect	Not supported; replaced by Action FX

Timewarp

NOTE To make sure the conform is accurate with the creative editorial decisions from the Avid editor, the conformed timewarp speed value seen in Flame timewarp editor can be different from the one seen in the Avid application. But the actual Timewarp effect will be visually similar to the expected result, and be frame and keyframe accurate.

Avid	Flame
Animated Timewarp	Translated to TW FX, with keyframes.
Speed Bump	Translated to TW FX, with keyframes.
Timewarp Ramp	Translated to TW FX, with keyframes.

Miscellaneous

Avid	Flame
3D PIP	Translated to Action FX (Position / Scaling, ISO, Softness / Crop) + cue mark
Peel	Not supported; cue mark
Push	Not supported; cue mark
Spin	Not supported; cue mark
Squeeze	Not supported; cue mark
Video gap	Video gap
Video filler	Video gap
Video match frame edit	Match frame

General Audio

Only audio level keyframes are supported. All other audio keyframes are ignored.

Avid	Flame
Audio level	Audio gain
Audio dissolve	Audio dissolve
Audio fade in	Audio dissolve
Audio fade out	Audio dissolve

Avid	Flame
Audio gap	Audio gap
Audio filler	Audio gap
Audio match frame edit	Match frame splice
Audio warp	Not supported; replaced by Cue mark

Audio Effects

Flame does not support any of the RTAS Audio Effects. Every RTAS Audio Effect is replaced with a cue mark.

Video and Audio Transitions

The following tables describe how AAF transitions are supported in Flame.

Transitions marked with an ***** are also supported with the "Inverse" option set.

Blend

Avid	Flame
Dip to colour	Converted to Dissolve (linear animation)
Dissolve	Converted to Dissolve (linear animation)
Fade from colour	Converted to Dissolve; background is black only, reset manually
Fade to colour	Converted to Dissolve; background is black only, reset manually
Picture-in-picture	Not supported; replaced by Cue marks

Film

Avid	Flame
Film dissolve	Converted to Dissolve (bezier animation) + Cue mark
Film fade	Not supported; replaced by Cue marks

Box wipe

Avid	Flame
Bottom box*	Converted to SMPTE 025; softness not supported
Bottom left to top right*	Converted to SMPTE 006; softness not supported

Avid	Flame
Bottom right to top left*	Converted to SMPTE 005; softness not supported
Left box*	Converted to SMPTE 026; softness not supported
Right box*	Converted to SMPTE 024; softness not supported
Top box*	Converted to SMPTE 023; softness not supported
Top left to bottom right*	Converted to SMPTE 003; softness not supported
Top right to bottom left*	Converted to SMPTE 004; softness not supported

Edge Wipe

Avid	Flame
Horizontal*	Converted to SMPTE 001; softness not supported
Horz open*	Converted to SMPTE 021; softness not supported
Bottom left diagonal*	Converted to SMPTE 042; animation is inverted, softness not supported
Bottom right diagonal*	Converted to SMPTE 041; animation is inverted, softness not supported
Upper left diagonal*	Converted to SMPTE 041; softness not supported
Upper right diagonal*	Converted to SMPTE 042; softness not supported
Vert open*	Converted to SMPTE 022; softness not supported
Vertical*	Not supported; replaced by a Cue mark

Shape Wipe

Avid	Flame
4 corners*	Converted to SMPTE 007; softness not supported
Horizontal bands	Converted to Dissolve + cue mark
Horizontal blinds	Converted to Dissolve + cue mark
Vertical blinds	Converted to Dissolve + cue mark
Center box*	Converted to SMPTE 101; softness not supported

Avid	Flame
Circle*	Converted to SMPTE 119; softness not supported
Ellipse*	Converted to SMPTE 120; softness not supported
Clock*	Converted to SMPTE 201; softness not supported
Diamond*	Converted to SMPTE 102; softness not supported

Sawtooth Wipe

Avid	Flame
Horizontal sawtooth*	Converted to SMPTE 071; softness not supported
Horz open sawtooth*	Converted to SMPTE 073; softness not supported
Vert open sawtooth*	Converted to SMPTE 074; softness not supported
Vertical sawtooth*	Converted to SMPTE 072; softness not supported

Matrix Wipe

Avid	Flame
Grid*	Not supported; replaced by SMPTE 008
One-way row	Not supported; replaced by SMPTE 001
Speckle	Not supported; replaced by SMPTE 001
Spiral	Not supported; replaced by SMPTE 001
Zig-zag	Not supported; replaced by SMPTE 001

S3D

Avid	Flame
S3D Depth Transition	Not supported; replaced by a Cue mark

Xpress 3D Effect

Avid	Flame
3D ball	Converted to Dissolve (linear) + cue mark

Avid	Flame
3D page fold	Converted to Dissolve (linear) + cue mark
3D slats	Not supported; replaced by a Cue mark
3D PIP	Translated to Action FX (Position / Scaling, ISO, Softness / Crop) + cue mark

Miscellaneous

Avid	Flame
Conceal	Converted to Action FX (bottom left to top right); softness not supported
Illusion FX	Converted to Dissolve + cue mark
L-Conceal	Converted to Action FX (bottom left to top right); softness not supported
Squeeze	Converted to Action FX (bottom centered); softness not supported
Peel	Converted to Dissolve + cue mark
Plasma	Converted to Dissolve + cue mark
Push	Converted to Dissolve + cue mark
Spin	Converted to Dissolve + cue mark
Video gap	Video gap
Video filler	Video gap

Importing a Final Cut Pro XML Sequence

Flame supports both FCP 7 and FCP X file formats.

To simplify the conform process, create the FCP XML project on a volume accessible toFlame. And when exporting the FCP XML, save the FCP XML to the root of the media used in that timeline: the media should either be with the FCP XML, or within a folder alongside the FCP XML.

To import a sequence using the MediaHub:

1. Open MediaHub, and set it to Browse for Files.
2. Review the AAF & XML Import Options.

   NOTE By default, the sequence is imported at the resolution specified in the FCP XML, not the project's. You can set a resolution with the Sequence Resolution box.

3. Using the file browser, navigate to the FCP sequence to import.

4 Drag the file from the browser to the Media Library.

Flame converts the FCP XML to its sequence format. According to the Media and Relative Search options, the sequence can be relinked or not, and the sources also imported, or not. The media itself is imported using the option file format options defined in the Format Specific Options tab.

You can also import a sequence from Conform:

1 Open Conform.
2 Select in the Media Panel where to put the imported sequence.
3 Right-click the Events list, and select Load New FCP XML/AAF/EDL...
4 Set the AAF & XML Import Options.
5 In the Media Import window, locate the sequence to import using the file browser, and click Import.

You can select multiple sequences to import, using `Shift-click` and `Ctrl-click`.

If a source is used multiple times in a sequence, or across multiple sequences, and Save Sources Separately is enabled, only one instance of the source is imported. And the sources are saved in a sources folder.

If you plan on perform a Connected conform, you must import your sequences into a Reel group. Make sure to drag and drop the sequences on top of a Reel group, or to import from Conform. When you import your sequences through Conform, they are always added to the Sequences reel of the default Reel group, making sure you will be able to use the Sources and Shots sequences required for the Connected conform workflow.

NOTE Bit depth and scan mode are media properties and not sequence properties: set these options in the Resolution options if you plan on using them as conform criteria.

Link to Media Files and Search and Import Files

The options *Search and Import Files* and *Link to Media Files* are performed sequentially:

1 If Link to Media Files is enabled, Flame locates the media files based file paths and names found in the FCP XML.
2 For the media that cannot be found, and if Search and Import Files is enabled, Flame tries to locate the media files in the location defined by the Directory Up field, trying to match media to segments based on the defined Match Criteria.

FCP 7 Sequence Import: Supported Transitions and Effects

Sections in this topic:

Supported Data

Flame allows you to import multi-track compositions from Apple Final Cut Pro (FCP). Flame reads XML exported from FCP (up to version 7.x of FCP, exported as XML version 2.0) and recreates a timeline accordingly.

The following tables describe the data that is output from FCP and input into Flame.

General Data

FCP composition(s) data maps to Flame timeline data.

Final Cut Pro	Flame
Name	Name
Frame rate	Frame rate
Duration	Duration

Editorial Data

FCP Source media data and Record side data maps to Autodesk clip data.

Final Cut Pro	Flame
Source media data: ■ Tape name ■ Source TC in/out ■ Edge code ■ Log notes ■ Aspect ratio ■ Comments	Source clips: ■ Tape name ■ Source TC in/out ■ Keycode ■ Elements comments ■ Aspect ratio ■ Elements Comments
Record side data: ■ In/Out ■ Transition type ■ Number of video tracks ■ Number of audio tracks ■ Marker ■ In/Out marker	Record clip: ■ Segment ■ Cut/Dissolve/Wipe/Action ■ Video tracks ■ Audio tracks ■ Track marks ■ In/Out marks

Effect Data

FCP transitions map to Flame transitions, while FCP Filter effects and FX Script data map to Flame Timeline FX.

Final Cut Pro	Flame
Filter effects	Timeline FX
Transitions	Transitions

Animation Interpolation

FCP animation interpolation maps to Autodesk interpolation.

Final Cut Pro	Flame
Corner	Linear
Smooth	Hermite

Motion

The following table describes how motion from FCP is mapped to Flame Action Timeline FX parameters.

Final Cut Pro	Flame
Basic Motion	Action (Axis)
Crop	Action (Crop)
Distort	Action (Surface)
Opacity	Action (Surface) (partially supported)
Drop Shadow	Action (Shadow)
Motion Blur	Not supported
Time Remap	Time Warp (see below)

About Time Remap to Timewarp

To make sure the conform is accurate with the creative editorial decisions from Final Cut Pro 7, the conformed timewarp speed value seen in Flame Time Warp editor can be slightly different from the one seen in FCP7. But the actual Time Warp Timeline FX will be visually similar to the expected result, and be frame and keyframe accurate. There are exceptions:

- Frame blending used with this effect is not translated.
- Negative constant timewarp speeds are not applied to the audio tracks of the segment being timewarped.
- Variable time warps are not applied to the audio tracks of the segment being timewarped.

Video Transitions

The names of FCP transitions are preserved in Flame and are visible in the timeline.

3D Simulation

Final Cut Pro	Flame
Cross Zoom	Dissolve (partially supported)
Cube Spin	Dissolve (partially supported)
Spin 3D	Action transition (partially supported)
Spinback 3D	Not supported; replaced by Dissolve
Swing	Action transition (partially supported)
Zoom	Action transition (partially supported)

Dissolve

Final Cut Pro	Flame
Additive Dissolve	Not supported; replaced by Dissolve
Cross Dissolve	Dissolve Additive
Dip to colour Dissolve	Dissolve To/From colour (partially supported)
Dither Dissolve	Not supported; replaced by Dissolve
Fade in/fade out Dissolve	Dissolve To/From Black (partially supported)
Non-Additive Dissolve	Dissolve Non-Additive
Ripple Dissolve	Not supported; replaced by Dissolve

Iris

Final Cut Pro	Flame
Cross Iris	SMPTE 007 (partially supported)
Diamond Iris	SMPTE 102 (partially supported)
Oval Iris	SMPTE 119, 120, or 121 (partially supported)

Final Cut Pro	Flame
Point Iris	SMPTE 047 (partially supported)
Rectangle Iris	SMPTE 101 (partially supported)
Star Iris	SMPTE 127, 128, or 129 (partially supported)

Map

Final Cut Pro	Flame
Channel Map	Not supported; replaced by Dissolve
Luminance Map	Not supported; replaced by Dissolve

Page Peel

Final Cut Pro	Flame
Page Peel	Not supported; replaced by Dissolve

QuickTime

Final Cut Pro	Flame
Channel Compositor	Not supported; replaced by Dissolve
Chroma Key	Not supported; replaced by Dissolve
Explode	Not supported; replaced by Dissolve
Gradient Wipe	Not supported; replaced by SMPTE 002 without Softness
Implode	Not supported; replaced by Action transition
Iris	SMPTE Wipes (partially supported): ■ Rectangle = SMPTE 101 ■ Diamond = SMPTE 102 ■ Triangle = SMPTE 103 ■ Triangle Right = SMPTE 104 ■ Triangle Upside Down = SMPTE 105 ■ Triangle Left = SMPTE 106 ■ Arrowhead = SMPTE 107 ■ Arrowhead Right = SMPTE 108 ■ Arrowhead Upside Down = SMPTE 109

Final Cut Pro	Flame
	■ Arrowhead Left = SMPTE 110 ■ Pentagon = SMPTE 111 ■ Pentagon Upside Down = SMPTE 112 ■ Hexagon = SMPTE 113 ■ Hexagon Side= SMPTE 114 ■ Circle = SMPTE 119 ■ Oval = SMPTE 120 ■ Oval Side = SMPTE 121 ■ Cat Eye = SMPTE 122 ■ Cat Eye Side = SMPTE 123 ■ Round Rect = SMPTE 124 ■ Round Rect Side = SMPTE 125 ■ 4 Point Star = SMPTE 127 ■ 5 Point Star = SMPTE 128 ■ 6 Point Star = SMPTE 129 ■ Heart = SMPTE 130 ■ Keyhole = SMPTE 131
Matrix Wipe	Not supported; replaced by SPMTE 001 with a comment indicating the type of FCP Matrix Wipe that had been at this mark
Push	Not supported; replaced by Dissolve
Radial	SMPTE Wipes (partially supported): ■ Rotating Top = SMPTE 201 ■ Rotating Right = SMPTE 202 ■ Rotating Bottom = SMPTE 203 ■ Rotating Left = SMPTE 204 ■ Rotating Left Bottom = SMPTE 205 ■ Rotating Left Right = SMPTE206 ■ Rotating Quadrant = SMPTE 207 ■ Top to Bottom 180 degree = SMPTE 211 ■ Right to Left 180 degree= SMPTE 212 ■ Top to Bottom 90 degree= SMPTE 213 ■ Right to Left 90 degree = SMPTE 214 ■ Top 180 Degree = SMPTE 221 ■ Right 180 Degree = SMPTE 222 ■ Bottom 180 Degree = SMPTE 223 ■ Left 180 Degree = SMPTE 224 ■ Counter Rotating Top Bottom = SMPTE 225 ■ Counter Rotating Left Right = SMPTE 226

Final Cut Pro	Flame
	■ Double Rotating Top Bottom = SMPTE 227 ■ Double Rotating Left Right = SMPTE 228 ■ V Open Top = SMPTE 231 ■ V Open Right = SMPTE 232 ■ V Open Bottom = SMPTE 233 ■ V Open Left = SMPTE 234 ■ V Open Top Bottom = not supported; replaced by SMPTE 001 ■ V Open Left Right = not supported; replaced by SMPTE 001 ■ Rotating Top Left = SMPTE 241 ■ Rotating Bottom Left = SMPTE 242 ■ Rotating Bottom Right = SMPTE 243 ■ Rotating Top Right = SMPTE 244 ■ Rotating Top Left Bottom Right = SMPTE 245 ■ Rotating Bottom Left to Top Right = SMPTE 246 ■ Rotating Top Left Right = SMPTE 251 ■ Rotating Left Top Bottom = SMPTE 252 ■ Rotating Bottom Left Right = SMPTE 253 ■ Rotating Right Top Bottom = SMPTE 254 ■ Rotating Double Center Right = not supported; replaced by SMPTE 001 ■ Rotating Double Center Top = not supported; replaced by SMPTE 001 ■ Rotating Double Center Top Bottom = not supported; replaced by SMPTE 001) ■ Rotating Double Center Left Right = not supported; replaced by SMPTE 001
Slide	Action transition
Wipe	SMPTE Wipes (various partially supported): ■ Slide Horizontal = SMPTE 001 ■ Slide Vertical = SMPTE 002 ■ Top Left = SMPTE 003 ■ Top Right = SMPTE 004 ■ Bottom Right = SMPTE 005 ■ Bottom Left = SMPTE 006 ■ Four Corner = SMPTE 007 ■ Four Box = SMPTE 008 ■ Barn Vertical = SMPTE 021 ■ Barn Horizontal = SMPTE 022 ■ Top Center = SMPTE 023 ■ Right Center = SMPTE 024 ■ Bottom Center = SMPTE 025

Final Cut Pro	Flame
	■ Left Center = SMPTE 026 ■ Diagonal Left Down = SMPTE 041 ■ Diagonal Right Down = SMPTE 042 ■ Vertical Bow Tie = SMPTE 043 ■ Horizontal Bow Tie = SMPTE 044 ■ Diagonal Left Out = SMPTE 045 ■ Diagonal Right Out = SMPTE 046 ■ Diagonal Cross = SMPTE 047 ■ Diagonal Box = SMPTE 048 ■ Filled V = SMPTE 061 ■ Filled V Right = SMPTE 062 ■ Filled V Bottom = SMPTE 063 ■ Filled V Left = SMPTE =064 ■ Hollow V = SMPTE 065 ■ Hollow V Right = SMPTE 066 ■ Hollow V Bottom = SMPTE 067 ■ Hollow V Left = SMPTE 068 ■ Vertical Zig Zag = SMPTE 071 ■ Horizontal Zig Zag = SMPTE 072 ■ Vertical Barn Zig Zag = SMPTE 073 ■ Horizontal Barn Zig Zag = SMPTE 074
Zoom	Action transition (partially supported)

Slide

Final Cut Pro	Flame
Band Slide	Not supported; replaced by Dissolve
Box Slide	Not supported; replaced by Dissolve
Center Split Slide	Not supported; replaced by Dissolve
Multi Spin Slide	Not supported; replaced by Dissolve
Push Slide	Not supported; replaced by Dissolve
Spin Slide	Not supported; replaced by Dissolve
Split Slide	Not supported; replaced by Dissolve

Final Cut Pro	Flame
Swap Slide	Not supported; replaced by Dissolve

Stretch

Final Cut Pro	Flame
Cross Stretch	Not supported; replaced by Dissolve
Squeeze	Action transition (partially supported)
Squeeze and Stretch	Not supported; replaced by Action Transition
Stretch	Action transition (partially supported)

Wipe

Final Cut Pro	Flame
Band	Not supported
Center Wipe	SMPTE 021 (partially supported)
Checker Wipe	Not supported
Checkerboard Wipe	Not supported
Clock Wipe	SMPTE 201 (partially supported)
Edge Wipe	SMPTE 001 (partially supported)
Gradient Wipe	Not supported
Inset Wipe	■ Upper Left = SMPTE 003 (partially supported) ■ Top = SMPTE 023 ■ Upper Right = SMPTE 004 ■ Right = SMPTE 024 ■ Lower Right = SMPTE 005 (reverse) ■ Bottom = SMPTE 025 ■ Lower Left = SMPTE 006 (reverse) ■ Left = SMPTE 026
Jaws Wipe	SMPTE 073 (partially supported, FCP's is smaller than Flame's)

Final Cut Pro	Flame
Random Edge Wipe	Not supported; replaced by Wipe 001 transition
V Wipe	■ Right = SMPTE 064 (partially supported) ■ Down = SMPTE 61 ■ Left = SMPTE 62 ■ Up = SMPTE 63
Venetian Blind Wipe	Not supported; replaced by SMPTE 001
Wrap Wipe	Not supported; replaced by SMPTE 001
Zig-Zag Wipe	Not supported; replaced by SMPTE 001

Video Filters

Blur

Final Cut Pro	Flame
Gaussian Blur	Action (Surface + Axis) (partially supported)
Radial Blur	Not supported
Wind Blur	Not supported
Zoom Blur	Not supported

Border

Final Cut Pro	Flame
Basic Border	Resize
Bevel	Not supported

Channel

Final Cut Pro	Flame
Arithmetic	Not supported
Channel Blur	Not supported
Channel Offset	Not supported

Final Cut Pro	Flame
Colour Offset	CC (partially supported)
Compound Arithmetic	Not supported
Invert	CC (partially supported)

Colour Correction

Final Cut Pro	Flame
Broadcast Safe	Not supported
Colour Correction	Not supported; replaced by CC
Colour Correction 3-way	Not supported; replaced by CC
Desaturate Highlights	Not supported
Desaturate Lows	Not supported
RGB Balance	Not supported; replaced by CC

Distort

Final Cut Pro	Flame
Bumpmap	Not supported
Cylinder	Not supported
Displace	Not supported
Fisheye	Not supported
Pond Ripple	Not supported
Ripple	Not supported
Wave	Not supported
Whirlpool	Not supported

Image Control

Final Cut Pro	Flame
Brightness & Contrast (Bezier)	Not supported; replaced by CW
Colour Balance	Not supported; replaced by CC
Desaturate	CC
Gamma Correction	Not supported; replaced by CW
Levels	Not supported
Proc Amp	Not supported; replaced by CC
Sepia	Not supported; replaced by CC
Tint	Not supported

Key

Final Cut Pro	Flame
Blue and Green screen	Not supported
Chroma Keyer	Not supported; replaced by Action
Colour Smoothing 4:1:1	Not supported
Colour Smoothing 4:2:2	Not supported
Colour Key	Not supported; replaced by Action
Difference Matte	Not supported
Luma Key	Not supported
Spill Suppressor - Blue	Not supported; replaced by Action
Spill Suppressor - Green	Not supported; replaced by Action

Matte

Final Cut Pro	Flame
8-Point Garbage Mask	Not supported; replaced by Action

Final Cut Pro	Flame
Extract	Not supported
4-Point Garbage Mask	Not supported; replaced by Action
Image Mask	Not supported; replaced by Action
Mask Feather	Not supported
Mask Shape	Action (partially supported)
Matte Choker	Not supported; replaced by Action
Soft Edges	Not supported; replaced by Action
Widescreen	Action (partially supported)

Perspective

Final Cut Pro	Flame
Basic 3D	Action (Axis) (partially supported)
Curl	Not supported
Flop	Action (Axis)
Mirror	Not supported
Rotate	Action (Axis)

QuickTime

Final Cut Pro	Flame
Gaussian Blur	Action (Axis)
Brightness/Contrast	Not supported; replaced by CC
Colour Style	Not supported
Colour Tint	CC (partially supported)
Colour Sync	Not supported
Edge Detection	Not supported

Final Cut Pro	Flame
Emboss	Not supported
General Convolution	Not supported
HSL Balance	Not supported; replaced by CC
Lens Flare	Not supported
RGB Balance	Not supported; replaced by CC
Sharpen	Not supported

Sharpen

Final Cut Pro	Flame
Sharpen	Not supported
Unsharp Mask	Not supported

Stylize

Final Cut Pro	Flame
Anti-Alias	Not supported
Diffuse	Not supported
Emboss	Not supported
Find Edges	Not supported
Posterize	Not supported
Replicate	Not supported
Solarize	Not supported

Video

Final Cut Pro	Flame
Blink	Action
De-Interlace	Resize (partially supported)

Final Cut Pro	Flame
Flicker Filter	Not supported
Image Stabilizer	Not supported; replaced by Action
Stop Motion Blur	Not supported
Strobe	Time Warp
Timecode Generator	Not supported
Timecode Reader	Not supported
Viewfinder	Not supported

Video Generators

The following tables describe generated effects that are translated into equivalent effects in Flame.

Once imported into Flame, effects created with the FCP video generator use the project's default resolution, regardless of their original resolution in FCP. This matches the behaviour of FCP: XML files with these effects that are reimported into FCP projects with different resolutions similarly inherit the project's resolution.

Video Generator

Final Cut Pro	Flame
Bars and Tone HD108060i	Colour Source SMPTE Bars (partially supported)
Bars and Tone HD720p60	Colour Source SMPTE Bars (partially supported)
Bars and Tone (NTSC)	Colour Source SMPTE Bars (partially supported)
Bars and Tone (PAL)	Colour Source SMPTE Bars (partially supported)
Slug	Black colour Source SMPTE Bars (partially supported)

Matte

Final Cut Pro	Flame
Colour	Colour Source (Fill Colour): ■ R = 0-255 ■ G = 0-255 ■ B = 0-255

Others

Final Cut Pro	Flame
More Bars and Signals	Not supported

Render

Final Cut Pro	Flame
Custom Gradient	Not supported
Gradient	Not supported
Highlight	Not supported
Noise	Colour Source Noise
Particle Noise	Not supported

Shapes

Final Cut Pro	Flame
Circle	Not supported
Oval	Not supported
Rectangle	Not supported
Square	Not supported

Text

Final Cut Pro	Flame
Crawl	Text (partially supported)
Lower 3rd	Text (partially supported)
Outline Text	Text (partially supported)
Scrolling Text	Text (partially supported)
Text	Text (partially supported)
Typewriter	Not supported; replaced by Colour Source

Audio

Some FCP audio elements are mapped to Flame equivalents. Audio keyframes are not supported.

Audio Transitions

Final Cut Pro	Flame
Cross Fade (0dB)	Audio fade (partially supported)
Cross Fade (+3dB)	Not supported; replaced by Audio Fade

Audio Controls

Final Cut Pro	Flame
Stereo	Audio Gain (partially supported)

Apple

Final Cut Pro	Flame
AuBandPass	Not supported
AuDelay	Not supported
AuDynamicProcessor	Not supported
AuGraphicsEQ	Not supported
AuHighShelfFilter	Not supported
AuHighPass	Not supported
AuLowPass	Not supported
AuLowShelfFilter	Not supported
AuMultibandCompress	Not supported
AuParametricEQ	Not supported
AuPeakLimiter	Not supported

Final Cut Pro HD

Final Cut Pro	Flame
3 Band Equalizer	Not supported
Band Pass Filter	Not supported
Compressor/Limiter	Not supported
DC Notch	Not supported
Echo	Not supported
Expander/Noise Gate	Not supported
High Pass Filter	Not supported
High Shelf Filter	Not supported
Hum Remover	Not supported
Low Pass Filter	Not supported
Low Shelf Filter	Not supported
Notch Filter	Not supported
Parametric Equalizer	Not supported
Reverberation	Not supported
Vocal DeEsser	Not supported
Vocal DePopper	Not supported

Composite Modes

Some FCP composite modes are mapped to Flame Action surface blend modes.

Final Cut Pro	Flame
Modify	■ Add = Add (partially supported) ■ Subtract = Subtract ■ Difference = Negate ■ Multiply = Multiply ■ Screen = Screen

Final Cut Pro	Flame
	■ Overlay = not supported ■ Hard Light = not supported ■ Soft Light = not supported ■ Darken = Min ■ Lighten = Max ■ Travel Matte - Alpha = not supported ■ Travel Matte - Luma = not supported ■ Normal = not supported

FCP X Sequence Import: Supported Transitions and Effects

Sections in this topic:

Supported Data

Flame supports Final Cut Pro X (up to 10.2) XML exports, including ones referencing directly R3D, MXF, and ARRI Alexa ProRes files.

NOTE Compounded (nested) clips are not supported. As a workaround, remove any compound clip before exporting the sequence from FCP.

Video Effects

The following effects are supported when conforming XML sequence from FCP X.

Apple Final Cut Pro X Effect	Translation in Flame
Position	2D Transform with X and Y position
Scaling	2D Transform with X and Y scaling
Rotation	2D Transform with Z scaling
Transparency	Comp with transparency
Compositing mode	Comp with matching Blend mode

Retime Effects

A clip retime is translated to a Soft Timewarp with matching speed up or down.

NOTE To make sure the conform is accurate with the creative editorial decisions from Final Cut Pro X, the conformed timewarp speed value seen in Flame timewarp editor can be slightly different from the one seen in FCPX. But the actual Timewarp effect will be visually similar to the expected result, and be frame and keyframe accurate.

Final Cut Pro X Retime	Translation in Flame
Slow	Slower than real-time linear timewarp, at same speed as in Final Cut Pro X
Fast	Faster than real-time linear timewarp, at same speed as in Final Cut Pro X
Normal 100%	Not Timewarp
Hold	0% Soft Timewarp, using same frame as the one defined in Final Cut Pro X
Reverse	Reverse speed linear timewarp, at same speed as in Final Cut Pro X
Speed Ramp (to 0% / from 0%)	Animated speed ramp timewarp
Rewind (1x / 2x / 4x)	Animated speed ramp timewarp

Title Effects

All text effects are translated to a Text Timeline FX for the text itself, and a Comp Timeline FX for the compositing:

- Text Timeline FX:
  - RGBA mode
  - Colour: White
  - Font: Discreet
  - Size: 50
  - Alignment: Centered
- Comp Timeline FX set to premultiplied.

Audio

Only audio fades are supported. Audio keyframes are not supported.

Flame correctly imports FCP X XML using multi-track audio.

Transitions

In cases where the video and audio are imported and treated as a single entity in the Final Cut Pro X sequence, applying a video transition affects both video and audio and is translated in Flame as follows:

- Dissolve transition: Flame applies a dissolve to video and audio.
- Wipe transition: Flame applies a matching wipe to video, and a dissolve transition to audio.
- DVE transition: Flame applies a dissolve to video and audio.

Apple Final Cut Pro X Transition	Translation in Flame
Blur (any)	Dissolve
Dissolve	Dissolve except: ■ Cross Dissolve changes to Dissolve (no keyframes) ■ Fade to Colour changes to Fade To/From Black (no keyframes)
Lights	Dissolve
Movements	Dissolve
Objects	Dissolve
Replicator/Clones	Dissolve
Stylized	Dissolve

Apple Final Cut Pro X Wipes	Translation in Flame
Bands	SMPTE Wipe 001 + cue mark
Center	Supported
Checker	SMPTE Wipe 001 + cue mark
Circle	Supported
Clock	Supported
Gradient Image	Dissolve + cue mark
Inset Wipe	Supported
Letter X	Supported
Wipe	Supported

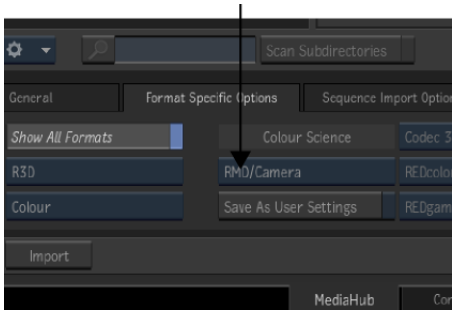
Loading RMD Files from FCP in Flame

NOTE The following only applies when using Final Cut Pro X 10.0.8 (or later).

With Final Cut Pro, you can work directly with R3D files, created by RED cameras, instead of using the QuickTime files. This means that when you import a FCP X XML sequence in Flame, you can relink the sequence to the R3D files, providing you with the best image quality.

Another benefit of using the R3D files is the RAW parameters set in Final Cut Pro X are saved by FCP as an RMD file. This file can be read by Flame on import.

To read FCP X Color settings for R3D files, you have to select an RMD option from **MediaHub ➤ Browse for Files ➤ Format Specific Options ➤ Colour Settings box** for R3D files.



MediaHub Reference: Browsing for Files

The MediaHub consists of two panels: the browser and the MediaHub tabs. Use the browser to locate the files to import, or the location where to export your clips. Use the MediaHub tabs to set you media and sequence import options.

MediaHub options are saved on a project-basis.

The MediaHub also monitors the file system and automatically updates the display of a browsed folder whenever its contents change: addition/removal of media files or folders, renaming, etc. There is no need to manually refresh the MediaHub. If you encounter an issue during the Preview, you can disable this feature in the Wiretap Gateway configuration file.

NOTE This feature is only available on Linux.

MediaHub Browser

The MediaHub file browser displays two sections:

- The Autodesk Network lists other Smoke, Flame, or Lustre workstations on the network, and can be used to import files from those workstations.
- The Local Devices displays the hard disks, external disks, and other storage devices that appear under the Devices category in the Finder. Use the Local Devices to access local storage.

TIP Network volumes connected to your workstation appear in Local Device.

General Tab

Cache Source Media button Enable to create a managed copy of the media in the application storage; this copy is a transcoded version of the original media, using the Cached Media Compression Format as the transcode target. Disable to link to the media of the imported clip, without transcoding or copying the media.

Enabling Cache Source Media ensures that the application is the sole owner of the media, preventing the media from being modified by an external source. With Cache Source Media disabled, the application decodes the clip on-the-fly and there is no transcoding.

You can always change your mind after importing a clip: right-click the clip and select **Media ➤ Cache Source Media** to transcode the media and copy it to your local storage. Right-click and select **Media ➤ Flush Source Media Cache** to get rid of the transcoded copy and refer back to the original media.

Generate Proxies button Enable to generate proxy media for imported clips.

Clip Options for Versions box Applies only to multi-versions clips. Sets if Generate Proxies and Cache Source Media settings apply to the current version or to all versions of a multi-version Open Clip.

Multi-Channel Processing box Sets how multi-channel clips are processed on import. Multi-Channel Clip creates a single multi-channel clip. Matte Container (Full) creates a matte container containing every channel. Other options create one clip or matte container per channel, with alpha channel included or discarded depending on the selected option.

Cached Media Compression Format field Displays the compression format applied to clips imported with Cache Source Media enabled. Defined for the project, in the Cache and Renders tab of the Project dialogue box.

You access your project's settings in **Flame ➤ Project and User Settings**.

Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

Fill box Select a fit method to be applied to the selected clip.

Select:	To fit:
Centre/Crop	The source image, centred, over the destination resolution frame. If the source is larger than the destination, it is cropped. If the source is smaller than the destination, it is surrounded by a black border.
Crop Edges	One edge of the source into the destination resolution frame without stretching or squashing the frame. Excess parts of the source frame after resizing are cropped. If the source—after the one edge is resized—is wider than the destination, its overhanging left and right edges are cropped. If the source is taller than the destination, the upper and lower edges are cropped.
Fill	The source, width, and height, into the destination resolution frame. This process, if the source and destination resolutions do not have the same aspect ratio, can distort the image.

Select:	To fit:
Letterbox	The source to the destination resolution frame without squashing or stretching it, and without cropping the source. If the source is wider than the destination, black bars fill the top and bottom of the destination frame. If the source is narrower than the destination, black bars fill the right and left sides of the frame. In all cases, the entire source frame is fit into the destination frame.

Resize Filter box Select the resize filter to apply to the clip; all but the Impulse filter are rendered using the GPU. This box does not appear if you select the Centre/Crop fit.

Select:	For:
Lanczos	Excellent and sharp results. Recommended for upscale and downscale. Expensive to compute.
Shannon	Excellent and sharp results. Results are sharper than Lanczos in small details. Recommended for upscale and downscale. Expensive to compute.
Gaussian	Medium quality and softer results.
Quadratic	Medium quality and softer results.
Bicubic	High-quality results, but not as sharp as Shannon. Use for both upscale and downscale.
Mitchell	High-quality results, but not as sharp as Shannon. Use for both upscale and downscale.
Triangle	Low quality results that are fast to compute. Use for downscale.
Impulse	Very low quality results that are fast to compute. Use for downscale.

Width field Displays the custom width resolution of the clip. Editable.

Height field Displays the custom width resolution of the clip. Editable.

Aspect Ratio Presets box Select a standard frame aspect ratio. Select the Set to w:h option to set the clip to use square pixels. Select Custom to define a custom frame aspect ratio in the Aspect Ratio field.

Aspect Ratio field Displays the custom render/output aspect ratio. Editable

Bit Depth box Select the render/output bit depth of clips.

Scan Mode box Select the scan mode of clips.

Still Duration field Specify a duration for stills, repeating the last frame as required. The result is the same as if you had trimmed out the stills in the Timeline to the specified duration.

Pixel Ratio box Use From Source to import using the information specified by the file. Use Square Pixels to ignore the file specification and use the Width x Height ratio, when file metadata is not set correctly.

Colour Management Mode box Select the type of colour management to apply: Tag Only (tag the media with a colour space but do not process the pixels); Auto Convert (convert from an input colour space to a working space); or Use LUT (manually apply a LUT or Colour Transform and then tag the result).

Format box Select Colour Transform unless applying a legacy format 1D or 3D LUT.

Applied LUT field Displays the type of conversion LUT applied to the clip, either imported using Import, or edited using Edit.

Import button Opens the Import LUT browser. Navigate to the 1D LUT file to import, and select it to load it into the list. This button is available when the LUT Type box is set to LUT File.

Colour Management Access button Click to configure colour management settings.

Format Specific Options

Show All Formats button Disable to display only the format of the file selected in the MediaHub browser. Enable to view or edit import options for any of the available file formats.

File Format box With Show All Formats disabled, File Format displays the format of the file selected in the browser; if more than one file is selected, File Format displays the formats of the selected files. With Show All Formats enabled, select the file format to view or edit its import options.

Options Category box Select the set of options to view or edit.

AAF & XML Import Options

Media Options

Link to Files button Enable to create a sequence with segments that link to the original media. Disable to import an unlinked sequence. If Search and Import Files is also enabled, the application first tries to relink to the media using the file paths, and then searches for what could not be relinked using only the file paths.

Preferred Media button Select which one of either offline intermediates or original sources to import when both are found by the application. Only used when importing AAF sequences.

Consolidate on Import button Enable to limit the handles to the amount specified in Maximum Handles. Disabled if Search and Import Files is enabled but the Match Criteria is not Tape-Source Timecode.

Maximum Handles field Displays the maximum number of handles (head and tail) allowed for each event in the sequence. If the sequence also sets the amount of handles, the application imports the sources using the lowest number of handles set between Maximum Handles and the sequence. Editable.

Save Sources Separately button Enable to import each source referred by the sequence, and to save them in a Sources folder.

Relative Search Options

Search and Import Files button Enable to search and import the media listed in the sequence, using the selected Match Criteria options but not the file paths the sequence might contain. The media found is imported as segments of the sequence. If Link to Media Files is also enabled, the application first tries to relink to the media using the paths, and then searches for what could not be located.

Directories Up field Use to expand the search to parent directories. The application searches for media to match by going down any folder structure, starting with the directory from where the sequence file is imported.

NOTE When setting the Directories Up field, keep in mind that the application navigates through the whole directory structure. This means that the higher up you go in the folder structure, the longer the conform takes.

Match Criteria box Opens the list of criteria used to identify the sources of the imported sequences.

Available Match Criteria

Name The name of the clip referenced by the sequence.

Filename The filename specified in the sequence as a match criteria.

Source Timecode The source timecode specified in the sequence as a match criteria. Enables Consolidate on Import.

Tape The tape name specified in the sequence as a match criteria.

UMID The starting SMPTE UMID in the sequence as a match criteria. This is only used with MXF files and is ignored in all other cases.

Resolution The resolution specified in the sequence as a match criteria. If this option is disabled, the Flame applies a Resize FX to the media found to make it match the resolution specified in the imported sequence, if required.

Frame Rate The frame rate specified in the sequence. Disable to disregard the frame rate; try slipping the clip and using a timewarp in the timeline after loading the timeline to correct any frame rate discrepancy.

Be careful when using the Use Frame Rate search option with FCP XML files, especially if the FCP sequence is using multiple frame rates. With Use Frame Rate enabled, Flame uses the frame rate of the sequence's *edits* as a match criteria to relink to the correct sources: if the frame rate of the considered source is 24 fps while the edit is at 30 fps, that source is not a potential candidate. Whether or not Flame relinks the considered source to the edit has nothing to do with the frame rate of the *sequence*.

But, if you are importing an FCP sequence, Flame timewarps the *edits* so that they match the frame rate of the imported sequence. For example, an FCP sequence @60fps contains edits @50fps: Flame timewarps the edits to 60 fps as it imports the sequence. Whether or not edits are relinked to their sources has nothing to do with this: if Use Frame Rate is enabled, Flame only relinks the above edits if the sources are matching the *edits' original frame rate*, in this case 50fps.

EDL Import Options

Media Options

Link to Files button Enable to create a sequence with segments that link to the original media. Disable to import an unlinked sequence. Only applies to DLEDL.

Consolidate on Import button Enable to limit the handles to the amount specified in Maximum Handles. Only applies to DLEDL.

Maximum Handles field Displays the maximum number of handles (head and tail) allowed for each event in the sequence. If the sequence also sets the amount of handles, the application imports the sources using the lowest number of handles set between Maximum Handles and the sequence. Editable. Only applies to DLEDL.

Save Sources Separately button Enable to import each source referred by the sequence, and to save them in a Sources folder. Only applies to DLEDL.

EDL Options

EDL Frame Rate box Select the frame rate applied to the imported EDL. Defaults to the project's.

EDL Conversion button Enable to convert the EDL to another frame rate using the settings defined by the Conversion box.

Conversion box Select the option that corresponds to the type of conversion that you want to apply to the EDL. The conversion scripts that appear in this list depend on the frame rate of the EDL you are loading.

Fix Timewarp Match Frames Errors button Enable to fix match frame errors. When you import an EDL that contains dissolves or timewarps, a match frame error may occur. This can cause an unwanted cut at the point where the timewarp begins in your EDL. Match frame errors occur when the in point of the second edit in a dissolve is not the same timecode as the out point of the previous shot.

Pulldown Removal button Enable to remove 2:3 pulldown when loading the EDL. The resulting sequence is @ 23.976 or 24 fps, depending on the setting of the Pulldown Original FCM box.

Pulldown Original FCM box Select the Frame Code Mode that matches the EDL that you are loading. If you are loading multiple EDLs, the same mode is used for all of them.

Multi-Assemble button Enable to assemble into one sequence multi-selected EDLs. In the imported sequence, each EDL will appear on a different video track, based on the selection order.

If you drag-and-drop a folder with multiple EDL, it creates either a single video track sequence or a single sequence with multiple video tracks, based on the Multi-Assemble option. With Multi-Assemble enabled, the alphabetical order defines the video track assigned to each EDL.

EDL Detection Mode button Select Automatic Detection to let the application multiple EDL into a single sequence. Select Selection Order to construct a the sequence based on the selection order. Select Custom Grouping to build the sequence through an EDL assembly window.

Varicam button Enable if the EDL is links to material with a Varicam. You must also select a frame rate from the Varicam Frame Rate box.

Jobs

The Jobs tab displays ongoing and completed background processes. Use the Jobs tab to monitor the import and export processes.

Actions box Select from the list an action to perform on the selected job.

When aborting the processing of a file sequence, frames already processed are retained. For example, if the Status column reads 20 of 44 when you click Abort, 20 of the 44 frames of the clip remain. Aborting a streaming file cancels the whole process, there is no partial processing.

Customizing your File Imports Options

Flame includes pre-defined import options for each available file format. While these presets should answer most of your needs, there are times you might want a different debayering settings for .r3d files, or rename a clip on import.

NOTE Whenever you change an import option, the new setting is saved to the project structure. This means that archiving a project archives the import options. Or this ensures that when transferring a project to another workstation, it is possible to import new media in the transferred project using the same import options settings.

To customize the import options of a selected file:

1. In the MediaHub, click the Format Options tab.
2. In the file browser, navigate to, and double-click the media file to import.
   The Selected Format field displays the format of the selected file.
3. From the Options box, select a category of options to edit.
4. Edit the options as needed.
   The import options are saved automatically; no need for a manual save. Flame will import media of that format using the new options.

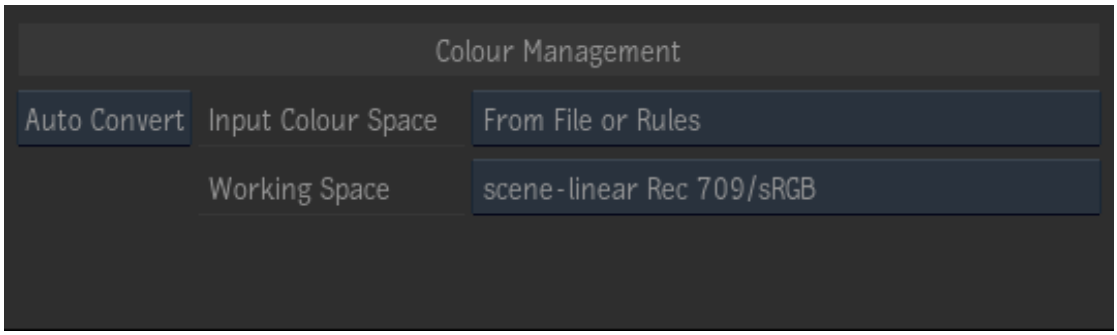
5 Drag and drop the media file from the browser to the Media Library. A new clip is created, using the new import options.

To customize the import options at any time:

1 Click the MediaHub tab.
2 In the MediaHub menu, click the Format Options tab.
3 Enable Show All Formats.
4 From the Selected Format box, select the media format to edit.
5 From the Options box, select the category of options to change.
6 Edit the options as needed.

The import options are saved automatically; no need for a manual save. From now on, media of that format are imported using the options you have set.

Applying Colour Management on Import



From the General tab of the MediaHub, you can manually apply a LUT or Colour Transform or automatically convert different media types to an appropriate working space on import.

All imported media is also tagged with a colour space identifier. See also: Tagging Clips with a Colour Space (page 1756)

Start by selecting the appropriate option from the Colour Management mode box. Available modes are:

- **Auto Convert:** Auto Convert mode will convert media in various input spaces to a common working space and tag them.
- **Use LUT:** Use the Colour Management tools to manually import or create a LUT or colour transform to apply. Then tag the result with a colour space.
- **Tag Only:** Tags the media with a colour space but does not modify the pixels.

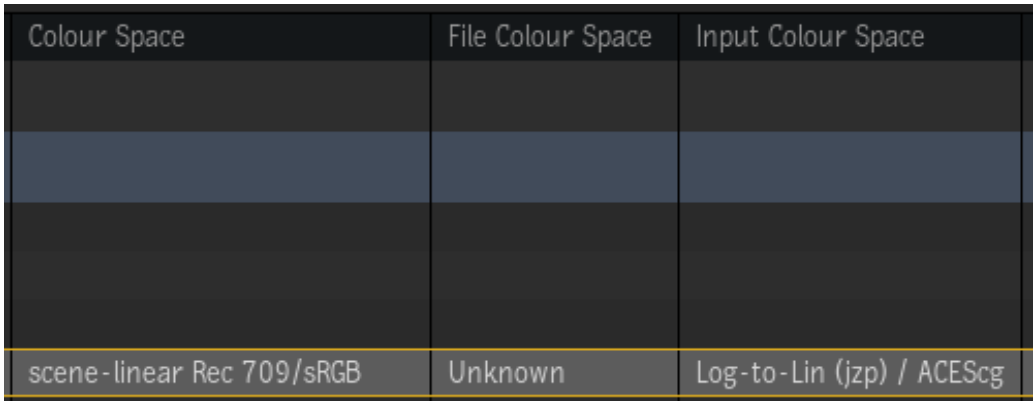
The "From File or Rules" option will attempt to automatically deduce the colour space of the source by looking at the file header and using the colour management Input Rules.

For ARRI, RED, and Sony RAW formats, using Auto Convert with the "From File or Rules" option converts directly to the chosen Working colour space using the highest quality path (i.e. you do not have to worry about which camera-specific Gamma and Colour primaries to specify in each format's SDK options). Auto mode is also supported in MediaReactor, so you can add Rules for file types that are imported in that way.

The Auto Convert mode is equivalent to the Input Transform mode in the Colour Management Timeline effect and Batch node.

See also: Applying Colour Management to Clips (page 1760)

The Media List view optionally shows various colour space metadata relevant to import. (Columns may be enabled by right clicking in the table header.)



The interpretation of the columns is as follows:

- **Colour Space:** This is the colour space the clip will be tagged with after import. When in Tag Only or Use LUT mode, it is equivalent to the Tagged Colour Space. When in Auto Convert mode, it is equivalent to the Working Colour Space.
- **File Colour Space:** This is the colour space that would be used for the "From File or Rules" option.
- **Input Colour Space:** When using the "From File or Rules" option, this is equivalent to the File Colour Space. However, if another option is selected to override the rules, that will be shown in this column. In Auto Convert mode, this is equivalent to the Input Colour Space. In Tag Only mode, this is equivalent to the setting of the Tagged Colour Space. In Use LUT mode, this is set to Unknown (since the input colour space of the LUT is generally unknown).

The Input Colour Space and Colour Space are also shown in the MediaHub Preview panel Clip Info.

You may also search for a colour space using the Media Panel search feature.

Available Colour Space Options

The following is an overview of the stock colour space choices in the Autodesk collection. (Other options are available via creating User Colour Spaces.)

Background info: "Gamma and Gamut"

Many of the colour spaces use a "Gamma / Gamut" naming convention, for example, "gamma 2.4 / Rec.709". The "Gamma" describes the relationship between the encoded values and linear values. The "Gamut" specifies the colour primaries used.

- The term "Gamma" is used very loosely since many of the colour spaces use a logarithmic function rather than a power law.
- Some colour spaces say "(no primaries)" for the "Gamut". This means that the transform to the working colour space is only a 1d-LUT -- there is no adjustment for primaries. One way of interpreting this is that the source values use whatever primaries the selected working space does.
- There are pros and cons of correcting for Gamut (that is, accounting for differences in the primaries between capture, working space, and output). One disadvantage is that it can create negative values. Some of the advantages are that you can convert various sources into a common working space with a gamut that is larger than the intended deliverable(s).
- When you don't want to correct for Gamut, use transforms that primarily all use the same Gamut values. Choose the Gamma based on the actual encodings.

Broadcast and Displays options

- The Broadcast category contains the conventional video colour spaces. For the Gamma (EOTF), all the SDR options use ITU-R BT.1886 (essentially a 2.4 gamma).
- The Rec.2100 options are for HDR monitors.
- The Display category has the other video-like colour spaces.
- sRGB, AdobeRGB, and ProPhotoRGB are often used for tagging images from Photoshop, etc.
- DCDM is the space used for digital cinema distribution masters.
- DCI-P3 is the gamut for the SMPTE Reference Projector and is widely used in grading theaters.
- The Eizo and HP options are for high-end monitors in their Native mode.
- The ST-2084 (PQ) options are for HDR monitors.
- These colour spaces are all in the Video family.

Camera options

- Includes ARRI, Canon, Panasonic, RED, Sony, VisionResearch, and generic ITU OETFs.
- Mostly use Gamma / Gamut naming
- The "(safeBlack)" option applies a slight toe in the response curve to avoid negative values.
- These options are in the Log or Scene-linear families.

Log options

- ACEScc and ACEScct logarithmic working spaces for colour correction. ACEScct has a slight toe in the shadows to avoid negatives and make for easier grading.
- ADX10 is useful for typical Cineon-style film scans. ADX16 is a wider range encoding capable of handling more recent film stocks.
- Lots of Gamma / Gamut colour spaces useful when not wanting to correct for Gamut.
- Two standard Log-to-Lin curves provided: Cineon is the classic 95/685/0.6 curve (be careful since values below 95 become negative); JZP is the Josh Pines (aka "pivoted") curve which is similar except 445 maps to 0.18 and all log values map to positive linear values.
- These options are in the Log family.

Scene-linear options

- All are scene-linear colour spaces, the only difference is the Gamut (primaries).
- ACEScg is a recommended space for CGI and compositing with a gamut that is wide, but not too wide.
- These options are in the Scene-linear family.

Textures options

- Large set of various Gamma / Gamut combinations.
- Intended for Texture or "Linear" Workflows where the intention is that a gamma-corrected value of 1.0 maps to a scene-linear value of 1.0.
- These options are in the Video family.

Data options

- The options in the Data category are to tag Mattes, Normals, Z-Depth, AOVs, etc.
- This can be useful to assign a viewing transform appropriate for the type of data (e.g. since Normals are half negative, it can shift everything up; for Z-Depth it could compress the range logarithmically).
- Keyer outputs are tagged as Matte.
- Action outputs use many of the other Data options.
- These options are in the Data family.

View Transform options

Two Viewing Transform options deserve special discussion: "Video (colorimetric)" and "Linear (gamma-corrected)". The goal of these two transforms is to take advantage of the Graphics Monitor and Broadcast Monitor colour space settings to simply present an accurate 1:1 display of the source colours on the given hardware. These transforms essentially represent an identity View Transform that does no "picture rendering" for display. They are very similar except the former expects as input colour spaces in the Video family and the latter expects as input colour spaces in the Scene-linear or Log families.

Here are some more notes on the View Transforms options:

- View Transforms convert from a scene-linear colour space to a video colour space. Using the terminology from ITU-R BT.2100, this is an OOTF (Opto-Optical Transfer Function).
- The ACES Output options essentially calculate a custom RRT+ODT for the given display. The choices are for the dynamic range, viewing conditions, and desired white point options supported in the ACES 1.0 system.
- The Cameras category has renderings for common digital cinema cameras.
- The Diagnostic category has options for camera-style exposure metering, flagging out-of-range values, etc.
- The Legacy category has traditional simple Gamma curves that do not apply any correction for Gamut.
- The Rec.2100 and Rec.709 options are various OOTFs from the standards.

Miscellaneous options

- "From File or Rules" first tries looking at the file header, then applying the Input Rules
- "From Project" uses the default project Working Space
- "From Source" uses the incoming colour space
- "Raw" never applies any conversion. Essentially the source values are just retagged as whatever the Working Space is set to.
- "Unknown" is used for Archive restores and Wire transfers from older projects. It essentially means "not tagged yet".
- "+Add New" allows you to create a User-defined Colour Space.

ARRIRAW Format Settings - Import

Metadata Settings

Clip Name box Select how the clip is named when imported.

Options are:
Clip Name from File Name
Enter Clip Name (manual entry)

Clip Name field Enter the name to use when importing the clip. Enabled if the Clip Name box is set to Enter Clip Name.

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Header	Read the tape name from the header of the imported file.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as the tape name when importing the clip. Enabled if the Tape Name is set to Enter Tape Name.

Orientation box Flip (vertical) or flop (horizontal) the media when importing the clip. Camera uses the orientation defined on the camera during the shoot. Select Disable Flip to use the true orientation of the image, as it was shot.

Timecode box Select which timecode to use, as ARRIRAW clips can have multiple timecodes. If the selected timecode is not available for the selected clip, the clip displays timecode 00:00:00.

Selection:	ARRIRAW Definition
Clip Code	Internally generated timecode frame count increasing with each new take.

Selection:	ARRIRAW Definition
Edge Code	Internally generated timecode frame count increasing with each new media.
Record Run	Internally generated record run timecode starting at a value defined by the user.
Free Run	Internally generated free run timecode starting at a time defined by the user.
External LTC	Longitudinal Time Code from an external sync.
External VITC	Vertically Integrated Time Code from an external sync.
Time of Day	Internally generated free run timecode starting at time of day.
Master TC	Internally generated timecode.

IMPORTANT The timecode of some ARRIRAW files was not correctly read in versions prior to Flame 2013 20th Anniversary Edition Extension 2 SP3 or Smoke 2013 Extension 1. Loading that material today shows the same visual results but with the source timecode is automatically updated. This ensures that both the ProRes and the ARRIRAW files have the same timecode, making it easier to offline in ProRes and relink to the ARRIRAW files.

Resolution Settings

Resolution box Select the resolution of the imported clip. These are debayering resolutions, not resize operations.

Options are:

- SD
- HD
- 2K
- UHD
- 4K
- Native

Debayer from 2868px button Enable to match ProRes files. Applies only to 2K and Native 2.8K in processing version 4.

Aspect Ratio box Set the frame aspect ratio used when importing clips, or select Ratio from File Header to read the ratio from each clip.

Aspect Ratio field Enter the frame aspect ratio (Width:Height) to use when importing clips. Enabled when the Aspect Ratio box is set to Enter Aspect Ratio.

Cropping Aspect box Select the frame aspect ratio used for cropping imported clips as part of the debayering process.

Anamorphotic Variant Toggle to enable a different cropping offered by the debayering engine.

Debayering Settings

NOTE Some combinations of options are not supported by the ARRIRAW SDK. If you create such a combination, the Preview displays an error message.

Debayering Mode box Select the ARRI debayer algorithm to use when importing files. HW (hardware) and SW (software) refer to the method ARRI cameras use when decoding RAW media. Note that there is no hardware debayering in Flame for ARRIRAW clips. All debayering is done in software for ARRIRAW material.

ADA stands for ARRI Debayer Algorithm.

Option:	Description:
ADA-1 HW	Reproduces the hardware optimised debayering that is realised in cameras. Named Camera in previous versions. Included for legacy.
ADA-2 SW	A more efficient debayering algorithm than ADA-1. Named AMC-1 in previous versions. Included for legacy.
ADA-3 HW	Visually matches the hardware optimised debayering realised in-camera with ARRI SUP 7.0: more complex debayering than ADA-1, with extended edge and color handling.
ADA-3 SW	Enhanced software debayering provided by the ARRI SDK, based on ADA-3 HW. Should deliver more image detail than ADA-3 HW.
ADA-5 HW	Improved image quality, sharper images, more details, less noise in red and blue channels, less aliasing and has a new regulation of parameters. Especially useful for VFX work when keying blue or green screens. Default setting.
ADA-5 SW	Enhanced software debayering, based on ADA-5 HW.
Proxy	A low quality debayering mode designed for performance over quality.

Crispness field Set the sharpness used when importing files.

ADA-5 SW Red field Set the sharpness of the ADA-5 SW Red channel. Only available when the Debayering Mode box is set to ADA-5 SW.

ADA-5 SW Green field Set the sharpness of the ADA-5 SW Green channel. Only available when the Debayering Mode box is set to ADA-5 SW.

ADA-5 SW Blue field Set the sharpness of the ADA-5 SW Blue channel. Only available when the Debayering Mode box is set to ADA-5 SW.

Colour Settings

NOTE When the MediaHub General colour management settings are in Auto Convert mode and the Input Colour Space is set to "From File or Rules", the Format Specific Colour Settings are ignored. The media will be decoded into ACES colour space and then convverted to the selected Working Space. Otherwise, make sure to set the tagging in General colour management settings to correspond to your colour settings below.

NOTE Some combinations of Source Camera (under Processing) and Colour Space are not supported.

Colour Settings box Select what colour information Flame uses when importing ARRIRAW clips.

Select:	To import ARRIRAW clips using:
User	The options you set in the Colour, Processing, and Image menus.
Camera	The look created on the ARRI camera and stored in the ARRIRAW file. This option disables the Processing and Image menus, as well as the Colour Rendering box.

Save as User Settings button Enable to make the Camera colour settings editable in the timeline FX Format Options.

Save as User Settings is implicitly enabled when Colour Settings is set to User.

Look Selection box Select from where Flame reads the Look information, when available.

ALEXA allows for a *look* creation workflow. In this workflow, a look is created in the ARRI Look Creator application and is either embedded in the source files during the shoot, or saved to a look file. And Flame can read and use this look information.

When working with ARRI Look information, you must set the Colour Space to one of the Video combinations or Flame will display a checkerboard instead of ARRI media. This is due to the fact that ARRI Look Creator only works Video encoding.

NOTE When used, the Look file must be placed in the folder containing the source material and named *look.xml*.

Select:	To:
Do not apply Look	Not apply any Look information.
From File Header	Apply Look information contained in the ARRIRAW file header.
From Look File	Apply Look information contained in an XML Look file located in the same folder as the media to import.
Header or Look File	Apply Look information from the header of the media to import. If none is present, use the information from an XML Look file located in the same folder as the file to import.
Look File or Header	Apply Look information from an XML Look file located in the same folder as the file to import. If none is present, use the information from the header of the file to import.

Apply Look LUT button Enable to use the LUT stored within the Look information read by Flame from the location determined by the Look Selection box.

Colour Space box Select one of the colour space and encoding combinations to apply to ARRIRAW clips.

When working with ARRI Look information, you must set the Colour Space to one of the Video combinations or Flame will display a checkerboard instead of ARRI media. This is due to the fact that ARRI Look Creator only works Video encoding.

Processing Settings

Processing Version box Select the version of the ARRI colour science used to process the file, or use From File Header to have Flame read the version of the colour science from the header of the imported file.

Use From File Header unless you are trying to recreate a look based on a specific colour science.

Source Camera box Select the camera that created the file, or have Flame read from the file header the camera used. Leave at Camera from File Header unless you are troubleshooting a file.

Image Settings

Exposure box Select how exposure is set. Choose Exposure From File to use the camera settings.

ISO field Select the ISO rating that is applied to the imported clips. Supported ALEXA values: 50 to 3200. Supported D-21 values: 50 to 500. Enabled when Exposure is Set ISO.

White Balance box Select how the white balance is defined. Select White Balance From File to use the camera settings burned in the clip.

NOTE To determine if white balance information is already present in a clip, look at *White bal burnt in* in the Metadata tab of the Previewer. If it displays *true*, there is white balance camera settings present in the clip and you can use the White Balance from File.

Kelvin field Set the perceptual colour temperature of the imported file. In Kelvin. Possible ALEXA values: 2000 to 11000. Possible D-21 values: 3200 to 7000. Available if White Balance is set to Set Temperature.

Green/Magenta Tint field Set the green/magenta tint balance: positive value for green tint, negative value for magenta tint. Available if White Balance is set to Set Temperature.

RGB Gain fields Set the RGB gains of the imported clip. Only available if White Balance is set to Set RGB White Balance.

Getting the Right Resolution

The actual resolution of an ARRIRAW file is determined by the sensor mode used at the time of the shoot. Most ALEXA cameras can use different sensor modes, but since ALEXA sensors are mostly identical resolution-wise, ARRIRAW files always end up with one of the following resolution.

Sensor Mode	Native Resolution
16:9 HD	2880x1620
16:9 2K	2868x1612
4:3	2880x2160
Open Gate	3414x2198

Now, ARRIRAW files being RAW files, one cannot just import the files in Flame. They need to be processed by ARRI's algorithms. This process, called debayering, allows you to re-interpret the file. This can be done using ARRI's *ARRIRAW Converter 3.0* (ARC), or directly in Flame. Flame uses the algorithms provided by ARRI to debayer ARRIRAW files, with debayering options named similar to the ones found in ARC. This ensures that in Flame you see ARRIRAW media as it was intended to be seen.

But the number of available options can make it hard to get the right resolution. Use the tables in the following sections to get the resolution you need. But before proceeding, make sure to reset the following options.

In the ARRIRAW Debayering menu:

- Resolution: Native

- Debayer from 2868px: Disabled
- Aspect Ratio: Ratio from File Header

This ensures that Flame displays a resolution that matches the sensor mode used and makes it easier to find the right information in the tables below.

NOTE The following information is correct for ALEXA footage. For D20/D21, some of these resolutions might not be available, in which case an error message is displayed in the Previewer's Clip Info tab.

Native 2880 x 1620 Resolution (16:9 HD Sensor Mode)

- Aspect Ratio of 1.78:1

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
768 x 432	SD	Disabled	From non-anamorphotic	Disabled
1920 x 1080	HD	Disabled	From non-anamorphotic	No Cropping
2048 x 1152	2K	Disabled	From non-anamorphotic	No Cropping
3840 x 2160	UHD	Disabled	From non-anamorphotic	No Cropping
4096 x 2304	4K	Disabled	From non-anamorphotic	No Cropping
2880 x 1620	Native	Disabled	From non-anamorphotic	Disabled

- Aspect Ratio of 1.85:1

  Set the Resolution box to Native.

  Apply a 2880 x 1558 centre crop resize, with a 1.85 ratio, using the Custom setting in **MediaHub ➤ General tab ➤ Resolution**.

- Aspect Ratio of 2.39:1

  Set the Resolution box to Native.

  Apply a 2880 x 1206 centre crop resize, with a 2.39 ratio, using the Custom setting in **MediaHub ➤ General tab ➤ Resolution**.

- Aspect Ratio 2.39 with 1.3x Anamorphotic

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
2048 x 858	2K	Disabled	From 1.3 Anamorphotic	2.39
4096 x 1716	4K	Disabled	From 1.3 Anamorphotic	2.39

■ Aspect Ratio 2.39 with 2x Anamorphotic

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
2880 x 810	Native	Disabled	From 2.0 Anamorphotic	2.39
2868 x 807	Native	Enabled	From 2.0 Anamorphotic	2.39

Native 2868 x 1612 Resolution (16:9 2K Sensor Mode)

The following resolutions and settings are essentially used when conforming ARRIRAW media to a sequence layed down using ALEXA ProRes 2K media.

■ Aspect Ratio of 1.78:1

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
2048 x 1152	2K	Enabled	From non-anamorphotic	No Cropping
2868 x 1614	Native	Enabled	From non-anamorphotic	Disabled

■ Aspect Ratio of 1.85:1

Set the Resolution box to Native and enable Debayer from 2868px.

Apply a 2868 x 1550 centre crop resize, with a 1.85 ratio, using the Custom setting in **MediaHub ➤ General tab ➤ Resolution**.

■ Aspect Ratio of 2.39:1

Set the Resolution box to Native.

Apply a 2868 x 1200 centre crop resize, with a 2.39 ratio, using the Custom setting in **MediaHub ➤ General tab ➤ Resolution**.

■ Aspect Ratio 2.39 with 2x Anamorphotic

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
2880 x 810	Native	Disabled	From 2.0 Anamorphotic	2.39
2868 x 807	Native	Enabled	From 2.0 Anamorphotic	2.39

Native 2880 x 2160 Resolution (4:3 Sensor Mode)

■ Aspect Ratio of 1.33:1

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
768 x 576	SD	Disabled	From non-anamorphotic	Disabled
1920 x 1440	HD	Disabled	From non-anamorphotic	No Cropping

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
2048 x 1536	2K	Disabled	From non-anamorphotic	No Cropping
3840 x 2880	UHD	Disabled	From non-anamorphotic	No Cropping
4096 x 3072	4K	Disabled	From non-anamorphotic	No Cropping
2880 x 2160	Native	Disabled	From non-anamorphotic	Disabled

- Aspect Ratio of 1.33:1 (specifically for ProRes matching)

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
2048 x 1536	2K	Enabled	From non-anamorphotic	No Cropping
2868 x 2152	Native	Enabled	From non-anamorphotic	Disabled

- Aspect Ratio of 1.78:1

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
1920 x 1080	HD	Disabled	From non-anamorphotic	1.78
2048 x 1152	2K	Disabled*	From non-anamorphotic	1.78

NOTE *When matching to ProRes 2048 x 1152, enable Debayer from 2868px.

- Aspect Ratio of 1.85:1

  Set the Resolution box to Native and enable Debayer from 2868px.

  Apply a 2868 x 1550 centre crop resize, with a 1.85 ratio, using the Custom setting in **MediaHub ➤ General tab ➤ Resolution**.

- Aspect Ratio 1.78 with 1.3x Anamorphotic

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
1920 x 1080	HD	Disabled	From 1.3 Anamorphotic	1.78
3840 x 2160	UHD	Disabled	From 1.3 Anamorphotic	1.78

- Aspect Ratio 1.85 with 1.3x Anamorphotic

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
1998 x 1080	2K	Disabled	From 1.3 Anamorphotic	1.85

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
3996 x 2160	4K	Disabled	From 1.3 Anamorphotic	1.85

- Aspect Ratio 2.39 with 2x Anamorphotic

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
1920 x 804	HD	Disabled	From 2.0 Anamorphotic	2.39
2048 x 858	2K	Disabled	From 2.0 Anamorphotic	2.39
3840 x 1608	UHD	Disabled	From 2.0 Anamorphotic	2.39
4096 x 1716	4K	Disabled	From 2.0 Anamorphotic	2.39

Native 3414 x 2198 Resolution (Open Gate Sensor Mode)

- Aspect Ratio of 1.55:1

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
3414 x 2198	Native	Disabled	From non-anamorphotic	Disabled

- Aspect Ratio of 1.78:1

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect
1920 x 1080	HD	Disabled	From non-anamorphotic	1.78
2048 x 1152	2K	Disabled	From non-anamorphotic	1.78
3840 x 2160	UHD	Disabled	From non-anamorphotic	1.78
4096 x 2304	4K	Disabled	From non-anamorphotic	1.78

- Aspect Ratio of 1.85:1

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect	ARRI ARC Resolution Name
1920 x 1080	HD	Disabled	From non-anamorphotic	1.85	HD
1998 x 1080	2K	Disabled	From non-anamorphotic	1.85 with *Anamorphotic*	2K DCP

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect	ARRI ARC Resolution Name
				Variant enabled	
2048 x 1108	2K	Disabled	From non-anamorphotic	1.85	2K
3996 x 2160	4K	Disabled	From non-anamorphotic	1.85 with *Anamorphotic Variant* enabled	4K DCP
4096 x 2214	4K	Disabled	From non-anamorphotic	1.85	4K

- Aspect Ratio of 2.39:1

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect	ARRI ARC Resolution Name
1920 x 804	HD	Disabled	From non-anamorphotic	2.39	HD
2048 x 858	2K	Disabled	From non-anamorphotic	2.39	2K
3840 x 1608	UHD	Disabled	From non-anamorphotic	2.39	UHD-1
4096 x 1716	4K	Disabled	From non-anamorphotic	2.39	4K

- Aspect Ratio of 2.39:1 with 2x Anamorphotic

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect	ARRI ARC Resolution Name
1920 x 804	HD	Disabled	From 2.0 anamorphotic	2.39 with *Anamorphotic Variant* enabled	HD
2048 x 858	2K	Disabled	From 2.0 anamorphotic	2.39 with *Anamorphotic Variant* enabled	2K
3840 x 1608	UHD	Disabled	From 2.0 anamorphotic	2.39 with *Anamorphotic*	UHD-1

Target Resolution	Resolution box	Debayer from 2868px	Aspect Ratio	Cropping Aspect	ARRI ARC Resolution Name
				Variant enabled	
4096 x 1716	4K	Disabled	From 2.0 anamorphotic	2.39 with *Anamorphotic Variant* enabled	4K

Audio File Settings - Import

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Timecode is read from the file header, when present. Not editable.

Sampling Rate box Project assigns to the audio clip a sampling rate equivalent to the project's timing. Use Select Rate to set the frame rate with the Frame Rate box.

Frame Rate box Select the frame rate of the imported clip. Enabled if the Rate box is set to Select Rate.

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Only applies to clips running at 29.97 and 59.94 frame rates. Enabled if the Rate box is set to Select Rate.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

AVCHD Format Settings - Import

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Aspect Ratio box Select the aspect ratio assigned to imported clips. Enter Pixel and Enter Aspect affect pixel and frame a pixel and frame aspect ratios respectively. You might need to change the aspect ratio as many applications erroneously write to exported files a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires you to enable, in the Player, **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips using the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Enter the aspect ratio of the imported frames, as a Width:Height ratio. Enabled when the Aspect Ratio box is set to Enter Aspect Ratio or Square Pixels.

YUV Decoding box Select the YUV colour space used by the clip to import. Because Flame works in the RGB colour space, it needs to translate YUV information into RGB. This setting ensures that the right decoder is used for this. The Auto setting selects the colour decoder based on the resolution of the clip: Rec. 601 for clips with a resolution lower than 720 lines, Rec. 709 for everything else. Specifying the wrong colour space results in colours that are off.

Per Recommendation BT. 2020, UHD media should use the Rec. 2020 colour space. But this is rarely the case at the moment: UHD clips use Rec. 709. The Rec. 2020 option is there is case you come upon a correctly encoded UHD clip.

Include YUV Headroom button Enable Include YUV Headroom when importing content that has Full Range video levels. Content with Full Range includes original media from cameras, such as:

- Sony S-Log
- ARRI Log-C
- Canon Log
- Panasonic Varicam Log

Include YUV Headroom should also be enabled when importing from grading systems content that has not been legalized. Disable this setting when importing content with legal video levels, like masters or final grade media.

NOTE This setting has no impact on media that is not YUV.

Canon Format Settings - Import

The following settings apply to all of Canon camera files.

Notes regarding Canon's XF-AVC codec.

- Canon XF-AVC clips are OP-1a clips with LPCM 16- or 24-bit 48 kHz audio.
- Canon XF-AVC media is recorded to a folder structure. Flame interprets that structure to present in MediaHub a clip that includes both audio and video tracks, in a manner similar to its processing of Panasonic P2 media. Enable **Browsing ➤ Essence Mode** to browse the folder structure and access directly the MXF media files.
- Flame supports both Intra and Long GOP flavors, as well as spanned clips.

Tips when importing Canon spanned clips:

- Canon cameras can split the recording of a long clip in a series of smaller clips, either across many memory cards (spans) or within a single memory card (breaks). Flame rebuilds, from related spans or breaks, a single clip, making it easier to import and manage. But this only works if you preserve the camera original folder structure to be able to rebuild the spanned clips as the MXF files do not contain enough information for that.
- You can browse the individual segments by enabling **Browsing ➤ Essence Mode**.
- For AAF Conform using the camera original media files, see the AAF Conform topics.

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Essence	Use the tape name inferred from the directory structure of the clip.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Timecode is read from the file header, when present. Not editable.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Select:	To:
Clip Name from XML	Use the clip name listed in the XML file accompanying the file.
Clip Name from Essence	Use the clip name inferred from the directory structure of the clip.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Aspect Ratio box Select the aspect ratio assigned to imported clips. Enter Pixel and Enter Aspect affect pixel and frame a pixel and frame aspect ratios respectively. You might need to change the aspect ratio as many applications erroneously write to exported files a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires you to enable, in the Player, **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips using the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Enter the aspect ratio of the imported frames, as a Width:Height ratio. Enabled when the Aspect Ratio box is set to Enter Aspect Ratio or Square Pixels.

YUV Decoding box Select the YUV colour space used by the clip to import. Because Flame works in the RGB colour space, it needs to translate YUV information into RGB. This setting ensures that the right decoder is used for this. The Auto setting selects the colour decoder based on the resolution of the clip: Rec. 601 for clips with a resolution lower than 720 lines, Rec. 709 for everything else. Specifying the wrong colour space results in colours that are off.

Per Recommendation BT. 2020, UHD media should use the Rec. 2020 colour space. But this is rarely the case at the moment: UHD clips use Rec. 709. The Rec. 2020 option is there is case you come upon a correctly encoded UHD clip.

Include YUV Headroom button Enable Include YUV Headroom when importing content that has Full Range video levels. Content with Full Range includes original media from cameras, such as:

- Sony S-Log
- ARRI Log-C
- Canon Log
- Panasonic Varicam Log

Include YUV Headroom should also be enabled when importing from grading systems content that has not been legalized. Disable this setting when importing content with legal video levels, like masters or final grade media.

NOTE This setting has no impact on media that is not YUV.

Browsing Settings

Essence Mode button Enable to browse the directory structure of Canon clips. This allows you to import specific video and audio media contained within a Canon clip.

DPX Format Settings - Import

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Header	Read the tape name from the header of the imported file.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Select an option to specify how the timecode information is set when importing clips.

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.
Timecode from File Name	Use a numerical file name (for instance, *100000.xyz*) and translate it into timecode for the resulting clip (based on the selected frame rate). This is useful when working with files that do not have embedded timecode.

Rate box Select Auto to use the frame rate specified by the clip; with no rate specified, the application assigns a rate from the Default Resolutions table, or the project's rate if there is no match in the table. Use Select Rate to set the frame rate using the Frame Rate box.

The Default Resolutions table uses a frame's height and width to assign it an aspect ratio, a scan mode, and a frame rate. You can find the Default Resolutions table in **MediaHub ➤ Browse for Archives ➤ Default Resolutions**. It is also used when restoring legacy archives.

Frame Rate box Select the frame rate of the imported clip. Enabled if the Rate box is set to Select Rate.

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Only applies to clips running at 29.97 and 59.94 frame rates. Enabled if the Rate box is set to Select Rate.

A DPX file can specify its frame rate in its file header, using either the TV or Movie rate fields. This is where the frame rate information is read when you set the Rate box to Auto Rate. If both fields are defined, Flame uses the Movie rate.

For 29.97 fps DPX files, Flame also reads the drop frame mode from the DPX file header if you enable Auto Rate and if that information is defined.

Also note that DPX SMPTE timecode does not support 50 fps or 60 fps timecodes, which means that the file header timecode is not accurate or reliable for these rates. Set Flame to read the timecode from the file name instead of the file header: set the Timecode Selection box to Timecode from File Name.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Sequence Detection box Select how Flame displays image sequences. Sequence builds and displays a clip out of multiple files, using their numerical suffix to rebuild the sequence. Frames displays each file independently and disables the Tape Name box from the Metadata menu.

An image sequence consists of image files, such as .psd, named identically except for a numeric suffix.

- myPhotoshopFile_version_001.psd
- myPhotoshopFile_version_002.psd
- myPhotoshopFile_version_003.psd
- myPhotoshopFile_version_004.psd

Keycode Settings

NOTE The keycode supplied here is only used for information purporses. In case of a discrepancy between the information supplied in the Metadata menu and the Keycode menu, the former is used to determine the timecode and frame rate used.

Keycode Scan Mode box Select an option to determine how keycode data is applied when importing the clip.

Select:	To:
File Header Keycode	Use the keycode information embedded in the image file header.
No Keycode	Discard the keycode information.

Keycode FCM box Select the frame code mode of the tape. Select File FCM to read the frame code mode from the file.

Film Gauge box Select a film gauge for the keycode.

About Keycode

Keycode (also known as edge code) tracks frames of film from source reels through the post-production pipeline using code. Keycode is a variation of timecode designed to uniquely identify frames in filmstock. Keycode is a valuable tool for any project that originates on film, evolves through digital post production, and eventually references the original film material for final image capture. Keycode is printed on film in both a human-readable and machine-readable form. It indicates the manufacturer and film emulsion, a unique identification for each reel, as well as the footage and frame number.

Keycode appears in a varying number of increments along the film strip depending on the film gauge. For example, on 35 mm film, keycode appears every foot, which translates to every 16 frames of film.

Keycode Syntax

Keycode in Autodesk Creative Finishing applications uses the following syntax:

KQ123456 7890+12

Where:

- K is the film manufacturer.
- Q is the film emulsion.
- 123456 (6 digits) is the film reel unique identifier.
- 7890 (4 digits) is the footage number.
- 12 (2 digits) is the frame number.

In 35mm / 3 perf keycode, a reference foot value is also set. The value can be 1, 2, or 3 between parenthesis:

KQ123456 1234+00(1)

Supported Film Emulsions

The available film emulsions are supported in keycode in Flame as follows.

Kodak

- 5274/7274 Vision 200T
- 5620/7620 Primetime 640T
- SFX 200T
- 5287/7287 EXR Ultra Lat
- 5244/7244
- 5279/7279 Vision 500T
- 5298/7298
- 5272/7272
- 5289/7289 Vision 800T
- 5277/7277 Vision 320T
- 5600/7600 Primetime
- 5249/7249
- 5292/7292
- 5248/7248
- 5293/7293
- 5245/7245
- 5296/7296
- 5246/7246 Vision 250D
- 5231/7231
- 5294/7294
- 5295/7295
- 5222/7222
- 5234/7234
- 5297/7297
- 5247/7247
- 5243/7243

Fuji

- VELVIA col rev 8540
- F-64D/125/250D/500
- FCI

Eastman

- 5242/7242 Vision Color I
- 5205/7205 Vision2 250D
- 5212/7212 Vision2 100T
- 5217/7217 Vision2 200T
- 5201/7201 Vision2 50D
- 5219/7219 Vision3 500T
- 5299/7299 Vision2 HD
- 5218/7218 Vision2 500T
- 5260 Vision2 500T
- 5284/7284 Vision Exp 500T
- 5263/7263 Vision 500T
- 7266 TRI-X
- 7265 PLUS-X
- 5229/7229 Vision2 Exp 500T
- 5285/7285 Ektachrome 100D
- 5203/7203 VISION3 50D
- 5223/7223 Vision3 640T
- 5227/7227 Vision3 500T
- 5230/7230 500T
- 5273/7273/2273/3273 Internegative

Agfa

- XTS 400
- XT 100
- XTR 250
- XT 320

Supported Film Gauges

The following film gauges are supported in keycode for Autodesk Visual Effects and Finishing applications:

- 16 mm and Super 16 mm / 1 perf
- 35mm / 8 perf
- 35mm / 3 perf
- 35mm / 4 perf
- 65mm / 5 perf

The increment offset of a particular frame from the keycode dot (which indicates the zero offset point on the film) can be recorded in one of two modes: frame offset and perforation offset. In general, film scanners can write keycode in either of these two modes.

For 35 mm / 3 perf and 35 mm / 4 perf film, both frame and perforation offset keycode are supported when importing files. For all other gauges, only frame offset mode is supported. In these latter cases, set the telecine to use frame offset keycode when you are scanning.

HDR Format Settings - Import

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips. Disabled if Sequence Detection from the Clip Options menu is set to Frames.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.

Select:	To:
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Timecode is read from the file name, when present. Not editable.

Rate box Select Auto to use the frame rate specified by the clip; with no rate specified, the application assigns a rate from the Default Resolutions table, or the project's rate if there is no match in the table. Use Select Rate to set the frame rate using the Frame Rate box.

The Default Resolutions table uses a frame's height and width to assign it an aspect ratio, a scan mode, and a frame rate. You can find the Default Resolutions table in **MediaHub ➤ Browse for Archives ➤ Default Resolutions**. It is also used when restoring legacy archives.

Frame Rate box Select the frame rate of the imported clip. Enabled if the Rate box is set to Select Rate.

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Only applies to clips running at 29.97 and 59.94 frame rates. Enabled if the Rate box is set to Select Rate.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Sequence Detection box Select how Flame displays image sequences. Sequence builds and displays a clip out of multiple files, using their numerical suffix to rebuild the sequence. Frames displays each file independently and disables the Tape Name box from the Metadata menu.

An image sequence consists of image files, such as .psd, named identically except for a numeric suffix.

- myPhotoshopFile_version_001.psd
- myPhotoshopFile_version_002.psd
- myPhotoshopFile_version_003.psd
- myPhotoshopFile_version_004.psd

Image Settings

Exposure Adjustement field Set the exposure of the HDR file, in units of f-stop.

Image Sequence Settings - Import

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips. Disabled if Sequence Detection from the Clip Options menu is set to Frames.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Select an option to specify how the timecode information is set when importing clips.

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.
Timecode from File Name	Use a numerical file name (for instance, *100000.xyz*) and translate it into timecode for the resulting clip (based on the selected frame rate). This is useful when working with files that do not have embedded timecode.

Rate box Select Auto to use the frame rate specified by the clip; with no rate specified, the application assigns a rate from the Default Resolutions table, or the project's rate if there is no match in the table. Use Select Rate to set the frame rate using the Frame Rate box.

The Default Resolutions table uses a frame's height and width to assign it an aspect ratio, a scan mode, and a frame rate. You can find the Default Resolutions table in **MediaHub ➤ Browse for Archives ➤ Default Resolutions**. It is also used when restoring legacy archives.

Frame Rate box Select the frame rate of the imported clip. Enabled if the Rate box is set to Select Rate.

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Only applies to clips running at 29.97 and 59.94 frame rates. Enabled if the Rate box is set to Select Rate.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Sequence Detection box Select how Flame displays image sequences. Sequence builds and displays a clip out of multiple files, using their numerical suffix to rebuild the sequence. Frames displays each file independently and disables the Tape Name box from the Metadata menu.

An image sequence consists of image files, such as .psd, named identically except for a numeric suffix.

- myPhotoshopFile_version_001.psd
- myPhotoshopFile_version_002.psd
- myPhotoshopFile_version_003.psd
- myPhotoshopFile_version_004.psd

MP4 Format Settings - Import

Sony XAVC-S supported rates and resolutions.

- 1920x1080p (HD)
  - 23.97 fps (50 Mb/s)
  - 25 fps (50 Mb/s)
  - 29.97 fps (50 Mb/s)
  - 50 fps (50 Mb/s)
- 3840x2160p (4K UHD):
  - 23.97 fps (60 Mb/s)
  - 25 fps (60 Mb/s)
  - 25 fps (100 Mb/s)
  - 29.97 fps (60 Mb/s)
  - 29.97 fps (100 Mb/s)
  - 50 fps (150 Mb/s)
  - 59.94 fps (150 Mb/s)

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Essence	Use the tape name inferred from the directory structure of the clip.
Tape Name from Header	Read the tape name from the header of the imported file.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Timecode is read from the file header, when present. Not editable.

Rate box Select Auto to use the frame rate specified by the clip; with no rate specified, the application assigns a rate from the Default Resolutions table, or the project's rate if there is no match in the table. Use Select Rate to set the frame rate using the Frame Rate box.

The Default Resolutions table uses a frame's height and width to assign it an aspect ratio, a scan mode, and a frame rate. You can find the Default Resolutions table in **MediaHub ➤ Browse for Archives ➤ Default Resolutions**. It is also used when restoring legacy archives.

Frame Rate box Select the frame rate of the imported clip. Enabled if the Rate box is set to Select Rate.

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Only applies to clips running at 29.97 and 59.94 frame rates. Enabled if the Rate box is set to Select Rate.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.

Select:	To:
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from Header	Read the clip name from the header of the imported file.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Aspect Ratio box Select the aspect ratio assigned to imported clips. Enter Pixel and Enter Aspect affect pixel and frame a pixel and frame aspect ratios respectively. You might need to change the aspect ratio as many applications erroneously write to exported files a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires you to enable, in the Player, **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips using the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Enter the aspect ratio of the imported frames, as a Width:Height ratio. Enabled when the Aspect Ratio box is set to Enter Aspect Ratio or Square Pixels.

YUV Decoding box Select the YUV colour space used by the clip to import. Because Flame works in the RGB colour space, it needs to translate YUV information into RGB. This setting ensures that the right decoder is used for this. The Auto setting selects the colour decoder based on the resolution of the clip: Rec. 601 for clips with a resolution lower than 720 lines, Rec. 709 for everything else. Specifying the wrong colour space results in colours that are off.

Per Recommendation BT. 2020, UHD media should use the Rec. 2020 colour space. But this is rarely the case at the moment: UHD clips use Rec. 709. The Rec. 2020 option is there is case you come upon a correctly encoded UHD clip.

Include YUV Headroom button Enable Include YUV Headroom when importing content that has Full Range video levels. Content with Full Range includes original media from cameras, such as:

- Sony S-Log
- ARRI Log-C
- Canon Log
- Panasonic Varicam Log

Include YUV Headroom should also be enabled when importing from grading systems content that has not been legalized. Disable this setting when importing content with legal video levels, like masters or final grade media.

NOTE This setting has no impact on media that is not YUV.

MXF Format Settings - Import

Material at 1080i sometimes uses a thin raster frame format. It uses a frame size of 1440x1080or 1280x1080. When importing, enable Scale to Full HD in **Format Specific Options ➤ Image** to scale the material to the full 1920x1080 frames. Disable Scale to Full HD to import the clip using the thin raster frame format.

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Header	Read the tape name from the header of the imported file.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Timecode is read from the file header, when present. Not editable.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from Header	Read the clip name from the header of the imported file.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Aspect Ratio box Select the aspect ratio assigned to imported clips. Enter Pixel and Enter Aspect affect pixel and frame a pixel and frame aspect ratios respectively. You might need to change the aspect ratio as many applications erroneously write to exported files a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires you to enable, in the Player, **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips using the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Enter the aspect ratio of the imported frames, as a Width:Height ratio. Enabled when the Aspect Ratio box is set to Enter Aspect Ratio or Square Pixels.

YUV Decoding box Select the YUV colour space used by the clip to import. Because Flame works in the RGB colour space, it needs to translate YUV information into RGB. This setting ensures that the right decoder is used for this. The Auto setting selects the colour decoder based on the resolution of the clip: Rec. 601 for clips with a resolution lower than 720 lines, Rec. 709 for everything else. Specifying the wrong colour space results in colours that are off.

Per Recommendation BT. 2020, UHD media should use the Rec. 2020 colour space. But this is rarely the case at the moment: UHD clips use Rec. 709. The Rec. 2020 option is there is case you come upon a correctly encoded UHD clip.

Include YUV Headroom button Enable Include YUV Headroom when importing content that has Full Range video levels. Content with Full Range includes original media from cameras, such as:

- Sony S-Log
- ARRI Log-C
- Canon Log
- Panasonic Varicam Log

Include YUV Headroom should also be enabled when importing from grading systems content that has not been legalized. Disable this setting when importing content with legal video levels, like masters or final grade media.

NOTE This setting has no impact on media that is not YUV.

Scale to Full HD button Enable to have media with a 1280x1080 or 1440x1080 resolution display at a standard 1920x1080 resolution. Disable to display the media at its native (1280x1080 or 1440x1080) resolution.

Open Clip Format Settings - Import

When you import an Open Clip, the media it references is still imported using that media's Format Specific Options: a referenced .mov is imported using QuickTime settings, while a referenced .dpx sequence is imported using DPX settings.

If you cannot access an Open Clip, it is probably because its different channels are of mixed frame rate. Use the Rate box to assign a uniform frame rate, which should solve the issue.

Metadata

Rate box Select Auto to use the frame rate specified by the clip; with no rate specified, the application assigns a rate from the Default Resolutions table, or the project's rate if there is no match in the table. Use Select Rate to set the frame rate using the Frame Rate box.

The Default Resolutions table uses a frame's height and width to assign it an aspect ratio, a scan mode, and a frame rate. You can find the Default Resolutions table in **MediaHub ➤ Browse for Archives ➤ Default Resolutions**. It is also used when restoring legacy archives.

Align to Zero button Enable to ignore the timecodes defined in the different versions of the Open Clip file, and align every track (and their versions) to timecode 00:00:00:00.

Working With Open Clip Files

About Open Clip Files

In its simplest form, an Open Clip file consists of two parts:

- A .clip file
- Media files

The media is any supported media files, from DPX sequences to RED files, including multi-channel OpenEXR renders. The other component, the .clip file, is written in the XML format; you can open a .clip in any word processor (kedit, notepad, TextEdit) and decipher its content.

In essence, the .clip file contains all the metadata and references to media that are required to define a source and its versions. The .clip file does nothing by itself, but is essential to recreate the sources, similar to how a wrapper works, whereas a source is similar to the essence of a P2 clip.

One of the strengths of the open .clip is that anyone can create .clip files. With one, you can manage media outside of the Flame application.

Using an Open Clip file, you can define many aspects of a clip, including:

- Source information
- Versioning
- Multiple channels of media
- Paths to media

To create an Open Clip:

- Use the Batch Write File node with Create Clip enabled.
- Export a sequence and enable **Show Advanced Options ➤ Clip Options ➤ Create Clip Data**.
- Use a third party application or utility.
- Create one from scratch.

NOTE Starting with version 2017 extension 1, Flame supports an Open Clip with versions that mix frame rates.

Why Use an Open Clip?

For open clip history and data exchange.

Open clip history Open clip history provides you with the ability to save with a processed clip the Batch setup used to create that clip. Later on, to open that clip in Batch or the Timeline and edit previous nodes and their settings.

Data exchange Data exchange requirements with a facility's shot management system. In such a case, the .clip is a file created by a third party system, allowing something else to define a source using instructions contained within the file. In this case, the clip can contain a list of render passes for you to composite in Flame through the Read File node.

Working with Multi-Version Clips

Multi-version workflow requires the use of Open Clip files, and is an extension of the publish workflow. The following features are available when you use multi-version clips. Open Clip allows marrying versions of similar content like CG renders from Autodesk Maya, and integrates well with Autodesk Shotgun-based workflows.

A multi-version clip is a construct where a single file (using the Open Clip .xml format) contains the information required to assemble media files into different versions. Export's Publish and Batch's Write File node can both generate Open Clips with versions. Or you can build your own Open Clip using the developer available developer documentation.

It is possible to have versions of different start, end or duration: versions are aligned internally based on their timecode. The duration is then derived from the earliest versions' start time to the latest versions' end time. Different versions can reference different media formats.

Multi-Version Clips in the MediaHub

In the MediaHub, you can identify a multi-version clip by its **V** badge.

The MediaHub displays the latest available version of multi-version clip. But you can use the Preview Panel for additional information: the Versions tab displays all the versions contained in the Open Clip, as well as the location of the Batch Setup, when one is available.

Multi-Version Clips and Media Format Import Options

While there are no Import Options specific to the Open Clip, the media referenced in the different versions is imported using the import options specified by the Format Specific Options.

Using the MediaHub's **General ➤ Clip Options for Versions** box, you decide when media cache and proxies are generated automatically on import: only for the version displayed in the MediaHub, or for every version of a multi-versions clip.

Multi-Version Clips in the Media Panel and on the Desktop

The following operations are available from the contextual menu:

- Update Source: Reloads the multi-version clip, ensuring you have the latest version of the Open Clip loaded.
- Select Current Version: Reloads the multi-version clip, and then displays the version defined as current in the Open Clip by the `currentVersion` property of the `<versions>` element. Pipeline tools such as Autodesk Shotgun can change the Current Version based on approval policies.
- Select Latest Version: Reloads the multi-version clip, and then displays the latest version.
- Mark All Updates as Viewed: Dismisses the segment's white outline that indicates updated segments.
- Viewing the names of the available versions.
- Selecting a specific version.

Shared Libraries

Multi-versions clips cannot change version when located in a Shared Library since there are no Media Options available in a Shared Library. Move the multi-version out of the Shared Library to switch between versions.

Multi-Version Clips on the Timeline

To select a version for a multi-version clip:

- Clip Versions box: Allows you to select a version for the selected segment. Available from the Timeline FX ribbon, with either Format Options or Pre-Processing selected. Whenever you click Source Versions, its contents are automatically updated to display the versions currently available to the selected segment.
- From the contextual menu, under Source Versions:
  - Select Latest Version: Reloads the multi-version clip, and then displays the latest version.
  - Select Current Version: Reloads the multi-version clip, and then displays the version defined as current in the Open Clip by the `currentVersion` property of the `<versions>` element. Pipeline tools such as Autodesk Shotgun can change the Current Version based on approval policies.
  - Version Name: If you right-click a clip (not a sequence), you can select a version directly from Source Versions.

You can perform Select Latest Version, Select Current Version, and Update Source on a segment, a sequence, multi-selected sequences, and even a reel, Reels Group, a library, multiple libraries to updated all clips and sequences within.

NOTE Switching away from a version discards any frames rendered for that version.

The following shortcuts can be used on Timeline segments, Media Library, as well as in Freeform and Reels views. They are unassigned by default, so you need to enable them in the Flame menu ➤ Keyboard Shortcut editor.

- Select Current Version (can be used on multi-selection)
- Select Latest Version (can be used on multi-selection)
- Select Next Version (single selection only)
- Select Previous Version (single selection only)

Segments with White Outline

When you are working with a multi-version clip, and that clip is updated by an external application, any segment referencing this clip is outlined in white. This outline is there indicate that a new version is available for this segment. It appears in the following cases:

A white outline appears whenever you:

- Use Update Source, Select Latest Version, or Select Current Version and the segment actually updates to display a version different from the one already displayed.
- Perform Select Latest Version, Select Current Version, and Update Source on a sequence, multi-selected sequences, a reel, Reels Group, or a library: every segment within that changed version is outlined with white.

To clear the outline, do one of the following:

- For individual segments: In the Timeline FX ribbon, with Format Options or Pre-Processing selected, click Mark As Viewed.

- For multi-selections or a whole sequence: Select, from the contextual menu, **Source Versions ➤ Mark All Updates As Viewed**

Adding New Versions

From the timeline, you can create a new version of a multi-version segment with the New Version button. To be able to create a new version from the Timeline, the currently selected version in the timeline must have been created in Batch, with the Write File node's Include Setup enabled.

NOTE The New Version button is only displayed if a Batch setup is available.

Create New Version Provides access to the open setup in a fashion similar to History. Like History, exiting automatically triggers a new render, however in this case it will create all of the required media and metadata in the appropriate paths in the job tree, and appends a version to the timeline segment.

Open as Batch Group Load the currently selected version in Batch. This will include the Write File node needed to append versions to the clip, the job tree, and the segments in the timelines where it is used. This method is not available in Flame Assist due to the lack of Batch environment.

Promote to Batch FX Ingests the shot directly in the current timeline, and converts any included setup into a BFX, discarding the source clip and other versions. In most cases, you cannot demote from a BFX.

Matte Containers

If the multi-version clip is in a container (or a Matte container), you can switch between versions from the Timeline using the contextual menu. But if the contextual menu shows *Out of Sync*, the components of the container are actually different clips: you have to enter the Container, and then update the select the desired versions.

Multi-Version Clips and Conform

If you unlink a multi-version clip and relink it at a later time, as long as you use an Open Clip, it will relink to the original version.

it is possible to conform sequences using Open Clips, either already imported in the Media Library or from a file system location.

If you use Conform view to define the media location, in this case no Open Clip is used but the actual referenced media files so the resulting clips will be a single version. This is a limitation since Conform view does not relink to Open Clips. Use Set Search Location instead.

Multi-Version Clips and Managing Media, Cache, and Proxy

Hard Commit

Hard committing multi-version clips preserves all available versions, but keeps the link to handles that are located outside the range of the timeline segment at the moment on the Hard Commit.

Pre-Processing Options

LUT and Resize options set in the Pre-Processing menu apply to all versions of a multi-version Open Clip.

- If proxies exist before entering the Pre-Processing editor, and you change the current version from within, new proxies are generated on exiting the editor. The same is true for existing media cache.
- The Player in the Pre-Processing editor is not aware of changing timecodes or durations between versions: this means that switching between version with different start timecodes or durations start might result in out the Player going out of sync. Exiting the editor fixes this.

Managing Cache and Proxies

From the contextual menu, you can manage proxies and source media cache for the current version, or for all versions:

- **Media ➤ Cache Source Media(Current)**
- **Media ➤ Cache Source Media (All Versions)**
- **Media ➤ Generate Source Proxies (Current)**
- **Media ➤ Generate Source Proxies (All Versions)**

You can also flush that source or those proxies, either for all versions, only the current version, or all but the current version:

- **Media ➤ Flush Source Media Cache (Current)**
- **Media ➤ Flush Source Media Cache (All Versions)**
- **Media ➤ Flush Source Media Cache (All but Current)**
- **Media ➤ Flush Source Proxies (Current)**
- **Media ➤ Flush Source Proxies (All Versions)**
- **Media ➤ Flush Source Proxies (All but Current)**

NOTE If proxies existed before entering in the Pre-Processing editor on the Timeline, and the version changes from within the editor, new proxies are generated on exiting the editor. The same is true for existing media cache.

Multi-version Clips and Archives

If your project, or the clips you are archiving, contains multi-version clips, use the Cache Versions box to manage what is cached. This setting can impact the size of the archive.

Limiting the multi-version caches:

- All Versions: Every single version of a clip is cached on being archived.
- Used Version: Only the currently selected versions of multi-version clips are cached.

Note that in both cases, the actual clips are archived: the Cache Versions box only manages the media, not the clips.

No Media Slate

The No Media slate is displayed whenever a segment has more assigned frames then there is available media. This prevents the structure of a timeline from changing when changing versions.

- Open Clip files: In a multi-version Open Clip, every version is timecode aligned. This means that there are cases where at a given timecode there is no media to display. In this case, the No Media slate is displayed.
- .mov files: When both audio and video tracks are present, it can happen that one track is shorter than the other one. To preserve the integrity of the source, if the audio track is shorter than the video one, silence is added to fill the gap. If the video track is shorter, No Media slates are added instead.

How No Media frames are handled is defined in different locations:

Export The Export Preferences tab, available from the Export window's Advanced Options, allows you to define the behavior of the export. Always export black when No Media slate is output, or ask what to do whenever a clip contains a No Media slate.

Tools Tab The tools available in the Tools tab all share a No Media setting. The No Media Handling box, available in any tools Setup menu, sets the behavior whenever a No Media slate is encountered: render a No Media slate, or render a black frame.

Batch Read File and Clip Nodes Read File and Clip nodes allow you to control Head, Tail, and Gap media. This includes repeating frames, using black, or simply outputting no media at all. In this case, due to the requirements of the Batch pipeline, instead of a No Media slate, nothing is sent through the Batch pipeline.

OpenEXR Format Settings - Import

- In the Previewer, the Metadata tab displays all the metadata available from OpenEXR files, including the file attributes. Open the Metadata tab, and display the section **Clip Information ➤ Attributes**.
- Flame can also read from the OpenEXR header the timecode, tape name, frame rate, and the keycode.
- The colour space of the image will be set to ACES2065-1 if the import is in "From File or Rules" mode and the header contains the acesImageContainerFlag attribute set to 1, per SMPTE 2065-4.

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips. Disabled if Sequence Detection from the Clip Options menu is set to Frames.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Header	Read the tape name from the header of the imported file.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Select an option to specify how the timecode information is set when importing clips.

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.

Select:	To:
Timecode from File Name	Use a numerical file name (for instance, *100000.xyz*) and translate it into timecode for the resulting clip (based on the selected frame rate). This is useful when working with files that do not have embedded timecode.

Rate box Select Auto to use the frame rate specified by the clip; with no rate specified, the application assigns a rate from the Default Resolutions table, or the project's rate if there is no match in the table. Use Select Rate to set the frame rate using the Frame Rate box.

The Default Resolutions table uses a frame's height and width to assign it an aspect ratio, a scan mode, and a frame rate. You can find the Default Resolutions table in **MediaHub ➤ Browse for Archives ➤ Default Resolutions**. It is also used when restoring legacy archives.

Frame Rate box Select the frame rate of the imported clip. Enabled if the Rate box is set to Select Rate.

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Only applies to clips running at 29.97 and 59.94 frame rates. Enabled if the Rate box is set to Select Rate.

Clip Settings

Clip Name box

Flame imports the channels of multi-channel files as individual clips. Select how to name, at import, the created clips.

Select:	To:
Filename	Use the filename of the container for all the imported channels.
Channel	Use the channel name as the clip name. Default setting.
Channel + Filename	Combine the channel name and the filename of the container to form the clip name.
Filename + Channel	Combine the filename of the container and the channel name to form the clip name.

Sequence Detection box Select how Flame displays image sequences. Sequence builds and displays a clip out of multiple files, using their numerical suffix to rebuild the sequence. Frames displays each file independently and disables the Tape Name box from the Metadata menu.

An image sequence consists of image files, such as .psd, named identically except for a numeric suffix.

- myPhotoshopFile_version_001.psd
- myPhotoshopFile_version_002.psd
- myPhotoshopFile_version_003.psd
- myPhotoshopFile_version_004.psd

Pixel Space Window

Use the Pixel Space Window box to import an OpenEXR using either the Data window or the Display window. The OpenEXR display window is similar to a viewport, whereas the Data window is similar to a region of interest. This concept is often used in 3D, where the rendered object might not occupy the full space of the

displayed frame. Or the data window can be bigger than the Display window, similar to an overscan, to provide edge-accurate blurring.

Panasonic Format Settings - Import

Material at 1080i sometimes uses a thin raster frame format. It uses a frame size of 1440x1080or 1280x1080. When importing, enable Scale to Full HD in **Format Specific Options ➤ Image** to scale the material to the full 1920x1080 frames. Disable Scale to Full HD to import the clip using the thin raster frame format.

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Essence	Use the tape name inferred from the directory structure of the clip.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Timecode is read from the file header, when present. Not editable.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Select:	To:
Clip Name from XML	Use the clip name listed in the XML file accompanying the file.
Clip Name from Essence	Use the clip name inferred from the directory structure of the clip.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Aspect Ratio box Select the aspect ratio assigned to imported clips. Enter Pixel and Enter Aspect affect pixel and frame a pixel and frame aspect ratios respectively. You might need to change the aspect ratio as many applications erroneously write to exported files a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires you to enable, in the Player, **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips using the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Enter the aspect ratio of the imported frames, as a Width:Height ratio. Enabled when the Aspect Ratio box is set to Enter Aspect Ratio or Square Pixels.

YUV Decoding box Select the YUV colour space used by the clip to import. Because Flame works in the RGB colour space, it needs to translate YUV information into RGB. This setting ensures that the right decoder is used for this. The Auto setting selects the colour decoder based on the resolution of the clip: Rec. 601 for clips with a resolution lower than 720 lines, Rec. 709 for everything else. Specifying the wrong colour space results in colours that are off.

Per Recommendation BT. 2020, UHD media should use the Rec. 2020 colour space. But this is rarely the case at the moment: UHD clips use Rec. 709. The Rec. 2020 option is there is case you come upon a correctly encoded UHD clip.

Include YUV Headroom button Enable Include YUV Headroom when importing content that has Full Range video levels. Content with Full Range includes original media from cameras, such as:

- Sony S-Log
- ARRI Log-C
- Canon Log
- Panasonic Varicam Log

Include YUV Headroom should also be enabled when importing from grading systems content that has not been legalized. Disable this setting when importing content with legal video levels, like masters or final grade media.

NOTE This setting has no impact on media that is not YUV.

Scale to Full HD button Enable to have media with a 1280x1080 or 1440x1080 resolution display at a standard 1920x1080 resolution. Disable to display the media at its native (1280x1080 or 1440x1080) resolution.

Browsing Settings

Essence Mode button Enable to browse the directory structure of P2 clips. This option allows you to import specific audio or video media contained within a P2 clip.

Photoshop Format Settings - Import

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips. Disabled if Sequence Detection from the Clip Options menu is set to Frames.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Clip Settings

Clip Name box

Flame imports the channels of multi-channel files as individual clips. Select how to name, at import, the created clips.

Select:	To:
Filename	Use the filename of the container for all the imported channels.
Channel	Use the channel name as the clip name. Default setting.
Channel + Filename	Combine the channel name and the filename of the container to form the clip name.
Filename + Channel	Combine the filename of the container and the channel name to form the clip name.

Sequence Detection box Select how Flame displays image sequences. Sequence builds and displays a clip out of multiple files, using their numerical suffix to rebuild the sequence. Frames displays each file independently and disables the Tape Name box from the Metadata menu.

An image sequence consists of image files, such as .psd, named identically except for a numeric suffix.

- myPhotoshopFile_version_001.psd
- myPhotoshopFile_version_002.psd
- myPhotoshopFile_version_003.psd
- myPhotoshopFile_version_004.psd

Layers Settings

Resolution box Select how Flame sets the resolution of .psd clips.

Select:	To:
Background	Import the layers using the same resolution as the background layer.
Native	Import the layers at their original resolution.

PNG Format Settings - Import

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips. Disabled if Sequence Detection from the Clip Options menu is set to Frames.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Timecode is read from the file name, when present. Not editable.

Rate box Select Auto to use the frame rate specified by the clip; with no rate specified, the application assigns a rate from the Default Resolutions table, or the project's rate if there is no match in the table. Use Select Rate to set the frame rate using the Frame Rate box.

The Default Resolutions table uses a frame's height and width to assign it an aspect ratio, a scan mode, and a frame rate. You can find the Default Resolutions table in **MediaHub ➤ Browse for Archives ➤ Default Resolutions**. It is also used when restoring legacy archives.

Frame Rate box Select the frame rate of the imported clip. Enabled if the Rate box is set to Select Rate.

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Only applies to clips running at 29.97 and 59.94 frame rates. Enabled if the Rate box is set to Select Rate.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Sequence Detection box Select how Flame displays image sequences. Sequence builds and displays a clip out of multiple files, using their numerical suffix to rebuild the sequence. Frames displays each file independently and disables the Tape Name box from the Metadata menu.

An image sequence consists of image files, such as .psd, named identically except for a numeric suffix.

- myPhotoshopFile_version_001.psd
- myPhotoshopFile_version_002.psd
- myPhotoshopFile_version_003.psd
- myPhotoshopFile_version_004.psd

QuickTime Format Settings - Import

ProRes from Panasonic VariCam 35 and VariCam HS If footage was recorded as QuickTime ProRes (as opposed to MXF), the .mov files are not displayed in the CONTENT folder of the P2 structure: you have to open CONTENT/AVCLIP/ to display them.

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.

Select:	To:
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Header	Read the tape name from the header of the imported file.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Timecode is read from the file header, when present. Not editable.

Rate box Select Auto to use the frame rate specified by the clip; with no rate specified, the application assigns a rate from the Default Resolutions table, or the project's rate if there is no match in the table. Use Select Rate to set the frame rate using the Frame Rate box.

The Default Resolutions table uses a frame's height and width to assign it an aspect ratio, a scan mode, and a frame rate. You can find the Default Resolutions table in **MediaHub ➤ Browse for Archives ➤ Default Resolutions**. It is also used when restoring legacy archives.

Frame Rate box Select the frame rate of the imported clip. Enabled if the Rate box is set to Select Rate.

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Only applies to clips running at 29.97 and 59.94 frame rates. Enabled if the Rate box is set to Select Rate.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from Header	Read the clip name from the header of the imported file.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Aspect Ratio box Select the aspect ratio assigned to imported clips. Enter Pixel and Enter Aspect affect pixel and frame a pixel and frame aspect ratios respectively. You might need to change the aspect ratio as many applications erroneously write to exported files a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires you to enable, in the Player, **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips using the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Enter the aspect ratio of the imported frames, as a Width:Height ratio. Enabled when the Aspect Ratio box is set to Enter Aspect Ratio or Square Pixels.

YUV Decoding box Select the YUV colour space used by the clip to import. Because Flame works in the RGB colour space, it needs to translate YUV information into RGB. This setting ensures that the right decoder is used for this. The Auto setting selects the colour decoder based on the resolution of the clip: Rec. 601 for clips with a resolution lower than 720 lines, Rec. 709 for everything else. Specifying the wrong colour space results in colours that are off.

Per Recommendation BT. 2020, UHD media should use the Rec. 2020 colour space. But this is rarely the case at the moment: UHD clips use Rec. 709. The Rec. 2020 option is there is case you come upon a correctly encoded UHD clip.

Include YUV Headroom button Enable Include YUV Headroom when importing content that has Full Range video levels. Content with Full Range includes original media from cameras, such as:

- Sony S-Log
- ARRI Log-C
- Canon Log
- Panasonic Varicam Log

Include YUV Headroom should also be enabled when importing from grading systems content that has not been legalized. Disable this setting when importing content with legal video levels, like masters or final grade media.

NOTE This setting has no impact on media that is not YUV.

R3D Format Settings - Import

When importing R3D media files from the MediaHub, you can use the file header information for various settings. This addition makes importing multi-selection of R3D files easier since you can define manual settings for some parameters while preserving camera original settings for non-modified options. You can set it up in the MediaHub, in the R3D format options.

Using file header information to set import options:

- The following boxes from the Colour menu have a From File option:
  - Colour Science
  - Colour Space
  - Gamma Curve
  - HDRx Offset

- The following toggles, located in the Image menu, set the behavior of associated fields:
  - Image Settings From File: Enable to use information from the imported R3D file to set the options displayed in the the Image menu. Disable to use the MediaHub values.
  - Advanced Settings From File: Enable to use information from the imported R3D file to set the options displayed in the the Advanced Colour menu. Disable to use the MediaHub values.
  - The 'Save As User Settings' option is now enabled by default and allows you to use the camera settings as a starting point in the Pre Processing Options.

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Header	Read the tape name from the header of the imported file.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Orientation box Flip (vertical) or flop (horizontal) the media when importing the clip. Camera uses the orientation defined on the camera during the shoot. Disable Flip to use the true orientation of the image, as it was shot.

Timecode box Select which timecode to use as RED camera records both edge code (run record) and time-of-day (free run) timecodes during a shoot. Or Use Primary to use the timecode defined at the time of the shoot.

Debayering Settings

Debayering box Select the resolution of the imported clip. Premium and Good refer to quality of the debayer, especially the anti-aliasing performance. Full is always Premium quality, while Quarter, Eighth and Sixteenth are always Good.

NOTE The debayering setting is the most resource-intensive setting. Try using the level of debayering the most appropriate for your work.

Bit Depth box Set the bit depth of the imported image. This setting is bypassed and implicitly set to 16 bits Floating Point when importing an HDRx clip as High Dynamic Range media, when using a Gamma Curve set to Scene Linear (16-bit fp), or when using the ACES colour space setting.

Detail box Select the level of detail extraction required. This is not a sharpen setting. Only processed when importing clips with Debayering set to Full.

This setting is ignored when the clip is imported through a RED Rocket.

Denoise box Select the level of noise reduction applied. Only processed when importing clips with Debayering set to Full.

This setting is ignored when the clip is imported using a RED Rocket.

Dragon Enhanced Black Enable to remove red noise in low light contents. Only available for DRAGON footage and if Debayer Mode is set to Full or Half Premium.

This setting is ignored when the clip is imported on a workstation with a RED Rocket.

OLPF Compensation box Select the level of Optical Low Pass Filter (OLPF) to compensate for the optical anti-aliasing filter. Only processed when importing clips with Debayering set to Half Premium or Full.

OLPF is a type of sharpening used to compensate for the optical anti-aliasing filter, which can induce softening of the image during recording.

This setting is ignored when the clip is imported through a RED Rocket.

Colour Settings

NOTE When the MediaHub General colour management settings are in Auto Convert mode and the Input Colour Space is set to "From File or Rules", the Format Specific Colour Settings are ignored. The media will be decoded into ACES colour space and then convverted to the selected Working Space. Otherwise, make sure to set the tagging in General colour management settings to correspond to your colour settings below.

Colour Settings box Select how Flame uses the colour information stored within an R3D file.

Select:	To have:
User	Flame import RED clips using the options you set in the Image, Gain, and Curve menus.
Camera	Flame import RED clips using the look created on the RED camera and stored in the RED clip. Disables the Image, Gain, and Curve menu options.
RMD/Camera	Flame import RED clips using the RMD look created in REDCINE-X, or fall back on the camera settings if there is no RMD settings. Disables the Image, Gain, and Curve menu options.
RSX Only	Flame import RED clips using the RSX look created in RED Alert!. The RSX file of a clip must reside in the same directory as the R3D file of that clip. Disables the Image, Gain, and Curve menu options. With this option selected, only clips with an RSX profile can be imported. Clips without an RSX profile appear to be missing media.

Select:	To have:
RSX or RMD/Camera	Flame import RED clips using the RSX look. If a clip has no RSX file, Flame imports it using the RMD look created in REDCINE-X, or fall back on the camera settings if there is no RMD settings. Disables the Image, Gain, and Curve menu options.
RSX or User	Flame imports RED clips using the RSX look. If a clip has no RSX file, Flame imports it using the options you set in the Image, Gain, and Curve menus. Enables the Image, Gain, and Curve menu options.

Save as User Settings button Enable to make the Camera colour settings editable in the timeline FX Format Options. Save as User Settings is implicitly enabled when the Colour Settings box is set to *User*, or *RSX or User*.

Colour Science box Set the version of the RED decoder to use. Using the version 3 of the decoder gives you access to the FLUT and the Shadow options in the Image menu, as well as version 3-only colour spaces and gamma curves.

Disabled when Colour Space is set to ACES.

NOTE As a rule, always set Colour Science to Codec 3.x, unless you are working with a footage shot using a RED camera with firmware 30, and that file was imported in Flame prior to version 2011.

Colour Space box Select the colour space of the imported clips.

ACES requires specific menu settings for R3D media to decode properly. If you select the ACES option, Flame disables the Gamma Curve and the Colour Science boxes, as well as the Advanced Colour and Curve menu options; it sets implicitly the Bit Depth (Debayering menu) to 16-bit fp. It also disables the Brightness, Contrast, DRX and Shadow sliders in the Image menu.

Gamma Curve box Set the value of the output gamma curve that is applied to the imported clips. Selecting Scene Linear (16bit fp) implicitly sets the Bit Depth to 16-bit floating point. If you set HDRx Settings to High Dynamic Range, HDRx clips are imported as Scene Linear, ignoring the Gamma Curve setting. If set to Linear, the ISO, Exposure, Brightness and FLUT controls, in the Image menu, are disabled.

Disabled when Colour Space is set to ACES.

HDRx Settings box Select which track of a RED HDRx file to import, or how to merge the two tracks together. A RED HDRx media file is made of two tracks, track A (the main exposure) and track X (the highlight protection exposure). Use HDRx Settings to set how you want to use those two tracks.

Select:	To have:
Primary Exposure	Only the main exposure (A track).
Highlight Exposure	Only the highlight protection exposure (X track).
Blend Exposures	A single clip resulting from blending together the Primary and Highlight tracks. Blend Exposures behaves similarly to the option of the same name found in REDCINE-X by RED. Use the Blend field to set the blend value to use.

Select:	To have:
Magic Motion	A single clip resulting from the merge of the Primary and Highlight tracks using an algorithm provided and developed by RED. This algorithm tries to match and blend together the motion blurs of the two tracks.
High Dynamic Range	A single clip resulting from the merge of the Primary and Highlight tracks using an HDR merging algorithm developed by Autodesk. It converts the two exposures into a single 16-bit float image. Importing an HDRx clip as High Dynamic Range forces the Bit Depth of the imported clip to 16-bit floating point, and the Gamma Curve to Scene Linear. Use the Blend, Highlight Threshold, and Exposure Offset fields to refine the blend.
RMD	A clip using blending options read from the RMD file. If there is no RMD file, the Flame loads the main exposure (A track).

Blend field Set how to blend the two exposures of a RED HDRx clip.

With HDRx Settings set to Blend Exposures, this field behaves like the blend in REDCINE-X; the blend also attenuates the artifacts created by the scene operation, which is a blend of the two exposures: -1 shows only the Highlight (X frame), 1 only the Primary (A Frame), and 0 a 50-50 mix.

With HDRx Settings set to High Dynamic Range, Blend also attenuates the artifacts created by scene motions; set to 1 unless you are troubleshooting motion artifacts.

Highlight Threshold field Set the threshold when pixels from the Highlight exposure are used instead of the over-exposed pixels from the Primary exposure. Only available when HDRx Settings box is set to High Dynamic Range. Set Highlight Threshold last because import options such as ISO or FLUT lighten or darken the image. Too high and you get clipping (often including a magenta-coloured cast), too low and the midtones and shadows have noise leaking in from the Highlight track.

Exposure Offset field Set how much greater the Primary exposure was when compared to the Highlight exposure, in units of stops. Only available when HDRx Settings box is set to High Dynamic Range. The Exposure Offset should be set to match the setting on the camera for how many stops separate the Primary and Highlight tracks. This is typically 2 or 3 stops.

Offset From File button Enable to use the Exposure Offset read from the R3D file. Enabled when HDRx Settings is set to High Dynamic Range.

Image Settings

Options from the Gamma Curve box and from the Colour Space box might disable some of the fields below.

Image Settings from File Enable to read the Image settings from the r3d file header. Disables all of the Image settings fields.

Advanced Settings from File Enable to read the Advanced Colour settings and the Curve settings from the r3d file header. Disables Advanced Colour settings and Curve settings.

ISO Select the value of the linear gain operation. Disabled if Gamma Curve is set to Linear.

Exposure Set the exposure value, an equivalent to f-stops. Disabled if Gamma Curve is set to Linear.

Brightness Set the brightness value. Disabled if Colour Space is set to ACES, or if Gamma Curve is set to Linear.

Saturation Set the saturation value.

Contrast Set the contrast value. Disabled if Colour Space is set to ACES.

DRX Set the Dynamic Range Extension, which sets how much pixel data is copied from non-saturated channels into saturated channels. Disabled if Colour Space is set to ACES.

Kelvin Set the perceptual colour temperature of the image, in Kelvin.

Tint Set the tint value.

Shadow Set the Shadow level. Disabled if Colour Space is set to ACES.

FLUT Set the FLUT to refine of the ISO level. As FLUT units are in stops, a +1 FLUT value is the same as doubling the ISO. Disabled if Gamma Curve is set to Linear.

Advanced Colour Settings

NOTE Disabled when **Image ➤ Advanced Settings from File** is enabled.

Use the Advanced Colour menu to set the RGB Lift, Gamma, and Gain of RED clips.

Use the Legacy Gain options if you import clips shot with a RED codec that uses a pre-3.0 Colour Science.

NOTE We recommend that you do not change the default settings unless you have prior experience with colour management.

Colour Curves Settings

NOTE Disabled when **Image ➤ Advanced Settings from File** is enabled.

Use this menu to set the RGB and Luma curves for RED clips.

NOTE We recommend that you do not change the default settings unless you have prior experience with colour management.

Sony Format Settings - Import

The following settings apply to all of XDCAM and XDCAM HD files. It is also used for XAVC footage shot with Sony cameras.

Tips when importing Sony XAVC:

- The guiding principle that applies to Panasonic P2 cards also applies to XQD cards: preserve the original XQD-card structure when using these media files. That is, only browse to the XDROOT folder, and import the clips from there. Do not browse within the folder hierarchy to import the essence directly.
- A split clip, that is a clip recorded over two XQD cards, is not seen as a single, continuous, clip, but as multiple clips.

Material at 1080i sometimes uses a thin raster frame format. It uses a frame size of 1440x1080or 1280x1080. When importing, enable Scale to Full HD in **Format Specific Options ➤ Image** to scale the material to the full 1920x1080 frames. Disable Scale to Full HD to import the clip using the thin raster frame format.

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Essence	Use the tape name inferred from the directory structure of the clip.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Timecode is read from the file header, when present. Not editable.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from XML	Use the clip name listed in the XML file accompanying the file.
Clip Name from Essence	Use the clip name inferred from the directory structure of the clip.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Aspect Ratio box Select the aspect ratio assigned to imported clips. Enter Pixel and Enter Aspect affect pixel and frame a pixel and frame aspect ratios respectively. You might need to change the aspect ratio as many applications erroneously write to exported files a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires you to enable, in the Player, **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips using the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Enter the aspect ratio of the imported frames, as a Width:Height ratio. Enabled when the Aspect Ratio box is set to Enter Aspect Ratio or Square Pixels.

YUV Decoding box Select the YUV colour space used by the clip to import. Because Flame works in the RGB colour space, it needs to translate YUV information into RGB. This setting ensures that the right decoder is used for this. The Auto setting selects the colour decoder based on the resolution of the clip: Rec. 601 for clips with a resolution lower than 720 lines, Rec. 709 for everything else. Specifying the wrong colour space results in colours that are off.

Per Recommendation BT. 2020, UHD media should use the Rec. 2020 colour space. But this is rarely the case at the moment: UHD clips use Rec. 709. The Rec. 2020 option is there is case you come upon a correctly encoded UHD clip.

Include YUV Headroom button Enable Include YUV Headroom when importing content that has Full Range video levels. Content with Full Range includes original media from cameras, such as:

- Sony S-Log
- ARRI Log-C
- Canon Log
- Panasonic Varicam Log

Include YUV Headroom should also be enabled when importing from grading systems content that has not been legalized. Disable this setting when importing content with legal video levels, like masters or final grade media.

NOTE This setting has no impact on media that is not YUV.

Scale to Full HD button Enable to have media with a 1280x1080 or 1440x1080 resolution display at a standard 1920x1080 resolution. Disable to display the media at its native (1280x1080 or 1440x1080) resolution.

Browsing Settings

Essence Mode button Enable to browse the directory structure of a Sony clip (XDCAM, XDCAM HD, and XAVC). This allows you to import specific video and audio media contained within that clip.

SonyRAW Format Settings - Import

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.

Select:	To:
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Header	Read the tape name from the header of the imported file.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Orientation box Flip (vertical) or flop (horizontal) the media when importing the clip. Camera uses the orientation defined on the camera during the shoot. Disable Flip to use the true orientation of the image, as it was shot.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Resolution Settings

Debayering box Select the resolution at which the media is imported. Higher resolutions require more processing and impact real-time playback. Other resolutions offer better performance at the cost of lower image resolution.

You cannot use a debayering setting higher than native resolution of the camera. Using a higher setting prevents the media from being displayed, with an error message in the Preview panel (*Cannot initiliaze codec*). Try lowering the debayering setting until the media debayers correctly.

QFHD (Quad-Full High Definition) is Sony's term for 3480x2160 UHD resolution, often times referred to as 4K HDTV, or 4K UHD. UHDTV (Ultra High Definition Television) is the 7680x4320 resolution, or 8K UHD.

Debayering options have been renamed to reflect what is used in other applications. The table below highlights the changes.

Debayering option	Resolution
0.25K	256x135
0.5K	512x270
1K	1024x540
HD (1920x1080)	1920x1080
2K	2048x1080
QFHD (3840)	3840x2160
4K	4096x2160
6K	6144x3240
UHDTV (7680)	7680x4320
8K	8192x4320

Quality box Select one of two qualities for the decoding of SonyRAW footage. While debayering resolution result in lower resolution clips, Quality affects the quality of the fully debayered pixels. Depending on your system configuration, Standard provides better decoding performance at the cost of lower image quality compared to High quality. The actual image degradation depends on the footage being decoded, but because of the Bayer pattern, expect to see differences in the red and blue channels.

The Quality box is only available to the following Debayering settings: HD, 2K, and 4K.

Colour Settings

NOTE When the MediaHub General colour management settings are in Auto Convert mode and the Input Colour Space is set to "From File or Rules", the Format Specific Colour Settings are ignored. The media will be decoded into ACES colour space and then convverted to the selected Working Space. Otherwise, make sure to set the tagging in General colour management settings to correspond to your colour settings below.

Colour Encoding box Select the colour space of SonyRAW media. Select Raw to disable this transformation and work with the raw media. All other settings implicitly convert the 16-bit RAW media to 16-bit half-float. If you select the Raw option, you can always apply a colour transform later from the General tab, using one of the SonyRAW colour transforms available in*camera/Sony*.

The list displays the newer options on top (ACES), legacy ones at the bottom (starting with ACES(tungsten)). These legacy options are included to provide compatibility with previously imported media. See Choosing a Working Color Space (page 1794) and Applying Colour Management to Clips (page 1760).

NOTE Some of the colour space options—*Scene-linear / Rec709 - legacy, ACES (daylight), ACES (tungsten)*—were updated in Flame 2013 20th Anniversary Edition Extension 2 SP3 and Smoke 2013 Extension 1, to solve an issue with black levels. This impacts SonyRAW media imported before the update: the black levels will not match SonyRAW media imported after the update. You can fix these old clips by viewing them on the Timeline, and in the Format Options Editor, select **Basic ➤ Colour**, and then re-apply the required colour space.

SonySStP Format Settings - Import

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Header	Read the tape name from the header of the imported file.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

XDCAM EX Format Settings - Import

About XDCAM EX

Support for Sony XDCAM EX media includes support for clips recorded as multiple segments on single SxS card (split clips) or multiple cards (span clips). Flame imports a serie of split & span segments as a single clip.

NOTE Clips spanning two card only become available in Flame when browsing the card that contains the last segment.

To ensure that Flame is able to see the clips organisation, copy the full folder structure from the SxS cards to a single folder on your local storage.

For example, with two SxS cards (named Card1 and Card2), you would recreate the following structure:

- storage
  - folder
    - Card1
    - Card2

IMPORTANT Do not modify the file structure inside the cards folders. For example, copying only the media files (.mp4) instead of the whole card structure will prevent Flame from recreating the correct clip structure.

AAF Conform

You can conform AAF files referencing XDCAM EX content in Flame. Make sure to import the XDCAM EX clips in Avid products through AMA (the Avid Media Architecture). If you prefer to work with transcoded media in Avid products, but still want to conform with the original media fils, enable in Flame the Use Original Media option when importing the AAF file. That way the conform uses the original media, instead of the transcoded MXF generated by the Avid products.

Metadata Settings

Tape Name box

Select an option to determine how the tape name is set when importing clips.

Select:	To:
Enter Tape Name	Activate the Tape Name field where you enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from File Name	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. Use the Level field below to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Essence	Use the tape name inferred from the directory structure of the clip.

Level field Set from which directory the tape name is taken, relative to the location of the clip in the directory structure.

Although available for all types of files, the Level field is intended for directory structures output by film scanners. A typical image file directory structure looks like this: *./<tape>/<resolution>/clip.######*. In this case, selecting Levels Up 2 in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Enter the name to use as tape name when importing the clip. Enabled if Tape Name is set to Enter Tape Name.

Timecode box Timecode is read from the file header, when present. Not editable.

Rate box Select Auto to use the frame rate specified by the clip; with no rate specified, the application assigns a rate from the Default Resolutions table, or the project's rate if there is no match in the table. Use Select Rate to set the frame rate using the Frame Rate box.

The Default Resolutions table uses a frame's height and width to assign it an aspect ratio, a scan mode, and a frame rate. You can find the Default Resolutions table in **MediaHub ➤ Browse for Archives ➤ Default Resolutions**. It is also used when restoring legacy archives.

Frame Rate box Select the frame rate of the imported clip. Enabled if the Rate box is set to Select Rate.

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Only applies to clips running at 29.97 and 59.94 frame rates. Enabled if the Rate box is set to Select Rate.

Clip Settings

Clip Name box

Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from Essence	Use the clip name inferred from the directory structure of the clip.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Aspect Ratio box Select the aspect ratio assigned to imported clips. Enter Pixel and Enter Aspect affect pixel and frame a pixel and frame aspect ratios respectively. You might need to change the aspect ratio as many applications erroneously write to exported files a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires you to enable, in the Player, **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips using the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Enter the aspect ratio of the imported frames, as a Width:Height ratio. Enabled when the Aspect Ratio box is set to Enter Aspect Ratio or Square Pixels.

YUV Decoding box Select the YUV colour space used by the clip to import. Because Flame works in the RGB colour space, it needs to translate YUV information into RGB. This setting ensures that the right decoder is used for this. The Auto setting selects the colour decoder based on the resolution of the clip: Rec. 601 for clips with a resolution lower than 720 lines, Rec. 709 for everything else. Specifying the wrong colour space results in colours that are off.

Per Recommendation BT. 2020, UHD media should use the Rec. 2020 colour space. But this is rarely the case at the moment: UHD clips use Rec. 709. The Rec. 2020 option is there is case you come upon a correctly encoded UHD clip.

Include YUV Headroom button Enable Include YUV Headroom when importing content that has Full Range video levels. Content with Full Range includes original media from cameras, such as:

- Sony S-Log
- ARRI Log-C
- Canon Log
- Panasonic Varicam Log

Include YUV Headroom should also be enabled when importing from grading systems content that has not been legalized. Disable this setting when importing content with legal video levels, like masters or final grade media.

NOTE This setting has no impact on media that is not YUV.

Browsing Settings

Essence Mode button Enable to browse the directory structure of XDCAM EX clips. This allows you to import specific video and audio media contained within an XDCAM EX clip.

About Pixel Aspect Ratio

Some video and film formats use rectangular instead of square pixels. This explains why NTSC and PAL video formats have a different frame dimension in terms of pixels, but are ultimately broadcast to the same 4:3 aspect ratio screen.

- NTSC pixels are narrower than they are high, which allows for the 720x486 (1.481) aspect ratio frame to fit into a 4:3 (1.333) aspect ratio broadcast screen.
- PAL pixels are wider than they are high, which allows for 720x576 (1.25) aspect ratio frame to fit into a 4:3 (1.333) aspect ratio broadcast screen.

When you capture video or import film frames that use rectangular pixels into Flame, the pixels become square because computer graphics work only with square pixels. This is why the NTSC and PAL frames appear as though they are different size—displayed with square pixels, they are.

You can set the Player to display clips originating from rectangular pixel formats using **View Settings ➤ Use Ratio** to simulate the use of rectangular pixels. This provides a display that corresponds with the delivery format.

Using the aspect ratio display affects system performance, so if you are using the aspect ratio filter in the Player and notice a performance slowdown, you can disable the option and return to square-pixel display.

NOTE Many HD video formats such as 1920x1080 and 1280x720 use square pixels, so enabling this option has no effect. One HD video format that does use “slightly” rectangular pixels is 1920x1035.

The following formats have an aspect ratio setting that allow you manage a clip's ratio, available in **Format Specific Options ➤ Image**.

About Video Field Management

About Interlaced Video

Interlaced video formats capture each frame of a moving picture as two separate images known as fields. One field consists of the first active and subsequent odd horizontal lines in the frame, while the other field consists of the second and subsequent even lines. During the recording process, the two fields are captured consecutively at slightly different moments. The field recorded first is called the dominant field. For example, when recording NTSC using Field 1 as the dominant field, Field 2 is recorded a 1/60th of a second after Field 1. When the interlaced video is played, the two fields of each frame are then displayed with a temporal offset, causing the human eye to perceive a smooth, continuous moving image.

Because of this time difference however, interlaced video can sometimes display jagged horizontal edges, particularly when showing subjects in fast motion. In addition, not all such video formats interlace frames in the same way. When different interlaced formats are mixed, the field dominance of the clips must be changed to avoid visual artifacts and jitter. In Flame, use the video field management tools to do so.

By default when editing or compositing clips that have conflicting scan formats, Flame displays warning messages and requires confirmation of potentially problematic operations. These warnings can be disabled if desired by enabling Ignore Scan Format in the Clip Select section of the General Preferences (page 1936).

About Field Dominance

Most interlaced video formats are Field 1 dominant, with a few exceptions. This means that for a given frame, Field 1 is recorded earlier in time, and therefore should be displayed first during broadcast. Field 2 is recorded after Field 1 and is displayed second.

Interlaced video formats may also differ in the first active line. The first active line is the uppermost line in the frame. In some interlaced video formats, this uppermost line belongs to Field 1, while in others it belongs to Field 2.

Format:	First active line belongs to:	Displayed in Flame as:	Displayed in Avid as:	Displayed in FCP as:	Other applications:
NTSC	Field 2	■ F1 if matching project's field dominance ■ F2 if not matching project's field dominance	Even	Lower	f2 bottom
PAL	Field 1		Odd	Upper	f1 top
HD (and other interlaced)					

All clips in Flame have a metadata tag showing their scan mode: F1 indicates that the clip conforms to the same dominant field requirement in the current project settings. F2 *shows a mismatch*, while P meanwhile indicates a progressive scanned, non-interlaced clip. This tag is assigned when the material is first imported. If the tag scan mode appears incorrectly for any reason, it can be changed to match the actual scan mode of the clip. This is different from other softwares that expose the actual spatiality of a clip's fields.

IMPORTANT Do not confuse Field 1 or Field 2 with the F1 and F2 scan mode tags. The actual dominant field of a clip showing F1 or F2 in its scan mode metadata may be either the upper or lower depending on the clip's video format.

In projects containing mixed-resolution clips, interlaced video formats may still be combined despite having different dominant fields. The differences between the two formats must be addressed however to avoid outputting a result that displays one of the interlaced video formats improperly and introduces undesirable visuals. Use the procedure below to correct such interlacing issues.

NOTE The effects are only viewable on an interlaced broadcast monitor.

Reversing Field Dominance

When working in a project that uses clips of various interlaced video formats, the field dominance of clips that do not correspond with the project settings should be reversed. For example, if a Field 1 dominant PAL clip is to be included in an NTSC project that is Field 2 dominant, simply reverse the field dominance of the PAL clip.

Mismatched clips show F2 in their metadata.

When reversing the dominance of a clip, the first and last fields of the clip are dropped to account for the offsetting required to reverse the field dominance. The resulting clip becomes a frame shorter. It is therefore a good practice to always capture enough head and tail frames so that the fields dropped on both ends do not adversely affect the visible frames.

Both the media and the scan mode metadata tag are changed by this process.

To reverse the dominance of clips:

1. Select Tools, and select the Clip tab.
2. Click Reverse Dominance.
3. Select the clip to process.
4. Select the destination.

   The field dominance of the clip is reversed.

Deinterlacing and Interlacing

Deinterlacing and interlacing operations are used primarily to aid in the rotoscoping of field-based material, or in the application of filters to field-based material. Deinterlacing and interlacing are not necessary for processes that support field-based rendering.

Because there is a line-based difference in time per frame of interlaced video, artifacts may be introduced when field-based frames are painted or filtered. For example, if a blur is applied over fields, colour information from the pixels on Field 1 will be smudged into the pixels on Field 2.

Deinterlacing clips prior to applying any paint or filtering operations always provides better results. Once the effects have been added, the clip can be interlaced again, if required.

To deinterlace and interlace clips, see the sections below. Clips can also be deinterlaced and interlaced using Batch and Batch FX. For more information, see Deinterlace (page 1530) and Interlace (page 1576) in the Effects and Tools Reference (page 1433) chapter.

Deinterlacing Clips

Deinterlace clips to separate each field to its own frame. The result clip is twice as long as the original.

To deinterlace clips:

1. Select Tools, and select the Clip tab.
2. Click DeInterlace.
3. If necessary, select Field 1 or Field 2 from the Field Dominance box. By default, Field Dominance is set to Auto, which automatically determines whether the clip is Field 1 or 2 dominant. Manually set a field if Auto does not work.

4 Enable or disable Interpolation based on the desired result:
   - If clips are deinterlaced with interpolation off, the process repeats each isolated line to fill in the scan lines that would otherwise be vacant in the resulting clip's frames.
   - If clips are deinterlaced with interpolation on, the process interpolates a blend between adjacent lines to fill in the isolated scan lines. This option reduces interlacing artifacts.
5 Select the clip to deinterlace.
6 Select the destination.
   The result clip appears, now with double the duration.

Interlacing Clips

Interlace clips to combine de-interlaced fields into interlaced frames. The resulting clip is half as long as the original.

To interlace clips:

1 Select Tools, and select the Clip tab.
2 Click Interlace.
3 From the Interlace Parameter box, select Field 1 or Field 2 to establish the resulting clip's field dominance.
4 Select the clip to interlace.
5 Select the destination.
   The resulting clip appears, now with half the duration.

Merging Fields

Merging fields blends pixels in neighbouring field lines to reduce artifacts caused by motion jitter. Merge fields when there is interlaced material to be output to a progressive scan format.

To merge fields:

1 Select Tools, and select the Clip tab.
2 Click Field Merge.
3 Set the percentage of blending between fields in the Level field.

Value	Result
100%	The average pixel value is replicated to both fields in an F1/F2 merge-pair, resulting in total field merging.
50%	Pixels on each line in an F1/F2 merge-pair are replaced by pixels defined by 50% of their original value and 50% of the blend, or field merge result.
0%	No change is applied to the clip.

4 Select the clip to process.
5 Select the destination.
   The field dominance of the clip is reversed.

Changing the Scan Mode of a Project

The project scan mode dictates how interlaced video formats are captured from tape, sent to the broadcast monitor in-session, and output to tape. The project scan mode also determines field-based processing.

You can change the scan mode of your project by selecting an alternative scan mode option. Do this, for example, to view a PAL clip in an NTSC project properly before reversing its field dominance. Generally, changing the scan mode of a project is a temporary measure performed only to allow for the monitoring of clips whose format does not match the scan mode of the current project.

See To edit a project (page 48).

Changing the Scan Mode Metadata Tag of a Clip

When material is brought into Flame, it is assigned a particular scan mode that becomes part of the clip metadata.

You can correct this problem using the Dominance option part of the Reformat tool, accessible from the right-click menu. This option changes the scan mode metadata tag for a clip without changing the actual scan mode of the media.

Working with Others

Flame offers you two methods to work in collaboration with another Flame.

The first one is the Projects browser in the MediaHub. Use it to transfer clips between your project and another one. This other project can be a local, such as a previous project you worked on, or a remote project, such as a a sequence being put together in Flame Assist. Or use the Projects browser to transfer clips from a project created with an older version of the software.

To work on the same shots, clips, sequences, use shared libraries. This other second method integrates directly in the Media panel, and are best used when you are working with other users on the same project.Flare Workflow (page 1951) has more about working with Workspaces, and even if it targets mostly a Flame-Flare workflow, it applies to any setup where shared libraries are used.

Import Settings and Lustre

When importing media files and using the Cache Source Media, Flame bakes in import options like resize, LUT, and format specific options like R3D settings. Of course, you always have access to the import settings, and edit it if need be. Open the Pre-Processing editor from the Timeline FX ribbon.

But Lustre does not have access to the import history for clips imported in Flame with an enabled Cache Source Media option. This means that if you import in Lustre a Flame-created sequence that uses such clips, you do not have access to their import history. Flushing the media cache prior to importing the clip in Lustre restores access to import history.

To restore access to the import history for Lustre:

1. From Flame, right-click the material to grade in Lustre.
2. Select **Media ➤ Flush Source Media Cache**.
   You can now edit the import history of the clip from Lustre.

Important Compatibility Information

The Flame 2017 family of products (Flame Premium, Flare, Flame Assist) can now read contents created in Smoke Desktop Subscription (DTS), with the following limitations:

- Projects from Smoke DTS can be opened (and converted) by the Flame Family 2017 products.

- Projects can be seen and imported through MediaHub, using Browse for Projects.
- Archives created in Smoke DTS can be read in Flame Family 2017 products.

IMPORTANT Contents created in Flame Family 2017 products cannot be read or opened in Smoke DTS.

		Will import in...				
		Flame 2017	Flare 2017	Flame Assist 2017	Smoke DTS 2017	Training Edition 2017
Projects Created In...	Flame 2017	Yes	Yes	Yes	No	No
	Flare 2017	Yes	Yes	Yes	No	No
	Flame Assist 2017	Yes	Yes	Yes	No	No
	Smoke DTS 2017	Yes	Yes	Yes	Yes	No
	Training 2017	No	No	No	No	Yes
	Flame 2016	Yes	Yes	Yes	No	No
	Smoke DTS 2016	Yes	Yes	Yes	Yes	No
	Training 2016	No	No	No	No	Yes
	Smoke DTS 2015	Yes	Yes	Yes	No	No

Sharing Clips Between Users

Clips placed in a special area of the Media panel, called Shared Libraries, becomes accessible to remote workstations that connect to the project. Shared libraries enable different Creative Finishing workstations to quickly and conveniently share media.

NOTE Shared libraries are part of the project's structure: clips stored within are included when you archive a project.

Placing media in a shared library automatically grants read access to any workstation connected to the project. The remote operator can then access the media and start working immediately after connecting to the project. Once finished, the remote operator needs to save the resulting media back to the shared library. To do this he must acquire exclusive access.

Write access locks the folder so that only the user having acquired write access can modify the folder contents (i.e. write to the folder). Read access is still available to other remote workstations connected to the project. Once the media is saved, the user can disable write access, enabling other remote workstations to obtain exclusive access and modify the contents.

To acquire exclusive access:

1. Select the shared library you want to acquire exclusive access to in the Media panel.
2. From the contextual menu, select Acquire Exclusive Access.

NOTE By default, after 600 seconds of inactivity, your exclusive access will be released. You can customize this setting in the user preferences (in the General tab).

To release exclusive access:

1. Select the shared library you want to acquire exclusive access to in the Media panel.
2. From the contextual menu, select Release Exclusive Access.

You can lock the top level of the Shared Libraries to prevent anyone but you from creating or deleting any shared library.

The Shared Libraries also display the Lock icon in the following circumstances:

- When moving a Shared Library
- When deleting a Shared Library

This locks prevents other users on the network from editing the hierarchy of the Shared Libraries. The user that moved or deleted the shared library can remove the lock by saving the project (Ctrl+S). Users can still modify the contents of each Shared Library, just not rearrange the order in which they appear in the Shared Libraries area of the Media Panel.

Important Compatibility Information

The Flame 2017 family of products (Flame Premium, Flare, Flame Assist) can now read contents created in Smoke Desktop Subscription (DTS), with the following limitations:

- Projects from Smoke DTS can be opened (and converted) by the Flame Family 2017 products.
- Projects can be seen and imported through MediaHub, using Browse for Projects.
- Archives created in Smoke DTS can be read in Flame Family 2017 products.

IMPORTANT Contents created in Flame Family 2017 products cannot be read or opened in Smoke DTS.

		Will import in...				
		Flame 2017	Flare 2017	Flame Assist 2017	Smoke DTS 2017	Training Edition 2017
Projects Created In...	Flame 2017	Yes	Yes	Yes	No	No
	Flare 2017	Yes	Yes	Yes	No	No
	Flame Assist 2017	Yes	Yes	Yes	No	No
	Smoke DTS 2017	Yes	Yes	Yes	Yes	No
	Training 2017	No	No	No	No	Yes
	Flame 2016	Yes	Yes	Yes	No	No

Smoke DTS 2016	Yes	Yes	Yes	Yes	No
Training 2016	No	No	No	No	Yes
Smoke DTS 2015	Yes	Yes	Yes	No	No

Open / Close

You can open and close shared libraries. Closing shared libraries turns off access to the media within the shared libraries as well as freeing up your system memory.

To open / close shared libraries:

1. Select the top level shared library you want to open or close in the Media panel.
2. From the contextual menu, select Open Library or Close Library.

Transferring Material Between Projects

Elements you can transfer when browsing for Projects:

- Clips
- Folders
- Libraries
- Batch groups
- Reel groups
- Desktops

To transfer material between your project and one located on another workstation:

1. Select **MediaHub ➤ Browse for Projects**.
2. In the Autodesk Network list, double-click the workstation where the project is located.
   This displays the list of projects hosted by that workstation.
3. Double-click the project to open it.
   This opens the project and displays its Workspaces and shared libraries.
4. Navigate the project to find what you want to transfer.
5. Drag-and-drop from the Projects browser to the Media panel. Or from the Media panel to the Projects browser: you can import or export between projects.

To transfer material between local projects:

1. Select **MediaHub ➤ Browse for Projects**.
2. In the Local Projects list, double-click the project to open it.
   This opens the project and displays its Workspaces.
3. Navigate the project to find what you want to transfer.
4. Drag-and-drop from the Projects browser to the Media panel. Or from the Media panel to the Projects browser: you can import or export between projects.

NOTE Proxies are only transferred when both projects have identical proxy settings, as defined in the Projects settings window.

Important Compatibility Information

The Flame 2017 family of products (Flame Premium, Flare, Flame Assist) can now read contents created in Smoke Desktop Subscription (DTS), with the following limitations:

- Projects from Smoke DTS can be opened (and converted) by the Flame Family 2017 products.
- Projects can be seen and imported through MediaHub, using Browse for Projects.
- Archives created in Smoke DTS can be read in Flame Family 2017 products.

IMPORTANT Contents created in Flame Family 2017 products cannot be read or opened in Smoke DTS.

		Will import in...				
		Flame 2017	Flare 2017	Flame Assist 2017	Smoke DTS 2017	Training Edition 2017
Projects Created In...	Flame 2017	Yes	Yes	Yes	No	No
	Flare 2017	Yes	Yes	Yes	No	No
	Flame Assist 2017	Yes	Yes	Yes	No	No
	Smoke DTS 2017	Yes	Yes	Yes	Yes	No
	Training 2017	No	No	No	No	Yes
	Flame 2016	Yes	Yes	Yes	No	No
	Smoke DTS 2016	Yes	Yes	Yes	Yes	No
	Training 2016	No	No	No	No	Yes
	Smoke DTS 2015	Yes	Yes	Yes	No	No

Troubleshooting

Read Only (In use by...)

Appears in one of two cases:

- When the workspace of the project you are browsing is opened and used by the user listed in the Read Only message.
- When someone else is browsing that workspace with the button Exclusive Write Access enabled.

You can still copy items from a Read Only workspace. If you absolutely need to write to the workspace, you will need to locate the user and ask him to switch to another project.

Owned by...

The workspace you are trying to write to is owned by another Flame, Smoke, or Flare. Enable Administrator mode to write to the workspace.

Unmanaged media contained in selection. Cache media on Wire?

When transferring media from a remote project, you can get the following message: "Unmanaged media contained in selection. Cache media on Wire?"

This means that the clip you are importing from the remote project has no media cached. Answer Yes to cache the media during the transfer of the clip. Answer No to only keep a reference to the media.

In fact, answering Yes is identical to importing media with the Cache Source Media enabled, and answering No is identical to importing media without Cache Source Media.

No route to host

The workstation you are trying to connect to is not available because it is shutdown, or no longer connected to the network.

Publishing Shots

Publishing a Shot

Before distributing shots to others, verify that the structure of the sequence, name the shots, and then distribute them.

To create a shot:

1. Verify that the shot has a "layered cake" structure.

   This structure is the best layout when publishing a shot: Flame uses a bottom-most left-most segment part of the shot to create the shot's timing structure.

2. Identify segments that are part of the shot by assigning them an identical shot name. For Flame, a shot is a group of segments that have the same Shot Name.

   You can edit the shot name from the Conform view, or with the Rename Shot option from either the Timeline gear menu or contextual menu.

3. Repeat for each shot on your timeline. Make sure to assign each shot a unique name.

   TIP The Conform tab provides your with a great overview of your timeline, including assigned shot names.

Possible strategy for naming shots:

1. Select the bottom video track.
2. Select Rename Shot from the Timeline gear menu.
3. From the Add Token, select Segment Index. This assigns a unique shot name to each segment.

4 Now go from segment to segment, assigning each segment the segment id of the bottom-most segment. You can do this in Conform, to edit Shot names in bulk.

When you are ready to distribute the shots, publish them.

To publish the shots to be used by other applications:

1 Export the sequence.

2 In the Media Export window, navigate to the root folder.

The root folder is the published material destination where the published shot structure is created. When you publish shots, you do not only export the media, you also export additional files that provide information crucial to the shot publish workflow, such as Open Clip XML files and Batch setups for each shot.

3 Set Export to Sequence Publish.

4 Set Format Preset to one of the Shot Publish existing presets.

For this procedure, we will use *Shot Publish - Flame Root Directory (16-bit OpenEXR and No Audio)*.

5 Click Export.

Once the export is completed, you can see, next your original sequence, the published timeline named as your sequence but appended with *_publish*.

When you open a shot published sequence, the versioned shot appears on a different Version layer of the sequence. The bottom Version contains the original sequence.

The versioned shot actually contains all of the media of the shot and effects, keeping the timing from the original sequence.

Create New Version

- Provides access to the open setup in a fashion similar to History. Like History, exiting automatically triggers a new render, however in this case it will create all of the required media and metadata in the appropriate paths in the job tree, and append a version to the timeline segment.

Open as Batch Group

- Load the currently selected version in Batch. This will include the Write File node needed to append versions to the shot, the job tree, and the segments in the timelines where it is used.

Promote to Batch FX

- Ingests the shot directly in the current timeline. This removes the connection to the job tree, as well as its versions.

Opening the Shot in Flare

1. In Flare's Batch view, click Load.
2. Navigate to the folder where the shot's Batch setups are stored.

   Batch setups for shots are placed in the path defined by **Clip Options tab ➤ Setup Pattern field** in the Media Export window. In the Format Preset *Shot Publish - Flame Root Directory (16-bit OpenEXR and No Audio)*, Batch setups are located in *flame/batch/*.
3. Double-click the Batch setup to open it.
4. Edit the setup.
5. When done, click Render.

   New media and version information are written to the paths set in the Write File node. The Open Clip XML is updated to include this new version information. And a new version of the Batch setup is automatically saved in the appropriate paths in the job tree.

Viewing a version in Flame:

1. Right-click the published sequence and select **Source Versions ➤ Update Source**. *_publish* identifies a published sequence.

   This reloads the associated Open Clip, making sure the latest versions are available for viewing. An updated segment is highlighted in white to indicate that is contains new versions.
2. Open the published sequence.

   The original sequence is displayed on the bottom Version of the timeline. A new Version contains the track with the published shots and referenced media.
3. On the Timeline, locate the shot segment on one of the Published shots tracks.
4. On the Timeline FX ribbon, select a version from the Clip Versions box. Click Pre-Processing if at first you don't see it.

   You can now playback the new version, open it in Batch, promote it to BFX, output it, or simply revert back to a previous version.

Regarding Source Versions options:

- Update Source: Reloads the Open Clip, making sure all the versions are available for viewing.
- Select Current Version: Reloads the Open Clip, and then displays the version defined as current in the Open Clip by the `currentVersion` property of the `<versions>` element. Pipeline tools such as ShotGun can change the Current Version based on approval policies.
- Select Latest Version: Reloads the Open Clip, and displays the latest version.
- Mark All Updates as Viewed: Clears from all the selected clips the white halo that highlights segments that were updated and which help you track down updated segments.
- Version Name: If you select a clip, you can select a version directly from within the Source Versions menu.

NOTE You can perform these operations on a segment, a sequence, multi-selected sequences, and even a reel, Reels Group, a library, multiple libraries to updated all sequences within.

Naming With Tokens

You can make it so a segment's name, shot identifier, or comment uses dynamic tokens.

To rename a segment so it uses tokens:

1. Right-click the segment in the timeline.

2 Select Rename.

3 Enter the new name for the segment. Use tokens to use existing metadata.

TIP Enable the Dynamic button to have a name that automatically updates its tokens.

To add the clip's height and width to the segment's name:

1 Right-click the segment in the timeline.

2 Select Rename.

3 Click the Pattern field, and then Esc to empty it.

4 From the Add Token box, select Clip Name.

Pattern now displays: <*name*>.

5 From the Add Token box, select Width.

Pattern field now displays: <*name*><*width*>.

6 From the Add Token box, select Height.

Pattern field now displays: <*name*><*width*><*height*>.

7 Add characters to clear up the name: <*name*>_<*width*>x<*height*>.

8 Click Rename. The segment's name now contains its name and dimensions.

You can edit multiple segments in a single operation by first selecting them and then selecting, from the contextual menu, Rename, Comment, or Rename Shot. And what information you type in the dialog box overwrites whatever was already present in those segments.

To add information to multiple segments' shot name, comment, or segment name without overwriting what is already there:

1 Multi-select the segments.

2 From the contextual menu, select Rename, Comment, or Rename Shot, depending what you need to update.

If you've never edited this information, both the Preview and Pattern are empty. If this is not the first time, they will contain some information.

3 Edit the Pattern field. Add the required text and tokens, but make sure that the relevant token is included. That token is a placeholder for the original information and makes sure. The tokens to include are:

- For Rename: <*segment name*>
- For Comment: <*comment*>
- For Rename Shot: <*shot name*>

4 Click the required button to close the window. Now each segment contains the old information (thanks to the placeholder) and the new information that was added.

Example:

1 Two segments need to be renamed to display their width and height: *Segment_1* and *Segment_2*.

2 Select both segments, and from the contextual menu, select Rename.

In the Rename dialog box, because this is the first time these segments are renamed, both Preview and Pattern fields are empty.

3 In the Pattern field, add the segments' name placeholder. The Pattern field is now:<*segment name*>

4 Add the width token, with a colon and space. The Pattern field is now:<*segment name*>: <*width*>

5. Add the height token, with an x and a space. The Pattern field is now:*<segment name>: <width> x <height>*
6. Click Rename.
7. The two segments are now named:
   - *Segment_1: 1920 x 1080*
   - *Segment_2: 1920 x 1080*

Regarding Batch Setups for Shots

Batch setups created with the Shot Publish workflow possess some interesting features that help minimize record keeping.

- The Write File node is already configured to use the correct render paths and the correct Format and Settings options.
- The version number automatically increases every time you render.
- The Batch timing view provides you with the required layer information.

Shot Publish Presets

Note that there is a limited amount of presets for Shot Publish. These presets are provided mainly as examples of naming patterns, examples you can customize to your specific needs. But as you customize those settings, make sure you enable:

- Copy Exported Clip in Media Library: This option creates in your Flame the special _publish sequence with the Version that contains all the individual shots.
- Create Clip Data: This option *is* the Shot Publish workflow by creating the Open Clip XML files, essential to this feature.
- Clip Versions: This option is required if you want to track the different versions for each shot.
- Create Shot Setup: This option exports, together with the clip, the batch setups required for the FX. Especially useful when working within a Flame-Flare setup.

But you can always edit the different patterns naming patterns to fit your facilities requirements.

Integration with Third Parties

You can integrate Flame in third-party asset management workflows, but it does require some additional work.

This additional work consists of configuring Nuke (or any third-party application) to update the Open Clip XML file. The Open Clip XML file stores version, media, and shot structure information. To leverage that information, the third-party app must be able to decode (and encode) Open Clip XML. You can find information about its format *here*.

You can integrate this info to a script in the Write node in Nuke, or to a post-render script in Maya. Rendering in such an application automatically runs the script to wrap the footage in the Open Clip XML format and automates this process. If any new renders are output from these applications, they automatically update in the Open Clip XML file.

If you additional monitoring or automation needs, you can use available Python hooks to integrate Flame into a shot management system.

And from the Flame, the process is identical to working with a Flare.

Using Subtitles Files

With Flame, you can load a subtitle file to quickly add subtitles to a sequence. This is the most effective method as it allows a third party to create subtitles without requiring access to a Flame.

The format of this file is described in Subtitle XML Elements (page 248), with a sample provided in Subtitle XML Sample (page 252).

Other methods for subtitling include creating Text Timeline FX on the sequence's segments, or adding a new layer to create the subtitles in Gap effects.

Importing Subtitles to a Timeline Sequence

The procedure below adds subtitles from an imported xml file to an opened sequence.

1. From the MediaHub tab, import the subtitle file to the Media Library. There are no Specific Import Options to set.

   The subtitle XML file is imported as a single-track clip, where the subtitles are editable Text Timeline FX applied to Gap effects.

   NOTE The subtitle clip is renamed to the name defined by the `<name>` element in the file. It also takes the timing and resolution defined for the project.

2. From the Timeline tab, `right-click` the sequence that requires subtitles and select Open or Open as a Sequence.
3. Add a track to the opened sequence.
4. Drag and drop the subtitle clip to the new top track.

   Each text element appears as a editable Text Timeline FX, at the timecode and length specified in the xml file.

TIP You can add multiple subtitle clips to a sequence, then mute all but the track you want to process. With this method you can quickly version a clip with different captions.

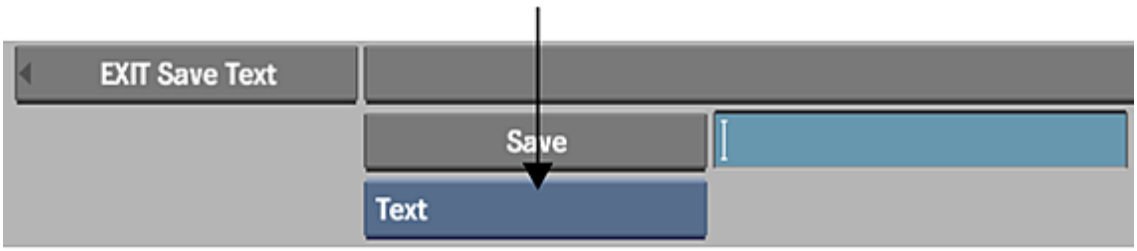
Defining Subtitles Looks Using Text Setup Files

A text setup file can be referenced from the subtitle XML file. The text setup file overrides the font style and position parameters indicated in the current `<title>` element.

Using a text setup file allows you to apply your preferred text setup to the subtitles in the file, or even have a different text setup for each subtitle.

To create the text setup file:

1. In a Text tool, create a text setup.
2. Click Save.
3. In the Save menu, select Selected Layers from the Save box.



4 Choose the directory into which you want to save the file.

5 Type a name for the file and press `Enter` or click the Save button.

The file is saved and you return to the Text tool.

To associate the text setup file to each subtitle element:

1 Open the subtitle XML file in a text editor.

TIP Consider using a text editor with syntax coloring for XML elements. It makes for easier navigation of files.

2 Within each `<title>` element, indicate the path to the text setup file in the `<setup>` element.

The parameters in the text setup file override the following text parameters within the `<title>` element of the subtitle: font, size, font colour, vertical, and horizontal.

Once you import the subtitle file in Flame, you can see the impact of the text setups on the imported subtitles. Changes to the text setup do not propagate to an imported subtitle file: if you modify the saved text setup after importing a subtitle file, you need to re-import that same file to update the subtitles referencing that setup.

Search for subtitles that are not properly linked to a setup file:

1 Place the positioner on the subtitle track.

2 Click the magnifying glass icon at the bottom right side of the timeline.

The Find and Select in Timeline window appears.

3 Enable

- Gaps
- Every Criteria Below
- Comments. Also enter in the Comments field *Unable* (make sure to enter the asterisk wildcards).

4 Click Select on Current Track.

The Find and Select in Timeline windows closes, and subtitles not properly linked to a text setup are highlighted. You need to fix the subtitles file to use the correct text setup path, and then import the file again.

Subtitle XML Elements

Subtitle XML files are text files that use the following elements. Use the following syntax to work with Subtitle XML files external to the Flame workstation.

In Flame-compliant subtitle XML file, the <!DOCTYPE> is `subtitle`.

<name></name>

Description	An element that defines the name of the Subtitle XML file. This name appears as the name of the clip created when importing the subtitle XML file.
Attributes	none
Sample	`<name>French version: Day at the Races</name>`

<rate></rate>

Description	A format element that describes the frame rate of the Subtitle XML clip that appears in Flame after importing the Subtitle XML file. The frame rate must be that of the target sequence or the subtitles might slip timecode.
Attributes	none
Sample	`<rate>59.94 NDF</rate>`

<resolution></resolution>

Description	A unique element that wraps around all of the attributes determining the format of the clip when importing a Subtitle XML file. This clip can then be resized within Flame like any other clip, if necessary. If any of these values differ from the target sequence, Flame will apply Resize Timeline FX to the subtitles.
Mandatory children	**<width></width>** The width of the imported Subtitle XML clip. An integer. **<height></height>** The height of the imported Subtitle XML clip. An integer. **<depth></depth>** The bit depth of the imported Subtitle XML clip. **<aspect></aspect>** The aspect ratio of the imported Subtitle XML clip. **<scanformat></scanformat>** The scanformat of the imported Subtitle XML clip.

Sample	`<resolution>`
	`  <width>1920</width>`
	`  <height>1080</height>`
	`  <depth>8</depth>`
	`  <aspect>1.778</aspect>`
	`  <scanformat>default</scanformat>`
	`</resolution>`

<timecode></timecode>

Description	An element that defines the timecode in and out points during which the subtitles occur. All the titles in the Subtitle XML file will occur during the timecode indicated here, with each title also having its own start and end point indicated within the title elements.
Mandatory children	**<start></start>** The start timecode of the subtitle sequence. **<end></end>** The end timecode of the subtitle sequence.

Sample	`<timecode>`
	`<start>01:00:00;00</start>`
	`<end>01:00:08;00</end>`
	`</timecode>`

<video></video>

Description	The <video> element wraps around all of the <title> elements, which in turn contain the text and attributes of individual subtitles.
Mandatory children	**<title></title>** The <title> element wraps around each subtitle, with its associated start and end time, text, and font style and position parameters.
Sample	<video> <title> <start>01:00:01;00</start> <end>01:00:02;00</end> <text>This is a subtitle string</text> Garamond <size>20</size> <fontcolor> <alpha>255</alpha> <red>255</red> <green>255</green> <blue>255</blue> </fontcolor> <vertical>0</vertical> <horizontal>0</horizontal> <setup>/opt/Autodesk/project/test/text/paz.ttg</setup> </title> </video>

<title></title>

Each <title></title> element defines a single subtitle which will appear as a Text Timeline FX applied to a gap once imported into Flame, and added to a sequence.

The font style, colour, size, and position parameters are overridden if a valid text setup file is referenced within the <title> element.

<start></start>

Description	The start timecode of the current subtitle
Attributes	none
Sample	<start>01:00:01;00</start>

<end></end>

Description	The end timecode of the current subtitle
Attributes	none
Sample	<end>01:00:02;00</end>

<text></text>

Description	The text of the current subtitle To insert a carriage return, add <
> (without the brackets) to the text.
Attributes	none
Sample	`<text>This is a
subtitle string</text>`

****

Description	The font of the current subtitle. This value is overridden if a text setup file is referenced in the <setup> element.
Attributes	none
Sample	`<font>Garamond</font>`

<size></size>

Description	The font size of the current subtitle. This value is overridden if a text setup file is referenced in the <setup> element.
Attributes	none
Sample	`<size>20</size>`

<fontcolor></fontcolor>

Description	This element wraps around the font colour elements (<alpha>, <red>, <blue>, <green>) of the current subtitle. These values are overridden if a valid text setup file is referenced in the <setup> element.
Mandatory children	**<alpha></alpha>** The <alpha> colour component of the font of the current subtitle. **<red></red>** The <red> colour component of the font of the current subtitle. **<green></green>** The <green> colour component of the font of the current subtitle. **<blue></blue>** The <blue> colour component of the font of the current subtitle.

Sample	`<fontcolor>`
	`  <alpha>255</alpha>`
	`  <red>255</red>`
	`  <green>255</green>`

```
    <blue>255</blue>
  </fontcolor>
```

<vertical></vertical>

Description	The vertical text position of the current subtitle. This value is overridden if a text setup file is referenced in the <setup> element.
Attributes	none
Sample	`<vertical>0</vertical>`

<horizontal></horizontal>

Description	The horizontal text position of the current subtitle. This value is overridden if a text setup file is referenced in the <setup> element.
Attributes	none
Sample	`<horizontal>0</horizontal>`

<setup></setup>

Description	The <setup> element can point to a text setup file from which font style and position parameters will be used. The parameters from the text setup file override the formatting elements of the current <title> element. Leave empty when not using a text setup file.
Attributes	none
Sample	`<setup>/opt/Autodesk/project/test/text/paz.ttg</setup>`

Subtitle XML Sample

Use the following XML sample as a starting point for your own subtitle XML files.

The following is an example of Subtitle XML which creates three Text soft effects applied to gaps at these timecodes:

- 01:00:01;00 until 01:00:02;00
- 01:00:03;00 until 01:00:04;00
- 01:00:05;00 until 01:00:06;02

```
<?xml version="1.0" encoding="UTF-8"?>
<!DOCTYPE subtitle>
<subtitle version="1">
```

```
<name>SampleSubtitles</name>
<!-- Rate should be one of : -->
<!-- -->
<!-- * '23.976'-->
<!-- * '24'-->
<!-- * '25'-->
<!-- * '29.97 DF'-->
<!-- * '29.97 NDF' -->
<!-- * '30'-->
<!-- * '59.94 DF'-->
<!-- * '59.94 NDF' -->
<rate>24</rate>
<!-- Set width and height to default -->
<!-- if you want project default res -->
<!-- depth should be : -->
<!-- -->
<!-- * '8', '10' or '12'.-->
<!-- * default : for project default depth -->
<!-- aspect should be : -->
<!---->
<!-- * '1.77777'-->
<!-- * '1.333'-->
<!-- * default-->
<!-- scanformat should be : -->
<!---->
<!-- * 'f1' -->
<!-- * 'f2' -->
<!-- * default-->
<resolution>
  <width>1920</width>
  <height>1080</height>
  <depth>8</depth>
  <aspect>1.778</aspect>
  <scanformat>default</scanformat>
</resolution>
<timecode>
  <start>01:00:00;00</start>
  <end>01:00:08;00</end>
</timecode>
<video>
  <title>
    <start>01:00:01;00</start>
    <end>01:00:02;00</end>
    <text>This is a subtitle string</text>
    <font>Garamond</font>
    <size>20</size>
    <fontcolor>
      <alpha>255</alpha>
      <red>255</red>
      <green>255</green>
      <blue>255</blue>
    </fontcolor>
    <vertical>0</vertical>
    <horizontal>0</horizontal>
    <setup>/opt/Autodesk/project/test/text/paz.ttg</setup>
  </title>
```

```
    <title>
      <start>01:00:03;00</start>
      <end>01:00:04;00</end>
      <text>This is another subtitle string</text>
      <font>Garamond</font>
      <size>20</size>
      <fontcolor>
        <alpha>255</alpha>
        <red>255</red>
        <green>255</green>
        <blue>255</blue>
      </fontcolor>
      <vertical>100</vertical>
      <horizontal>100</horizontal>
      <setup>/opt/Autodesk/project/test/text/paz.ttg</setup>
    </title>
    <title>
      <start>01:00:05;00</start>
      <end>01:00:06;02</end>
      <text>This is a third subtitle string</text>
      <font>Garamond</font>
      <size>20</size>
      <fontcolor>
        <alpha>255</alpha>
        <red>255</red>
        <green>255</green>
        <blue>255</blue>
      </fontcolor>
      <vertical>-100</vertical>
      <horizontal>-200</horizontal>
      <setup>/opt/Autodesk/project/test/text/paz.ttg</setup>
    </title>
  </video>
</subtitle>
```

Exporting from the Command Line

Flame includes a command line export utility, `flame_export`. It can be found in `/opt/Autodesk/io/bin/`.

This utility allows you to couple a Wiretap Node ID and an export preset to export material from the command line. Here is an example:

```
./flame_export -e
/projects/projectOne/638010ac_540f1f4c_0000b4d4/638010ac_5421becf_0008e888/638010ac_5421beea_000cb50c
 -p /tmp/qt_review.xml -d /path/to/export/
```

Although this utility could be coupled with a small program that leverages the C or Python APIs for Wiretap, there are simple tools bundled with the application to browse Flame project structures, using command line tools:

```
./wiretap_get_children
./wiretap_print_tree
```

We can get parts of the hierarchy from a project by limiting the depth of the tree to show:

```
 /opt/Autodesk/wiretap/tools/current/wiretap_print_tree -h localhost:IFFFS -n
/projects/projectOne/ -d 1
```

This will output:

```
projectOne <node> (PROJECT) Node ID: /projects/projectOne/
Workspace <node> (WORKSPACE) Node ID:
/projects/projectOne/638010ac_540f1f5c_000a0265
Shared Libraries <node> (LIBRARY_LIST) Node ID:
/projects/projectOne/638010ac_540f1f4c_0000b4d4
```

The clip we want to export is located in a Shared Library, so we can just look at this part of the project hierarchy:

```
 /opt/Autodesk/wiretap/tools/current/wiretap_print_tree -h localhost:IFFFS -n
/projects/projectOne/638010ac_540f1f4c_0000b4d4 -d 2
```

This will output:

```
Shared Libraries <node> (LIBRARY_LIST) Node ID:
/projects/projectOne/638010ac_540f1f4c_0000b4d4
LIB1 <node> (LIBRARY) Node ID:
/projects/projectOne/638010ac_540f1f4c_0000b4d4/638010ac_5421becf_0008e888
sg050 <clip> (CLIP) Node ID:
/projects/projectOne/638010ac_540f1f4c_0000b4d4/638010ac_5421becf_0008e888/638010ac_5421beea_000cb50c
```

The clip we want to export is `sg050` and the preset we want to use is `/tmp/qt_review.xml`.

```
./flame_export -e
/projects/projectOne/638010ac_540f1f4c_0000b4d4/638010ac_5421becf_0008e888/638010ac_5421beea_000cb50c
 -p /tmp/qt_review.xml -d /path/to/export/
```

This will export the clip using the defined preset.

For more detail about the available options of the command line, just type the following in a shell:
`/opt/Autodesk/io/bin/flame_export`

You can use flame_export to export content from an opened project by using the `-f` switch (foreground export). This exports the contents in the foreground.

Exporting from a currently opened project does come with some limitations:

- Do not delete the clip being exported by flame_export before the process finishes, or the export will fail.
- flame_export with the foreground option does not create a Backburner-managed task, and so cannot be managed by the Backburner queue.
- Effects and transitions must be rendered prior to being exported by flame_export. If not rendered, these effects and transitions appear as black frames in the exported media files.
- The option `Copy Exported Clip in Media Library` used in a Sequence Publish export preset is not supported by flame_export. The export will succeed but without creating clips referencing the exported media.

NOTE You can only use flame_export to export sequences from projects created in a Flame that is of the same release version. You cannot use flame_export to export sequences from projects created in a previous version. So flame_export installed with Flame 2017 can export from a project created in version 2017, but not 2016.

Conform Overview

Use Conform to troubleshoot imported timelines and to link segments to sources.

Events List Displays the events that make up the sequence displayed in the Timeline area. Selecting an event in the list also selects it in the Timeline. You can sort the list using by clicking the column headers. Linked events are usually displayed, but if you are only interested in unlinked events, you can hide the linked events

from the Display Options box. If the Events list displays the message All Sources Linked, the selected sequence is already conformed.

The data displayed in the Event list is one of the following colours:

Red This information is used as a match criteria, and the Potential Matches contain no source that matches this criteria.

Yellow This information is used as a match criteria, and the Potential Matches contain more than one source that matches this criteria.

White This information is used as a match criteria, and the Potential Matches contain only once source that matches this criteria.

Grey This information is not used as a match criteria.

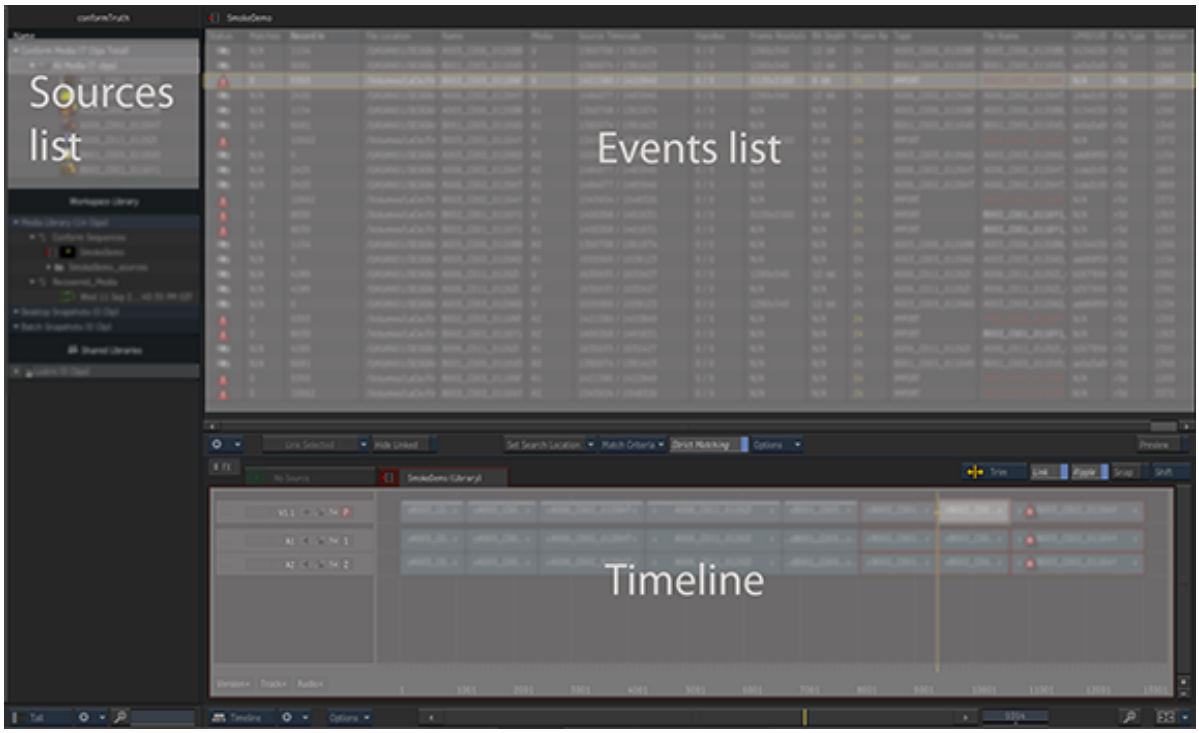
NOTE Lock a track and select Filter Locked Tracks to exclude that track's segments from the Events list.

Sources list The Sources list display all the information about the sources found when using Set Search Location or Set as Conform Search Location, from the contextual menu. From all the potential sources, Flame uses the criteria you set using the Match Criteria box to filter in potential matches. From there, you can select the right source and link it to the selected event.

Disable **Display Options ➤ Filter Potential Matches** to show all the sources. If you select an event with no matches, it shows the full list, in order to allow for force linking.

In the Media Panel, click the Conform tab to display only the Sources list.

Timeline The Timeline displays an open sequence, with controls similar to the ones found in the Timeline tab. Unlinked segments are boxed in red.



Editing Events for Conforming

Whenever you import an AAF, XML, or EDL, you can encounter conforming issues. Most of these issues are problems that can be resolved by editing the contents of the sequence: a file path (files were moved), a file type (different transcoding), a tape name, or even a source timecode.

You can edit any of the following columns, unless the track containing the segment is locked. You edit most of these fields directly in the events list, unless otherwise noted. And modifications to the timeline updates the Events list, so slipping a segment updates its Handles in the Events list.

Event # Only EDLs specify a Event number value, for reference. Or you can set manually an Event number, if needed, as this data can be used as a number for Publish.

Segment # A sequential index (from 1) indicating the relative position of each segment; gaps are not indexed unless they have a Timeline FX applied. You cannot edit this field directly, but changing the position of the segment in the timeline updates this

Media Displays the segment's track. You can only edit the audio tracks, for audio patching, using either the calculator (click the field), or by `click-drag`.

Tape Tape name if present in the imported sequence file.

Source In and Source Out You can edit these fields using either the calculator (click the field), or by `click-drag`.

- If the selected segment is unlinked, you can edit Source In and Source Out without affecting the head & tail. Use his to fix source timecode issues in AAF and XML sequences.
- If the selected segment is linked, the modifications are interactive and similar to a slip: heads & tails are updated based on the linked media. In this case, you cannot change the timecode outside of the media boundaries.

Name The name of the segment.

Shot Name Used in the Shot Publish workflow, this links segments with the same Shot Name together to form a Shot.

Comment The contents can come from a variety of sources, but in an EDL conform this field contains additional conform instructions.

File Location The path to the referenced file.

File Name The file name, without the extension. When you import a sequence, a file's name is broken in two: the name in the File Name column, the extension in the File Type. To edit the extension, you edit the File Type column.

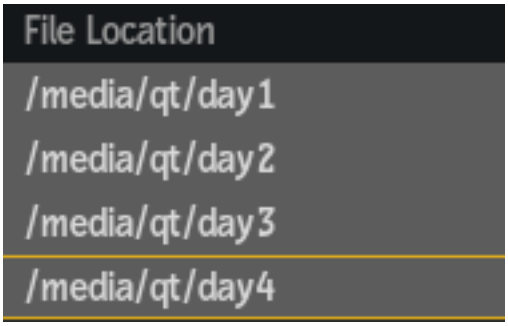
File Type The file extension, to edit if the referenced files were actually transcoded to a different format.

Bulk Editing in Conform

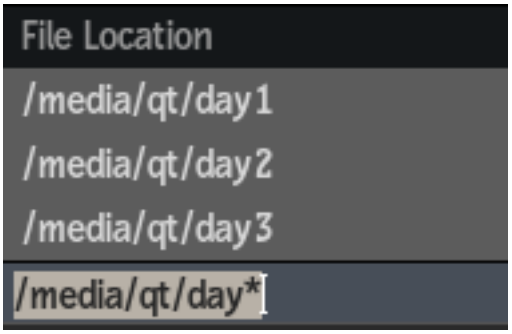
You can edit each field one at the time. But sometimes you need to update multiple file paths (because the files are not where the XML says they are), change the start timecode of a multiple sources.

Bulk editing multiple file locations:

1. `Shift-click` or `Ctrl-click` segments to select the ones that need their file location to be updated.

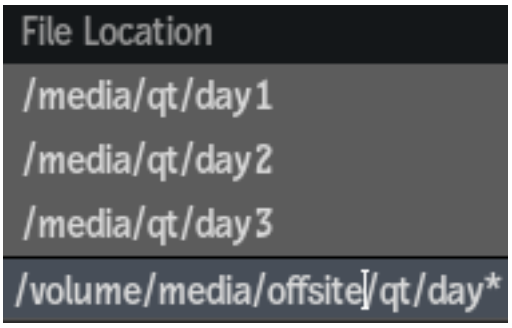


2 Click any one of the selected file location.



Note how non-matching characters are replaced by an asterisk.

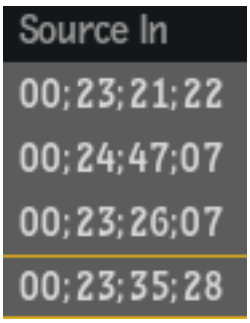
3 Edit the file location to the correct path.



4 Press enter to update all the selected locations.

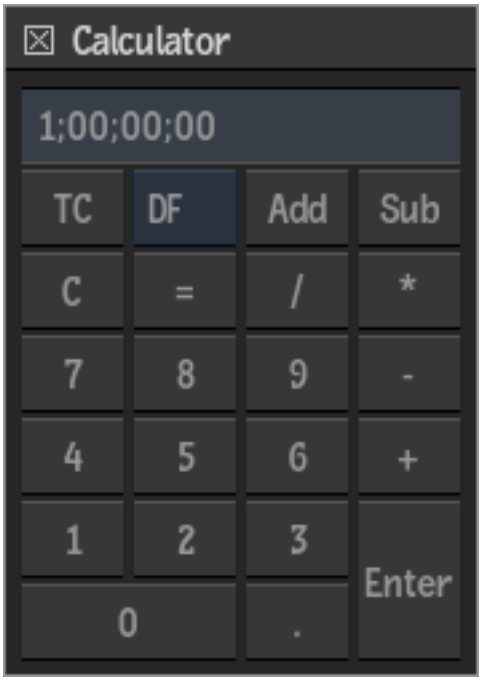
Bulk editing Source In timecodes with an offset:

1 `Shift-click` or `Ctrl-click` segments to select the ones that need Source In to be updated.

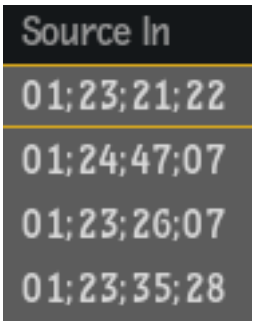


2 Click any one of the selected source in to display the calculator.

3 In the calculator, enter the offset.



4 Click Add to offset the selected source in by the specified value.



Adding Sources for Linking to the Conform Media List

Adding sources from an external location:

1. In Conform, click Search Location.
2. Using the Set Directory window, select the directory where the sources are located.
3. Click Set.

   Flame clears previous sources from the Conform Media folder, and then displays all the media files from the selected location, including any media files located in sub-directories.

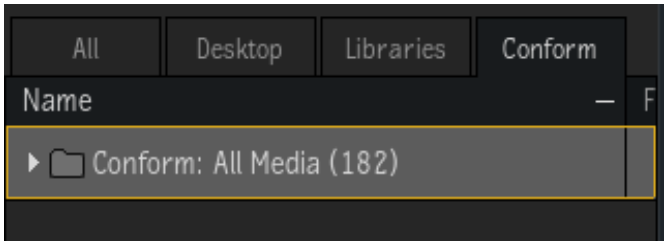
Adding sources that are already in Flame:

1. In Conform, select a folder, a library, a reel, or a shared library.
2. Right-click and select Set As Conform Search Location.

   Flame clears previous sources from the Conform Media folder, and then displays all the media files from the selected location, including any media files located in sub-directories.

You can also manually add clips from the Media Panel to the Conform Media folder: drag-and-drop them to the Conform Media folder. This does not clear already present sources.

TIP Clip the top-left Conform tab for the Media Panel to display only the sources.



Notes

- Flame only loads files it can use in a conform, so do not worry about non-media files being loaded to the Conform Media folder.
- Use the options found in the General and Format Specific Options tabs of the Set Directory window to modify the characteristics of the source files found: edit their resolution, bit depth, debayering settings, etc. Keep in mind that this will either facilitate or hamper the linking, depending on the settings and match criteria you set during the conform.
- When you load sources using the Set Search Location button, Flame does not import the media files at that time: it creates only a list of references to the clips. This means that Flame does not cache source media loaded to the Conform Media folder. To cache the source media once relinked, right-click the segment and select **Media ➤ Cache Source Media**.

Relinking Media

In Conform, you can easily relink an unlinked sequence that stores the path to the media (display the path by `Alt-click` the segment). Most AAF, FCP XML, DLEDL, as well as native sequences, store these file paths.

To relink segments of a sequence to their media:

1. Display the sequence in the Conform tab.
2. Do one of the following:
   - Select Link to Media File from the Set Search Location drop-down menu.
   - Click Link to Media File if the button is already displayed.

Flame relinks all unlinked segments to their media.

To relink a segment to its media:

1. In the Events list, select the unlinked segment that you need to relink.
2. Click one of File Location, File Name, or File Type to select it.
3. Press Enter.

   Flame scans the File Location for files matching the File Name and File Type fields, and displays the result of the scan in the Conform folder.
4. Click Link to Media File to relink the segment to the file.

Available Options

Limit Handles Enable Limit Handles when using Link to Media Files to limit the number of relinked handles, performing a consolidate on relinking. And if you ever need to access more than the consolidated handles, unlink the segment, disable Limit Handles, and click Link to Media Files: the full media is now relinked to the segment.

Save Sources Enable Save Sources to save a copy of the source in the Sources reel. The source saved is what is relinked to the segment and thus respects the Limit Handles setting.

TIP When you import a sequence (AAF, FCP, XML), consolidate the handles on import, and enable Cache Source Media to improve performance. When you need the original full-resolution media, or the media with the discarded handles, unlink the sequence, and then relink using the previous procedure. Every segment is now restored to the full-resolution, not cached, original media.

Multi-Channels and Relinking

You can relink content to multi-channel clips (OpenEXR or Photoshop .psd) using Conform. The next procedure is useful if your sequence contains segments using for source a layer of a multi-channel clip. It is still applicable, but less useful, for alphas; in this case you should look into Conforming with Matte Containers.

TIP `Alt-click` the clip or the segment to display the path to its media.

If the media of the unlinked OpenEXR or .psd is at the location specified by the clip:

1. From the contextual, open the clip as a sequence.
2. Go to the Conform tab.
3. Select the segment to relink.
4. Click Link to Media File.

   The segment is now relinked to its media.

If the media of the unlinked OpenEXR or .psd is at a location differing from the one specified by the clip:

1. Open the clip as a sequence.
2. Click the Conform tab.
3. Set only File Name as Match Criteria.

4 Click Set Search Location to locate the media files.

Make sure that **Clip Options** is set to **Include Alpha Clip** so Conform also finds the different channels in the media files.

NOTE You no longer have to use both File Name and Name as in previous releases to find the media. You can still use Name for advanced matching but it is no longer required for standard filename Conform.

5 Once you have located the files, click Set.

You can now proceed with matching sources and segments as usual.

Conforming With Matte Containers

You can promote a segment to an RGBA matte container when conforming. It allows you to keep comp information in your conformed sequence.

When importing a sequence (EDL, AAF, and FCP XML):

- Set, in the MediaHub, **General ➤ Clip Options ➤ Alpha Channel Processing box** to **Matte Container with Comp**: every segment referencing an RGBA media file is promoted to a matte container with one RGB track and one alpha track.
- Select only the File Name match criteria in the Conform tab, or Flame cannot conform the RGBA content.

NOTE

- When importing an FCP XML sequence, all content created from Photoshop .psd layers is conformed as a matte container.
- The contents in Final Cut Pro should have been imported and the segment needs to use the layer name for clip name. If the clip name is derived from the PSD media file, the match is done using the flat PSD layer.

To relink a segment to RGBA material and promoting that segment to a matte container:

1 In the Conform tab, click Set Search Location.

2 In the Set Directory, set the Alpha Channel Processing box to one of the following:
   - Include Alpha Clip: This imports RGB and alpha channels as separate clips. In the Conform list, paired clips (RGB clip + its alpha) are displayed as a Multiple, yellow, entry.
   - Create Matte Container: Creates a Matte Container for the clip. And segments referencing linked clips are promoted to Matte Containers when matched.
   - Ignore Alpha Channel: Only the RGB part of the media is used to create the clip, and the conform sequence shows a regular (non-container) segment.

3 Once you have located the files, click Set.

You can now proceed with matching sources and segments as usual.

NOTE If a segment is already a matte container, using the Create Matte Container effectively create a matte container within that original matte container. Select the Include Alpha Clip and Flame automatically assigns the correct channel to each of the matte container tracks.

Working with Path Translation

Path Translation allows you to fix problems with media that was relocated after it was imported in Flame.

Use path translation for:

- Clips in the Workspace: If the file system location of the clip's source has been modified, add a translation path to fix the issue without having to use Conform or re-import the contents. Restore a broken connection between a clip and its media. This break can happen when the media referenced by the clip is moved from its location. The original paths to the media are not changed, but translated by Flame. This means that if you open the workspace on the original workstation, the sources remain linked.
- Batch setups: Loading a Batch setup with Import, Read File, and Write File nodes uses the path translation to modify the location of media files, Open Clip files and Batch setup. The data from the setup files is not modified, to protect its integrity: a user on the original file system will not be impacted.
- Conform: When you import an AAF or an FCP XML file with Link to Media Files enabled, Flame tries to reconnect the segments to their sources, using the paths defined within the sequence file. Oftentimes in a collaborative environment, with varying OSes and volume mount points, or with dead drops, the paths in the imported sequence cannot be resolved by Flame.

  Usually, the AAF is produced on a Mac OS X or Windows, where external volumes are mounted under */Volumes/* or a drive name. On Linux, such volumes can be mounted on any point. Path translation allows you to convert a path from one format into another. Once path translation is set up, linking and relinking to sources should be seamless.

NOTE This mechanism is similar to Lustre's Wiretap path translation function.

You set up the path translation in a configuration file, where you define pairs of paths.

To set a path translation pair:

1. Open a shell window. You do not have to close Flame.
2. As root, open the following file in a text editor:
   /opt/Autodesk/cfg/pathTranslation.cfg
3. Define a Source path and a Destination.
   For example, we need Flame to search for media files seen from an offlining Mac at /Volumes/SAN, but seen from the conforming Linux workstation at /ProductionSAN:

   ```
   <PathTranslationTable>
   <PathTranslation src="/Volumes/SAN" dst="/ProductionSAN" />
   </PathTranslationTable>
   ```

4. Save and close the file.
   Flame can immediately start translating a source path found in pathTranslation.cfg to its destination path when you import a sequence or use Link To Media File in Conform.

Conforming an EDL

A DLEDL, such as one produced by the Flame when exporting a Sequence as an EDL, does contain the paths to the media. If after having imported a DLEDL some segments are still not linked to their media, use the Conform tab to relink both segments and media.

A regular EDL does not contain the paths to its media. This means that in most cases you will need to perform the following conform operations.

1. Display the EDL to conform in the Conform tab. Do one of the following:
   - If you imported the EDL using the MediaHub: switch to the Conform tab, right-click the EDL and select Open as a Sequence.

- If you have not already imported the EDL: switch to the Conform tab, right-click the Event list and select Load New FCP XML/AAF/EDL.

  From the window that appears, locate and select the EDL to conform, and then click Load.

2 Add potential sources to the Conform Media (page 260) folder.

You now have two lists: the Events list, displaying all the events making up the sequence, and the Conform Media list, displaying the potential sources you selected.

NOTE By default, the Conform Media list displays only the sources matching the currently selected event. Disable **Options ➤ Filter Potential Matches** to view all the possible sources.

3 With **Options ➤ Filter Potential Matches** selected, click a event from the event list. The sources matching that event are displayed in the Potential Matches list.

NOTE You can also select a segment directly in the timeline. Both the timeline and the event list update to reflect the current selection.

4 What you do next depends on the Status column of each event.

Match Select Link Sources from the Linking combo box. Link Matched Sources links all events with the Match status to their unique sources. Match indicates that Flame found only one source from the Sources list that fit the selected Match Criteria.

Multiple Matches Do one of the following:

- Select the event, the source to relink, and then select Link Selected from the Linking combo box.
- Select the event from the event list, right-click the source to relink, and then select Link.

If there are too many sources to choose from, set additional criteria using the Match Criteria drop-down menu; by trial and error you can reduce the number of matching sources. The event list displays in yellow any criteria with multiple matches.

No Match Found Either add more sources to the Media folder or modify the criteria selected in the Match Criteria drop-down box. Not Found indicates that either the criteria are too restrictive for Flame to find a match in the available sources, or that the source is simply not there. The event list displays in red any criteria that cannot be matched.

Unlinked Add sources to the Media folder. Unlinked indicates that the event is not linked to any source. It appears only when no sources are available in the Media folder, and the Conform section of the Media panel displays 0 clip total.

Linked Nothing to do: the event is already linked to source. Linked events appear in the Events list only if **Options ➤ Hide Linked** is not enabled.

5 Once there all the events are linked to sources, you are done with the conform. You can leave the Conform tab.

IMPORTANT No matter the selected criteria, a source cannot be linked to an event if it is not long enough to cover the whole event.

Available Options

Limit Handles To limit the amount of relinked handles and perform the equivalent of a consolidate on link, enable Limit Handles when linking to the sources. When you need to access more than the consolidated media:

1. Right-click the segment.
2. Select **Media ➤ Unlink**.
3. Disable Limit Handles.
4. Click Link to Media File.

   The full media is now relinked to the segment.

Save Sources To save a copy of the source in the Sources reel, enable Save Sources. The source saved is what is relinked to the segment, including the Limit Handles setting.

Caching Sources Flame does not automatically cache the relinked source media. To cache the source media: right-click the target and select **Media ➤ Cache Source Media**. You can target a segment on the timeline to cache only that segment, or the sequence itself to cache every segment in the sequence.

Filtering Out Tracks Filter out tracks from the Conform list by locking them on the timeline and selecting Filter Locked Tracks from the Options menu. This removes the segments of the locked track from the Conform Events list, and excludes matching sources from the Media Panel's Conform area.

About the Load and Edit New EDL Menu

Use the menu to load an EDL and edit it before conforming it.

EDL Type box Select an EDL type to load. Available options are: ALE, FLX, ATN, TLC, EDM, R23, ETL, and OMF.

File Extension field Displays the file extension for the associated file type. Editable.

EDL Frame Rate box Displays the frame rate of the current EDL. Editable.

EDL Conversion button Enable to convert the EDL to another frame rate.

Conversion Option box Select the option that corresponds to the type of conversion that you want to apply to the EDL. The conversion scripts that appear in this list depend on the frame rate of the EDL you are loading.

Fix TW Match Frames button Enable to fix match frame errors. When you import an EDL that contains dissolves or timewarps, a match frame error may occur. This can cause an unwanted cut at the point where the timewarp begins in your EDL. Match frame errors occur when the in point of the second edit in a dissolve is not the same timecode as the out point of the previous shot. This feature is enabled by default.

Frame Code Mode box Select the option that corresponds to the EDLs you are loading. If you are loading multiple EDLs, the same Frame Code Mode is used for all of them.

2:3 Removal Mode button Enable to remove pulldown when loading the EDL.

Varicam button Enable if the EDL is used to capture material shot with Varicam.

Varicam Frame Rate box Select the Varicam frame rate.

Editing an EDL

Once you import an EDL, you can edit any value except the event number. This includes:

- The tape name for single or multiple events
- The audio patching information
- The transition duration and speed value of dissolves
- A cut to a dissolve or wipe
- The source and record timecodes of events

These tasks are described in the following sections. You can open multiple EDLs and copy and paste events between them. You can also use Auto Edit mode to quickly make changes to the entries you specify.

About the Import EDL Menu

Use the Import EDL menu to create, import, edit, auto-capture, assemble, and save EDLs.

EDL List

The EDL list is divided into 12 columns.

Event # The event number, followed by a capture indicator.

Tape The name of the tape containing the source clip.

A. Patch The Audio Patch information.

Tracks The track for the edit (shown as a combination of: V, 1, 2, 3, 4, 5, 6, 7, 8).

Transition The type of transition between the clips: C for cuts, D <duration> for dissolves, and W <wipe code> <duration> for SMPTE wipes.

Source In The starting timecode of the element in the source clip.

Source Out The ending timecode of the element in the source clip. (`Ctrl`-click the heading to view Source duration.)

Keycode In The starting keycode of the element in the source clip.

Keycode Out The ending keycode of the element in the source clip.

Speed The speed at which the source clip is timewarped in the edit. The value is preceded by a negative sign if the effect is a reverse. The column is blank if the speed value is 100% (no timewarp).

Record In The starting timecode of the element in the result clip.

Record Out The ending timecode of the element in the result clip. (`Ctrl`-click the heading to view Record duration.)

Operations Group

The Operations group contains the commands that allow you to import an EDL.

(a) File Name field

File Name box Select a previously loaded EDL file to display in the EDL work area.

Load button Opens the Load EDL menu where you set options for loading EDLs.

Save button Opens the Save EDL menu where you set options for saving EDLs.

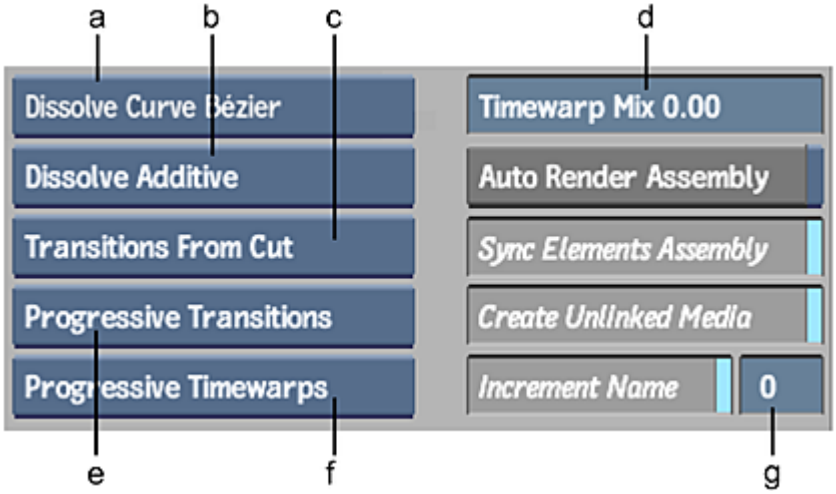
Capture button Opens the Auto-Capture menu where you capture media from a VTR using an EDL.

Close box Select Close to close the currently displayed EDL file, or Close All to close all EDL files.

Assemble box Select Assemble to combine the captured or imported material into a clip. Select M-Assemble to combine multiple EDLs into a multitrack clip.

Assembly Options Menu

The Assembly Options menu contains options that you can set that affect EDL assembly.



(a) Dissolve Curves box (b) Dissolve Type box (c) Transitions Alignment box (d) Timewarp Mix field (e) Transition Render Option box (f) Timewarp Render Option box (g) Increment field

Dissolve Curves box Select the type of interpolation to use for all dissolves.

Select:	To:
Dissolve Curves Linear	Create dissolves with linear interpolation curves.

Select:	To:
Dissolve Curves Bézier	Create dissolves with Bézier interpolation curves.

Dissolve Type box Select a rendering option for dissolves during EDL assembly.

Select:	To:
Dissolve Additive	Render dissolves as additive.
Dissolve Non Additive	Render dissolves as non additive.
Dissolve Inv Non Additive	Render dissolves as inverse non additive.

Transitions Alignment box Select an option for the placement of transitions in assembled EDLs.

Select:	To:
Transitions From Cut	Align transitions after the cut.
Transitions Centred	Centre transitions on the cut.
Transitions Up To Cut	Align transitions before the cut.

Transition Render Option box Select the rendering mode for transitions in assembled EDLs.

Select:	To:
Interlaced Transitions	Override the Preferences settings and render transitions in Interlaced mode.
Progressive Transitions	Override the Preferences settings and render transitions in Progressive mode.

Timewarp Render Option box Select an option for the rendering mode for timewarps in assembled EDL.

Select:	To:
Progressive TW	Render timewarps in Progressive mode.
Interlaced TW	Render timewarps in Interlaced mode.

Timewarp Mix field Enter the mix value for assembled timewarps. You can enter any value from 0.00 to 1000.00.

Auto Render Assembly button Enable to render transitions and timewarps when you assemble the final clip.

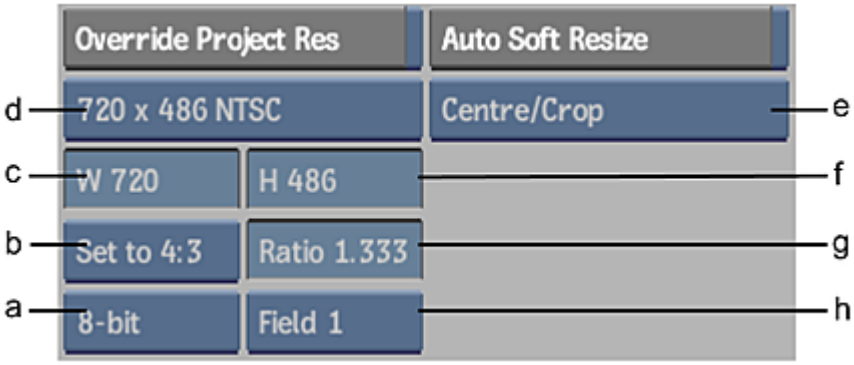
Sync Elements Assembly button Enable to automatically create Edit Sync groups for video and audio tracks that share the same timecode.

Create Unlinked Media button Enable to assemble an EDL, even if you do not have all the sources and, later, use the Recapture/Relink media feature to bring the missing media into your work-in-progress timeline clip.

When you disable this option, you get virtual sources with no indication of source timecode and tape name for the events when no recapture media exists for an event.

Increment Name button Enable to increment the assembled clip name (when assembling the same clip multiple times).

Increment field Enter a number by which the name is incremented. Active only when the Increment Name button is enabled.



(a) Frame Depth box (b) Aspect Ratio Presets box (c) Frame Width field (d) Resolution Presets box (e) Fit Method box (f) Frame Height field (g) Aspect Ratio field (h) Scan Mode box

Override Project Resolution button Enable to assemble pre-captured material of a different resolution than your project's default resolution. Additional controls become active that you use to specify the resolution options for assembly.

During assembly, the system searches the specified library for matching media. For media to match an event during EDL assembly, all the specified parameters must be the same (including frame rate, resolution, bit depth, and so on). When a match occurs for an event, the event is marked as captured.

By default, Override Project Resolution is disabled and the project's default resolution is used when matching EDL events to pre-captured media.

NOTE If an EDL refers to a clip that has the same tape name and timecode but is of a different resolution than the default resolution or that of the first matched event, the clip will not be marked as captured. For example, if you are assembling an NTSC EDL and the EDL refers to a captured HD clip with the same tape name and timecode, the HD clip will not be marked as captured.

Resolution Presets box Select one of many standard resolutions, as well as a Custom option that you can use to specify non-standard resolutions.

Auto Soft Resize button Enable to apply a soft resize when assembling any clip that was captured at a resolution that differs from the native project resolution. Options become active that allow you to specify how the soft resize should be applied.

Fit Method box Select a fit method option to be applied to the imported clip.

Select:	**To:**
Centre/Crop	Fit the source image, centred, over the destination frame. If the source is larger than the destination, it is cropped. If the source is smaller than the destination, it is surrounded by a black border.
Crop Edges	Fit one edge of the source into the destination frame without stretching or squashing the frame. Excess parts of the source frame after resizing are cropped. If the source—after the one edge is resized—is wider than the destination, its overhanging left and right edges

Select:	To:
	are cropped. If the source is taller than the destination, the upper and lower edges are cropped.
Fill	Fit the source, width and height, into the destination frame. If the source and destination frames do not have the same aspect ratio, the image can become distorted.
Letterbox	Fit the source to the destination frame without squashing or stretching it, and without cropping the source. If the source is wider than the destination, black bars fill the top and bottom of the destination frame. If the source is narrower than the destination, black bars fill the right and left sides of the frame. In all cases, the entire source frame is contained within the destination frame.

Frame Width field Displays the frame width of the selected resolution preset. If Resolution Presets is set to Custom then this field is active, allowing you to enter the frame width value that you want to use.

Aspect Ratio Presets box Select a standard frame aspect ratio. Select the Set to w:h option to set the clip to use square pixels. Select Custom to define a custom frame aspect ratio in the Aspect Ratio field.

Frame Depth box Select from one of five frame depth options: 8-bit, 10-bit, 12-bit, 12-bit u, or 16-bit fp.

Frame Height field Displays the frame height of the selected resolution preset. If Resolution Presets is set to Custom then this field is active, allowing you to enter the frame height value that you want to use.

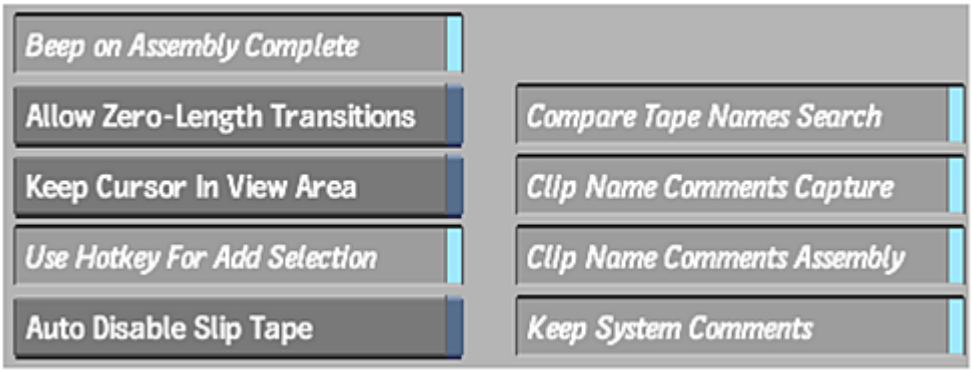
Aspect Ratio field Displays the aspect ratio of the imported clip. When Aspect Ratio Presets is set to Custom, this field becomes active so that you can enter a custom frame aspect ratio.

Scan Mode box Select an option to determine the order in which the fields of interlaced material are scanned.

Select:	To:
Progressive	Scan a frame-based clip with no interlacing.
Field 1	Scan Field 1 first, followed by Field 2.
Field 2	Scan Field 2 first, followed by Field 1.

Preferences Menu

Use the options in the Preferences menu to specify your EDL preferences.



Beep on Assembly Complete button Enable to hear an audible tone when the EDL is assembled.

Allow Zero-Length Transitions button Enable to allow zero-length wipes and dissolves. If this option is disabled and you change a transition duration to zero, the transition automatically becomes a cut.

Keep Cursor In View Area button Enable to keep the cursor from leaving the window when you scroll an EDL. When you scroll an EDL, the cursor will stop at the first or last event on the EDL page.

Compare Tape Names Search button Disable to ignore tape names during assembly.

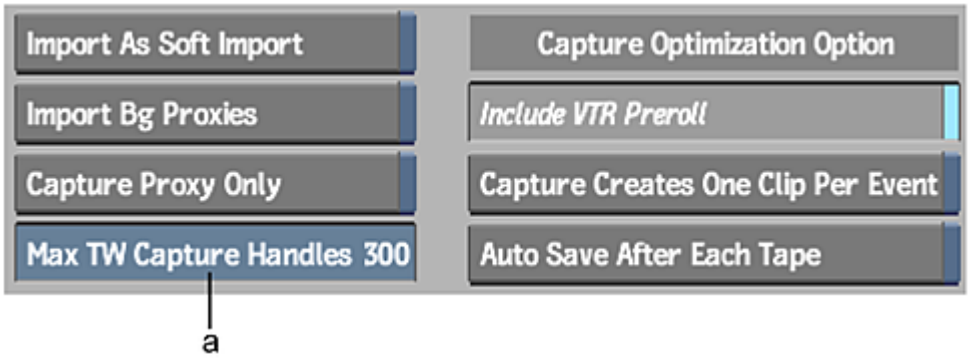
Clip Name Comments Capture button If your events contain clip names in their comments, enable this option to use those names during capture. If this option is disabled, the event number is used as the captured shot's name.

Use Hotkey For Add Selection button Use to change the functionality of selecting events. When this option is disabled, you add to your selection range by clicking the events. When this option is enabled, you must hold the `Ctrl` key or the pen button while clicking events to add them to your selection range.

Auto Disable Slip Tape button By default, the Slip Tape option remains enabled after you slip sources. Enable this option to disable the Slip Tape option automatically after you slip sources.

Clip Name Comments Assembly button If your events contain clip names in their comments, enable this option to use those names during assembly. If this option is disabled, the EDL title is used as the assembled clip's name.

Keep System Comments button Use the Keep System Comments option to protect system comments. When enabled, you can only delete user comments.



(a) Max TW Capture Handles field

Import As Soft Import button Enable to soft-import files with a file-based EDL that are located on a standard filesystem. If you disable this option, the files are hard-imported.

Import Background Proxies button Enable to allow proxies to be generated in the background.

Include VTR Preroll button Enable to capture the events in one pass and store them as individual clips in the clip library. By default, if events in an EDL are separated by a shorter duration than the VTR preroll and more than 10 frames, they are captured in a single pass.

Capture Proxy Only button Use this option in HD projects to auto-capture proxies only when conforming an EDL. This option does not appear in SD projects.

Max TW Capture Handles field You can set the maximum number of handles to be captured for source clips that are part of timewarped events. On timewarped events, the capture handles value is equal to the capture handles value multiplied by the speed of the timewarp. For example, for an EDL event with an increased speed of 500% (in NTSC), if the capture handles are set to 30 frames, 150 frames are captured as handles (30 frames multiplied by 5).

Standard handles may not be enough in the case of high-speed timewarps.

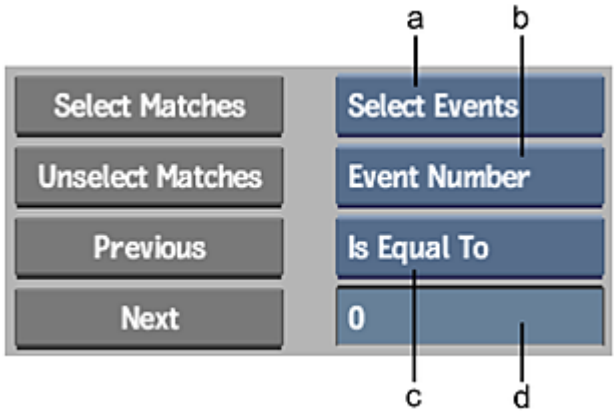
Capture Creates One Clip Per Event button Enable to create a clip in the clip library for every event in the EDL. When disabled, a single clip is created in the library for each group of events captured during the same pass. You can assemble the montage regardless of which option you chose to store EDL events in the clip library.

To reduce the size of clip libraries, you should disable this feature. However, if you need to reference back to individual shots, they may be easier to find if you enable the option and generate a clip for every event.

Auto Save After Each Tape button When this option is enabled, the system performs an AutoSave between each tape when auto-capturing.

Search Events Menu

Use the Search Events menu to search entries for text or timecode in an EDL. When you search the EDL, you need to specify the item you want to search, the filter type you want to use, and the criteria for which you want to search.



(a) Select Item box (b) Search Item box (c) Filter Type box (d) Search Criteria field

Select Matches button Highlight all items that match the search criteria.

Unselect Matches button Remove the highlight from all items that match the search criteria.

Select Item box Select the option that you want to search for.

Select:	To highlight:
Select Events	All events that match the search criteria.
Select Edits	All edits that match the search criteria.

Search Item box Specify the item that you want to search for in the EDL.

Select:	To search for:
Event Number	Event numbers.
Tape	Tape names.
Transition Length	A specific transition duration.
Source In	A source in timecode.

Select:	To search for:
Source Out	A source out timecode.
Speed	A specific timewarp speed.
Record In	A record in timecode.
Record Out	A record out timecode.
Comment	A comment or text within a comment.
Track	A track.

Previous button Move the cursor to the previous item that matches the search criteria.

Next button Move the cursor to the next item that matches the search criteria.

Filter Type box Specify the filter type. The filter type will differ depending on the item you are searching. For numerical searches, select one of the following options from the Filter Type box.

Select:	To find:
Is Equal to	All event numbers that match your search criteria.
Is Not Equal to	All event numbers that do not match your search criteria.
Is Less Than	All event numbers that are less than your search criteria.
Is Greater Than	All event numbers that are greater than your search criteria.

For alphabetical searches, select one of the following options from the Filter Type box.

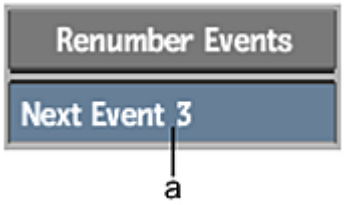
Select:	To search:
Matches	For all tapes that match your search criteria.
Does Not Match	For all tapes that do not match your search criteria.
Contains	For all tapes that contain your search criteria.
Does Not Contain	For all tapes that do not contain your search criteria.
Is Less Than	For all tapes lexicographically less than your search criteria.
Is Greater Than	For all tapes lexicographically greater than your search criteria.

Alphabetical searches are performed on Tape and Comment items by default. To perform a numeric search on these items, enable Treat As Numeric. To match results by case, enable Case Sensitive.

Search Criteria field Enter the search criteria.

Renumber Events Menu

Use the options in the Renumber Events menu to specify the value by which events are renumbered.



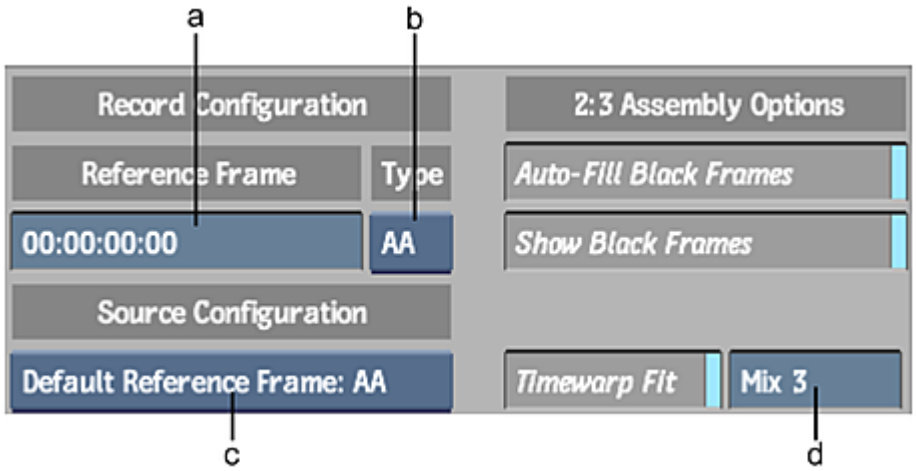
(a) Next Event field

Renumber Events button Renumbers the selected events according to the value specified in the Next Event field.

Next Event field Enter a value for the next event.

2:3 Pulldown Options Menu

Use the 2:3 Pulldown Options menu to set the parameters for 2:3 pulldown insertion upon import.



(a) Reference Frame field (b) Reference Frame Type box (c) Default Reference Frame box (d) Mix field

Reference Frame field Enter the timecode for the reference frame. This is only required for clips stored as files, and not for material on tapes.

Reference Frame Type box Set the reference frame type: AA or BB, corresponding to the timecode value entered in the Reference Frame field.

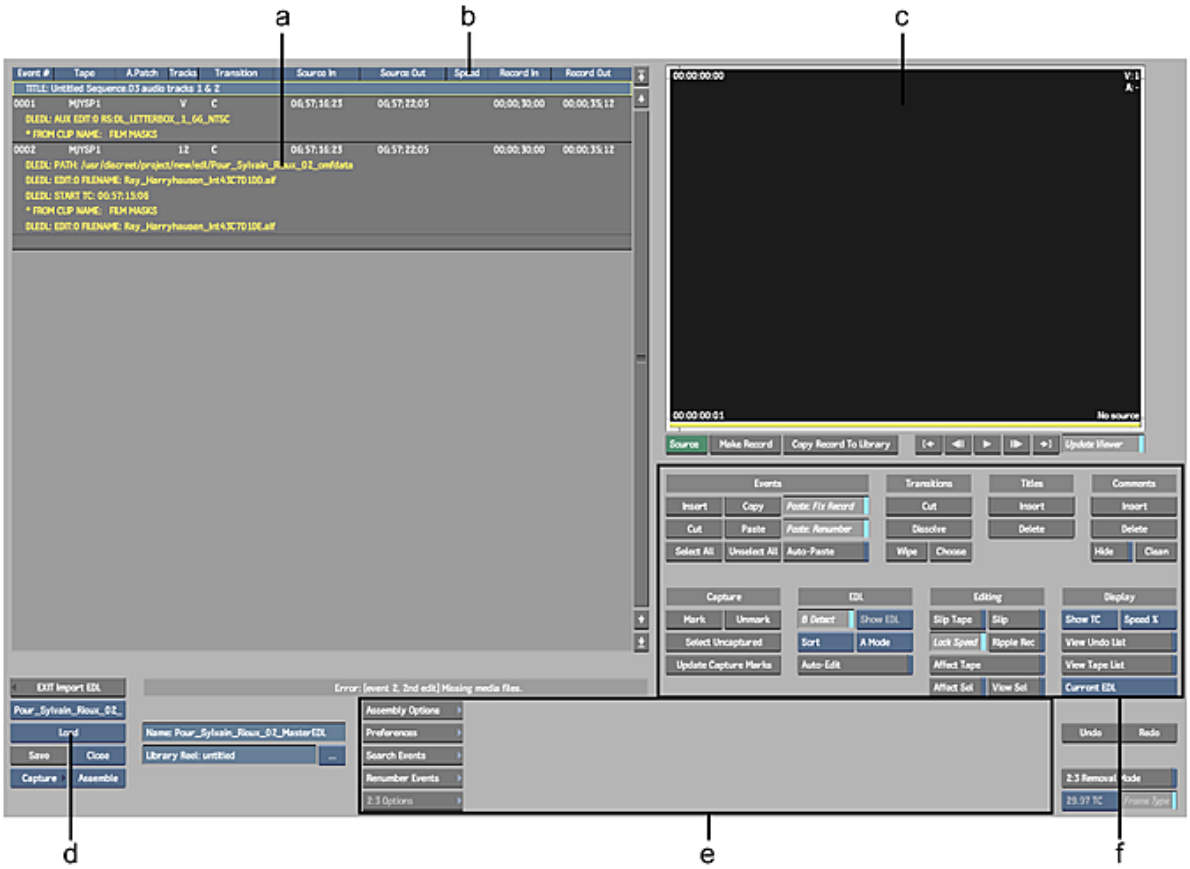
Auto-Fill Black Frames button Enable to fill black frames with the last frame of the outgoing event. By default, gaps occuring between EDL events are filled with black frames.

Default Reference Frame box Select AA or BB.

Show Black Frames button Enable to highlight the places in the currently loaded EDLs where gaps of black frames occur.

Timewarp Fit button Enable to automatically apply a timewarp to material on tapes referenced by EDLs where 2:3 pulldown removal was disabled (for example, when a tape contains native 30 fps interlaced video material), so that it can be easily integrated with 24 fps material.

Mix field Enter the number of frames to ensure that match frames on timewarped elements are maintained. This field is active when Timewarp Fit is enabled.



(a) EDL work area (b) Work area column headers (c) EDL Player (d) Load button (e) Import EDL menus (f) EDL editing tools

About EDL Editing Commands

Events Group

Use the commands in the Events group to cut or copy single or multiple events and paste them to a new location in any open EDL. You can also use special paste functions and selection tools for events.

Insert Event button Inserts a new event into the EDL.

Cut Event button Cuts the selected event. May be used with Paste Event button.

Copy Event button Copies the selected event. May be used with Paste Event button.

Paste Event button Pastes the previously cut or copied event to the selected location.

Paste: Fix Record button Enable to change Record In of the first pasted event to start at Record Out of the previous event. All pasted events are then rippled by the same amount.

Paste: Renumber button Enable to automatically renumber all events according to the Next Event value in the Renumber Events menu. If this option is disabled, the pasted events retain their original event numbers.

Select All button Click to select all events for capture.

Unselect All button Click to unselect all events.

Auto-Paste button Enable to quickly cut events from multiple EDLs and automatically paste them to a single EDL. For a selected EDL, when clicking the Paste button, any events that are cut or copied are automatically pasted to the first EDL.

Comments Group

You can add or delete comments from events using the commands in the Comments group of the EDL menu. Comments can contain up to 256 alphanumeric characters. You can also toggle comments on or off.

EDL comments become part of the source clip when assembling the EDL. Multiple comment lines are merged into one timeline comment line.

Insert Comment button Add a comment to the event selected in the EDL work area. You can add multiple comments to an event. To edit a comment, double-click it and enter the new comment.

Delete Comment button Delete a comment selected in the EDL work area.

Hide button When enabled, will hide all comments in the EDL work area.

Clean button Removes asterisks from comments.

Titles Group

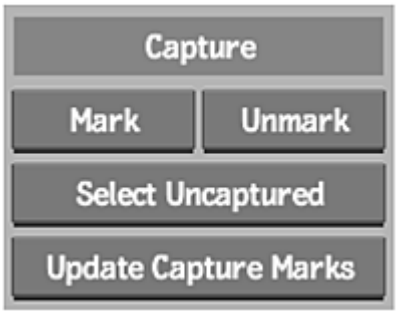
You can add or delete a title from an EDL using commands in the Titles group of the Import EDL menu.

Insert Title button Enter a new title in the Titles area of the EDL work area. You can add multiple titles. To edit a title, double-click it and enter the new title.

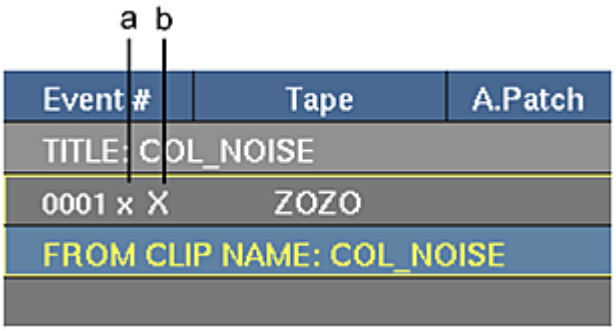
Delete Title button Delete a title in the EDL work area.

Capture Group

You can mark events for capture or select only the uncaptured ones for recapture.



Mark button Click to mark the selected events for recapture with an "X". This allows you to keep track of events that you want to recapture. Events that have already been captured are marked with a lower case x. Therefore, when you mark a captured event for recapture, a lower and upper case x appear (x X) to the right of the event number.



(a) Capture mark (b) Recapture mark

Unmark button Click to remove a recapture mark from the selected events.

Select Uncaptured button Click to select all uncaptured events in the EDL. The next time you click Auto-Capture, only the selected events are captured.

Update Capture Marks button Click to update the list of captured events.

Transitions Group

Use the commands in the Transitions group to change any transition to a cut, dissolve, or standard SMPTE wipe.

Cut button Changes the transition type to a cut for the selected transition, or range of transitions.

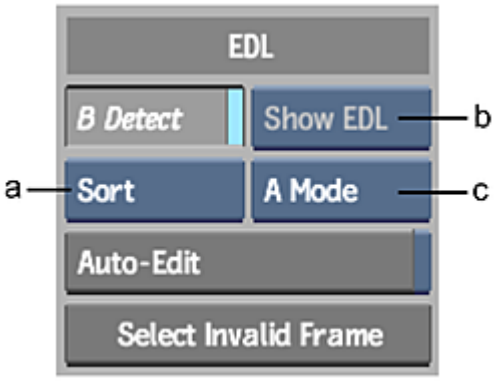
Dissolve button Changes the transition type to a dissolve for the selected transition, or range of transitions.

Wipe button Changes the transition type to a wipe for the selected transition, or range of transitions. The wipe type is set to SMPTE 001.

Choose button Opens the Choose Wipe menu from which you can select from a variety of standard SMPTE wipe types.

EDL Group

You can use any of these options in the EDL group when using EDLs to capture media.



(a) Sort box (b) Show Timecode box (c) Sort Mode box

BRoll Detect button Enable to detect BRolls. BRolls appear as the original tape. You are not prompted for the tape containing the BRoll when you capture the clips. When disabled, BRolls are indicated with a "B" following the tape name in the Tape entry.

Show Timecode box Select the type of timecode to display.

Select:	To:
Show EDL	Display EDL timecode.
Show VC	Display Varicam timecode.

Sort box Click Sort to sort the current EDL, or click Sort All to sort all the EDLs. EDLs are sorted according to the sort order specified in the Sort Mode box. You can change the way an EDL is sorted.

Auto-Edit button Enable to quickly modify only the entries in the selected columns of the EDL. Click the column headers of the columns that you want to auto-edit. Starting with the first entry that you edit, press `Enter` to move automatically to the next entry. If no further entries exist, a new event is added.

Sort Mode box Select the appropriate sort order.

Select:	To sort the EDL:
A Mode	By Record In timecode. If you want to view the EDL in the order of the final assembly, sort the EDL in A Mode.
B Mode	By Tape and Record In timecode.
C Mode	By tape number and Source In timecode. If you want to view the EDL in the order the clips are captured, sort the EDL in C Mode. EDLs are always captured in C Mode, regardless of the sort mode you selected.

Select:	To sort the EDL:
S Mode	By Source In timecode, regardless of tape number. This sort mode is useful for multi-camera real-time EDLs.
by Event#	By event number.
by Tracks	By tracks. In this sort mode, video tracks are placed at the bottom of the list, and the highest audio track is placed at the top of the list.
by Uncaptured	By placing all uncaptured events at the top of the list.

Select Invalid Frame button Click to select a frames of 0 length.

Editing Group

When you modify an entry in an EDL, several options control how other entries are affected. Make sure these options are either enabled or disabled, depending on how you want to edit the EDL.

Slip Tape button Enable to slip all source clips on the tape, or all record clips in the EDL list. All the in and out points of all source or record clips change without affecting their duration.

Lock Speed button Enable to edit the timecode without changing the speed value for the event.

Slip button Enable to slip a source or record clip in a single event. The in and out points of the selected clip change without affecting its duration.

Ripple Rec button Enable to move all Record entries, following an edited or pasted Record Out entry, forward or backward accordingly. This is useful when you want to edit the Record Out of an event without overwriting or creating a gap between the following shots.

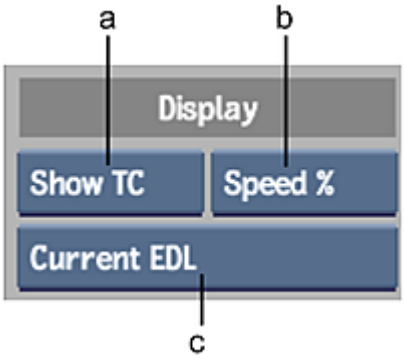
Affect Sel button Enable to affect only the selected events.

Affect Tape button Enable to affect every relevant entry on the same tape when you modify a single entry.

View Sel button Enable to view only the selected events.

Display Group

The Display group contains options for modifying the display of events in the EDL list.



(a) Show TC/KC box (b) Speed box (c) EDL Display box

Show TC/KC box Select whether to display keycode or timecode.

Select:	**To:**
Show TC	Display timecode.
Show KC	Display keycode.

Speed box Select a mode to display the speed change for timewarps.

Select:	**To:**
Speed %	Display the speed change as a percentage.
Speed FPS	Display the speed change in frames per second.

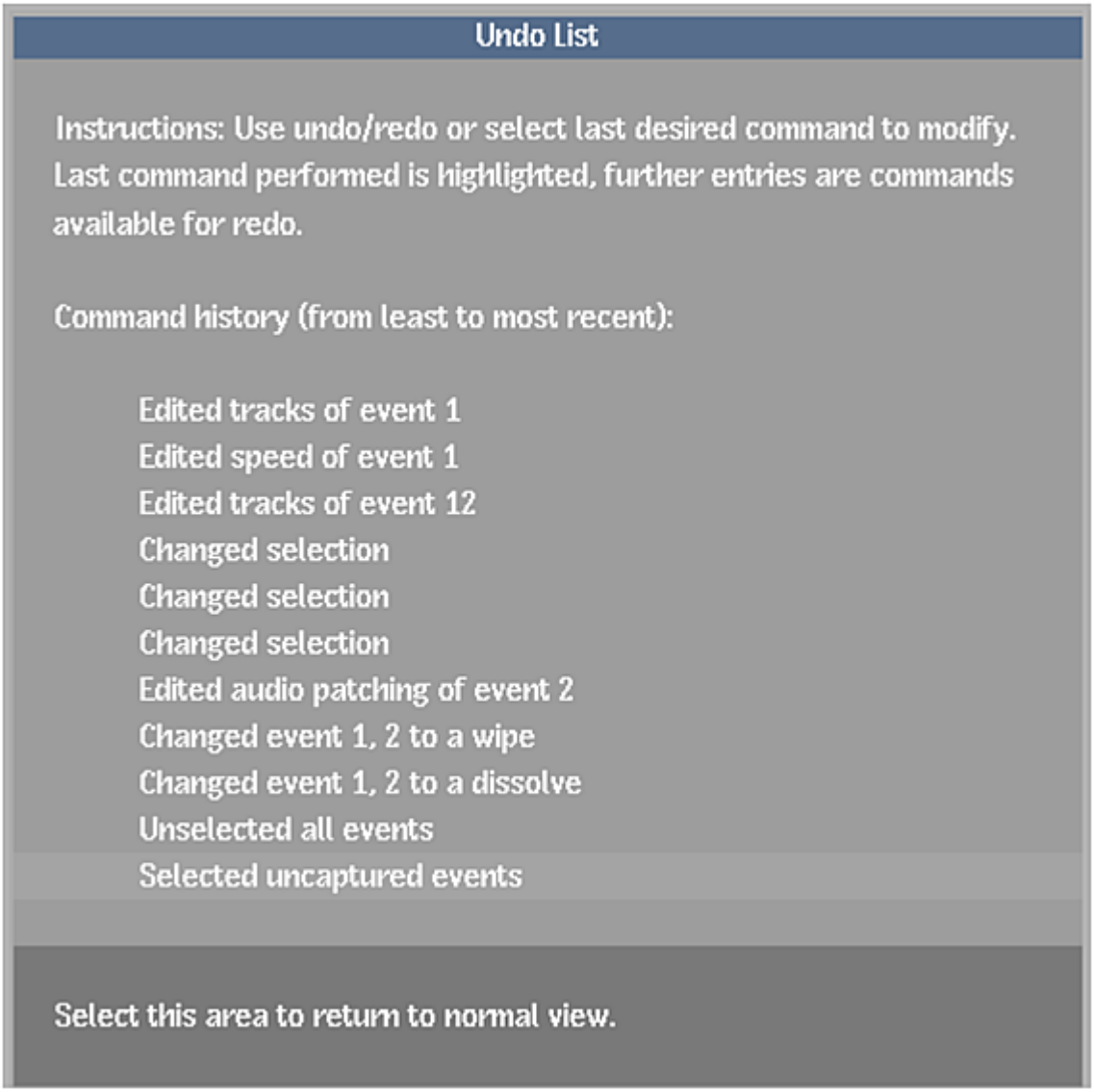
EDL Display box Select a display mode for EDLs, Tape List view, or Undo List view.

Select:	**To:**
EDL	Display timecode.
Undo List	View all modifications (up to the number of undo levels specified in the Preferences menu) that you made in the current session, and undo a single or series of commands. Click the item in the undo list that you want to undo. All operations are undone up to and including that modification. All operations prior to the highlighted modification are redone.
Tape List	Display the Tape List view, showing the last selected EDL.
Tape List (All EDLs)	Display the Tape List view, showing all selected EDLs.

Undo List View

Use the Undo List to view all modifications (up to the number of undo levels specified in the Preferences menu)that you made in the current session and undo a single or series of commands.

Click the item in the Undo List that you want to undo. All operations are undone up to and including that modification. All operations prior to the highlighted modification are redone.

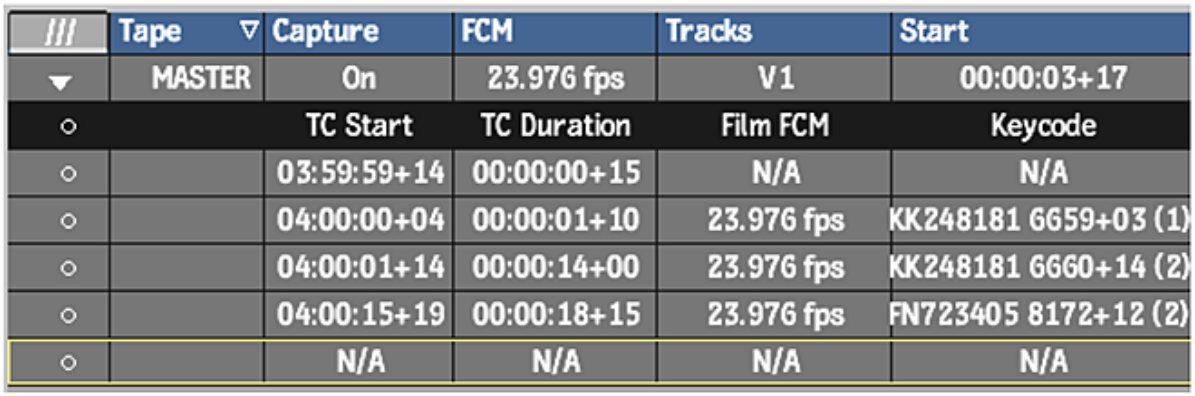


Tape List View

You can use the tape list to modify information pertaining to events for a given tape.

As well, each tape's entry can be expanded in Tape List view to show keycode and 2:3 pulldown information associated with the source timecode.

Click the column header to sort the tape list in descending order using that column. Click the column heading again to sort the tape list in ascending order.



///	Tape	Capture	FCM	Tracks	Start
▼	MASTER	On	23.976 fps	V1	00:00:03+17
○		TC Start	TC Duration	Film FCM	Keycode
○		03:59:59+14	00:00:00+15	N/A	N/A
○		04:00:00+04	00:00:01+10	23.976 fps	KK248181 6659+03 (1)
○		04:00:01+14	00:00:14+00	23.976 fps	KK248181 6660+14 (2)
○		04:00:15+19	00:00:18+15	23.976 fps	FN723405 8172+12 (2)
○		N/A	N/A	N/A	N/A

The work area of the Tape List view is divided into nine columns, and two sections for each tape.

The upper fields of each tape entry contain the following columns.

Tape The name of the tape that contains the source footage. To change the tape name, click and enter a new name.

Capture The capture flag for the tape. When set to On, the tape is captured during an auto-capture session. To set the capture flag for a given tape to On or Off, click the item under the Capture column and drag left or right to set it On or Off.

FCM The frame code mode for the tape. If the frame code mode is 29.97, you can drag over the item in this column to switch between DF/NDF modes.

Tracks The number and type of tracks that will be captured for the tape.

Start The starting timecode for the first event to be captured on the given tape.

Capture Time The duration of the material to capture for the given tape.

Telecine Indicates the telecine log associated with the tape.

Final Telecine Indicates the final telecine log associated with the tape.

Events The number of events to capture for the given tape. You cannot modify this value in Tape List view.

The lower fields of each tape entry contain the following columns.

TC Start Start timecode for each keycode sequence.

TC Duration Duration of time for the keycode sequence.

Film FCM Film frame code mode indicating the speed of the telecine when it scanned the film frames. Change the film FCM by clicking the Keycode field to access the calculator.

Keycode Keycode for the first frame of the sequence.

Removal Whether to apply 2:3 pulldown to the sequence.

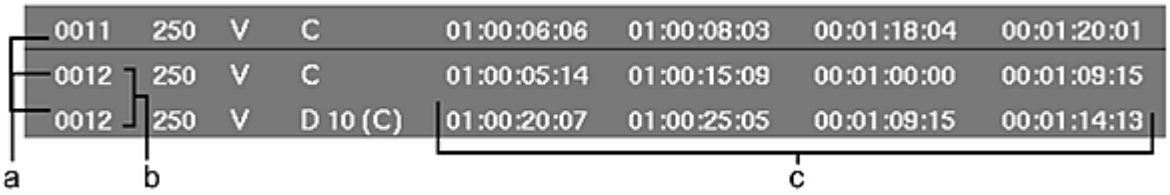
Ref Frame Timecode of the reference frame.

Ref Type Type of reference frame.

DF Ref Frame Timecode of the drop frame reference frame.

EDL View

An EDL contains events, edits, and entries. An event is a complete element in the EDL such as a shot, dissolve, or wipe. An edit is an individual component that makes up an event, such as an outgoing shot or an incoming shot. An entry is an individual value for any variable in the edit, such as dissolve length, Source In, or Speed.



(a) Edits (b) EDL events (c) EDL entries

Every event has a different event number. Some events, such as dissolves and wipes, consist of two edits. The two edits in a dissolve (outgoing and incoming shots) have the same event number.

Editing a Tape Name

You can change a single event's tape name, or the tape name for all events on the same tape. This is useful if the events you want to capture are on a different tape than that listed in the EDL.

Tape names can be up to 56 characters long. The EDL Editor displays only the first eight characters, but the clips maintain the original tape name.

To edit a tape name:

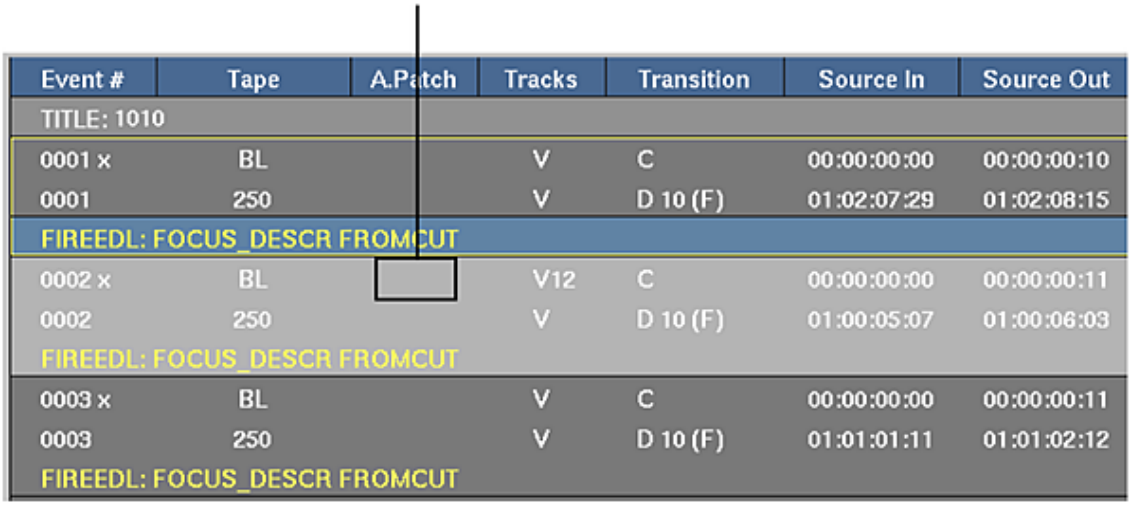
1. In the Import EDL menu, do one of the following:
   - To change the tape name for all events on the same tape, enable Affect Tape.
   - To change only the selected tape name, disable Affect Tape.
2. Click the Tape entry in the event you want to edit and enter the new name:
   - If the Affect Tape option is enabled, the tape name for all events on the same tape changes automatically.
   - If the Affect Tape option is disabled, only the tape name for the selected event changes.
   - If the Affect Selection option is enabled, only the tape names for selected events change.

Audio Patching

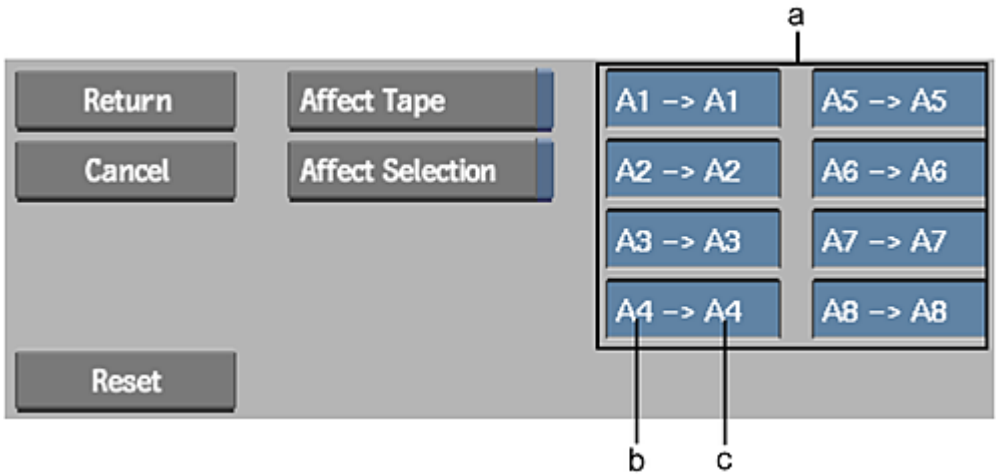
If you want to capture audio from a different track than that specified in the EDL, you can change the audio patching for a single event, selected events, or all events on the same tape. For example, if an edit specifies the audio tracks as 1 and 2, but you want to capture the audio from tracks 3 and 4, use the Audio Patching menu to reroute the audio during auto-capture.

To change audio patching:

1. In the EDL work area, select the events that you want to modify.
2. Click an Audio Patch field.



The Audio Patching menu appears.



(a) Audio Patch boxes (b) Source audio track (c) Destination audio track

You can edit the source audio tracks. Each one of the eight available source audio tracks can be patched to one of the eight available destination audio tracks. You can patch the same source audio track to multiple destination audio tracks.

3 Enable or disable the Affect Tape and Affect Selection buttons.

4 Reroute the audio in the Audio Patch boxes by doing one of the following:

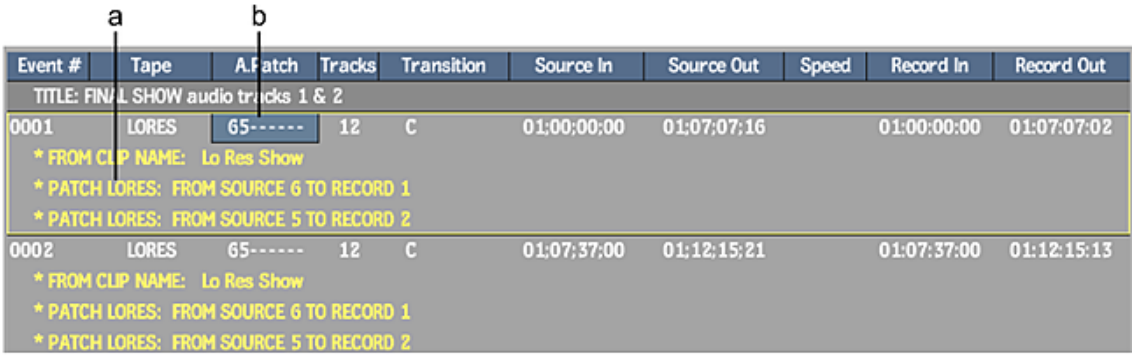
- Drag in an Audio Patch box to cycle through source audio tracks one through eight.
- Click an Audio Patch box to access the calculator, then type a value one through eight indicating the source audio track.

5 Click Return.

To exit the Audio Patching menu without changing the patching information, click Cancel.

The source tracks that have been patched to different audio tracks than the defaults are indicated in the affected event's Audio Patch Field. Dashes in the Audio Patch Field indicate the track is patched to its default. Comments are also added which indicate the name of audio tracks and their patching information.

When clips are captured, audio tracks are patched as indicated.



(a) Comments indicate patching information (b) Source 1 and 2 patched to destinations 5 and 6

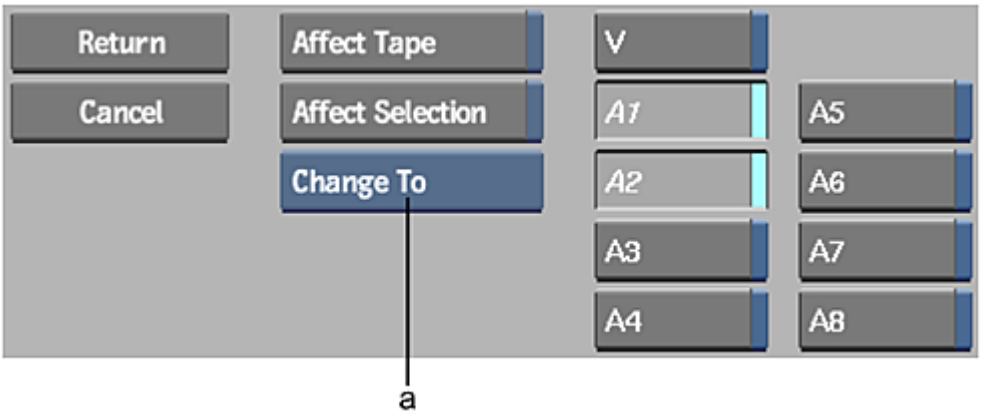
Editing Tracks

You can edit the tracks of an event in the EDL. Editing tracks is useful if you want to modify the tracks captured when you auto-capture or assemble the EDL.

For example, if the Track entry for an event is V1 and you want to capture only the video track (V), change the Track entry to V.

To edit the tracks for an event:

1 In the EDL work area, select the Track entries in the event that you want to edit from the EDL.
The Track menu appears.



(a) Track Tools box

2 Enable the tracks that you want to edit.

3 Enable or disable the Affect Tape and Affect Selection buttons.

4 From the Track Tools box, select the mode that you want to use.

Select:	To:
Change To	Replace tracks in the event with tracks you specified in the Track menu.
Add	Add the tracks you specified in the Track menu to the tracks in the event.

Select:	To:
Filter	Keep only the tracks you specified in the Track menu. For example, if an event is “V12” and you specify “V1,” audio track 2 is filtered out, and the resulting event has “V1.”
Flip	Reverse the status of tracks you specified in the Track menu. For example, if an event is “V23” and you specify “V34,” the resulting event has “V24.”

5 Click Return to confirm the change or Cancel to exit back to the EDL module without making any modifications.

Editing Transitions in EDLs

You can change any transition to a cut, dissolve, or SMPTE wipe. If the effect of an event is a dissolve, you can edit its duration (Transition Duration). If the effect of an event is a wipe, you can also select the type of wipe in the SMPTE Wipe library.

You can add or modify transitions based on a selection range.

You can also change a cut to a dissolve by clicking the Transition entry of the cut. You can change a dissolve to a cut by setting the Transition Duration to 0, as long as the Allow Zero-Length Transitions option is disabled in the EDL preferences.

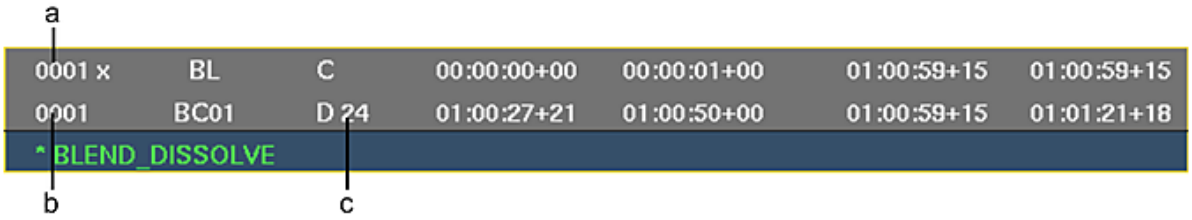
Dissolves in EDLs

Dissolves are represented in the EDL by two consecutive edits with the same event number. The first edit represents the outgoing shot of the dissolve and the second edit represents the incoming shot of the dissolve. The dissolve is listed in the Transition column of the second edit as D *n* (where *n* is the transition duration).

The Transition duration appears in the Transition entry of the second edit in the dissolve. The Start Location for the dissolve is also indicated in the Transition entry.

If the EDL was exported from Flame, the dissolve can be Centred, From Cut, or Up To Cut. The start location for the dissolve appears beside the Transition duration.

The following illustration is a typical dissolve in an EDL.



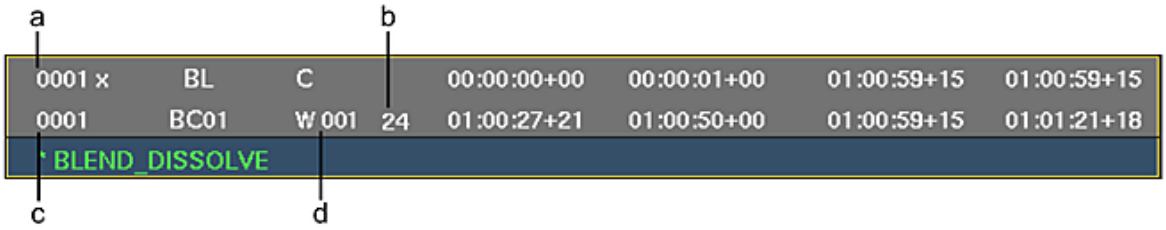
(a) Outgoing shot (b) Incoming shot (c) Transition duration

If you change the Transition duration of a dissolve, the Source Out of the incoming shot changes by the same duration automatically.

Wipes in EDLs

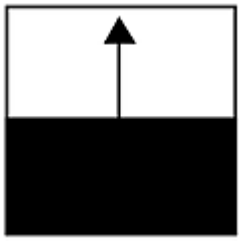
Wipes are represented in the EDL by two consecutive edits with the same event number. The first edit represents the outgoing shot of the wipe and the second edit represents the incoming shot of the wipe. The wipe is listed in the Transition column of the second edit as W *n m* (where *n* is the wipe code and *m* is the wipe duration).

The SMPTE wipe number appears in the Transition entry of the second edit in the wipe. The Start Location for the wipe is also indicated in the Transition entry. A wipe can be Centred, From Cut, or Up To Cut. The start location for the wipe appears beside the Transition duration. The following illustration is a typical wipe in an EDL.

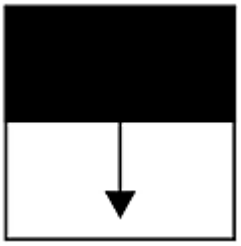


(a) Outgoing shot (b) Transition duration (c) Incoming shot (d) SMPTE wipe number

If you add 500 to the SMPTE wipe number, the wipe is inverted during assembly. For example:



Wipe 002



Wipe 502

If you change the Transition duration of a wipe, the Source Out of the incoming shot automatically changes by the same duration.

Editing Timecode

If you want to capture different material than what appears in the EDL, you can slip or trim the edits to specify the correct material for your final assembly. You can change any source or record timecode in the EDL.

You can also edit timecode and keycode in the Tape List.

To edit timecode:

1. If you want to edit source timecode, and keycode in and out values are displayed, toggle the Show Timecode/Show Keycode box to Timecode.
2. Do one of the following:
   - Click the source or record timecode entry you want to modify and drag right to increase the value or drag left to decrease the value. You can use hotkeys to change the values in varying increments.

Use:	To:
`Shift`-drag	Jump by seconds.
`Ctrl`-drag	Jump by minutes.

Use:	To:
Alt-drag	Jump by hours.

- Double-click the source or record timecode you want to modify to get the calculator, and then enter a new timecode. If the source or record timecode is already selected, click it again to get the calculator.

When you edit source timecode, the corresponding keycode changes to reflect the new frames to be captured. Toggle the Show Timecode/Show Keycode button to see the edited keycode in and out values. You cannot edit the keycode for events in the EDL work area.

NOTE Other entries in the same edit affected by your modifications become highlighted as you edit an entry.

Editing Tape List Timecode and Keycode

In Tape List view, you can slip all the start timecode values for a given tape. This is useful when you need to modify the start timecode of a tape that is used in multiple EDLs, because you edit the value only once.

You can also slip keycode and its associated start timecode and duration. This is similarly useful to alter keycode values correlated to the timecode for given sequences on the tapes.

To slip the start timecode value for a given tape:

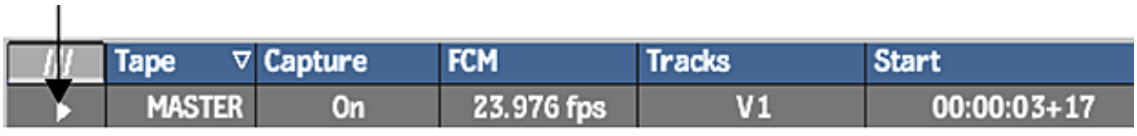
1. In the Import EDL menu, from the Display group, enable View Tape List.
2. Do one of the following:
   - Click the start timecode entry you want to modify and drag right to increase the value or drag left to decrease the value. You can use hotkeys to change the values in varying increments.

Use:	To:
Shift-drag	Jump by seconds.
Ctrl-drag	Jump by minutes.
Alt-drag	Jump by hours.

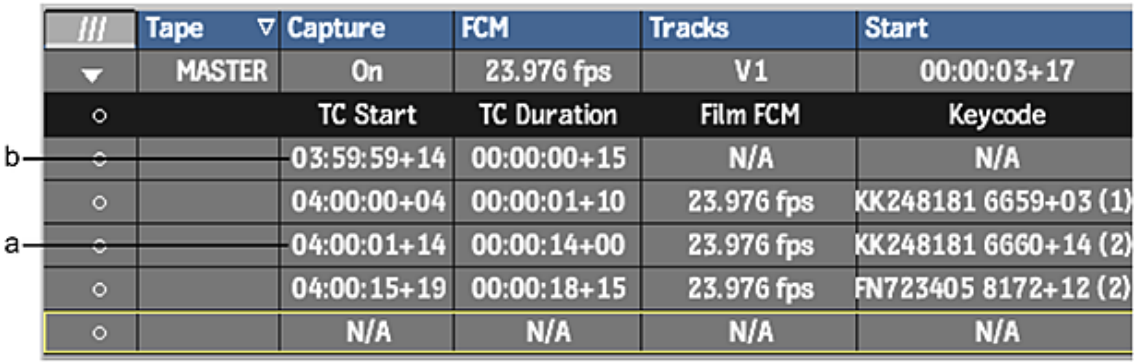
   - Click the start timecode entry you want to modify to get the calculator, then enter a new timecode.

To slip the keycode for a given tape:

1. In the Import EDL menu, from the Display group, enable View Tape List.
2. If necessary, click the triangle on the left of the tape entry to expand its contents.



Sequences of timecode on the tape are associated to keycode values. If no keycode is available from the telecine log(s) you loaded, N/A (not available) appears for the sequence.



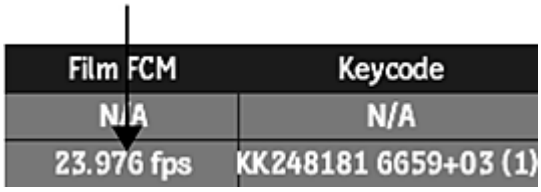
(a) Timecode start and duration defines sequence, with corresponding keycode (b) Timecode without corresponding keycode

3 To edit the keycode for a sequence, do one of the following:

- Drag the keycode entry you want to modify to the right to increase the value or to the left to decrease the value.
- Click the keycode entry you want to modify to get the keycode calculator, and then enter a new keycode.

  The keycode values associated to the timecode sequence (defined by the values in TC Start and TC Duration fields) are altered.

  If you change the frame rate in the keycode calculator, it is reflected in the Film FCM field.

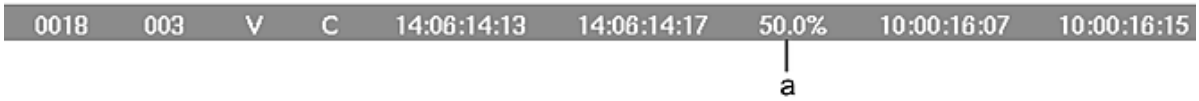


Editing and Creating Timewarps

You can edit an existing timewarp or create a timewarp in your EDL. When you edit a timecode entry of an event with the Lock Speed option disabled, no other entries are affected. This results in a speed change because the source duration differs from the record duration.

For example, if you increase the Source In entry of an event by 5 seconds, the record clip of the same event is 5 seconds shorter than the source clip. The result clip is timewarped when assembled. No other entries are affected.

Timewarps are represented in the Speed column of the EDL. If the Speed value is blank, the event is not timewarped. If there is a Speed value, the event is timewarped.



(a) Speed value

To edit or create a timewarp:

1 In the Import EDL menu, from the Editing group, disable Lock Speed.

2 In the EDL, click the source or record timecode entry for the event that you want to edit and enter the new value.

The value of the timewarp appears in the Speed entry. You may also click directly in the Speed box to change the entry. If the event is already a timewarp, editing any timecode of that event changes its speed value.

You can view the speed value in either percentage or frames per second. Select the View mode in the EDL Preferences menu.

Conforming an EDL with VTR Recapture

NOTE The following requires that the workstation is connected to a VTR.

1. In the Conform tab, right-click the Event list and select Load New EDL.
2. From the window that appears, locate and select the EDL to import, and then click Load.

   The EDL is loaded and opened as a sequence in the timeline view of the Conform tab.
3. Right-click the event list, and select Capture All From VTR.

   The VTR module opens and cues the VTR.
4. Proceed with the recapture of the cued events.
5. Once back to the Conform tab, continue with the conform process as you would with a file-based sequence.

Conforming an Avid AAF

If after having imported an AAF some segments are still not linked to their media, use the Conform tab to relink both segments and media.

1. Display the AAF to conform in the Conform tab. Do one of the following:
   - If you imported the AAF using the MediaHub: switch to the Conform tab, right-click the AAF and select Open as a Sequence.
   - If you have not already imported the AAF: switch to the Conform tab, right-click the Event list and select Load New FCP XML/AAF/EDL.

     From the window that appears, locate and select the AAF to conform, and then click Load.
2. Add potential sources to the Conform Media folder.

   You now have two lists: the Events list, displaying all the events making up the sequence, and the Conform Media list, displaying the potential sources you selected.

   NOTE By default, the Conform Media list displays only the sources matching the currently selected event. Disable **Options ➤ Filter Potential Matches** to view all the possible sources.
3. With **Options ➤ Filter Potential Matches** selected, click a event from the event list. The sources matching that event are displayed in the Potential Matches list.

   NOTE You can also select a segment directly in the timeline. Both the timeline and the event list update to reflect the current selection.
4. What you do next depends on the Status column of each event.

   Match Select Link Sources from the Linking combo box. Link Matched Sources links all events with

the Match status to their unique sources. Match indicates that Flame found only one source from the Sources list that fit the selected Match Criteria.

Multiple Matches Do one of the following:

- Select the event, the source to relink, and then select Link Selected from the Linking combo box.
- Select the event from the event list, right-click the source to relink, and then select Link.

If there are too many sources to choose from, set additional criteria using the Match Criteria drop-down menu; by trial and error you can reduce the number of matching sources. The event list displays in yellow any criteria with multiple matches.

In the event list, if the handles amount for a event is displayed in red, than none of the matching sources have enough handles: This usually happens when the AAF references more media than there is media actually available for conform. You can still force a match by manually linking the source to the sequence's event.

No Match Found Either add more sources to the Media folder or modify the criteria selected in the Match Criteria drop-down box. Not Found indicates that either the criteria are too restrictive for Flame to find a match in the available sources, or that the source is simply not there. The event list displays in red any criteria that cannot be matched.

Unlinked Add sources to the Media folder. Unlinked indicates that the event is not linked to any source. It appears only when no sources are available in the Media folder, and the Conform section of the Media panel displays 0 clip total.

Linked Nothing to do: the event is already linked to source. Linked events appear in the Events list only if **Options ➤ Hide Linked** is not enabled.

5 Once there all the events are linked to sources, you are done with the conform. You can leave the Conform tab.

IMPORTANT No matter the selected criteria, a source cannot be linked to an event if it is not long enough to cover the whole event.

Flame does not automatically cache the relinked source media. To cache the source media: right-click the target and select **Media ➤ Cache Source Media**. You can target a segment on the timeline to cache only that segment, or the sequence itself to cache every segment in the sequence.

RGBA media files and AAF sequences:

- Based on the MediaHub Clip Options for RGBA media files, a conformed sequence shows either a regular segment (if the option is set to Ignore Alpha Channel) or a Matte Container (if the option is set to Create Matte Container).
- Use already imported source clips: the status of the clip (regular or matte container) is then used. This applies to any segment in an AAF, including segments with Matte Key effects.

Available Options

Limit Handles To limit the amount of relinked handles and perform the equivalent of a consolidate on link, enable Limit Handles when linking to the sources. When you need to access more than the consolidated media:

1. Right-click the segment.
2. Select **Media ➤ Unlink**.
3. Disable Limit Handles.
4. Click Link to Media File.

   The full media is now relinked to the segment.

Save Sources To save a copy of the source in the Sources reel, enable Save Sources. The source saved is what is relinked to the segment, including the Limit Handles setting.

Caching Sources Flame does not automatically cache the relinked source media. To cache the source media: right-click the target and select **Media ➤ Cache Source Media**. You can target a segment on the timeline to cache only that segment, or the sequence itself to cache every segment in the sequence.

Filtering Out Tracks Filter out tracks from the Conform list by locking them on the timeline and selecting Filter Locked Tracks from the Options menu. This removes the segments of the locked track from the Conform Events list, and excludes matching sources from the Media Panel's Conform area.

Spanned Clips

You can conform AAF referencing Canon C series spanned MXF clips by using both the Original Sources media options and the Filename as a Match Criteria. Because AAF files use the name of the first file of the clip as file name, automatic linking of media does not work when importing AAF files.

To link your sequence to the original camera spanned clips:

1. Import the AAF file and set **Media Options ➤ Preferred Media** to *Original Sources*.

   If Conform is able to find the media files, events referring to spanned clip will not be matched.
2. Press Set Search Location.
3. Navigate to the location of the media files and press the Set button.
4. Conform matches the clips. You can now relink your sequence.

Conforming a Final Cut Pro XML

If after having imported an FCP XML some segments are still not linked to their media, use the Conform tab to relink both segments and media.

1. Display the FCP XML to conform in the Conform tab. Do one of the following:
   - If you imported the FCP XML using the MediaHub: switch to the Conform tab, right-click the FCP XML and select Open as a Sequence.
   - If you have not already imported the FCP XML: switch to the Conform tab, right-click the Event list and select Load New FCP XML/AAF/EDL.

     From the window that appears, locate and select the FCP XML to conform, and then click Load.
2. Add potential sources to the Conform Media (page 260) folder.

   You now have two lists: the Events list, displaying all the events making up the sequence, and the Conform Media list, displaying the potential sources you selected.

NOTE By default, the Conform Media list displays only the sources matching the currently selected event. Disable **Options ➤ Filter Potential Matches** to view all the possible sources.

3 With **Options ➤ Filter Potential Matches** selected, click a event from the event list. The sources matching that event are displayed in the Potential Matches list.

NOTE You can also select a segment directly in the timeline. Both the timeline and the event list update to reflect the current selection.

4 What you do next depends on the Status column of each event.

Match Select Link Sources from the Linking combo box. Link Matched Sources links all events with the Match status to their unique sources. Match indicates that Flame found only one source from the Sources list that fit the selected Match Criteria.

Multiple Matches Do one of the following:

- Select the event, the source to relink, and then select Link Selected from the Linking combo box.
- Select the event from the event list, right-click the source to relink, and then select Link.

If there are too many sources to choose from, set additional criteria using the Match Criteria drop-down menu; by trial and error you can reduce the number of matching sources. The event list displays in yellow any criteria with multiple matches.

In the event list, if the handles amount for a event is displayed in red, than none of the matching sources have enough handles: This usually happens when the FCP XML references more media than there is media actually available for conform. You can still force a match by manually linking the source to the sequence's event.

No Match Found Either add more sources to the Media folder or modify the criteria selected in the Match Criteria drop-down box. Not Found indicates that either the criteria are too restrictive for Flame to find a match in the available sources, or that the source is simply not there. The event list displays in red any criteria that cannot be matched.

Unlinked Add sources to the Media folder. Unlinked indicates that the event is not linked to any source. It appears only when no sources are available in the Media folder, and the Conform section of the Media panel displays 0 clip total.

Linked Nothing to do: the event is already linked to source. Linked events appear in the Events list only if **Options ➤ Hide Linked** is not enabled.

5 Once there all the events are linked to sources, you are done with the conform. You can leave the Conform tab.

IMPORTANT No matter the selected criteria, a source cannot be linked to an event if it is not long enough to cover the whole event.

Flame does not automatically cache the relinked source media. To cache the source media: right-click the target and select **Media ➤ Cache Source Media**. You can target a segment on the timeline to cache only that segment, or the sequence itself to cache every segment in the sequence.

Available Options

Limit Handles To limit the amount of relinked handles and perform the equivalent of a consolidate on link, enable Limit Handles when linking to the sources. When you need to access more than the consolidated media:

1. Right-click the segment.
2. Select **Media ➤ Unlink**.
3. Disable Limit Handles.
4. Click Link to Media File.

   The full media is now relinked to the segment.

Save Sources To save a copy of the source in the Sources reel, enable Save Sources. The source saved is what is relinked to the segment, including the Limit Handles setting.

Caching Sources Flame does not automatically cache the relinked source media. To cache the source media: right-click the target and select **Media ➤ Cache Source Media**. You can target a segment on the timeline to cache only that segment, or the sequence itself to cache every segment in the sequence.

Filtering Out Tracks Filter out tracks from the Conform list by locking them on the timeline and selecting Filter Locked Tracks from the Options menu. This removes the segments of the locked track from the Conform Events list, and excludes matching sources from the Media Panel's Conform area.

About Connected Conform

Connected Conform is a set of tools created to help you conform sequences that share similar sources and similar contents, either mono or stereoscopic. Aimed at helping you deliver multiple deliverables, it offers you two new types of specialized sequences, the Sources Sequence and the Shots Sequence. The former helps conforming multiple sequences in a single operation, the latter helps consolidate with shot distribution and processing.

Connected Conform is there to help you when you have multiple deliverables that share structure and source media but:

- Are of varying length.
- Require different finishing tasks such as visual effects or titling.
- Use the same media, but in different locations, or of different durations, or even uses different sections of said media.

The Connected Conform tools allow you to:

- Conform a group of imported sequences in a single operation.
- Consolidate media ingest and linking by tying together related metadata of the segments.
- Manage and distribute shots to be worked on from a central location.
- Integrate the worked on shots with the imported sequences for final delivery.

All this can be accomplished using the Connected Conform tools which include:

- Sources Sequence, a specialized sequence to conform multiple sequences in a single operation.
- Shots Sequence, a specialized sequence to help with shot distribution and processing
- Connected Segments, a concept to share information between related segments across different sequences.

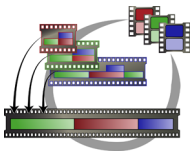
- Reel Group and the Sequences reel, which provides a central location for the Connected Conform workflow.

Connected Conform Workflow Overview

What follows is an example of a Connected Conform workflow. Also provided, links to online videos.

Step 1: Connecting Sequences

Online tutorial: ***Connected Conform Workflow - Conforming & The Sources Sequence***



1. Go to Conform.
2. Import your sequences. Make sure that they are all in a single Reel group's Sequences reel.
3. Click Create Sources Sequence.
4. Go to the created Sources Sequence, and conform it.
   - For added benefits:
   - Enable Limit Handles and specify a number of handles. This option limits the amount of media relinked, but can ensure that there are enough heads and tails to slip around effects.
   - Enable Save Sources. This option facilitates source management later on. And does not impact media cache usage unless you have enabled the Cache Source Media or Generate Proxies import options.

     NOTE If you plan on using the Sources Connection workflow, enable Save Sources.

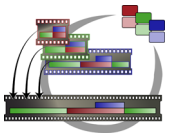
   - Lock a track to exclude its segments from being included in the Sources Sequence.

- Do one of the following
- To work on shots, continue to Step 2: Connecting Segments.
- To work on the sources, see Connecting Sources (page 298).

Step 2: Connecting Segments

Online Tutorial: ***Connected Conform Workflow - Metadata & The Shots Sequence***

Step 2: Connecting Segments



1. Click Create Shots Sequence.
2. From the Shots Sequence, prepare your shots by adding comments or renaming segments. Or, as in the following example, by assigning Shot Names to your segments.
   1. Select all the segments from the Shots Sequence.
   2. Right-click the timeline and select Rename Shot.
   3. Type in the shot name of your choice, but make sure to include the Background Index token. For example: ShotName_<background segment>
   4. Click Rename. And because the Shots Sequence connects the segments across the different sequences, every segment now has a shot name.

      By using the Background Index token you are assured that every background segment has a different shot name. Shot creation relies on segments using the same Shot Name: vertically laid segments with the same Shot Name are part of the same shot.

Lock a track to exclude its segments from being included in the Shots Sequence. Applied Batch FX gets carried over to the Shots Sequence, but might prevent shots from connecting between sequences.

NOTE If you defined a limited amount of handles but need more content from the original source media file:

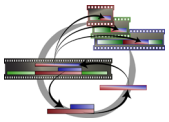
1. Unlink the segment. Notice how every connected segment in the connected sequences is unlinked from its media file.
2. Enable and set Limit Handles to the required number of handles.
3. Using Link to Media File, relink the segment to its media. The segment now sports the new handles.

   This operation is similar to an un-consolidate.

Step 3: Apply VFX and Distribute Shots

Online Tutorial: ***Connected Conform Workflow - Using Batch FX and Batch***

Step 3: Apply VFX and Distribute Shots



1 Work in context of the timeline:

1 Apply a Timeline FX to a segment in the Shots Sequence.

2 Right-click that segment, and select Sync Connected Segments. The Timeline FX is applied to every connected segment across the connected sequences.

- You can right-click any segment from any connected sequence. Metadata connection is not unidirectional from Shots Sequence to other sequences. There is no hierarchy, only links.
- Rendering a segment and then selecting Sync Connected Segments also connects the renders between connected segments.

1 Work on individual shots, in the context of Batch:

1 Select the shots in the Shots Sequence.

Each shot must have a Shot Name.

2 Click Create Batch Group. It creates a Batch Reel for each of the shots. It also creates a new Version track as a placeholder where your shots are rendered.

3 Work on the shot in Batch, and render the shot. It is now available in the Batch Renders reel.

4 Make sure the placeholder segment in highlighted in the timeline.

5 Select the rendered shot and select Media Replace from the Secondary Edit box in Timeline, or press `Ctrl-Shift-R`.

This operation updates every connected sequence with the new media.

NOTE This example assumes that there is only one artist working on the sequences. But you could well distribute the shots (Batch Groups) using the Shared Libraries. Or without creating Batch Groups, you could use the Shot Publish approach to distribute the shots. Both approaches create a Version track and use placeholder segments.

Creating Batch Groups

Once you have created a Shots Sequence, you can, from Conform, create Batch Groups. One batch group is created for each different Shot Name within the selected segments. Each batch group can be seen as a discrete unit of work encapsulated.

Creating a Batch Group:

1 Select the segments for you need to composite in Batch.

The Batch Group can be created from any sequence within a Sequences reel, including from the Shots Sequence.

2 Assign a Shot name to the segments, using the Events list in Conform View, or the contextual menu.

Segments without a shot name do not create a Batch group.

3 Create the Batch Group. Do one of the following:

- In Conform View, click Create Batch Group.
- In Timeline View, right-click the timeline and select Create Batch Group.

One Batch group is created for each unique Shot name, in the location defined by the Batch Group Destination box. Each segment of the shot becomes a clip node inside Batch, stored in a Schematic Reel.

NOTE A BFX applied to a segment becomes part of the clip node when the Batch Group is created. If you open the clip node options, you can use any of the regular Edit BFX, Explode FX, or Convert to 3D Comp options.

When you use Create Batch Group in Conform, that Batch group's Render node uses the following settings:

- Its attributes, such resolution, source timecode, or tape name, are taken from the back plate. The node is also connected to the back plate.
- Its In/Out marks are from the original timeline segment.
- Its name is set to the Shot Name, or by the Default Render Node Name preference.

When you create a Batch group, a related segment is created on a new Version track in every sequence connected to Shots Sequence by the connected segments. By using Replace Media, this related segment can be replaced with the result of the Batch group, with every connected sequence automatically inheriting the replacement.

Once the work is done in Batch:

1. Render the result.

   The rendered clip, with the original clip's metadata, can be found in the Batch Renders reel.
2. Open the Timeline view.
3. In the Shots Sequence, on the new Version track created previously, select the related segment to be updated.
4. Select the rendered clip from the Batch Renders reel.
5. Select **Replace Media** from the **Secondary Edit** box. The new media is now available across all connected sequences.

Selecting where Batch Groups are created:

- Use the Create Batch Group Destination box to decide where the Batch Groups are created.
- You can choose to create Batch Groups in a new library, a new shared library, or as a series of batch group of reels.
- Pick Destination allows you to select a destination when you create the Batch groups.

NOTE By default, the Batch Group created starts at the frame matching the source frame number of the background element. Enable **Preferences ➤ Timeline FX / Batch / BFX ➤ Ignore in Publish/Conform** to have the Batch Group start at frame 1.

Operations To Avoid When in Connected Conform

When working in the context of Connected Conform, avoid the following:

Applying a Gap FX Any created Gap FX is local to the sequence where it is created; a Gap FX is not connected between connected sequences.

Applying a Batch FX to vertically composited segments The flattening of the vertical structure require by a Batch FX disables segment connections since it destroys the spatial information for the concerned segments. Use instead the Create Batch Group workflow, where such information is preserved.

NOTE Applying a Batch FX to an individual segment is a supported operation in the context of Connected Conform.

Connecting Sources

Use Source Segment Connection to set Connected Segment from a source to other instances of this source within the same Reel Group. For example, perform your colour corrections from the Sources Sequence Once

done, propagate the changes to every instance of the source you worked on, to every sequence in the Sequence Reel.

1. In the Sources Sequence, right-click the segment with the source media that you need to connect and select Create Source Segment Connection.

   This connects together every segment, in the Sequences reel, that uses the same source media.
2. Use the Sources Sequence as a working sequence.
3. Use **Sync Connected Segments** from the contextual menu to propagate the effects to you deliverables' sequences.

The scope of **Create Source Segment Connection** can be set from the contextual menu's Source Segment Connection Options:

- Sequences Only: Enable this option to connect only segments located in the Sequences Reel. Disable this option to extend the scope of the connection to the whole Reel Group, including the Sources reel.
- Overwrite Existing Connections: Enable this option if connections already exist but you want to overwrite them with new ones. For example, you duplicated multiple times a segment referencing the same source, but now you need these connections to share the same source segment.
- Sync Up: Enable this option to synchronize the segments as soon as the connection is created. This automatic sync only happens when the connection is first created: after this, you need to manually synchronize the segments.

Adding a sequence to sequences already connected through Source Connection:

1. Verify that the sources for the already imported sequences are all saved in the Sources reel.

   If the sources are not available in the Sources reel,
2. Add a sequence to the Sequences reel.
3. Disable the Save Sources option.
4. Link to the sources located in the Sources reel. This automatically adds Source Segment Connections to the new sequence, complete with the link icon.

For this operation to work properly, you must make sure that Save Sources was enabled and that the sources were all imported to the Sources reel the first time you created the Sources Sequence. And when you import and relink the new sequence, the Save Sources option must be disabled. Because Flame assigns a different ID to a source every time it is imported, and because the Source Connection relies on that ID to create the connection,

---

NOTE Do not mix Sources and Shots connections. If a Shots Sequence exists in the Reel Group, take precedence over the Sources Sequences: it owns all of the Segment Connections. If you try to link media to a newly added sequence while a Shots Sequence exists in the Reel Group, there can be no Source connections, only Shot connections.

About Sources Sequences

The purpose of the Sources Sequence is to simplify the conform of multiple sequences: in a single operation, you can link all the sequences of the Reel group to their sources.

You create a Sources Sequence with the **Create Sources Sequence** button, in Conform. Only one Sources Sequence can exist in a given Reel Group. Creating a Sources Sequence where one already exists overwrites the existing one.

A Sources Sequence is the result of the analysis of the sequences located within the Sequences Reels of a Reel Group: it is an aggregate sequence of all the sources present in the sequences. The Sources Sequence enables the sharing of sources across the connected sequences.

The Sources Sequence contains a single video track (and as many audio tracks as required). This track contains the longest instance of source media referenced by the connected sequences, with the order of the segments based on Tape or File Name, then Timecode (also known as EDL C-Sort mode).

NOTE One could distribute the Sources Sequence for source grading; relinking to the new, graded, sources will update all of the connected sequences.

Information Shared Between Connected Sequences

The creation of a Sources Sequence connects together the sequences used to create it. This allows you to use the Sources Sequence to perform a number of source operations that are automatically carried to every connected sequence.

Operations that propagate to all sequences connected by the Sources Sequence:

- Linking and unlinking segments from their media.
- Caching and flushing source media cache.
- Generating proxies.
- Editing source clip metadata, such as source timecode, tape name, or any format option such as debayering settings.
- Editing format specific options such as R3D or ARRIRAW options.
- Editing source clip pre-processing options, such as LUT or resize.

NOTE The segments of the Sources Sequence are not connected to segments in the original sequences, only the sources are connected. You need the Shots Sequence to connect segment metadata.

How the Sources Sequence Is Created

Since the goal of the Sources Sequence is to simplify the conform of multiple related sequences, the Sources Sequence creation process tries to consolidate the different sources from the many sequences to simplify the Sources Sequence. It does this using the following steps.

1. It find the longest sequence of the group.
2. It determines how many different sources are referenced by the longest sequence.
   - The Sources Sequence creation process finds out which one of Tape Name, File Name, and Name has the most different occurrences. For example, if out of 10 segments there are 3 different Names, 4 different File Names, and 6 different Tape Names, then 6 different sources are considered to exist.
   - If two segments refer to the same source and:
     - If the start timecode of one event is within 100 frames of the end timecode of the other event, Flame consolidates those 2 segments into a single one in the created Sources Sequence.
     - If the start timecode of one event is at least 101 frames from the end timecode of the other event, Flame considers that the segments are unrelated, and creates 2 independent segments in the Sources Sequence.
3. Once the longest sequence has been analysed, the process analyses the other sequences, trying to consolidate the sources together using the same criteria as above.

When all the sequences and sources are consolidated and connected, the Sources Sequence is built, its segments ordered by Tape or File Name, and then Timecode (also known as EDL C-Sort mode).

In addition, if Limit Handles is enabled and the sequences are unlinked, Flame adds the specified amount of handles to the segments in the Sources Sequence, or less if the specified interval of a source's timecodes does not allow it by being too short.

Finally, the segment names used for the Sources Sequence are the one from the original sequences, not the media file name.

Managing Sources in Connected Conform

When working in Connected Conform, creating a Sources Sequence connects together the different sequence by the used sources. This means that changing the Format Options or the Pre-Processing settings on a segment in one sequence affects every other occurrence of that source.

To prevent this from happening, you can create an new and exclusive link between the segment you wish to isolate and its source by duplicating that source.

To duplicate a source:

- Right-click the segment with the source to isolate.
- From the contextual menu, select **Duplicate Source**.

Using Duplicate Source does not duplicate any media: it only ensures that the segment now has independent Format Options and Pre-Processing Options from the other connected segments. Also, this does not break Segment Connection enabled by Connected Conform: you can still Sync Segments across sequences. Finally, a clip derived from a segment to which you applied a Duplicate Source (through Copy & Paste, Duplicate, or simply a cut) behaves as usual, in that it shares the same source as the clip from which it is derived.

To see how Duplicate Source works:

1. In Connected Conform, with the Sources Sequence created, Link to Sources making sure Save Sources is enabled.
2. Select a segment connected across the connected sequences, and through the Pre-Processing options, apply a visible letterbox resize.

   Verify that this affects all connected segments across the connected sequences.
3. Apply Duplicate Source to that same segment.
4. Unlink that same segment.
5. With Save Sources still enabled, select Link To Media Files.

   In the Sources reel, you can now see a duplicate source gets added. This is because Duplicate Source creates a new link to the source, forcing Flame to consider it as a completely different source. And if you disable the Letterbox resize from the Pre-Processing options, it only affects this segment.

About Shots Sequences

A Shots Sequence is the result of the analysis of the sequences located in Sequences Reels of a Reel Group. It detects similar contents, and creates an aggregate sequence of all the segments from the connected sequences.

You create a Shots Sequence with the **Create Shots Sequence** button, in Conform. Only one Shots Sequence can exist in a given Reel Group. Creating a Shots Sequence where one already exists overwrites the existing one.

It allows you to break down shots for visual effects work: to name, to separate and to work on VFX shots, either for solo work, or for collaborative shot distribution. You work on the shots knowing that the work propagates, through connected segments, to the connected sequences. A Shots Sequence is required to use Create Batch Group in Conform.

A Shots Sequence is similar to a master timeline:

- It contains the longest instance of a shot from connected sequences.

- It maintains the vertical structure of shots, as well as the logical story order.
- It creates links between segments of connected sequences, enabling the connection of metadata (including VFX) between related segments.

TIP In Conform, when you modify a connected segment, it is selected in every sequence where it is found. This makes it easy to see the modified segment across the different sequences using the sequence tabs.

A Shots Sequence preserves original timewarps, Batch FX, as well as Containers and Matte Containers. Note that in the case of both containers, only the segments they contain are connected, not the containers themselves.

Information Shared Between Connected Segments

While the Sources Sequence connects only source clip metadata, a Shots Sequence connects source *and* segment metadata with all connected sequences.

Metadata connected between the Shots Sequence and the connected sequences:

- Source clip metadata
- Segment Name
- Shot Name

  NOTE If you use Shot Naming with the Segment Index token, the index used is that of the sequence where the Shot naming is being done. It propagates to all other sequences using the same segment, using the index of the sequence used for naming, not the index of each sequence.

- Comment
- Timeline FX (including Batch FX).

TIP Consider always doing any bulk operation from the Shots Sequence: this ensures that modifications include every segment because the Shots contains all the segments from every connected sequence.

How the Shots Sequence Is Created

In the Shots Sequence, the segments are ordered based on the story order of the longest sequence. Segments found in other connected sequences but not in the longest sequence are added sequentially at the end of the Shots Sequence. If two or more sequences share the title of "longest sequence", then the tied sequences are ordered alphabetically, and the first one is used as the reference sequence.

In addition, Batch FX are also used as a matching criteria when grouping segments from the different connected sequences: if two segments are match in every aspect except for the presence a Batch FX, they will result in different segments in the Shots Sequence (but those segments in the Shots Sequence will each have a version of the Batch FX).

In the Shots Sequence creation process, Shot names impact the structure of the Shots Sequence:

- If Shot naming is done *after* the creation of the Shots Sequence, its segments are organized based on their location in the original sequences. This is the recommended approach.
- If Shot naming is done *before* the creation of the Shots Sequence, the structure of the created sequence takes into account the Shot naming: segments sharing sources but with different shot names are duplicated for each differing shot name. If you use this approach, make sure that every segment of every connected sequence has a shot name to ensure predictability of the resulting Shots Sequence.

The approach to use depends on your goal: the latter is useful when working with sequences that may share the same media but require different visual treatment, such as delivery of localized content. The former creates a more compact structure, easier to manage, aimed at creating uniform visual treatment.

The segment names used in the Shots Sequence are the ones from the original sequences, not the media file name.

Working with Connected Segments

A Shots Sequence connects the segments of the sequences in the Reel group. This in turn allows you to modify metadata, like Shot Names, in one sequence and have that change propagate to the other sequences connected by the Shots Sequence.

Connected Segment Icon A segment connected across multiple sequences is identified with

.

Navigating Instances of Connected Segments `Right-click` a segment and select Jump to Connected Segment: the drop-down list displays the sequences with an instance of that segment. Selecting a sequence from the list opens that sequence, with the segment selected.

Syncing Connected Segments Modifications made to the Timeline FX, Batch FX, or History of a given segment are local to that segment until you sync it. For example, when working with the Shots Sequence.

1. In the Shots Sequence, apply a Timeline FX on a Connected Segment or on a selection of Connected Segments.
2. From the contextual menu, select Sync Connected Segments.
3. All corresponding Connected Segments located in the connected sequences in the Reel Group are updated.

   TIP Render all of the updated Timeline FX for all synced Connected Segments by rendering the original segments Timeline FX before syncing.

Notes:

- You can undo this sync.
- You can sync a selection of connected segments. You can sync a segment from any sequence part of the Reel group. When working with a Shots Sequence, sync from the Shots Sequence, since this sequence contains every connected segment.
- The number of connected segments that were synced appears in the Message Console.

Replacing Connected Segments To replace a connected segment across the connected sequences of a Reel group:

1. Open Timeline.
2. Select the segment to be replaced.
3. Select the clip that will replace the segment.
4. Do one of the following:
   - To replace the segment and overwrite the Timeline FX applied to the destination segment, use the Timeline Replace operation. Press `R`, or click the Replace button.

   - To replace the segment and preserve the Timeline FX applied to the destination segment, use the Timeline Replace Media operation. Press `Ctrl-Shift-R`, or click the Secondary Edit box and select Replace Media.

5 All instances of the connected segment are updated.

NOTE The media used must be long enough to cover the whole of the segment it replaces, or this will fail.

The Replace operation can be performed on any clips, whether they have Timeline FX applied, a Batch FX, or include History.

Removing Segment Connections Sometimes you need to isolate a segment to perform a unique edit, such as applying language-specific titling to multi-language deliverables; the connection of segments' metadata becomes a hindrance. In this case, you want to remove the segment's connection to other segments.

1 Select the segment to isolate.
2 From the contextual menu, select Remove Segment Connection.

IMPORTANT While you can Undo this operation, there is no Restore Connection operation that can be performed later. Although you can create new connected segments.

Connecting Segments Across Sequences

You can create a connected segment, use that segment across multiple timelines, and then sync the connected segments to propagate changes across timelines, without using the Shots Sequence and Connected Conform workflow.

1 Create a new Reel Group.
2 Move the sequences that will be using the same connected segment to that new Reel Group.
3 Open the sequences.
4 In one the sequences, select the segment you want to connect across your sequences.
5 Right-click and select **Create Segment Connection**.
6 Right-click and select **Copy**.
7 Paste the segment the other sequences.

You can now use the same tools available to you in Connected Conform: sync segments, navigate between connected segments, propagate Timeline FX across connected segments.

Shots Sequence is the best solution when working in a multi-sequence conform workflow, where you need to keep many segments in many sequences in sync.

Clip Input/Output Using a VTR

Flame allows you to perform numerous VTR-based clip input and output operations. You can capture individual clips or frames, or log clips for capture using an EDL. Similarly, the application allows you to output single clips or frames, as well as multiple clips to a VTR device.

Clip input and output general workflow:

1. Make sure all hardware devices involved in the clip input and output process are properly configured. If not, the VTR Input and VTR Output menu options are disabled.
2. Edit the software initialization configuration file or run the Flame Setup application to ensure the proper devices and settings are initialized on application start-up.
3. Do one of the following:
   - Input clips.
   - Output clips.

If you are using an NVIDIA graphics card with and SDI daughter card, Flame also supports real-time operations for some deliverables.

Configuring Hardware For Clip Input and Output Using a VTR

When preparing for a VTR session, confirm the following:

- Audio and video outputs of the VTR are connected to the audio and video inputs of the Flame system. Audio and video inputs of the VTR are connected to the audio and video outputs of the Flame system.
- The VTR is connected, using an RS-422 video I/O control cable, to enable its remote control from the workstation.
- A video sync signal is connected to the sync input of the video device to ensure frame-accurate capture. If there is a separate audio device, an audio sync signal must be connected to it as well.

Configuring Mac Hardware

Your Mac must be equipped with either an AJA or a Blackmagic Design device to connect to a VTR.

Consult the online *System Requirements* to see which devices are supported. The actual supported features depend on the device used, but generally includes SDI Capture, SDI Playout, and SDI / HDMI Preview, and 3G I/O when available.

Configuring Software For Clip Input and Output Using a VTR

Software configuration in preparation for clip input and output session involves editing the software initialisation configuration file on a Linux system, or using the Flame Setup utility on Mac system.

On a Mac system, in the Flame Setup application, in the VTR tab, enable every VTR you plan on using with Flame. Also verify the Preview tab for the timings you want to make available to your projects. If you do not have a sync connected to your workstation, you might want to define for your project some free run timings. They are turned off by default.

When you are preparing for a clip input and output session on a Linux system, consult the software initialization configuration file to confirm the following keyword sections are uncommented and properly configured:

- VIDEO KEYWORD

  You do not need to specify the default video I/O timing. The initial timing corresponds to the default resolution and frame rate of the project that you select on start-up. If you select a film or custom resolution for your project on start-up, video I/O timing corresponds to the resolution that was selected as the preferred resolution when the application was installed.
- VTR KEYWORD

You must assign a specific VTR timing, such as NTSC, PAL, or 1920x1080_5994i. Multi-format video input and output capability of your system allows you to modify the VTR timing, without exiting the project, to any timing supported by your system's video board.

If you plan to perform input and output operations using a live video signal from a broadcast feed or from a device that does not support remote control functionality using RS-422 interface, such as a VCR or camcorder, uncomment the corresponding Live video keyword at the bottom of the SD DECKS section.

NOTE When working in HD projects and capturing material from a VariCam (variable frame rate 720p material), ensure that the following VTR are enabled: *VTR DVCProHD 720 59p, SERIAL1, 1280x720_5994P* and *VTR DVCProHD 720 60p, SERIAL1, 1280x720_60p*.

Inputting Clips From a VTR

Once you have set up your hardware devices and configured the software initialization file, you are ready to perform clip input and output. The VTR Input module contains various controls to help you perform operations, such as capturing a single frame, capturing a single clip, capturing and logging multiple clips.

Accessing the VTR Input Module

1. Select a folder from the Media Library. This is where the captured clips will be created.
2. Open the Tools tab.
3. In the Utilities tab, select Clip VTR Input.

   The VTR Input module appears.

If VTR Input is not available, connect your workstation to a VTR, and configure Flame to use the VTR.

Capturing Media with a VTR

You can capture a single clip from a VTR. You can capture on the fly with your in and out points determined by mouse clicks during playback, or be more precise by setting in and out points in the corresponding fields.

A single frame capture, or a frame grab, is the simplest clip input operation. This is useful for extracting snapshot frames for a preview or promotion piece.

To capture a single clip:

1. Cue up the tape to the frame at which you want to begin capturing.
2. Do one of the following:
   - From the Capture Method box, select Start On Click/Stop On Click.
   - Enter clip input in and out points in the In and Out fields.
3. To begin capturing, click Process.

   The timecode field turns green, indicating that capture is in progress. Depending on your hardware configuration, the preview window may go black during capture.
4. To end the capture at any time, click the cursor anywhere over the preview window.

   Depending on your project proxy settings, a post-process may occur, generating proxies for each captured frame. You can see a notification on the process that is taking place in the message bar.

   Once all capture-related processes are complete, the clip is saved to the Workspace location that you selected before you entered the VTR Input module.

To capture a single frame:

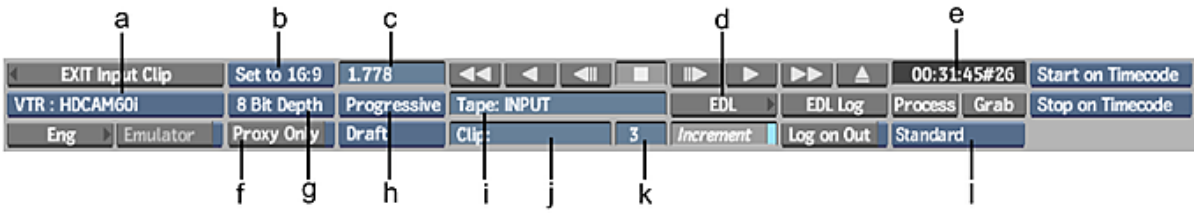
1. Cue up to the frame you want to capture using the VTR Transport controls or by scrubbing the preview window.
2. Click Grab.

   The captured frame is saved in the Workspace.
3. Click EXIT Input Clip to view the captured frame.
4. If you want to compare the captured frame against the frame in the preview window, load the clip into the VTR Input module and enable split-screen view.

   NOTE If you notice that the previewed image is different from the captured one, edit the Video Input Delay field found in the Engineering menu and recapture that frame. If the two images now match, the delay is fine. If they do not match, you need to tweak the Input Delay until they do.

VTR Input Menu Options

The basic VTR Input controls are described as follows.



(a) Device Name box (b) Aspect Ratio box (c) Aspect Ratio field (d) EDL button (e) Current Timecode field (f) Proxy Quality box (g) Bit Depth box (h) Scan mode box (i) Tape Name field (j) Clip Name field (k) Increment Name field (l) Tape Type box

Device Name box Select the VTR and timing combination to use to capture clips. The available VTR are defined in the software configuration file using the VTR keyword.The available VTR are defined in the Smoke Setup application.

Engineering button Opens the engineering menu.

Aspect Ratio box Select the aspect ratio of the clip to capture.

Aspect Ratio field Displays the aspect ratio of the clip to capture. Editable.

Bit Depth box Select the bit depth used to capture the clip.

Scan Mode box Selects how to flag the captured clip: Progressive, Field 1 dominant, or Field 2 dominant. The flag is only there as a reminder; for example, setting Field 2 does not prevent you from de-interlacing on Field 1. In most cases, select the option that matches the format you are capturing.

Proxy Only button Enable to capture proxies only. In this case, the high-resolution media is captured, proxies are generated, but the high-resolution media is discarded. As a result, much storage space is required.

Only available if the current project is configured to use proxies. If your proxies are set to be generated as a post process in the Engineering menu, you can see an estimated time required for proxies generation in the message bar. You can abort this process at any time by clicking anywhere on the screen. When you click

the screen, a message appears asking you to confirm the operation. Only the captured media that corresponds to completed proxy generation is preserved. Any captured frames for which no proxies have been generated are purged.

Proxy Quality box Proxies generated during capture are always of draft quality. Using the Proxy Quality box, you can set the default quality for proxy generation after capture. Results vary depending on the type of clips involved, so it is a good idea to try different settings. Only available if the current project is configured to use proxies.

Select:	To get:
Impulse	Quick, low-quality results.
Triangle	Moderate results with little processing overhead.
Mitchell	Best results when resizing a clip to a higher resolution.
Bicubic	Very good results for resizing soft-looking images. Use to sharpen the image.
Quadratic	Good results for resizing simple images with straight edges. Similar to Gaussian but with more blurring. Use to soften the image.
Gaussian	Excellent results when resizing a clip with no patterns and alot of straight edges to a lower resolution. Useful for softening some detail.
Shannon	Excellent results when resizing a clip to a lower resolution. Very similar to Lanczos, but results are a little softer.
Lanczos	Best results when resizing a clip containing a variety of patterns and elements to a lower resolution. It is the most complex with the longest processing time.

Emulator button Enable to have the application emulate a VTR.

Capture button Starts the capture.

Grab button Grabs the current frame.

Input Type box Select the type of footage found on the tape. Use Standard for regular capture. 2X can only be used with HDCAM SR. Dual Image is for stereo footage recorded side-by-side on the tape; use Slice to create a single Stereoscopic clip on capture, but with half the horizontal resolution; use Scale to create a single Stereoscopic clip, but resized to full horizontal resolution. With Scale, use Engineering > Dual Image Resizing Filter to select the quality of the resize.

EDL Log button Logs the clip to the EDL.

EDL button Opens the EDL menu where you can capture and edit EDLs.

Log On Out button Enable to log EDL events every time you enter an out point.

Tape Name field Displays the name of the tape from which you are capturing. This name is important for EDL assembly and media recapture procedures. Editable.

Clip Name field Displays the name to use for the clip to capture. Editable.

Increment field Displays the number automatically appended to the clip name. Active when Increment is enabled.

Increment button Enable to append numerical increments to the clip name automatically. For example, enter “My_Clip” in the Clip Name field and then enable Increment Name. The first clip you capture is named “My_Clip-1”, the second is named “My_Clip-2”, and so on.

Start Mode box Determines the start mode for clip input.

Select:	To:
Start On Click	Capture starting from the currently displayed frame on the tape. Click Process to activate the start-on-click trigger.
Start On Timecode	Capture starting from the timecode you enter in the In field. Click Process to start the clip input process.

Stop Mode box Determines the stop mode for clip input.

Select:	To capture until:
Stop On Click	You click anywhere on the screen.
Stop After Frames	A specific number of frames have been captured. When you select this option, a field appears in which you enter the number of frames you want to capture.
Stop On Timecode	A timecode on the tape has been reached (entered in the Out Timecode field).

Current Timecode field Indicates the current timecode of the tape in the VTR.

The following controls are found on the right side of the menu (not shown in preceding illustration).

In Timecode field Displays the timecode on the tape at which point the clip input begins. Editable.

Out Timecode field Displays the timecode on the tape at which point the clip input process ends. Editable.

Duration field Displays the duration, in timecode, between the clip in point and out point. Editable.

Tape EE button Click to toggles E-to-E on and off. When lit, indicates that the VTR is in E-to-E mode (electronic to electronic). This means that the VTR output is showing its input signal.

Standby button When lit, the VTR is in standby mode. Click to toggle between standby modes.

Cue In button Cues the VTR to the value of the In Timecode field.

In button Sets the In Timecode field to the current VTR timecode.

Cue Out button Cues the VTR to the value of the Out Timecode field.

Out button Sets the Out Timecode field to the current VTR timecode.

Toggle Audio Tracks button Switch between audio banks of 4 audio tracks.The actual number of tracks depends on your capture device.

Video Track button Enable to capture the video track.

VTR Status display Indicates the current status of the VTR.

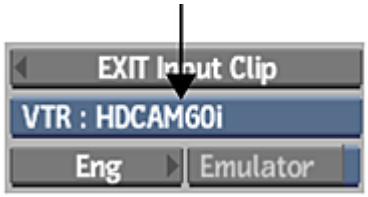
Selecting a VTR Device For Input

The VTR devices available depend on the VTR keyword lines uncommented in the software initialisation configuration file. Make sure the VTR you select is appropriately connected using the corresponding audio, video, and RS-422 connections.

In certain situations, Flame pre-selects a VTR with appropriate timing for you.

To select the VTR device for input:

1 From the Device Name box, select an option corresponding to your VTR device.



2 Make sure that the selected device is in Remote mode.

Naming Tapes and Clips

Proper media management is an essential part of the clip input and output process. One of the elements of efficient media management is methodical and consistent naming of the clips you capture and the tapes from which they originate. Before capturing a clip, assign a name to the tape and the resulting clip. This assignment makes it easier to organize your clips, trace them back to the source tape, and, when necessary, recapture them.

To name a tape and clip:

1 In the Tape field, enter the tape name.

The tape name is stored with the clip. In the Workspace, you can `Alt`-click the clip to view its information, including the tape name.

2 In the Clip field, enter a clip name.

3 Enable Increment to use the same name for multiple clips captured in sequence. This appends a numerical tag at the end of each clip so you can differentiate them.

4 To change the number of the numerical tag for the next captured clip, enter a number in the Clip Index field. By default, the increment begins at 1.

Selecting Tracks and Channels to Capture

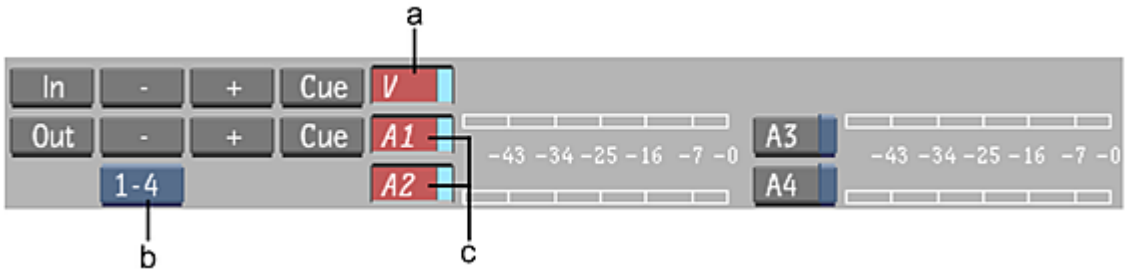
Whether you are inputting single frames, single clips, multiple clips or conforming EDLs, you must select the tracks and channels you want to capture from the VTR.

To select the tracks and channels to capture:

1 To capture the video track, enable V.

2 To switch between audio tracks banks, use the Toggle Audio Tracks button.

3 To capture audio channels, enable the corresponding channel selection buttons.



(a) Video track enabled for capture (b) Toggle Audio Tracks button (c) Audio channels 1 and 2 enabled for capture

Audio meters display the gain level being captured. You cannot change this level during capture. You can modify the gain of a clip once capture is complete.

HDCAM SR Double-Speed and Stereo Tape Capture

Using an HDCAM SR, you can capture material from specially formatted double-speed and stereoscopic tapes.

Double-speed tapes allows you to capture material twice as fast. Stereoscopic tapes essentially stores in an interlaced timing two progressive clips; a 60i (50i) "clip" contains two 30p(25p) clips.

This feature does have the following limitations:

- To use this feature, you must use specially formatted tapes. If you insert a regular tape in the HDCAM SR and try to capture it as double-speed or stereoscopic material, the capture fails.
- Audio monitoring is not available during capture.
- When capturing stereo tapes, only audio channels 1 through 8 are available.

To capture material recorded at double-speed:

1. Ensure that the HDCAM SR is connected to the AJA card using a dual-link.
2. Set the HDCAM SR VTR to DBL 422.
3. From the Device Name box, select the HDCAM SR VTR.
4. From the Input Type box, select 2x-DOUBLE.

NOTE If the player displays the clip with some colour bias, it is because the player falsely interprets the 4:2:2 signal from the VTR as a 4:4:4 signal. This does not impact the capture; the stereoscopic clip will be captured without that bias. To remove this bias, go to the Engineering menu and set the Input Connection box to Serial 1 4:2:2.

5. Capture the clip. See Capturing Media with a VTR (page 306).

To capture material recorded on stereoscopic tapes:

1. Ensure that the HDCAM SR is connected to the AJA card using a dual-link.
2. Set the HDCAM SR VTR to the stereoscopic setting.
3. From the VTR Input menu, select the HDCAM SR VTR from the Device Name box.

4 From the Input Type box, select 2x-STEREO.

NOTE If the player displays the clip with some colour bias, it is because the player falsely interprets the 4:2:2 signal from the VTR as a 4:4:4 signal. This does not impact the capture; the stereoscopic clip will be captured without that bias. To remove this bias, go to the Engineering menu and set the Input Connection box to Serial 1 4:2:2.

5 Capture the clip. See Capturing Media with a VTR (page 306).

The stereoscopic material is captured as a single, regular stereoscopic clip, with two layers, one for each eye.

Outputting Clips To a VTR

Once you have clips that are ready for output to tape, use the VTR Output module to perform this operation. Like capturing, you can output single clips or multiple clips, and enable a split view to preview the clip you want to output alongside the media on the tape simultaneously.

Another way to output multiple clips is to generate an EDL and then output the clips using the EDL.

Most of the procedures in this section assume that you have already blacked your tape. However, if necessary, you can output in assemble mode, which allows you to perform clip output to a tape that was only partially blacked. See Outputting Clips in Assemble Mode (page 323).

Accessing the VTR Output Module

1 Open the Tools tab.
2 In the Utilities tab, select Clip VTR Output.
3 Select the clip to output from the Media Library, `Ctrl+click` to select multiple clips. You can also select a folder if you wish to output multiple clips in one session.

The VTR Output module appears.

If VTR Output is not available, connect your workstation to a VTR, and configure Flame to use the VTR.

VTR Output Menu Options

You can view the VTR Output menu in large or small format depending on which tab is selected.

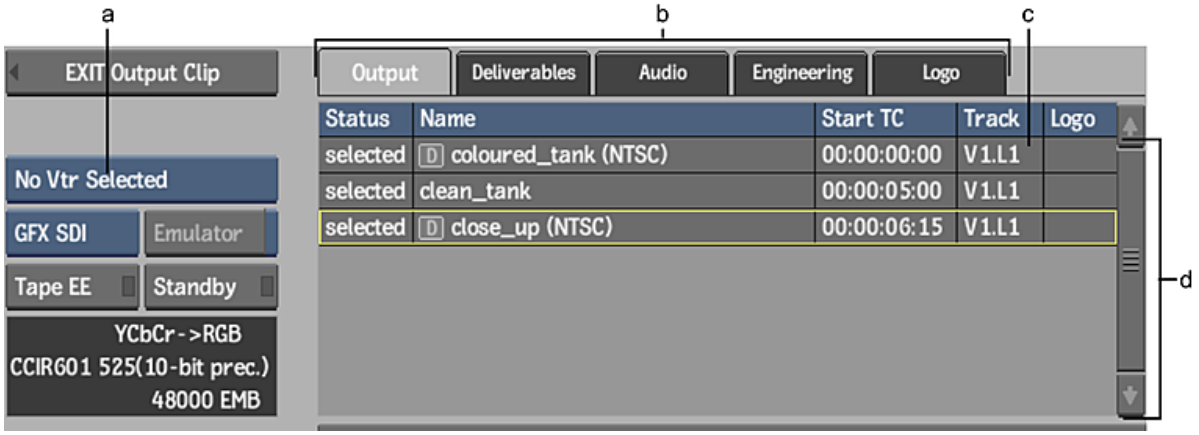
When one of the Output, Audio, or Engineering tabs is selected, both the large and small formats are available. To toggle between the large and small formats, Ctrl-swipe the bottom of the screen.

When the Deliverables or Logo tab is selected, only the large format is available, but additional Pan and Zoom controls are available.

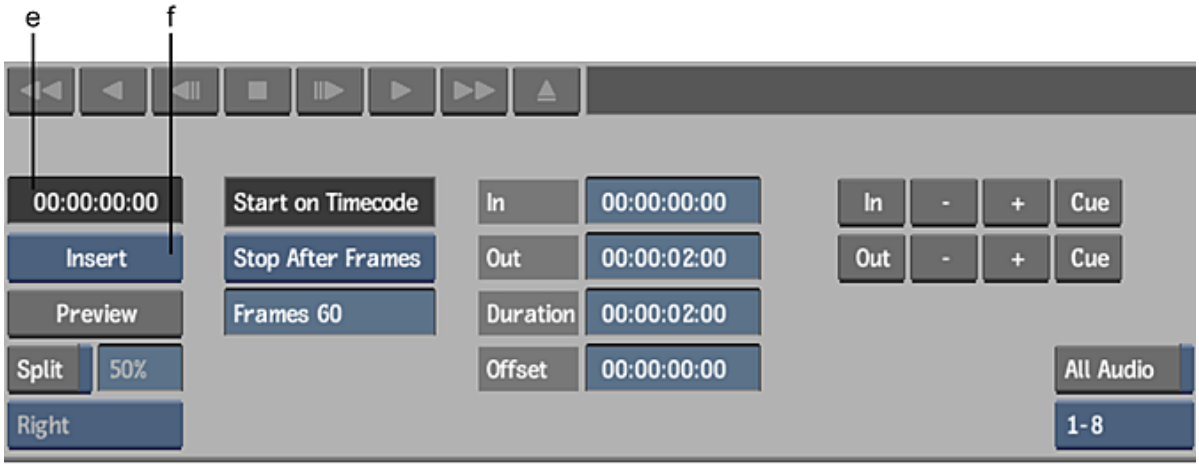
If you are using the large Output menu with an HD clip, the menu automatically switches to the smaller format during clip output, and then switches back when output is complete. This gives you an unobstructed view of the clip during output.

Small VTR Output module (left portion):

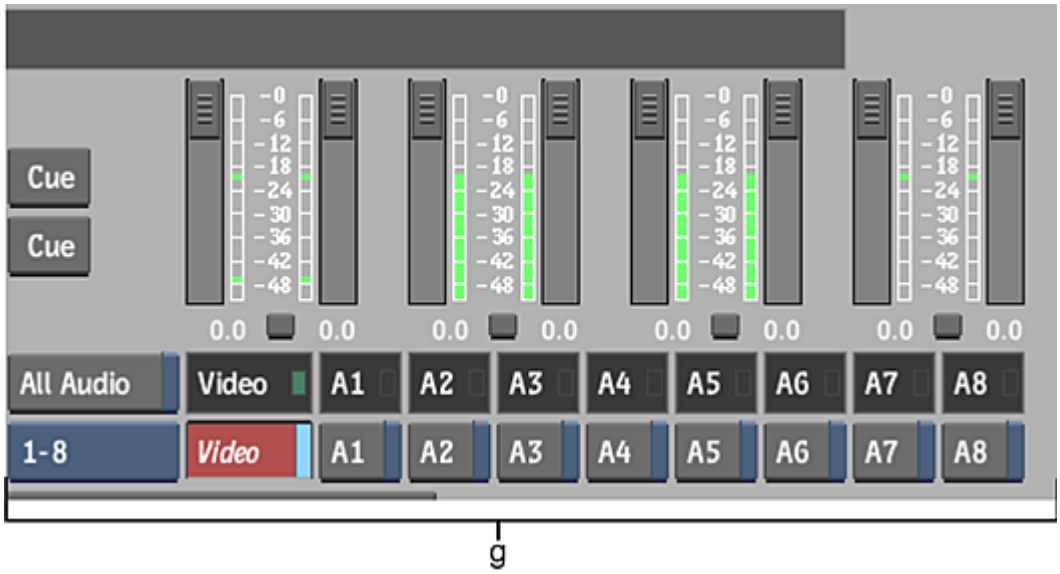
Large VTR Output module (broken into three parts):



(a) Device Name box (b) Navigation tabs (c) Video Layer field (d) Output list



(e) Current Timecode field (f) Output box



(g) Audio controls

In this documentation, the large menu is described. Differences present in the small menu are noted where applicable.

Device Name box Displays the options for each uncommented VTR keyword line in the software initialisation configuration file. Select the option corresponding to the VTR to which you want to output clips.

Navigation tabs Switch between different VTR Output tabs.

Select:	To:
Output	Configure the output settings described in this section. This is the default menu.
Deliverables	Set up Real-Time Deliverables on output, such as Letterboxes or LUTs.
Audio	Set Audio preferences. Changes are reflected in the Audio section of the Preferences menu, and vice versa.
Engineering	View the VTR Output module Engineering menu.
Logo	Set up a logo on output.

NOTE The Deliverables and Logo tabs are only available if the workstation uses an NVIDIA Quadro FX 5600 SDI graphics card.

Output list Displays information about the clip selected for output. If there are multiple clips, you can sort them by clicking the column headings. This changes the output sequence order. You can also edit the Timecode field in this list.

If the list includes Deliverables and you do not have the hardware required by Real-Time Deliverables, the Deliverables are greyed out.

Video Layer field Indicates the track to output in a multi-track clip. Drag the field to browse through the video tracks and versions. This field is red when the selected track is not the top track of the selected video version; this does not prevent output.

Graphics Card box If you have the NVIDIA Quadro FX 5600 SDI graphics card for using Real-Time Deliverables, switch between it and the standard AJA_OEM2K card using this box.

Tape EE button When lit, indicates that the VTR is in E-to-E mode (electronic to electronic). This means that the VTR output is showing its input signal. When E-to-E is off, the VTR shows the contents of the tape it contains. Click this button to toggle E-to-E on and off.

Standby button When lit, indicates that the VTR is in standby mode. Click this button to toggle between on and off.

VTR Status display Indicates the current status of the VTR.

Current Timecode field Indicates the current timecode of the tape in the VTR.

Output box Switch between insert or assemble mode. Click to perform the selected action.

Preview button Triggers a simulation of the output process. The VTR behaves as if it is inserting material, however no material is recorded to tape.

Split View button Enable to simultaneously monitor the clip selected for output and the contents of the tape.

Start On Timecode field A locked field indicating that clip output begins at the timecode entered in the In Timecode field.

Stop Mode box Determines the stop mode for clip output.

Select:	To output the current clip until:
Stop On Timecode	A timecode on the tape is reached (indicated in the Out Timecode field).
Stop After Frames	A specified number of frames is output. When you select this option, a field appears in which you enter the number of frames to output.

In Timecode field Indicates the timecode on the tape at which point the clip output process begins.

Out Timecode field Indicates the timecode on the tape at which point the clip output process ends.

Duration field Indicates the duration, in timecode, between the clip output in and out points.

Offset field Indicates the offset, in timecode, by which the selected clip is output. For example, an offset of 00:00:00:05 indicates that the first five frames of the clip to be output are skipped. The first frame to be output is frame 5 of the clip (counting frames from 0).

In/Out Point controls Use to enter, adjust, and cue the in and out points.

All Audio button Enable to output all audio channels for monitoring, even if only some audio channels are enabled for recording to tape.

Audio Channel buttons Enable and control audio output signals.

Selecting a VTR Device For Output

The VTR devices available depend on the VTR keyword lines uncommented in the software initialisation configuration file. Make sure the VTR you select is appropriately connected using the corresponding audio, video, and RS-422 connections.

Flame pre-selects a VTR with appropriate timing when entering the Output clip module.

To select the VTR device for output:

1. From the Device Name box, select a VTR device.
2. Make sure that the selected device is in Remote mode.

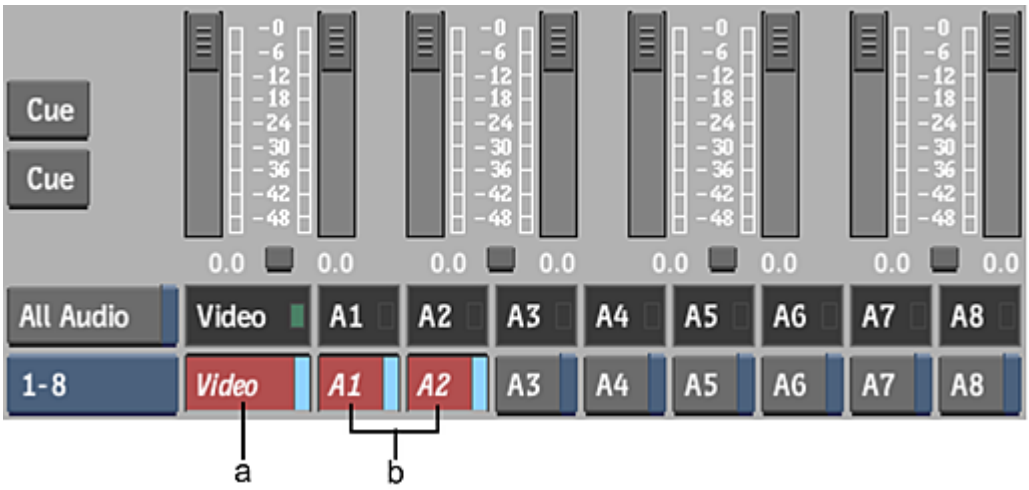
Outputting a Single Clip

To output a clip to a VTR, load a clip into the VTR Output module and then set the in and out points for clip output. If necessary, enable the options to offset the start frame for output and to apply a letterbox overlay. See VTR Output Menu Options (page 312).

If the duration marked by the in and out points that you set for output is greater than the duration of the clip you are outputting, the last frame of the clip is repeated to fill the gap between the end of the clip and the out point of clip output. For example, if you set an in point at 01:00:00:00 and an out point at 01:01:00:00, the clip output process has a duration of one minute. If the clip you are outputting is 30 seconds long, the last 30 seconds of the output result on tape are filled with the last frame of the clip you are outputting.

To output a single clip to a VTR:

1. Load the clip that you want to output into the VTR Output module. See Accessing the VTR Output Module (page 312).
2. Make sure that the correct video track and audio channel buttons are enabled so that the corresponding video track and audio channels are output to tape.



(a) Video track enabled for output (b) Audio channels 1 and 2 enabled for output

3. Enable or disable All Audio in the Clip Output menu (to the left of the audio controls).

   When All Audio is enabled, all audio channels are sent to the audio converter, not just the ones you enabled in the previous step.

   This means you can monitor all audio, even though you are only recording one or two tracks to the VTR. You can also route the other audio channels to a mixer or any other device capable of capturing an audio signal.
4. Make sure the appropriate VTR is selected in the Device Name box. See Selecting a VTR Device For Output (page 315).

   The preview window displays the contents of the tape currently in the selected VTR.
5. To preview the clip before outputting, click Preview.
6. To preview the clip you are outputting against the contents on the tape, enable Split View. See Monitoring Video During Clip Output (page 324).

7 To output the selected clip starting at any frame other than the first one, enter the start timecode in the Start Offset field.

8 Set the in and out points for clip output. See Setting Input and Output In and Out Points (page 324).

9 To output the clip to the VTR, select Insert from the Output box.

The clips with Status *selected* are output to tape. During output, the Status column is updated to reflect the status of each clip:

- Pending: the clip is waiting to be output to tape.
- Output: the clip is being output to ape.
- Done: the clip has been ouput to tape.

10 After the transfer is complete, verify that it was successful by playing the transferred clip. To do so, cue to the in timecode and click the Play button in the VTR Transport controls.

11 When you are done, click EXIT.

Outputting Multiple Clips

You can output more than one clip to a VTR in a single pass. When you load multiple clips into the VTR Output module, you must define in and out points (and any other output options) for each clip before starting the clip output process.

When multiple clips in the same session are selected for output, the application performs a validation check to determine if certain parameters match the project settings and whether or not the video device is capable of outputting these clips. The following outcomes are possible:

- If the clips have mixed timing settings, you are prompted to select which timing you want to use. Clips having timings different from the one you select are discarded.
- Clips with field dominance mismatch are discarded or kept, depending on your selection.
- Clips exceeding the maximum bit depth are discarded from the selection. A selection containing 8- and 10-bit clips can be output.
- If all the clips are discarded from the selection after the validation check, a message appears allowing you to confirm.

If some of the clips have overlapping timecodes, their timecodes are highlighted in red in the clip list. To fix overlapping timecodes, edit the timecodes in the clip list, or use the timecode fields in the Output tab.

To output multiple clips to VTR:

1 Load the clips that you want to output into the VTR Output module. See Accessing the VTR Output Module (page 312).

2 If the clips have conflicting timings, you are prompted to select the timing you want to use.

Clips that do not match this timing are discarded.

3 To ouput a clip, its status field must have the *selected* indicator. Click the Status field to alternate between the *selected* indicator and an empty field.

4 Set output options for each clip.

5 Optional: Click the column headers to sort the list. This changes the order of the output sequence.

6 To output the clip to the VTR, select Insert from the Output box.

The clips with Status selected are output to tape. During output, the Status column is updated to reflect the status of each clip:

- Pending: the clip is waiting to be output to tape.
- Output: the clip is being output to tape.

- Done: the clip has been ouput to tape.

7 After the transfer is complete, verify that it was successful by playing the transferred clips: cue to the in timecodes and click the Play button in the VTR Transport controls. Clips that have been output are highlighted in the clip list of the large VTR Output module.

8 When finished, click EXIT.

When outputting multiple clips with the small VTR Output module, note the following differences:

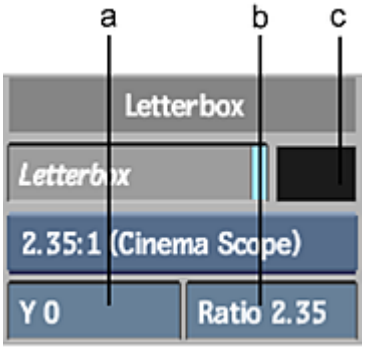
- Use the Previous and Next buttons to navigate the clip list.
- Once output options have been set for each clip, make sure you are viewing the first clip you want to output.
- When you select Insert in the Output box, the clip you are viewing, and all subsequent clips, are output to tape. Clips that precede the clip you are viewing are not output to tape.

Outputting Clips With a Letterbox Overlay

You can apply a letterbox overlay on output. The letterbox does not affect the clip itself; it is applied on the fly as part of the real time output process.

To output clips with a letterbox overlay:

1 In the Output Clip menu, click the Deliverables tab to see the Letterbox controls.



(a) Letterbox Offset field (b) Letterbox aspect ratio field (c) Colour selector box

2 Enable Letterbox.

3 Set the letterbox options:

- In the Aspect Ratio field, enter a value to set the aspect ratio of the letterbox overlay. Alternatively, select a preset from the Aspect Ratio box.
- In the Letterbox Offset field, enter a value to shift the letterbox overlay up or down, as required.
- Use the Colour Selector box to set the colour of the letterbox overlay.

Outputting Clips with a Logo Overlay

You can add a logo overlay on output, in real time. This feature is available only if your workstation is equipped with an NVIDIA Quadro FX 5600 SDI graphics card.

To add a logo, make sure the FX 5600 SDI card is selected in the engineering menu of the VTR Output module. If AJA_OEM2K is selected, you cannot use the Logo Overlay feature.

To select the FX 5600 SDI card in the Output Clip menu:

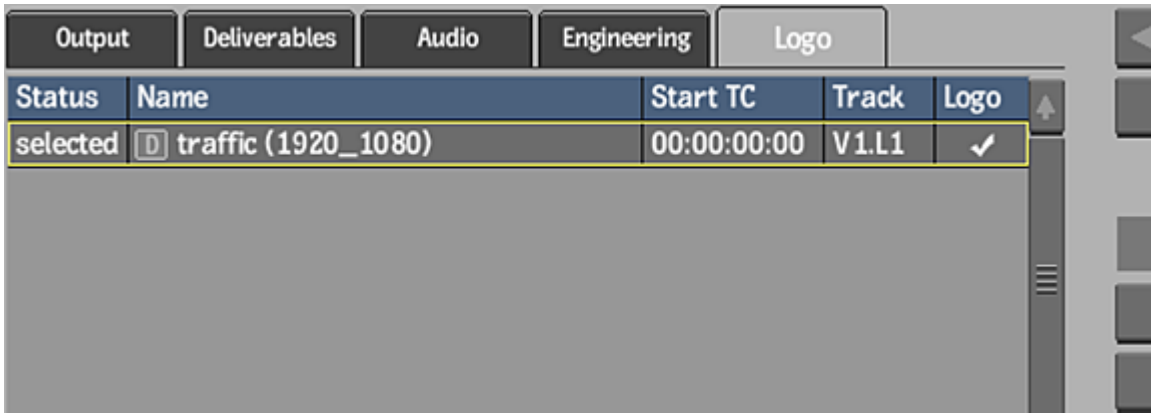
1. Open the Output Clip menu.
2. From the Graphics Card box, select GFX SDI.

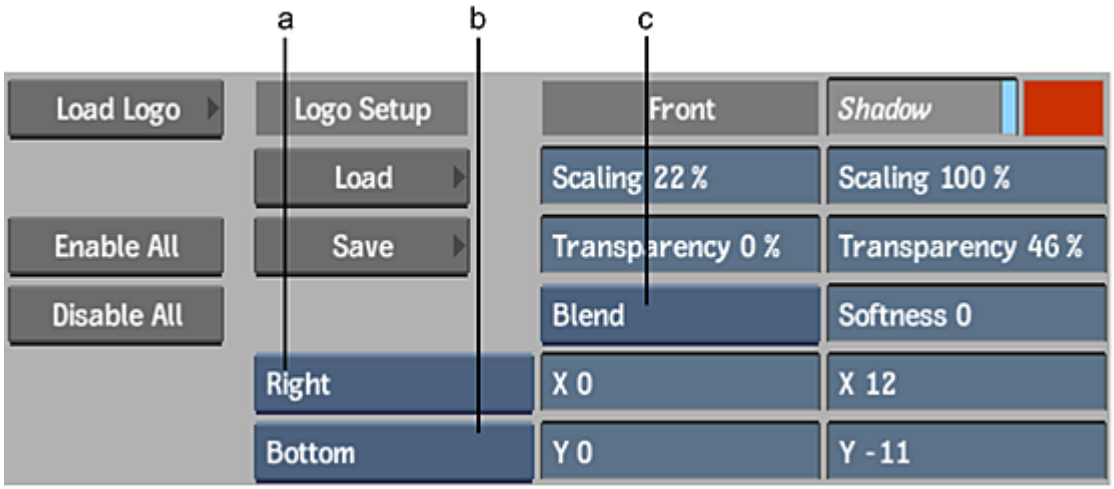
If you do not see GFX SDI as an available option in the Graphics Card box, it is either not installed, or it is not set up or configured properly.

About the Logo Controls

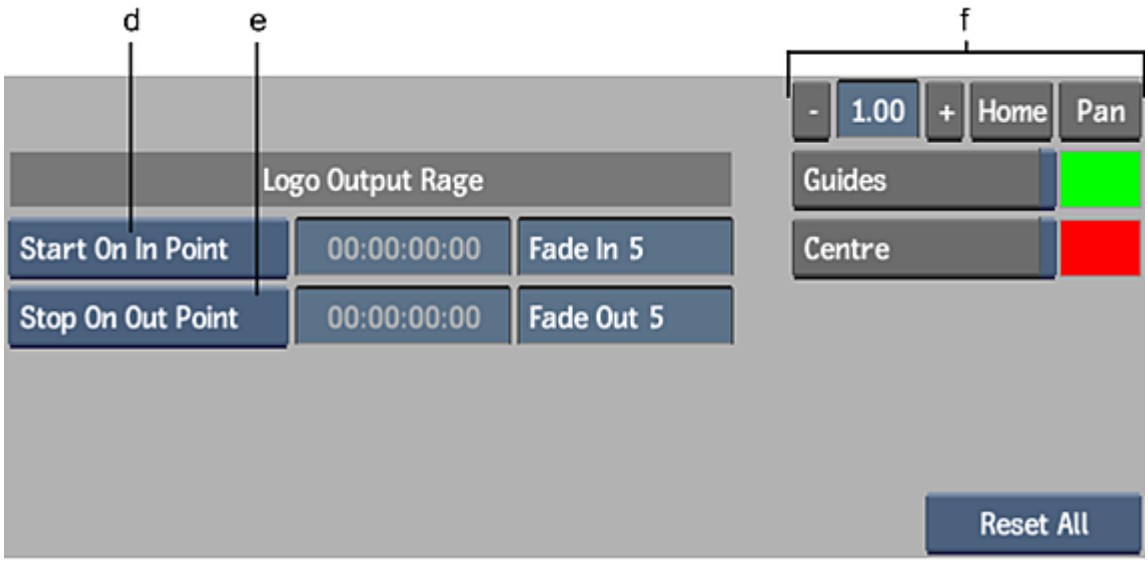
The logo controls are found in the VTR Output module, under the Logo tab.

NOTE When you select the Logo tab, you see the clip selected in the Output list instead of the VTR feedback.





(a) Horizontal alignment button (b) Vertical alignment button (c) Blend Mode box



(d) In point (e) Out point (f) Zoom controls

Logo Enables or disables burning of the selected logo on output.

Load Logo Opens the Clip Select menu where you can select logo clips.

Enable All Use to display the logo on all clips to output.

Disable All Use to remove the logo from all clips to output.

Scaling (Front) Changes the size of the Logo clip. The range of values is 0 to 100, with 100 being the actual size of the Logo clip.

Transparency (Front) Changes the transparency of the Logo (front clip). 0% is Opaque. 100% is Transparent.

Position controls (X and Y fields) Moves the logo along the X and Y axes.

Blend Mode box Selects how the front and matte clips or the front and back clips are combined.

Select:	To:
Add Trans	Compensate for the soft or anti-aliased edge on an object in a front and matte clip media, with transparency.

Select:	To:
Blend	Punch the matte through the front. This blends the edge of the front clip and adds additional softness to the media.

Shadow Uses the Matte clip to add a Shadow. The default position is set to X:-5 and Y:-5. A colour pot on the right allows for the shadow colour to be changed.

Scaling (Shadow) Changes the size of the shadow. The range of values is 0 to 100, with 100 being the actual size of the shadow.

Transparency (Shadow) Changes the transparency of the Shadow. 0% is Opaque. 100% is Transparent.

Softness Softness can be added to the Shadow. The range of values is 0 to 200.

Horizontal alignment Positions the logo along the X-axis using one of the following options.

Select:	To:
Horizontal Centre	Center the logo on the X-axis.
Left	Snap the left edge of the clip to the Safe Title guide on the left.
Right	Snap the right edge of the clip to the Safe Title guide on the right.

Vertical alignment Positions the logo along the Y-axis using one of the following options.

Select:	To:
Vertical Centre	Center the logo on the Y-axis.
Top	Snap the top edge of the logo to the Safe Title guide at the top.
Bottom	Snap the bottom edge of the clip to the Safe Title guide at the bottom.

Guides Enables the display of the Safe Action and Safe Title guides within the Clip Preview window. Use the colour pot to change the colour of the guides.

Centre Enables the display of the Centre guide within the Clip Preview window. Use the colour pot to change the colour of the Centre guide.

In point Select an option to specify when the logo appears on the clip.

Select:	To:
Start On In Point	Have the logo appear at the in point of the clip.
Start After Frames	Specify after how many frames the logo appears. Enter the number of frames in the adjacent field.

Out point Select an option to specify when the logo disappears from the clip.

Select:	To:
Stop on Out Point	Have the logo disappear at the out point of the clip.
Duration	Specify how long the logo remains on screen. Enter the duration in the adjacent field.
Stop Before Frames	Hide the logo before the end of the clip, by the number of frames specified in the adjacent field. For example, selecting this option and entering a value of 5 makes the logo disappear by frame 15 in a 20-frame clip.

Fade In Defines the length of the logo fade in. A value of 0 indicates there is no fade in.

Fade Out Defines the length of the logo fade out. A value of 0 indicates there is no fade out.

Zoom controls The small VTR Output module is not available from the Logo tab. Use the Zoom controls to view oversized clips.

Reset All Use to reset the Logo controls to default.

Select:	To:
Reset All	Reset every field, and delete the logo.
Reset	Reset every field, but keep the logo.

Inserting Logos for Real-Time Output

Keep in mind the following when selecting clips to use as logos:

- If the clip has more than one frame, only the first frame is used.
- If the clip is Burn™ pending, unlinked or unlinked HiRes, it cannot be used.
- The higher the resolution for the clip, the more impact it has on the capacity of the application to output in real time.

Logo settings are persistent, and previous logo settings are loaded when you open the VTR Output module.

NOTE The same logo, with its settings, is applied to all clips for which you have checked the Logo column in the VTR Output module list.

To insert a logo in real time:

1. Select clips from the Media List and select File > VTR Output...
2. In the VTR Output module, use the Logo column to indicate which clips are to be output with the logo.
3. Click the Logo tab to access the logo controls.
4. Select one or two clips to use as logo:
   - If you select one clip, it is used as the front clip and a luminance key is applied to it. If the clip has multiple frames, only the first frame is used.
   - If you select two clips, the first is used as the front and the second as the matte.

5 Use the logo controls to position and configure the appearance of your logo, and to define in and out points. See About the Logo Controls (page 319).

6 Output the clip.

Outputting Clips in Assemble Mode

If you do not have time to black an entire tape, you can black a small portion of the tape and output clips to the tape in assemble mode. In this mode, the timecode on the tape is generated by the VTR as part of the recording process. Make sure that timecode regeneration is properly configured on the VTR. The standing recommendation is blacking one minute at the beginning of the tape.

If outputting a clip in assemble mode, extend the duration of the output to avoid problems when you need to recapture the segment. For example, this can be done by adding 5 to 10 seconds of black at the beginning and at the end of the clip, before entering the output clip module.

To output a clip in assemble mode:

1 On the VTR, set the TC generator switch to Internal and Regen to make the VTR generate the timecode. Refer to your VTR manual for instructions on setting the TC generator.

TIP If you have just added black to your tape, the VTR should already have these settings.

2 Set clip output options. See Outputting a Single Clip (page 316).

3 With the clip you want to output in the VTR Output module, set the clip output in and out points. See Setting Input and Output In and Out Points (page 324).

4 If black was not added at the end of the clip, make sure the out point exceeds the duration of the clip by five to ten seconds.

5 Select the Assemble option from the Output box.

6 Click Assemble to output the clip in assemble mode.

HDCAM SR Stereo Tape Output

Using an HDCAM SR, you can output material to specially formatted stereoscopic tapes. Stereoscopic tapes essentially stores in an interlaced timing two progressive clips; a 60i (50i) “clip” contains two 30p(25p) clips.

To output stereoscopic material to stereoscopic tape:

1 Ensure that the HDCAM SR is connected to the output device using a dual-link.

IMPORTANT If you use an NVIDIA SDI card to output the stereoscopic material, the NVIDIA card downconverts the material to 8-bit.

2 Set the HDCAM SR VTR to the stereoscopic setting and insert a stereoscopic tape.

3 In the application, enter the VTR Output module with a stereoscopic clip. The presence of an **S** in the bottom right corner of a proxy indicates a stereoscopic clip.

NOTE You can output only clips at 23, 24, 50 or 59 fps, with a 1080i or 1080PsF resolution.

4 Output the clip. See Outputting a Single Clip (page 316).

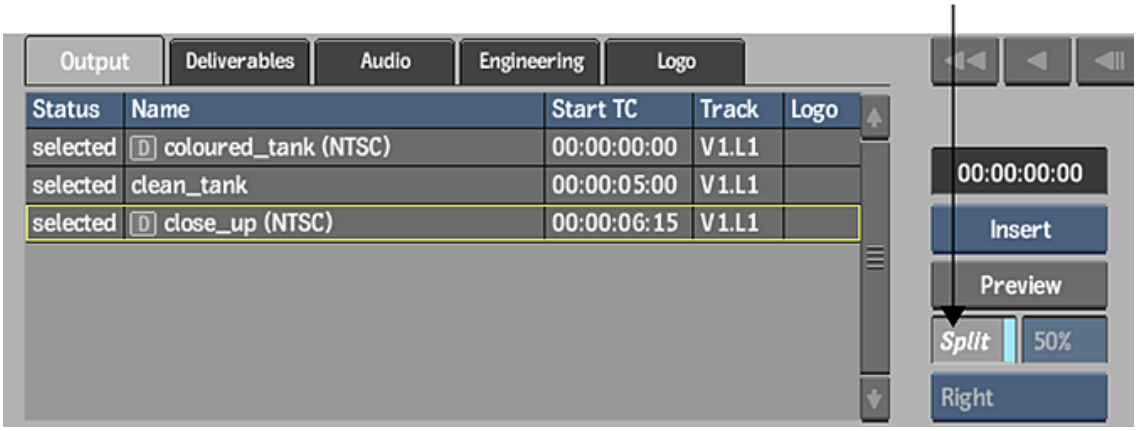
The player in the VTR Output module displays only the left-eye layer, but the application outputs both right- and left-eye tracks, in a 4:2:2 colour space.

Monitoring Video During Clip Output

When you output a clip to a tape that already has media on it, you can enable a split view to compare a clip with a portion of tape, in much the same way as split view allows to compare two video segments in the player. This can be especially useful if you are outputting a clip to seamlessly overwrite a section on the tape.

To use the split view preview option:

1 In the Output tab of the VTR Output module, enable Split.



The Split View controls are enabled and you can view the clip that is currently set for output alongside the content on the tape.

2 Set Split View preview options:
- In the Split field, enter a value to set the relative position of the split.
- From the Current Clip Display box, select an option to set the location of the current clip display.
- From the Split Orientation box, select Horizontal Split or Vertical Split to set the Split View orientation.

3 With Split View enabled, you can scrub the VTR to locate the frame that matches the frame of the current clip.

4 Set the In point to the timecode of the frame on tape that corresponds to the first frame of the clip.

5 Cue to the In point. The first frame of the clip and matching frame on tape are displayed in Split View.

6 Click Play to play the tape and the clip simultaneously in Split View.

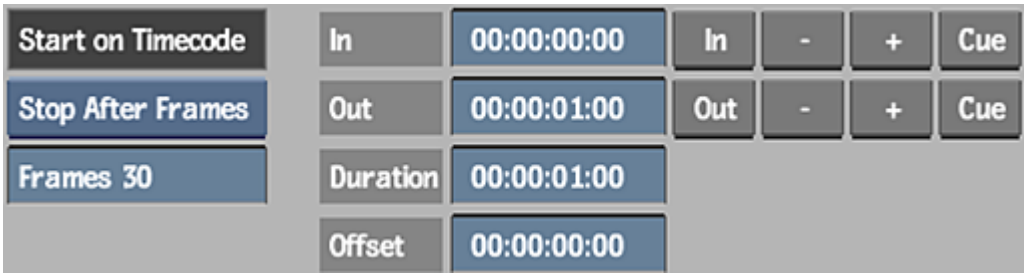
7 Click Stop, and disable Split View when you are finished.

Setting Input and Output In and Out Points

You can enter timecode values directly in the In, Out, and Dur fields to set in and out points for clip input and output. However, there are other methods of setting the in and out points that are more convenient if you are scrubbing the tape in the VTR to locate reference frames for clip input and output. For example, if you output a clip to overwrite a segment on a tape, you scrub the tape to locate the frame that matches the first frame of the clip you are outputting.

You can also enter in and out points, as well as duration values based on the timecode value of a clip selected for input. Simply click Input Clip, then while holding down the left `Ctrl` key, select a clip and a destination reel.

Besides the described methods of entering timecodes in the In, Out, and Dur fields, you can modify clip in and out points or cue the VTR to these points using the following controls on the right side of the Input or Output menu.



In button Click to set the In field to the VTR's current timecode.

Out button Click to set the Out field to the VTR's current timecode.

+ and - buttons Click to frame-step the In or Out timecodes forward or backward. If Stop After Frames is selected from the Stop Mode box, both the in and out points move because the duration between them is locked.

Cue buttons Click to cue the VTR to the in or out point.

Inputting and Outputting a Live Video Signal

On a Linux system, in the configuration file, if you uncomment the `live NTSC` or `live PAL` VTR lines, you can capture a live video signal or crash-record a clip using a click to start and stop the clip input or output process.

On a Mac system, in the Flame Setup application, in the VTR tab, if you enable the lines with `live NTSC` or `live PAL`, you can capture a live video signal or crash-record a clip using a click to start and stop the clip input or output process.

On input, use the Live NTSC or PAL option to capture directly from a camera, VCR, or any other device that does not support remote control via RS-422. You can also feed a live video signal from satellite or any other broadcast source.

On output, use the Live Video option to output clips to a device that does not support remote control via RS-422.

To input a live video signal:

1. Enter the VTR Input module.
2. From the VTR Device box, select Live NTSC or Live PAL.

   The incoming live video signal appears in the preview window. The Start Mode box is unavailable. With Live Video capture, you must use Start On Click mode. You can use Stop On Click or Stop On Frames to end the capture.

NOTE When Stop On Click is selected as the preferred capture stop mode, the out point and the duration timecode fields are updated to reflect the longest possible duration corresponding to the full capacity of a framestore. The capture stops either when you click anywhere on the screen or the timecode indicated is reached, meaning that the framestore is full.

3. Set input options. For example, enter the clip name and enable the video tracks and audio channels that you want to capture. See Inputting Clips From a VTR (page 306).
4. Make sure you are receiving the live video signal.
5. If you are capturing from a device that does not support remote control, press Play (allowing enough time to click Process in Flame).
6. To begin capturing, click Process.

   After a moment, you are prompted to click anywhere to begin the output process.
7. Click anywhere on the screen.
8. To end capturing in Stop On Click mode, click anywhere on the screen.

To output a live video signal:

1. Load a clip into the VTR Output module.
2. From the VTR Device box, select Live NTSC or Live PAL.

   The Start Mode box is unavailable. With Live Video output, you must use Start On Click mode. You can use Stop On Click or Stop On Frames to end the output.
3. Set output options. For example, enter the clip name and enable the video tracks and audio channels that you want to capture. See Outputting Clips To a VTR (page 312).
4. If you are outputting to a device, click Record, or take any action required to enable the device and capture the signal being output from Flame.
5. To begin capturing, click Process.

   After a moment, you are prompted to click anywhere to begin the output process.
6. Click anywhere on the screen.
7. To end capturing in Stop On Click mode, click anywhere on the screen.

Generating Proxies from VTR Input

With sufficient processing power, proxies are generated in real time. Otherwise, this occurs as a post-process. To achieve better performance, or to use higher quality proxy types, you might want to perform clip input without proxy generation, then edit project settings to generate proxies overnight.

The following guidelines refer to working on projects set to generate proxies.

You may create projects with proxy management options set to generate proxies for HD clips. If your hardware configuration supports on-the-fly proxy generation, your clips are captured and proxies are generated as part of the real-time input process. Otherwise, you may be required to wait for proxy generation once the capture process itself is complete.

If you are capturing many clips one-at-a-time, you may want to turn off proxy generation for you project. That way, you can capture your clips without having to wait for proxy generation.

Once you have completed the capture process, turn proxy generation back on. If you have captured many clips, this process could take a long time. You may want to do this when you can leave the workstation unattended, for example, overnight.

Capturing Material with Variable Frame Rate

To perform frame-accurate video input and output using Panasonic® variable frame rate VTR decks, you must use the following firmware versions:

- AJ-HD3700H D5-HD deck:

  SYSCON: 1.04 or later

  AV: 0.21 or later

  FRONT: 0.13 or later

- AJ-HD1200A DVCPRO HD deck:

  SYSIF: 1.30 or later

  AVDV: 1.39 or later

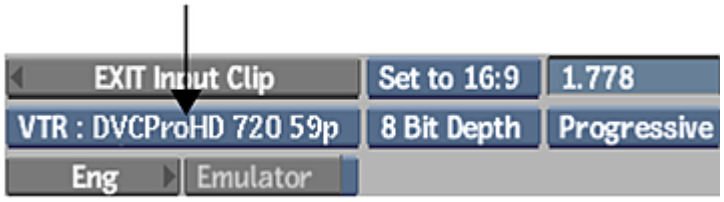
  SERVO: 1.22 or later

Regular video input and output is not affected by a firmware version.

NOTE Refer to your Panasonic documentation for information on verifying the firmware version.

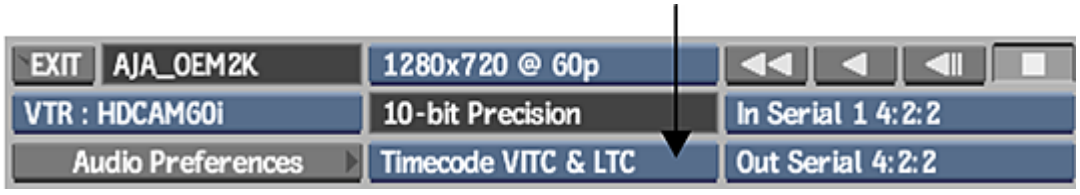
To capture material from a VariCam device:

1. Ensure your devices are properly connected and the corresponding VTR keywords are enabled in the software initialization configuration file. See Configuring Software For Clip Input and Output Using a VTR (page 305).
2. Open the VTR Input module. See Accessing the VTR Input Module (page 306).
3. From the Device Name box, select a video device, supporting variable frame rate (for example, one of the following options: VTR DVCProHD 720 59p or VTR DVCProHD 720 60p).



4. Enter the Engineering menu and enable Varicam (on the right side).

5. Make sure the Video Input Delay is set to zero or a positive value (negative video input delay is not supported on VariCam capture).
6. From the Timecode Mode box, select the required timecode.



7 Exit the Engineering menu and click Process.

Flame removes the redundant frames so that the frame rate of the captured clip corresponds to the frame rate of the project. You can monitor the process using the VTR Status display. The Current Frame Rate field is updated depending on the frame rate of the captured material.

(a) Current Frame Rate field

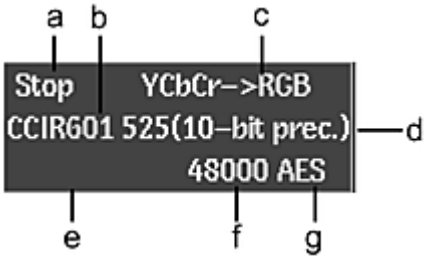
NOTE When performing operations in Varicam mode, audio/video synchronization largely depends on frame rate compatibility and can be guaranteed only if the following conditions are observed: (1) Material shot at 24 fps and captured into a 24 fps project (2) Material shot at 30 fps and captured into a 30 fps project (3) Material shot at 60 fps and captured into a 60 fps project.

Controlling a VTR

The RS-422 serial connection between the VTR and your workstation enables full remote control of the device from Flame. Connected VTR devices send their current status to Flame, and this status is displayed in the VTR Input and VTR Output modules.

Verifying the VTR Status

The following illustration shows a VTR status display.



(a) VTR Status (b) VTR Timing (c) Colour Space (d) Bit Precision (e) Control Mode (f) Audio Sample Rate (g) Audio Input/Output Type

Other VTR status values include.

VTR field:	Indicates:
VTR Status	Play, Play Lock, Stop, Jog, Fast Forward (FFW), Rewind (RWD), 3x, 4x, 8x, 15x, 30x. When the field is blank the VTR is disengaged.
VTR Timing	The video timing currently configured on the video device.
Colour Space	The colour space conversion method for both input and output of clips.
Control Mode (LOCAL, REMOTE, or REC INHIBIT)	Whether the VTR is in Local or Remote control mode. Local appears in yellow when the VTR is in Local mode. In Local mode, the VTR cannot be controlled from Flame. You can switch the VTR between Local and Remote mode only on the VTR. Rec Inhibit implies REMOTE mode, but you cannot output to the tape.

VTR field:	Indicates:
Audio Sample Rate	The audio sampling rate when you input or output from the VTR.
Audio Input/Output Type	The source audio interface type when inputting and the destination audio interface type when outputting.
Bit Precision	The accuracy used (SDI bit depth) when transferring data. This cannot be changed.

Using the VTR Transport Controls

The current frame on the cassette in the VTR appears in the image window and the Current Timecode field (the uppermost timecode field in the Clip Input and Output menus) displays the timecode of the current frame.

(a) Current Timecode field

Use the following VTR Transport controls to play the VTR.

NOTE The VTR must be in Remote mode to use the VTR Transport controls.

Click:	To:	Hotkey:
	Rewind the tape.	`Home`
	Play the tape backward.	-
	Move backward one frame. Hold down to slowly jog backward.	`left arrow key` (`down arrow key` for -5 frames)
	Stop the tape.	`spacebar`
	Move forward one frame. Hold down to slowly jog forward.	`right arrow key` (`up arrow key` for +5 frames)
	Play the tape forward.	`Enter`
	Fast-forward the tape.	`End`

Click:	To:	Hotkey:
	Eject the tape.	none

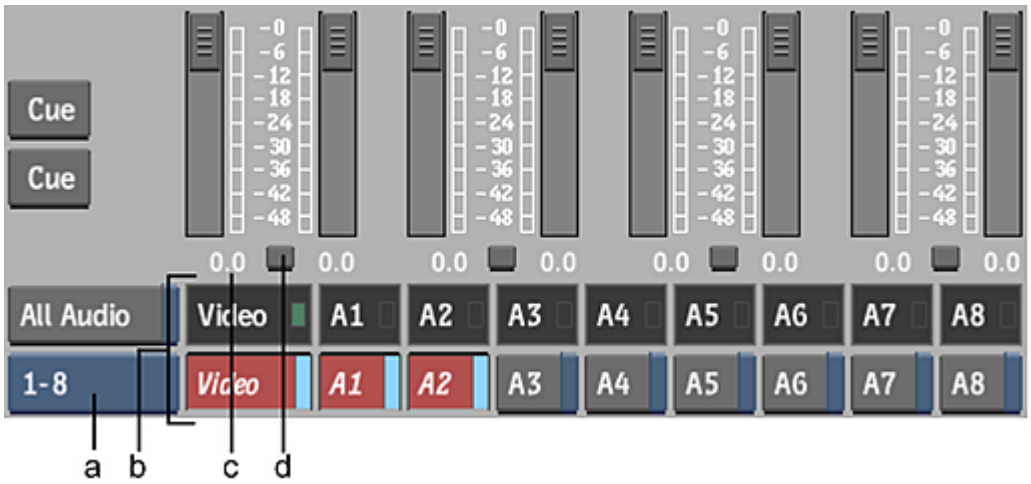
You can also shuttle the VTR by dragging the cursor in the image window. Place the cursor on the image, press the pen or mouse button, and drag the cursor to the right to shuttle forward and to the left to shuttle backward. The farther away you move the cursor horizontally from the middle of the image, the faster the speed.

You can also change the reverse or forward playback speed by pressing the `Up` or `DownarrowKeys` on your keyboard. Each press reduces or increases the speed by two.

Audio Gain Adjustment Settings

Audio gain adjustment is a part of the clip output process only. The clips you are outputting are unaffected by audio gain adjustments made using the VTR Output module.

On output, you can adjust the audio gain, for example, to restore the levels you had monitored on capture.



(a) Audio Tracks Toggle button (b) Channel Selection buttons and indicators (c) Audio Level fields (d) Fader Lock buttons

All Audio button When enabled, outputs to the audio monitor every audio channels. When disabled, outputs only the enabled audio channels to the audio monitor. The All Audio button has no impact on the audio tracks recorded by the VTR.

Audio Tracks Toggle button Toggles the Channel Selection buttons and indicators between audio tracks 1-8 and 9-16.

Channel Selection buttons and indicators Controls and displays which audio channels are recorded by the VTR. The black boxes with the green LEDs indicate video tracks and audio channels that are part of the clip that you want to output. The red buttons indicate the tracks and channels the VTR records on output.

Audio Level fields Displays the audio gain, in decibels. Adjust using the faders. In the small VTR Output module, increase or decrease the gain by dragging left or right on the fields. By default, audio gain is 0 db.

Fader Lock buttons When enabled, locks the faders for the corresponding pair of audio channels together.

Adjusting Audio Gain on Clip Output

To adjust the audio gain on output clip:

1. Enable the Fader Lock buttons (so that they are light grey) if you want to apply the identical value to pairs of audio channels.
2. Slide the faders to adjust the audio gain before you start processing. Use the All Audio button to monitor all the audio tracks that are output, regardless of what audio tracks the VTR records.
3. In the VTR Output module, enable Output All Audio.
4. Select or deselect channels for output by clicking the Channel Selection button for each channel as needed.

   NOTE In a multiple clip selection, channel selection is independent for each clip but the gain levels set with the faders are the same for all clips.

5. Process the clip.

   Selected channels are output.

Setting Video Input and Output Engineering Menu Controls

Clip input and output engineering options include video I/O settings such as pre-roll, post-roll, play delay, colour space conversion, and settings that define the process by which YUV video material on a tape is converted to the RGB format used by Flame, and vice-versa.

On a Linux system, each VTR device is associated with a set of default engineering settings that are specified in the software initialisation configuration file, in the VTR KEYWORD section.

To change the default Engineering menu settings for a VTR device on a Mac system, edit the VTR configuration files using the provided Flame Setup utility.

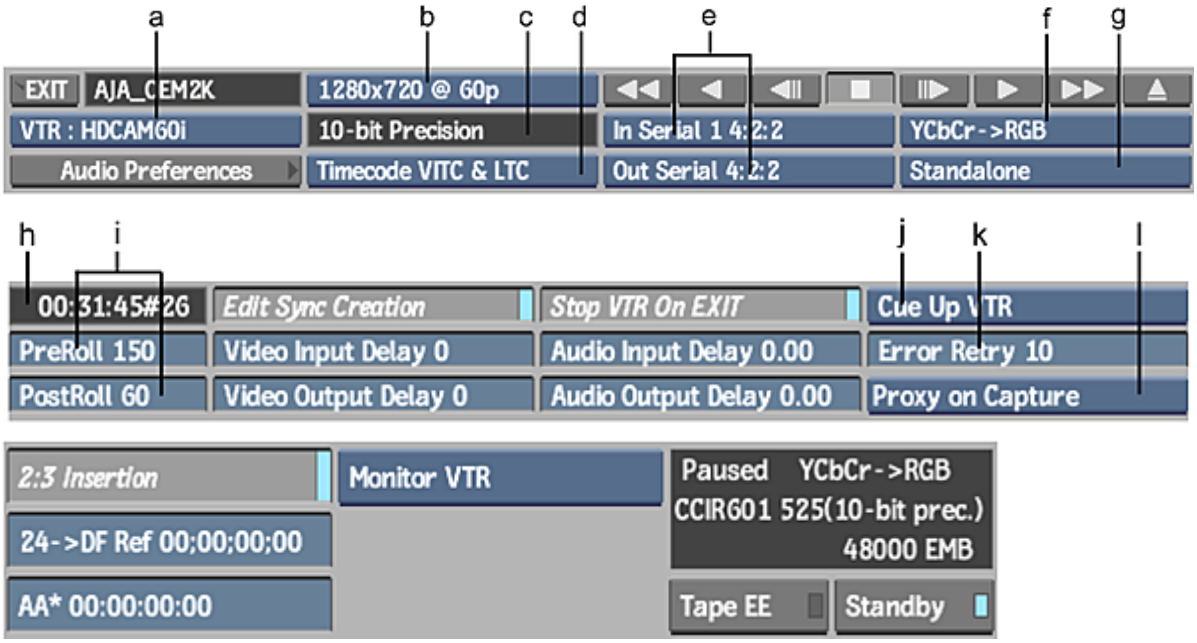
When you start Flame and set the video I/O timing for your project, enabled VTR devices are initialized, and the settings in the Engineering menu are populated accordingly. However, if necessary, you can modify these settings on a session-to-session basis.

To open the Engineering menu:

1. From the VTR Input module, click Eng, or from the VTR Output module, click the Engineering tab, to open the Engineering menu.

Engineering Menu Options

Consult the following illustration (broken into three parts) and explanations of the options in the Engineering menu. These illustrations are of the VTR Input Engineering menu. The VTR Clip Engineering menu contains a subset of these controls.



(a) Device Name box (b) Video I/O Timing box (c) Precision box (d) Timecode Source box (e) Input and Output Connection boxes (f) Colour Space box (g) Output Sync box (h) VTR Current Timecode field (i) PreRoll/PostRoll fields (j) Cue Up box (k) Error Retry field (l) Proxy Box

Device name box The Device Name box differentiates between 3G and non-3G capable VTR and provides an option for each enabled VTR. To modify settings for a specific VTR device, select the corresponding option from this box.

Video I/O Timing box Provides video timing options (resolution and frame rate) corresponding to different video formats supported by the video input/output board of the system.

Precision box Determines the video interface precision, or SDI bit depth used. This value cannot be changed.

Audio Preferences button Click to view the Audio Preferences menu.

Timecode Source box Determines which type of timecode is obtained from the VTR device. Timecode VITC & LTC is the default, and recommended option. You should only have to switch to Timecode VITC or Timecode LTC if one of the timecode tracks is corrupted.

Select:	**To obtain:**
Timecode VITC	Vertical interval timecode (VITC).
Timecode LTC	Longitudinal timecode (LTC).
Timecode VITC & LTC	Both types of timecode. At normal playback speed, Flame obtains LTC, but switched to VITC when the tape is rewinding, fast-forwarding, or otherwise moving at a non-playback speed.

Input and Output Connection boxes These boxes determine the connection by which the video signal is transferred. This box is automatically set to reflect the selected VTR.

- In Serial 1 3G / Out serial 1 3G: The video signal is transferred through one SDI 3G link. This is the only available option when you use a 3G VTR. The VTR Status field indicates any conversion that might happen because of the sampling used, 4:2:2 (conversion YCbCr <-> RGB) or 4:4:4 (no conversion RGB <-> RGB).

- In serial Dual 444 / Out serial Dual 444: The video signal is transferred through dual SDI links; one video field is sent through one SDI cable, the other field through the other cable.
- In Serial 1 4:2:2 / Out serial 422: Traditional VTR, single-SDI connection.

Colour Space box Determines the YCrCb colour space conversion method.

Select:	To perform clip I/O with:
YCrCb->RGB	A standard YCrCb-RGB conversion process that clips superblack and superwhite luma (Y). Use this option for typical clip I/O processes with VTR devices.
YCrCb->RGB + Headroom	A YCrCb-RGB conversion process that preserves superblack and superwhite colour information. Use this option when inputting or outputting greyscale mattes or other clips where preserving extremes in the luma channel is required.
No Conversion	This is available when using dual link for RGB input and output. Video black and white levels on the SDI stream are mapped to black and white values in RGB on the framestore. Use this option in conjunction with 4:2:2 input and output connections to input and output 4:4:4 video using dual-links (4:2:2 and 0:2:2).
No Conversion + Headroom	Also available when using dual link for RGB input and output. This mode uses all levels available and preserves all but a few RGB values. Use this option with the 4:4:4 input and output connections to input clips from and output clips to a Telecine.

Output Sync box Determines the output sync reference source. The reference signal may originate from several different sources. Select the source you are using from this box according to the following table.

Source type	Available on:	Description:
House	All systems	A centralized analogue reference signal, originating from a sync generator, sent to the genlock port on the video board or VBOB.
Digital 1 and Digital 2	Most HP® 8400s and all HP 8600s and 9400s	Same as Digital, except you can choose between two inputs: Digital 1 or Digital 2. On the HP 8400 with the AJA SD (OEM-LH) video board, only Digital 1 is available.
Standalone	All systems	The reference signal generated internally by the Flame workstation.

PreRoll field Indicates the pre-roll, in frames.

PostRoll field Indicates the post-roll, in frames.

Video Input Delay field Indicates the video delay on input, in frames. If this value is incorrect, the result clip when you click Frame Grab in the VTR Input module does not match the frame you see in the preview window.

Video Output Delay field Indicates the video delay on output, in frames. If this value is incorrect, the clip you output does not get recorded to the proper place on the tape.

Audio Input Delay field Indicates the video delay on input, in frames.

Audio Output Delay field Indicates the video delay on output, in frames.

Stop VTR on EXIT button When enabled, sends a stop command to the VTR when you exit the Input Clip, Output Clip, Auto-Capture, or Archiving menu. For example, if the VTR is playing a clip, or if it is cueing to an in point, the transport operation in-progress is interrupted.

Cue Up box Determines the speed of the cueing process.

Select:	To cue up the VTR:
Cue Up VTR	Using the internal cueing algorithm of the VTR.
Cue Up Fast Forward	Using Flame. Use this option if Cue Up VTR is too slow for far cue points, such as on the betacam SP.

Error Retry field Indicates the number of times Flame retries failed input or output processes.

2:3 Removal/Insertion button Enables automatic, real-time 2:3 removal on output and insertion on input. This is only available when the VTR is set to 29.97i or 59.94i.

Edit Sync Creation button Automatically create edit sync groups for clips with audio channels on input.

Proxy box Determines proxy management when inputting clips.

Select:	To generate:
Proxy in Post	Proxies as a post-processing step
Proxy on Capture	Proxies during capture

Generating proxy during capture is the quickest method. Depending on hardware configuration of your system, capture may be performed in real time with playback. Some extra required processing, however, may prevent the graphics board from updating the image window and broadcast monitor in real time, so you may not be able to view the clip being played as it is captured.

Monitor box Set this to Monitor VTR or Monitor Output. In Monitor VTR mode, the signal coming back from the VTR is displayed, and video may be appear to be late compared to the audio. In Monitor Output mode, the signal being output to tape is displayed, and the video and audio should be in sync.

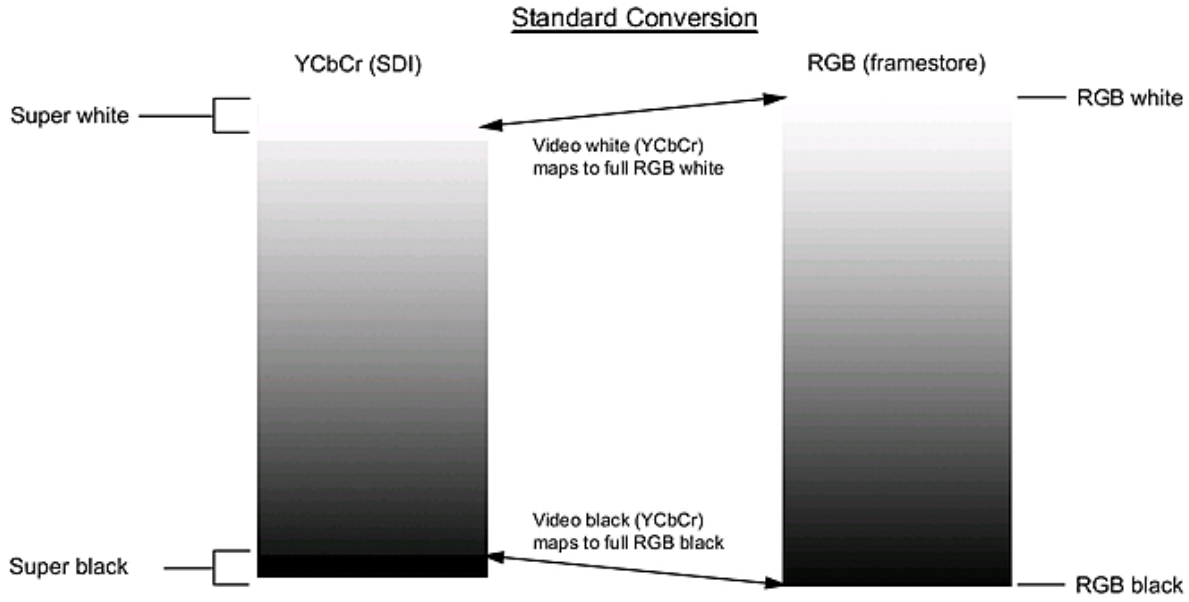
Dual Image Resizing Filter box Select the resize algorithm used to scale back to full resolution clips captured using the Dual Image - Scale resolution. Ordered from lowest quality (Triangle) to highest (Lanczos). On some lower-end workstation, you might need to select a lower quality algorithm to capture every frame.

Inputting and Outputting with Headroom

With 4:2:2 serial digital interface (SDI) input and output connections, the colour components of video signals are Y (luma), Cb (blue colour difference), and Cr (red colour difference). For standard video signals in 10 bits, black has a luma value of 64 and white has a luma value of 940. When performing standard captures to RGB values on Flame systems, black YCbCr (64,512,512) maps to RGB (0,0,0) and white YCbCr (940,512,512) maps to RGB (1023,1023,1023). SDI values 0 to 3 and 1020 to 1023 are reserved values for synchronization purposes.

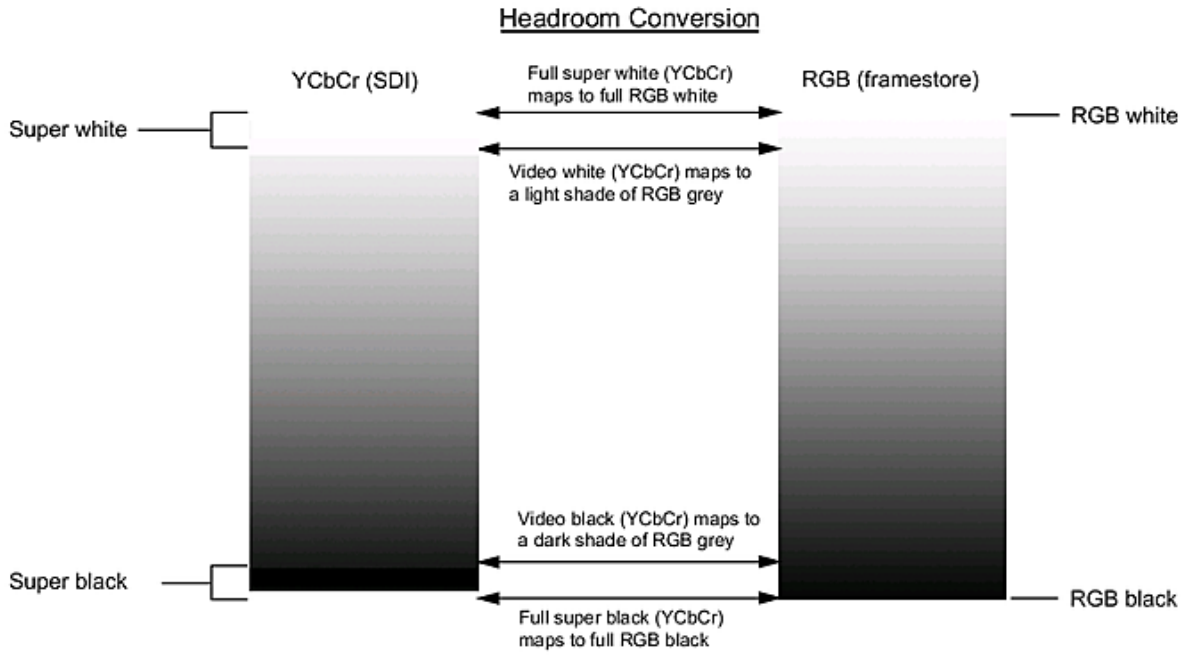
Some cameras record shadow details below the video luma black value of 64 and white detail above the video luma white value of 940. These details are called super blacks and super whites. These are also referred

to as “headroom” and “footroom.” Under normal circumstances, headroom is not converted during capture, and the super black and super white details are lost, as described in the following illustration:



In this illustration, headroom and footroom information is lost.

Using the YCbCr<->RGB with headroom colour space option during input on Flame systems, it is possible to capture these super black and super white values. In this case, video black YCbCr (64,512,512) maps to RGB (64,64,64) in 10 bits, and video white YCbCr (940,512,512) maps to RGB (940,940,940). YCbCr Luma values between 4 and 64 and between 940 and 1019 are converted to RGB on capture. While this gives you the advantage of being able to capture super black and super white values, it also means that video black will not map to full black in RGB, and video white will not map to full white in RGB. The following image illustrates this:



In this illustration, headroom and footroom information is conserved

For 4:4:4 RGB standard input and output, a similar explanation applies. On capture using the No Conversion setting, RGB 4:4:4 SDI video black at value (64,64,64) maps to RGB (0,0,0), and RGB 4:4:4 SDI video white at value (940,940,940) maps to RGB (1023,1023,1023).

With the No Conversion with Headroom option, the maximum possible SDI value range is used. RGB 4:4:4 SDI values ranging from (4,4,4) to (1019,1019,1019) map to the same RGB range on capture.

Enabling Colour Space Conversion on Clip Input

To enable colour space conversion on clip input:

1 In the VTR Input module, click Engineering.
The Engineering menu appears.

(a) Input Connection box (b) Colour Space box

2 From the Input Connection box, select the input connection.

3 From the Colour Space box, select YCbCr->RGB+Headroom.

4 Capture the material.
Material is captured with headroom and footroom.

To enable colour space conversion on clip output:

1 Open the clip library with the clip(s) you want to output.

2 Click Output Clip and prepare output clip settings.

3 In the VTR Output module, click Engineering.

4 In the Engineering menu, select the output connection from the Output Connection box.

5 From the Colour Space box, select YCbCr->RGB+Headroom.

6 Output the material.

Material is output with headroom and footroom.

Capturing 60i Material with 2:3 Pulldown Removal

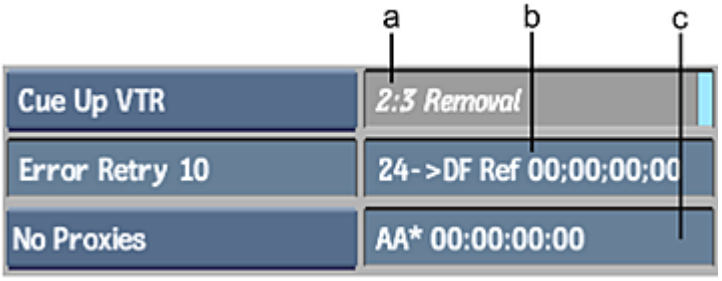
Using the VTR Input module, you can capture 30 fps film-based material and convert it to 24 fps material in real-time as you capture. The procedure for enabling automatic 2:3 removal differs depending on whether your tape uses non-drop or drop 60i timecode.

To capture 60i material with 2:3 pulldown removal:

1 Set up your capture.

2 In the VTR Input module, click Eng.

The Engineering menu appears. It contains options for removing 2:3 pulldown in real time while capturing.



(a) 2:3 Removal button (b) 24>DF Reference field (c) AA Reference field

3 Enable 2:3 Removal.

4 In the AA Reference field, enter a timecode that corresponds with an AA frame on the tape that is currently in the VTR.

TIP To determine an AA frame, frame-step (`Right arrow`) the tape and look closely at each frame. In 2:3 pulldown, two jitter frames with field artefacts are followed by three normal frames. The second frame after the two jitter frames is an AA frame.

5 When you capture material from a tape that uses 30 fps drop-frame timecode with 2:3 removal enabled, a slight discrepancy between timecodes is introduced due to the reference timecode used to determine the 30 drop frame to 24 timecode conversion. To optimize correspondence between the timecodes, enter the timecode in the 24 >DF Reference field that matches the start reel timecode for the tape.

TIP The start clip timecode can be deduced from the hour-mark of the timecode used by your clips. For example, 1;00;00;00 drop-frame would require a 1;00;00;00 value in the 24p>DF reference field to get a clip starting at 01;00;00+00.

6 Make sure the project's field dominance is set to field1. If the project's field dominance is set to field 2, you must switch the field dominance back to field 1.

7 Click EXIT to return to the VTR Input module.

8 Using the In and Out timecode fields, set the in and out points for the capture session.

(a) In and Out timecode fields (b) 2:3 In and Out pulldown display

With the in and out timecodes set, the corresponding 2:3 pulldown frame is indicated in the In and Out 2:3 pulldown display. Use the 2:3 Pulldown display to determine the 24 fps result in and out frames.

2:3 Pulldown Frame	24 fps Result Frame
In/Out AA	In/Out A
In/Out BB	In/Out B
In/Out DD	In/Out D
In BC	In C
In CD	In D
Out CD	Out C

9 Click Process to capture the material with 2:3 pulldown removal.

Configuring Dual-Serial Link I/O

Flame supports dual-serial link input and output connections for 4:4:4 clip I/O. Configure the type of I/O connection (single- or dual-link serial) using the Input Connection and Output Connection boxes in the Engineering menu.

When you select an input or output connection for clip I/O, you usually also select the colour space conversion method using the Colour Space box on the Engineering menu. The conversion methods that appear in this box depend on the type of I/O connection used. The following table shows the colour space conversion methods that are available for single-link and for dual-link I/O connections.

Colour Space Conversion Method	Supported for Single-Link Serial (4:2:2) I/O	Supported for Dual-Link Serial (4:4:4) I/O
YCbCR -> RGB	Yes	No
YCbCR -> RGB + Headroom	Yes	No
No Conversion	No	Yes
No Conversion + Headroom	No	Yes

VTR I/O Supported Timings

The following VTR I/O timings are supported on Flame.

AJA (Mac or Linux)

Format	Resolution	Supported Frame Rates
SD	720 x 486	29.97i
SD	720 x 576	25i
HD	1280 x 720	50p, 59p, 60p
HD	1920 x 1080	23p, 23psf, 24p, 24psf, 25p, 25psf, 50i, 29p, 59i, 30p, 50p, 59p, 60p
2k DCI	2048 x 1080	23p, 23psf, 24p, 24psf, 25p, 25psf

Blackmagic Design (Mac)

Format	Resolution	Supported Frame Rates
SD	720 x 486	29.97i
SD	720 x 576	25i
HD	1280 x 720	50p, 59p, 60p
HD	1920 x 1080	23p, 23psf, 24p, 24psf, 25p, 25psf, 50i, 29p, 59i, 30p

Clip Output Using Real-Time Deliverables

About Real-Time Deliverables (Linux only)

The Real-Time Deliverables mastering solution allows you to perform certain types of operations on material during clip output without having to first render the clips. For example, you can output from a master source format to many output formats directly to tape without rendering. Using the concept of the Deliverable—a virtual clip containing real-time operations and linked to its original source clip—you can create and manage multiple real-time output formats from one clip.

Supported operations include:

- Resize
- Pan and Scan
- Application of 3D and 1D LUTs
- Frame rate change with deceleration or acceleration with 2:3 insertion
- Audio timewarp
- Letterbox overlay

The Real-Time Deliverables feature supports source material up to 2K (10-bit) depending on the hardware setup. Good performance may be achieved beyond 2K (10-bit), but is not guaranteed. Soft effects are not supported for real-time output, unless processed before output. Embedded audio is also not supported.

Real-Time Deliverables sections explain important concepts and workflows for how to manage, preview, and output real-time operations, and references other sections of the user guide for more details, where appropriate.

Make sure to follow the hardware requirements described in Hardware Requirements for Real-Time Deliverables (page 340).

What Is a Deliverable?

The settings created in Real-Time Deliverables panel are saved as a Deliverable, which is central to the Real-Time Deliverables workflow.

A Deliverable is a virtual clip that is associated with the source clip from that point on, in the library or on the Desktop. A clip can have multiple Deliverables, and each Deliverable that you create is itself treated like a clip—a modified version of the original clip. As with clips, you can enter the Output Clip menu and output a Deliverable to tape. See Managing Deliverables (page 347).

Hardware Requirements for Real-Time Deliverables

To follow the Real-Time Deliverables procedures, you must be using an NVIDIA Quadro FX 5600, FX 5800, or FX 6000 SDI graphics card. The configuration for using these cards is described in the *Hardware Setup Guide* for your workstation.

When installing this release of the application for use with Real-Time Deliverables, it is important to use the settings in the new version of the *init.cfg* file. These settings specify the new hardware to be used as your preview device. For example:

```
VideoPreviewDevice 1920x1080@23976psf, nvidia, 1, 1920, 1080, 23976psf, DTVsync
```

If there are settings in your previous *init.cfg* file that you want to keep, port them to the new version during installation.

To use Real-Time Deliverables, make sure the appropriate SDI card is selected in the Output Clip menu.

To select the appropriate SDI card in the Output Clip menu:

1. Open the Output Clip menu as described in Outputting Clips To a VTR (page 312).
2. From the Graphics Card box, select GFX SDI.

When GFX SDI is selected, the installed SDI card is automatically used for video output.

When AJA_OEM2K is selected, the AJA OEM-2K is used for video output.

GFX SDI is not an available option in the Graphics Card box if:

- The SDI card is not installed.
- The SDI card is not set up or configured properly.

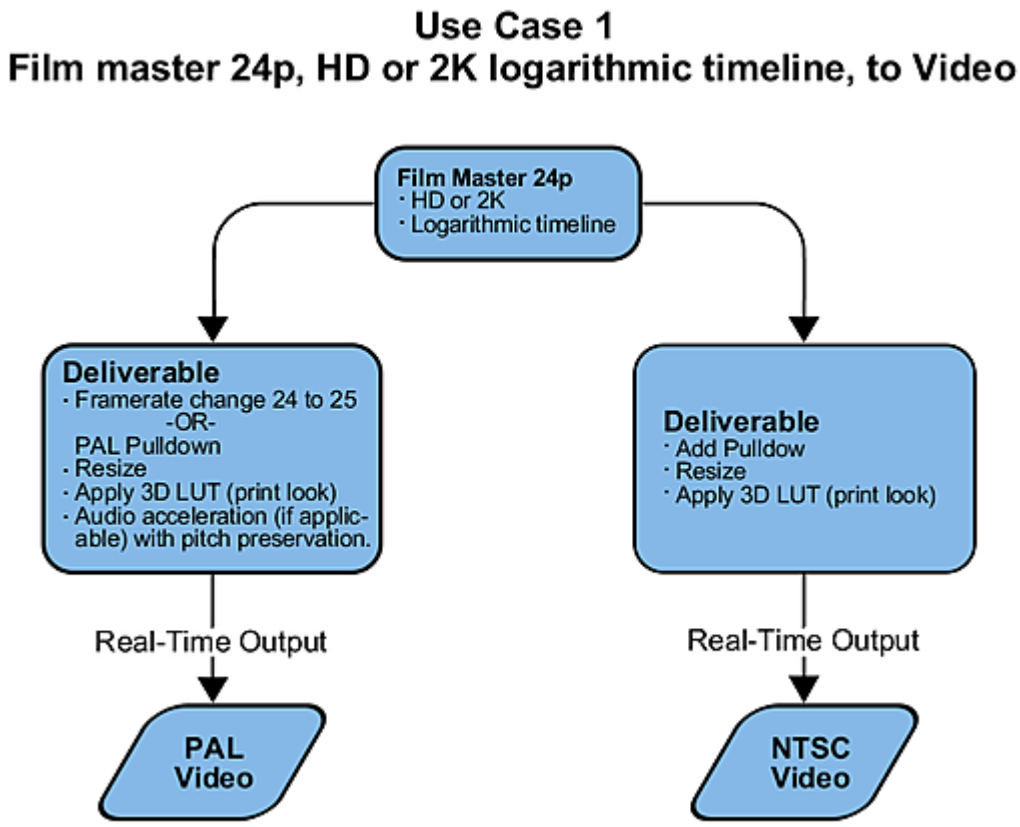
To install or configure an FX 5600, FX 5800, or FX 6000 SDI, see one of the hardware guides previously cited in this topic.

Real-Time Deliverables Use Cases

This section provides two examples of how Real-Time Deliverables can be applied.

Film Master 24p, HD, or 2K Logarithmic Timeline, to Video

An important case addressed by Real-Time Deliverables is when an organization wants to take a 24p film master with an HD or 2K logarithmic timeline, and create real-time output to PAL and NTSC video format.



To output to PAL video, you would create a Deliverable that does the following:

- Increases the frame rate from 24 to 25, or uses PAL Pulldown
- Resizes the clip
- Applies a print-look 3D LUT
- Accelerates the audio while preserving the pitch, if necessary

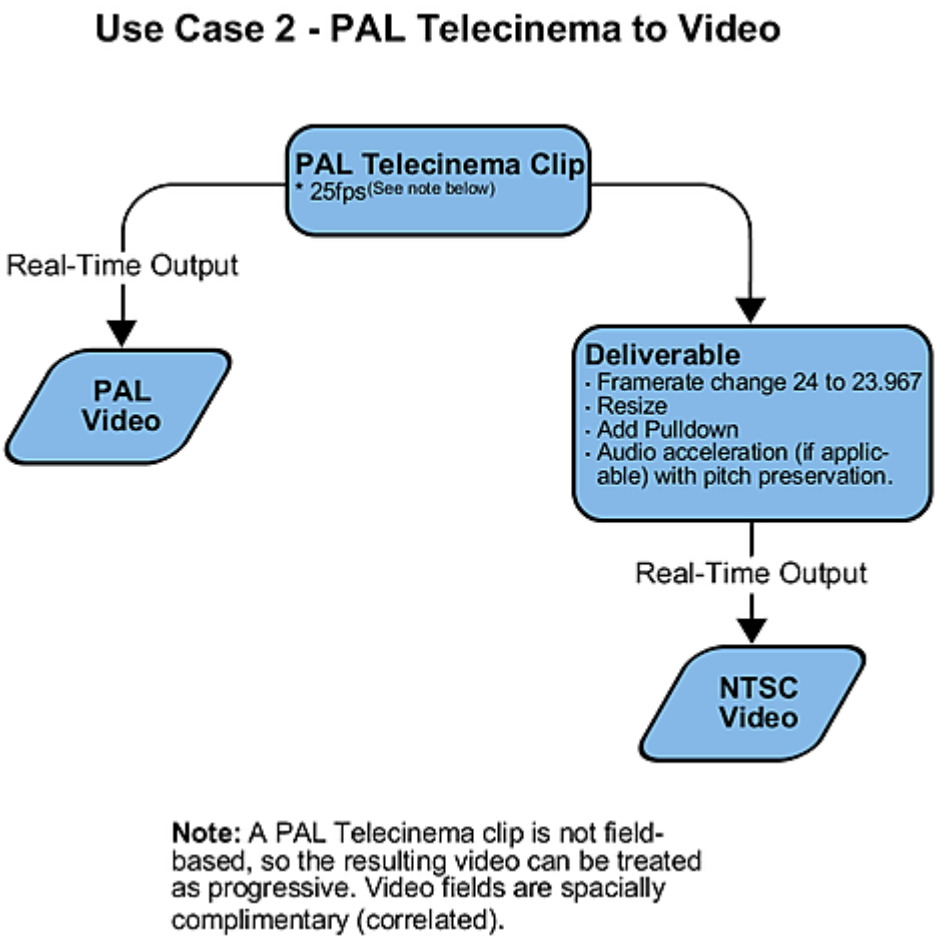
To output to NTSC video, you would create a Deliverable that does the following:

- Adds 2:3 pulldown
- Resizes the clip
- Applies a print-look 3D LUT

From the Clip Output menu, the clip is output to tape with no need to render the preceding settings.

PAL Telecinema to Video

In this example, NTSC and PAL video are output, but the source material is a PAL Telecinema clip. Because a PAL Telecinema clip is not field-based, the resulting video can be treated as progressive. Its video fields are spatially complementary.



To output to PAL video, you would not need to create a Deliverable. You can output the clip directly to tape from the Output Clip menu.

To output to NTSC video, you would create a Deliverable that does the following:

- Decreases the frame rate from 25 to 23.967
- Resizes the clip
- Adds 2:3 pulldown
- Accelerates the audio while preserving the pitch, if necessary

From the Clip Output menu, the clip is output to tape with no need to render the preceding settings.

Accessing the Real-Time Deliverables Panel

Use the Real-Time Deliverables Panel to manage Real-Time Deliverables, and to configure real-time operations such as resize, pulldown, and so on. Many of these settings can also be altered in the Output Clip menu.

To access the Real-Time Deliverables panel:

1. Select the clip you want to work on and open the Player.
2. From the Options box, select Show Real-Time Deliverables.
   The Real-Time Deliverables panel will open to the right of the Player.

Most of the controls in the Real-Time Deliverables panel are common to various parts of the software, and their functions are explained in their respective sections. The following does not describe how to use the controls, but rather how to manage the results using Deliverables and output them to tape in real time. The settings you create in the Real-Time Deliverables panel can be previewed in the Player and are applied in real time during clip output.

To configure real-time operations, see Managing Deliverables (page 347).

NOTE Settings in the Real-Time Deliverables panel override clip settings from other modules. These settings are overridden, not discarded. For example, if a Deliverable has LUTs set, the LUTs from the source clip are not used.

Source clip information Displays information about the source clip before real-time operations are applied.

Resize controls Displays values set in the Resize editor, accessible with the Editor button. You can edit the values directly in the Real-Time Deliverables panel. The available destination resolutions and timings depend on the format of the source material.

Audio controls Displays the audio speed. To preserve audio pitch, enable Preserve Pitch.

LUT controls (3D and 1D) Enables or disables the use of LUTs. You can specify a 3D LUT to use, and access the LUT editor.

Stereo controls Enables stereoscopic output settings, if a stereo track is detected. You can select a Stereo Mode option and a corresponding Method option.

Select:	To:
Anaglyph	Output the left eye and right eye as a red/cyan clip. You can reduce ghosting (Dubois), remove RGB values (Mono) or create custom anaglyph results.
Interlaced	Output the left eye and right eye as an interlaced RGB clip. You can output the left eye as field 1 or field 2.
Dual Image - Scale	Output the left eye and right eye so that they are adjacent to each other in the same clip, either in a left and right or top and bottom orientation.
Dual Image - Splice	Ouput the left eye and right eye so that they are side by side in the same clip. Only available when each eye track's horizontal resolution is half that of the output resolution (2 tracks at 960 for a 1920 output).
Dual Output	Output the left and right eye separately to dual streams of a tape.

Letterbox controls Enables or disables a letterbox overlay.

Start TC field Set the start timecode of the Deliverable. The start timecode is used during output, it indicates where on the tape the clip is recorded.

Previewing and Processing Material for Real-Time Output

Preview Deliverables to verify your settings. If necessary, process source clips or Deliverables before outputting in real time.

Previewing Deliverables

The real-time operations you create in the Real-Time Deliverables panel can be previewed in the Player.

To preview the result of a Deliverable in the Player:

1 From the Options box, select Show Real-Time Deliverables.
The Real-Time Deliverables panel will open to the right of the Player.

With this option selected, your real-time settings are displayed during playback or when you jog or scrub through the clip.

You can also preview real-time operations in the Clip Output menu. See Outputting Deliverables (page 345).

Processing Source Clips and Deliverables

The purpose of Real-Time Deliverables is to be able to output material without having to process first. However, there are exceptions where, before outputting, you must process:

- Source material that has an unrendered soft effect applied to it.
- A Deliverable whose settings place unusually high demands on hardware and cannot be output in real time.

If you open the Real-Time Deliverables panel with a clip that has an unprocessed soft effect applied to it, you can configure Deliverable settings for it, but you cannot output the Deliverable until the effect is processed. To do so, click Process in the source clip information section of the Deliverables menu.

Once you set up your Deliverable, it is possible that your hardware cannot output the Deliverable to tape in real time. The status of the GPU Benchmark button helps you decide whether to process the Deliverable.

Status:	Processing status:
Green	You can output the Deliverable in real time, without firs rendering.
Yellow	You should process the Deliverable before going to Output Clip, as real-time output might not be possible.
Red	Real-time output is not possible: you must process the Deliverable before going to Output Clip.

If the GPU Benchmark button indicates that real-time output is impossible, click Process in the resize and frame rate group of the Real-Time Deliverables panel.

A new clip is created with "_<Deliverable Name>" appended to the clip name. Any settings from the Real-Time Deliverables panel are burned into the clip. Because it is a new clip, it does not have any associated Deliverables associated. But the new clip uses the start timecode of the Deliverable. Enter the Output Clip menu with the clip and output it to tape.

Outputting Deliverables

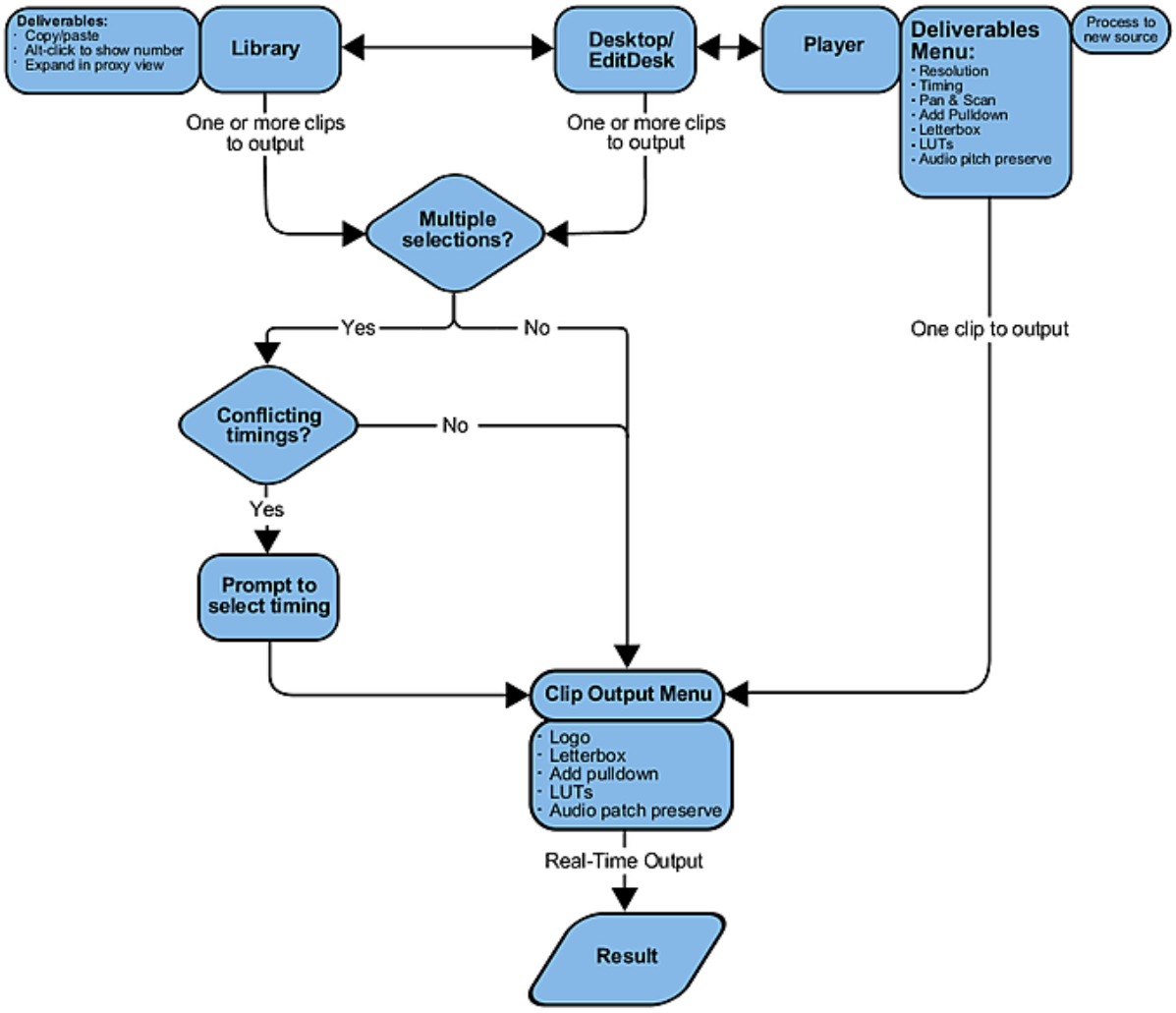
The workflow for outputting Deliverables is like the standard clip output workflow described in Outputting Clips To a VTR (page 312).

Each Deliverable that you create is treated like a clip. You can enter the Output Clip menu from the library with a Deliverable and output it to tape as you would any clip.

From the Real-Time Deliverables panel, click the Output Clip button to enter the Output Clip menu with the current Deliverable.

If you enter the Output Clip menu from the library with multiple items (clips, Deliverables, or both), the resulting behaviour is the same as when outputting multiple clips. If there are conflicting timings, you are prompted to select the timing you want to use, and items that do not match this timing are discarded. See Outputting Multiple Clips (page 317).

If you select a clip in the library to bring it into the Output Clip menu, all its Deliverables are also selected. If there are conflicting timings, you are prompted to select the timing to use. This can result in the exclusion of the original clip.



Once in the Output Clip menu, you can alter Letterbox and LUT settings in the Deliverables panel, as well as preserve audio pitch. You can also add a logo in the Logo tab, although it does not require a Deliverable.

These changes are saved to the Deliverable. The next time you work with that Deliverable in the library or Player, the changes you made in the Output Clip menu are loaded.

NOTE When you select the Real-Time Deliverables panel, you do not see the VTR feedback; instead you see the clip selected in the Output Clip list.

To create real-time settings for a clip and output to tape in real time:

1. From the Player, create a Deliverable. See Accessing the Real-Time Deliverables Panel (page 343).
2. Do one of the following:
   - From the Real-Time Deliverables panel, access the Output Clip menu.
   - From the library or Desktop, select the Deliverable or multiple clips and Deliverables, and access the Output Clip menu. If you selected multiple items and they have conflicting timings, you are prompted to select which timing you want to use.
3. In the Output Clip menu, make adjustments to real-time settings if necessary.
4. To preview real-time operations before outputting, use the Player controls in the Real-Time Deliverables panel.
5. Output the material to tape.

Managing Deliverables

A Deliverable results when you create real-time settings for a clip, in the Real-Time Deliverables panel. Deliverables are virtual clips associated with the original clip. You can:

- Rename or delete a Deliverable.
- Create multiple Deliverables for one clip, so that you can output many different formats of that clip.
- Save a Deliverable as a reusable Deliverable template.
- Load a Deliverable template onto another clip to apply the settings of that Deliverable to the clip.
- View information about Deliverables in the Desktop.
- Transfer clips and their Deliverables to other systems.

NOTE Any change to the frame rate (timing) or resolution of the clip deletes all Deliverables attached to that clip.

Creating and Modifying Deliverables

From the Real-Time Deliverables panel, you can create, rename, and delete Deliverables.

To create a Deliverable:

1. Access the Real-Time Deliverables panel, as described in Accessing the Real-Time Deliverables Panel (page 343).
2. From the Deliverable box, select <New Deliverable>.
3. Type a name for the Deliverable and press `Enter`.

Once the Deliverable is created, set up real-time operations in the Real-Time Deliverables panel. The operations you set up are automatically stored in the Deliverable you created.

You can create multiple Deliverables for a clip. If you create another Deliverable:

- The settings of the previous Deliverable are cleared from the Real-Time Deliverables panel.
- The new Deliverable name is added to the list in the Deliverables box.

To rename a Deliverable:

1. With the Deliverable loaded menu, click Name.
2. Type a new name for the Deliverable and press `Enter`.

To delete a Deliverable:

1. With the Deliverable loaded, click Delete.
2. Click Confirm.

The Deliverable is deleted, and the previously-loaded Deliverable (if there was one) is reloaded.

Exporting and Loading Deliverables as Templates

Use the Save and Load buttons in the Real-Time Deliverables panel to export a Deliverable as a template and load it on another clip.

You can only load a Deliverable on clips that have identical resolutions and timings with the original clip.

When you load a Deliverable template:

- Its settings are loaded to the Real-Time Deliverables panel.
- A new Deliverable is appended to the list in the Deliverables box.

Viewing Deliverables in the Desktop

Clips with Deliverables are indicated by a "D" overlay on their proxies.

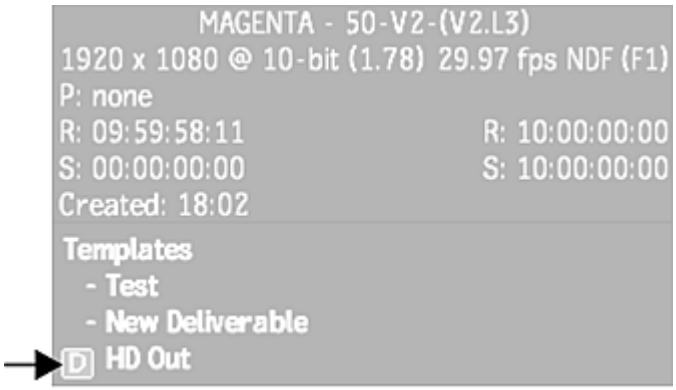
There are two types of "D" overlays, each indicating the selection in the Playback Resolution box:

Grey "D" The Player displays the Deliverable. See Previewing Deliverables (page 345).

Black "D" The Player displays Proxies or Full Res.

To open the Real-Time Deliverables panel from the clip proxy on the Desktop, double-click the "D" overlay.

To see quickly a list of Deliverables for a clip, `Alt`-click the clip proxy on the Desktop.



The list of Deliverables is included in the clip information. A "D" appears beside the active Deliverable (the Deliverable most recently selected in the Real-Time Deliverables panel).

In the library, Deliverable proxies are black and include the Deliverable name. Playing the Deliverable gives you the actual deliverable result.

Transferring Deliverables to Other Systems

When a clip is transferred to an Autodesk Visual Effects and Finishing system using Wire, its Deliverables are also transferred.

If the other system does not have the hardware needed for Real-Time Deliverables, users of that system will see that the clip has Deliverables, but will not be able to edit or preview them. The Deliverables will be greyed out in the Output Clip menu.

Users on such systems will be able to work with the original clip and transfer it back to the original system, with its Deliverables intact. As usual, changes to the frame rate (timing) or resolution of the clip deletes all Deliverables attached to that clip.

You cannot export Deliverables to other file formats. You cannot use Deliverables with EDLs.

Real-Time Deliverables Timing Specifications

When you set up a Deliverable to modify the timing of a clip during real-time output, the timing is changed according to the following tables.

NOTE In the following tables, N/A stands for *not applicable* and NC stands for *no change in timing*.

Destination timings 23.967psf through 25p:

Source					**Destination**			
	23.967psf	**23.967p**	**24psf**	**24p**	**PAL**	**50i**	**25psf**	**25p**
23.967psf	N/A	NC	Speed up	Speed up	Speed up to 25 or speed up to 24 + PAL Pulldown	Speed up	Speed up	Speed up
23.967p	NC	N/A	Speed up	Speed up	Speed up to 25 or speed up to 24 + PAL Pulldown	Speed up	Speed up	Speed up
24psf	Slow down	Slow down	N/A	NC	Speed up to 25 or apply PAL Pulldown	Speed up	Speed up	Speed up
24p	Slow down	Slow down	NC	N/A	Speed up to 25 or apply PAL Pulldown	Speed up	Speed up	Speed up
PAL	Slow down	Slow down	Slow down	Slow down	N/A	NC	NC	NC
50i	Slow down	Slow down	Slow down	Slow down	NC	N/A	NC	NC
25psf	Slow down	Slow down	Slow down	Slow down	NC	NC	N/A	NC

Source					Destination			
	23.967psf	**23.967p**	**24psf**	**24p**	**PAL**	**50i**	**25psf**	**25p**
25p	Slow down	Slow down	Slow down	Slow down	NC	NC	NC	N/A

NOTE When going from 23.967 to PAL, you can choose between increasing the frame rate to 25 or increasing to 24 and adding PAL Pulldown. When going from 24 to PAL, you can choose between increasing the frame rate to 25 or adding PAL Pulldown.

Destination timings NTSC through 60p:

Source					Destination		
	NTSC	**29.97p**	**30 psf**	**30p**	**60i**	**59.94p**	**60p**
23.967psf	Add Pulldown	Not supported	Not supported	Not supported	Speed up to 24 + Add Pulldown	Not supported	Not supported
23.967p	Add Pulldown	Not supported	Not supported	Not supported	Speed up to 24 + Add Pulldown	Not supported	Not supported
24psf	Slow Down to 23.976 + Add Pulldown	Not supported	Not supported	Not supported	Add Pulldown	Not supported	Not supported
24p	Slow Down to 23.976 + Add Pulldown	Not supported	Not supported	Not supported	Add Pulldown	Not supported	Not supported
PAL	Slow Down to 23.976 + Add Pulldown	Not supported	Not supported	Not supported	Slow Down to 24 + Add Pulldown	Not supported	Not supported
50i	Slow Down to 23.976 + Add Pulldown	Not supported	Not supported	Not supported	Slow Down to 24 + Add Pulldown	Not supported	Not supported
25psf	Slow Down to 23.976 + Add Pulldown	Not supported	Not supported	Not supported	Slow Down to 24 + Add Pulldown	Not supported	Not supported
25p	Slow Down to 23.976 + Add Pulldown	Not supported	Not supported	Not supported	Slow Down to 24 + Add Pulldown	Not supported	Not supported
NTSC	N/A	NC	Speed up	Speed up	Speed up	Not supported	Not supported
29.97p	NC	N/A	Speed up	Speed up	Speed up	Not supported	Not supported
30 psf	Slow down	Slow down	N/A	NC	NC	Not supported	Not supported

Source					Destination		
	NTSC	**29.97p**	**30 psf**	**30p**	**60i**	**59.94p**	**60p**
30p	Slow down	Slow down	NC	N/A	NC	Not supported	Not supported
60i	Slow down	Slow down	NC	Field merge	N/A	Not supported	Not supported
59.94p	Not supported	Not supported	Not supported	Not supported	Not supported	N/A	Speed up
60p	Not supported	Not supported	Not supported	Not supported	Not supported	Slow down	N/A

Archiving in Flame

8

Archiving in Flame consists of writing your media and project setups to external storage devices or to filesystem, to store your projects offline but in a restorable format.

A project archive includes all of a project's Media panel content, including the Media library, Shared Libraries, the Desktop, as well as all of a project's workspaces.

You can also archive individual clips from the Media panel, and the project setups.

Important Compatibility Information

The Flame 2017 family of products (Flame Premium, Flare, Flame Assist) can now read contents created in Smoke Desktop Subscription (DTS), with the following limitations:

- Projects from Smoke DTS can be opened (and converted) by the Flame Family 2017 products.
- Projects can be seen and imported through MediaHub, using Browse for Projects.
- Archives created in Smoke DTS can be read in Flame Family 2017 products.

IMPORTANT Contents created in Flame Family 2017 products cannot be read or opened in Smoke DTS.

		Will import in...				
		Flame 2017	Flare 2017	Flame Assist 2017	Smoke DTS 2017	Training Edition 2017
Projects Created In...	Flame 2017	Yes	Yes	Yes	No	No
	Flare 2017	Yes	Yes	Yes	No	No
	Flame Assist 2017	Yes	Yes	Yes	No	No
	Smoke DTS 2017	Yes	Yes	Yes	Yes	No
	Training 2017	No	No	No	No	Yes
	Flame 2016	Yes	Yes	Yes	No	No
	Smoke DTS 2016	Yes	Yes	Yes	Yes	No

Training 2016	No	No	No	No	Yes
Smoke DTS 2015	Yes	Yes	Yes	No	No

Choosing a Medium for Archiving

Choosing a medium or device for your archiving needs largely depends on your technical resources and overall needs. There are certain advantages and disadvantages to using each medium/device.

Flame can read and write filesystem archives. If your application is running on a Linux workstation, it can also read and create tape archives.

While Flame can read VTR archives created in a previous version, it cannot create nor write to one.

Filesystem Archive

A filesystem archive is an archive stored on a hard disk drive, such as external USB/FireWire® (IEEE 1394) hard drives offers, or shared storage such as a SAN. The device can use any of the formats supported by your workstation, but the recommended ones are ext2, ext3, or xfs for Linux, and HFS+ for OS X. NTFS is not supported.

Using a filesystem to archive your material provides the quickest method of archiving and restoring your material, which can be of any bit depth or resolution.

Tape Archives

A tape archive is an archive written directly to a device such as an LTO tape device. Tape provides fast and reliable read and write performance. Tape archives can be of any bit depth or resolution.

However, tape archives can only be restored to a Flame workstation and are unreadable by other applications. The procedure of archiving to a tape device is similar to archiving to file.

Flame only supports tape devices which the vendor confirms that:

- The device driver is compliant with standard UNIX tape device calls.
- The specific version of your operating system and kernel is supported.

The initialization file (*init.cfg*) for your Flame contains examples of the ClipMgtDevice Tape keyword to help you set up the appropriate block size value for your tape device and define a text label to identify the device in the Archiving module. Please refer to the documentation from your archiving device vendor for guidelines on the actual block size to use.

VTR Archives

Flame cannot create or write to a VTR archive. It can restore a VTR archive created in a previous version.

The following are VTR that were used to create VTR archives and which can be used to restore them:

- Uncompressed SD VTRs such as D-1 or lightly compressed SD VTRs such as Digital Betacam
- Uncompressed HD VTRs or lightly compressed HD VTRs such as Sony HDCAM SR or Panasonic HD D5
- Compressed HD VTRs such as Sony HDCAM (using "HDCAM" rather than "VTR" as the archive device type)

Archiving a Project to a File-based Archive

Archiving a project to a file archive consists of opening an existing archive file, or creating a new archive file, on storage such as an external hard drive, and archiving the project to it.

NOTE About *Archive and Close*. Previous versions of the application had an *Archive and Close* button. It was required because the archive header only updated when you explicitly closed the archive. But now, in the MediaHub, the archive header is systematically updated as the contents are written to the archive. This means that you no longer need to explicitly close an archive: as soon as the application is done writing to an archive, the whole archiving process is complete. The Close Archive button is there for you to dismiss an archive easily, and then open another one.

To archive a project to a new archive:

1. In the MediaHub, enable Browse for Archives.
2. In the Local Devices list, navigate to the location where to store the archive.
3. Click New Archive.
4. In the New Archive Creation dialog, enter the required information and click Create.

   Flame creates and opens the archive.
5. Set as required the options displayed in the Archive Options tab.
6. Click Archive Project.

   Flame starts archiving the project. The actual duration of the archive process depends on the size of the project and on the options you enabled or disabled. To cancel the process, click anywhere on the screen; you cannot use Flame while it archives your project.
7. Click Close Archive to finalize the archive. The project is now archived.

To add a project to an existing archive:

1. In the MediaHub, enable Browse for Archives.
2. Do one of the following:
   - In the Archives list, select the archive to which you want to add the project.
   - If you want to use an archive that is not displayed in the Archives list: open a volume displayed in the local devices and navigate to the archive. Select it.

   You can recognize an archive by its icon:

   .
3. Click Open Archive.

   The contents of the archive are displayed in the archive browser.
4. Set as required the options displayed in the Archive Options tab.
5. Click Archive Project. If the archive spans multiple segments, Flame might prompt you to locate a specific one.

   Flame starts archiving the project. The actual duration of the archive process depends on the size of the project and on the options you enabled or disabled. To cancel the process, click anywhere on the screen; you cannot use Flame while it archives your project.
6. Click Close Archive to finalize the archive. The project is now archived.

Errors When Archiving

In some cases, Flame warns that it cannot archive some of the elements you had selected for your archive. You can quickly identify what was archived, or what was not archived and why, using the Media Panel.

To see the archived status of clips:

1. Open Timeline.

If the archive is still opened, you need to close it before changing tab.

2. Set Media Panel's View Mode box to Full Width, or type `Shift-~`.
3. Set the Media Panel to the List View.
4. Use the scroll bar tot display the Archived column.
   If you cannot see the column, right-click a column header and select Archived from the list.

About Project Archives

You want to archive a project when you need to backup your project so that you can restore it in its entirety at a later time. A Project Archive stores all workspaces found in the project, and their contents:

- Project Setups (aka module setups)
- Desktop, including Batch and its snapshots, reels
- Libraries and their contents
- Shared Libraries

Important Compatibility Information

The Flame 2017 family of products (Flame Premium, Flare, Flame Assist) can now read contents created in Smoke Desktop Subscription (DTS), with the following limitations:

- Projects from Smoke DTS can be opened (and converted) by the Flame Family 2017 products.
- Projects can be seen and imported through MediaHub, using Browse for Projects.
- Archives created in Smoke DTS can be read in Flame Family 2017 products.

IMPORTANT Contents created in Flame Family 2017 products cannot be read or opened in Smoke DTS.

		Will import in...				
		Flame 2017	Flare 2017	Flame Assist 2017	Smoke DTS 2017	Training Edition 2017
Projects Created In...	Flame 2017	Yes	Yes	Yes	No	No
	Flare 2017	Yes	Yes	Yes	No	No
	Flame Assist 2017	Yes	Yes	Yes	No	No
	Smoke DTS 2017	Yes	Yes	Yes	Yes	No
	Training 2017	No	No	No	No	Yes
	Flame 2016	Yes	Yes	Yes	No	No
	Smoke DTS 2016	Yes	Yes	Yes	Yes	No
	Training 2016	No	No	No	No	Yes
	Smoke DTS 2015	Yes	Yes	Yes	No	No

Restoring a Project from a File-based Archive

No two projects can use the same name on a given workstation. If you attempt to restore a project with a name already in use, you are prompted to change its name before restoring it. You can see a list of all the projects in **MediaHub ➤ Browse for Projects ➤ Local Projects**. You can delete projects from the menu **Flame ➤ Project and User Settings**. See Working with Projects (page 47).

To restore a project from a file archive:

1. In the MediaHub, enable Browse for Archives.
2. Open the archive, recognizable by its icon:

   . Do one of the following:
   - In the Archives list, select the archive from which you want to restore the project. Click Open Archive.
   - If you want to use an archive that is not displayed in the Archives list: open a volume displayed in the local devices and navigate to the archive. Select it and click Open Archive.
3. Navigate the contents of the archive, and click the project to restore to select it. You can recognize a project by its icon:

   .

   Every archived project is created in a folder named Archive Session, time stamped with the date and time when the project was archived.
4. Click Restore Project. If the archive spans multiple segments, Flame might prompt you to locate a specific one.

   Flame starts restoring the project. The actual duration of the archive process depends on the size of the project.

   You cannot use the application while Flame restores your project. Cancel the process at any time by clicking anywhere on the screen.

Once restored, load the project through **Flame ➤ Project and User Settings**. Or browse the its contents using **MediaHub ➤ Browse for Projects**.

NOTE When you restore multi-version clips, Flame does not automatically resync to the original Open Clips to ensure the material is restored in the state it was archived. If you need updated versions, you need to use the contextual menu's **Media Versions ➤ Update Source**. This can be done on a single clip, or on containers like reel, folder, Reels groups. Note that segments that were updated are highlighted in white.

Important Compatibility Information

The Flame 2017 family of products (Flame Premium, Flare, Flame Assist) can now read contents created in Smoke Desktop Subscription (DTS), with the following limitations:

- Projects from Smoke DTS can be opened (and converted) by the Flame Family 2017 products.
- Projects can be seen and imported through MediaHub, using Browse for Projects.
- Archives created in Smoke DTS can be read in Flame Family 2017 products.

IMPORTANT Contents created in Flame Family 2017 products cannot be read or opened in Smoke DTS.

		Will import in...				
		Flame 2017	Flare 2017	Flame Assist 2017	Smoke DTS 2017	Training Edition 2017
Projects Created In...	Flame 2017	Yes	Yes	Yes	No	No
	Flare 2017	Yes	Yes	Yes	No	No
	Flame Assist 2017	Yes	Yes	Yes	No	No
	Smoke DTS 2017	Yes	Yes	Yes	Yes	No
	Training 2017	No	No	No	No	Yes
	Flame 2016	Yes	Yes	Yes	No	No
	Smoke DTS 2016	Yes	Yes	Yes	Yes	No
	Training 2016	No	No	No	No	Yes
	Smoke DTS 2015	Yes	Yes	Yes	No	No

Archiving Clips to a File-based Archive

Archiving clips to a file archive consists of opening an existing archive file, or creating a new archive file on storage such as an external hard drive, and archiving contents to it.

NOTE About *Archive and Close*. Previous versions of the application had an *Archive and Close* button. It was required because the archive header only updated when you explicitly closed the archive. But now, in the MediaHub, the archive header is systematically updated as the contents are written to the archive. This means that you no longer need to explicitly close an archive: as soon as the application is done writing to an archive, the whole archiving process is complete. The Close Archive button is there for you to dismiss an archive easily, and then open another one.

To archive material to a new archive:

1. In the MediaHub, enable Browse for Archive.
2. In the Local Devices list, navigate to the location where to store the archive.
3. Click New Archive.
4. In the New Archive Creation dialog, enter the required information and click Create.
   Flame creates and opens the archive.
5. Set as required the options displayed in the Archive Options tab.
6. From the Media panel, drag and drop the clips and folders to archive to the Archive browser.

What you add to the archive appears as greyed out in the Browser, indicating that it is yet to be committed to the archive: ready to be archived, but not yet written to the archive. The Pending Archive folder can also be used as a summary of what is being archived.

NOTE In the archive information panel, the Archive Size Pending field indicates the number of clips that will be archived, and the Archive Size Total field displays the size of the archive once those clips are archived.

7 Click Archive to start writing the clips to the archive.

The time taken to archive depends on the size of the material being archived and on the options you enabled. You cannot use Flame while it archives. You can always Cancel the process at any time: click anywhere on the screen.

To add material to an existing archive:

1 In the MediaHub, enable Browse for Archive.

2 In the Archives list, double-click the archive to use for archiving.

3 Drag and drop clips and folders to archive from the Media panel to the archive.

What you add to the archive appears greyed out in the archive, indicating that it is yet to be committed to the archive: ready to be archived, but not yet copied to the archive. The Pending Archive folder can also be used as a summary of what is being archived.

NOTE In the archive information panel, the Archive Size Pending field indicates the number of clips that will be archived, and the Archive Size Total field displays the size of the archive once those clips are archived.

4 Click Archive to start the archiving process. If the archive spans multiple segments, Flame might prompt you to locate a specific one.

The time taken to archive depends on the size of the material being archived and on the options you enabled. You cannot use Flame while it archives. Cancel the process at any time by clicking anywhere on the screen.

Errors When Archiving

In some cases, Flame warns that it cannot archive some of the elements you had selected for your archive. You can quickly identify what was archived, or what was not archived and why, using the Media Panel.

To see the archived status of clips:

1 Open Timeline.

If the archive is still opened, you need to close it before changing tab.

2 Set Media Panel's View Mode box to Full Width, or type `Shift-~`.

3 Set the Media Panel to the List View.

4 Use the scroll bar tot display the Archived column.

If you cannot see the column, right-click a column header and select Archived from the list.

Important Compatibility Information

The Flame 2017 family of products (Flame Premium, Flare, Flame Assist) can now read contents created in Smoke Desktop Subscription (DTS), with the following limitations:

- Projects from Smoke DTS can be opened (and converted) by the Flame Family 2017 products.
- Projects can be seen and imported through MediaHub, using Browse for Projects.
- Archives created in Smoke DTS can be read in Flame Family 2017 products.

IMPORTANT Contents created in Flame Family 2017 products cannot be read or opened in Smoke DTS.

		Will import in...				
		Flame 2017	Flare 2017	Flame Assist 2017	Smoke DTS 2017	Training Edition 2017
Projects Created In...	Flame 2017	Yes	Yes	Yes	No	No
	Flare 2017	Yes	Yes	Yes	No	No
	Flame Assist 2017	Yes	Yes	Yes	No	No
	Smoke DTS 2017	Yes	Yes	Yes	Yes	No
	Training 2017	No	No	No	No	Yes
	Flame 2016	Yes	Yes	Yes	No	No
	Smoke DTS 2016	Yes	Yes	Yes	Yes	No
	Training 2016	No	No	No	No	Yes
	Smoke DTS 2015	Yes	Yes	Yes	No	No

Restoring Material from a File-based Archive

To restore material from an archive:

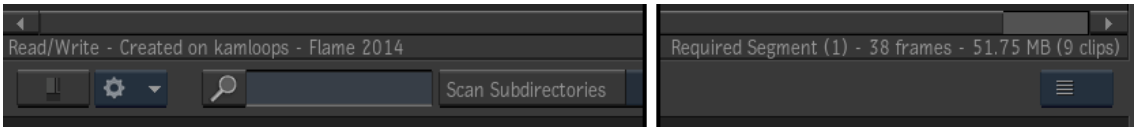
1. In the MediaHub, enable Browse for Archive.
2. In the Archives list, double-click the archive that contains the material to restore.
3. Drag and drop clips and folders from the archive to the Media panel.

   Material you drag and drop appears in the Media panel, flagged *(Pending)*, and is not actually restored until you click Restore.
4. Click Restore. If the archive spans multiple segments, Flame might prompt you to locate the required one.

   The length of the restoration process depends on the size of the material being restored. You cannot use the application for anything else while Flame restores the material. Cancel the process at any time by clicking anywhere in the MediaHub.

To see how much space a selected item will take on your storage, as well as the archive segment where it is stored:

1. In the archive, select the items to restore.
2. Click the MediaHub Gear menu and select Update Selection Information.
3. The right of the MediaHub status bar displays the number of frames and the space required to restore the selected items.



It also displays the archive segments that contain the selected items, which are required to restore the selection.

NOTE When you restore multi-version clips, Flame does not automatically resync to the original Open Clips to ensure the material is restored in the state it was archived. If you need updated versions, you need to use the contextual menu's **Media Versions ➤ Update Source**. This can be done on a single clip, or on containers like reel, folder, Reels groups. Note that segments that were updated are highlighted in white.

Important Compatibility Information

The Flame 2017 family of products (Flame Premium, Flare, Flame Assist) can now read contents created in Smoke Desktop Subscription (DTS), with the following limitations:

- Projects from Smoke DTS can be opened (and converted) by the Flame Family 2017 products.
- Projects can be seen and imported through MediaHub, using Browse for Projects.
- Archives created in Smoke DTS can be read in Flame Family 2017 products.

IMPORTANT Contents created in Flame Family 2017 products cannot be read or opened in Smoke DTS.

		Will import in...				
		Flame 2017	Flare 2017	Flame Assist 2017	Smoke DTS 2017	Training Edition 2017
Projects Created In...	Flame 2017	Yes	Yes	Yes	No	No
	Flare 2017	Yes	Yes	Yes	No	No
	Flame Assist 2017	Yes	Yes	Yes	No	No
	Smoke DTS 2017	Yes	Yes	Yes	Yes	No
	Training 2017	No	No	No	No	Yes
	Flame 2016	Yes	Yes	Yes	No	No
	Smoke DTS 2016	Yes	Yes	Yes	Yes	No
	Training 2016	No	No	No	No	Yes
	Smoke DTS 2015	Yes	Yes	Yes	No	No

Tips for Using Archives

Although the way you organize your archives is a matter of personal preference, the following guidelines may be helpful:

- After you have processed a clip, remove any material that you are no longer using before you create your archive.
- Try to keep your libraries clean at all times. Remove clips that are no longer being used and delete earlier but no longer used versions of your work.
- Use a consistent and clear naming technique so that you can easily find your material when you want to create or open an archive.
- Consolidate clips before archiving in order to save space.
- If you want to use the workstation that archived the material as an access point for the contents of the archive: in Archive Options, set Linked Archive Options to Use Archived Path, and then restore the archived project.

  TIP The path for media is displayed in the File Location column Media Panel.

NOTE Multiple users can connect to the same file archive and restore material, projects, and settings from it. But only the first user to open it can write to it! Once that user closes the archive, someone else can open it and write to it.

Archiving When Working On-location

For projects done on remote location and which are archived without their media, it is possible to relink later the archived clips to their media files, even if the media are not in the same location.

Archive the project using the following Archive Options:

- Cache Media on Archive: disabled.
- Archive Renders box: set to Exclude Renders and Cache.

These two archive options create the smallest possible archive: it contains only the clips of the imported media files (links), and the media generated by tools such as Paint. Renderings from Timeline FX are not archived with these settings.

Restore and relink:

1. Restore the archived project Flame.
   - If the media is stored at the path displayed in the Location column, then all the clips are automatically relinked to their media once restored. You have nothing else to do.
   - If the media is stored in a different location, then the restored clips are unlinked. Continue with this procedure.

   TIP Location column is visible in the MediaHub browser, when viewing the archive using the Workspace view.

2. In the Media panel, right-click the sequence and clips to relink. Select **Media ➤ Unlink Reloadable Media**.

   This deletes from the sequence the old paths that linked the clips to the media. Since the media is no longer accessible, these paths are obsolete.

3. Relink the clips to the media:

   1. Open the sequence and go to Conform tab.

2 Set the Match Criteria to *File Name*.

3 Click Set Search Location and navigate to the folder where you want to search from, and click Set.

4 When media files are found, press Link Matched Sources.

5 You sequence is now ready to be used.

NOTE Since audio files do not have a frame rate, you need to make sure to work within a project that has the same frame rate as the original project.

Managing the Size of an Archive

While the clips are always archived in a project archive, you can decide to not include the media, Timeline FX, and BFX renders and reduce the size of that archive. Batch renders and other renders, such as the result of the Paint tool, and virtual clips, are always archived in a project archive.

You need:	Archive Options Settings:		What is included in the archive:		
	Cache Media on Archive:	Archive Renders box:	Cached source media:	Uncached source media:	Timeline FX/ BFX renders:
The largest, most inclusive archive.	Enabled	Include Renders	Included	Included	Included
An archive which can easily rebuild the Timeline FX and BFX renders.	Enabled	Exclude Renders	Included	Included	Excluded
An archive where media already cached and renders are all included in the archive. Uncached media are not archived.	Disabled	Include Renders and Cache	Included	Excluded	Included
An archive where renders are archived, source media are not.	Disabled	Exclude Source Media Cache	Excluded	Excluded	Included
An archive where media already cached are included in the archive. Uncached media and renders are not archived.	Disabled	Exclude Renders	Included	Excluded	Excluded
An archive where media and renders are not archived, only the clips. This is the smallest possible project archive.	Disabled	Exclude Renders and Cache	Excluded	Excluded	Excluded

Flame also compresses the size of an archive by archiving a single copy of duplicate frames.

Multi-version Clips and Archives

If your project, or the clips you are archiving, contains multi-version clips, use the Cache Versions box to manage what is cached. This setting can impact the size of the archive.

Limiting the multi-version caches:

- All Versions: Every single version of a clip is cached on being archived.
- Used Version: Only the currently selected versions of multi-version clips are cached.

Note that in both cases, the actual clips are archived: the Cache Versions box only manages the media, not the clips.

Configuring a Data Tape Device

Before you start archiving to a data tape device, configure both the hardware and the software.

NOTE Data tape devices are only supported on Linux workstations.

Software Configuration

Before you start Flame to archive, edit the following keyword in the software initialization configuration file.

- CLIPMGTDEVICE: Define as many tape devices as you need; only those detected at start are actually listed in the Archives browser.

NOTE The software initialization configuration file is located here: */opt/Autodesk/<installed_application>/cfg/init.cfg*.

Hardware Configuration

Setting the Default Block Size for New Tape Archives

In **MediaHub ➤ Browse for Archives ➤ Archive Options** when you format a new tape archive, the default block size matches the value in the Block Size field.

Keep track of the block size value for each tape in case you need to reset the block size at a later time.

To verify the block size of your tape device:

1. In a command shell, log in as root.
2. Verify the block size of your tape device by typing:

   `mt -f /dev/st <archive device number> status`

Example:

```
mt -f /dev/st0 status
```

This command provides feedback similar to the following:

```
SCSI 2 tape drive:

File number=0, block number=0, partition=0.

Tape block size 65536 bytes.
```

```
Density code 0x40 (DLT1 40 GB, or Ultrium).

Soft error count since last status=0

General status bits on (41010000):

BOT ONLINE IM_REP_EN
```

Determining the Device Number for the Archive Device

If you receive an error message when verifying the block size, you may not be using the correct archive device number. Look in the software initialization configuration file to determine the correct number.

To determine the archive device number:

1. In a terminal, type:
   dlcfg
2. Look for the line that begins with the following:
   `ClipMgtDeviceTape, /dev/st<archive device number>`
   The archive device number should be listed on this line. If it is not, or appears to be incorrect, consult the administrator for your system.

Resetting the Block Size for an Existing Tape Archive

If you rebooted the machine or reformatted another archive with a different block size, you must reset the block size for your archive in the command shell.

To reset the block size of your tape:

1. In the command shell, log in as root.
2. Set the correct block size by typing:
   mt -f /dev/st`<archive device number>` **defblksize**`<block size of your tape device>`
   Example:
   mt -f /dev/st0 defblksize 4096
   The block size for your tape is reset and you can continue to archive to this tape.

Archiving a Project to a Tape Device Archive

Archiving a project to a tape device archive consists of opening an existing tape archive, or creating a tape archive using a tape device such as a DLT.

NOTE About *Archive and Close*. Previous versions of the application had an *Archive and Close* button. It was required because the archive header only updated when you explicitly closed the archive. But now, in the MediaHub, the archive header is systematically updated as the contents are written to the archive. This means that you no longer need to explicitly close an archive: as soon as the application is done writing to an archive, the whole archiving process is complete. The Close Archive button is there for you to dismiss an archive easily, and then open another one.

To archive a project to a blank tape:

1. Load the blank to the tape device.

2 In the MediaHub, enable Browse for Archives.

3 In the Local Devices list, double-click the tape device to open it.

4 Click Format to create a new archive.

WARNING Formatting the tape erases all data already present on it. If you want to add to an existing archive, click Open, and follow the procedure described below.

5 In the New Archive Creation window, enter the required information and click Create.

Flame creates and opens the archive in the MediaHub. You can now archive the project.

6 Set as required the options displayed in the Archive Options tab.

7 Click Archive Project.

Flame starts archiving the project. The actual duration of the archive process depends on the size of the project and on the options you enabled or disabled. To cancel the process, click anywhere on the screen; you cannot use the application for anything else while Flame archives your project.

8 Click Close Archive to finalize the archive.

To add a project to a tape already containing an archive:

1 Load to the tape device one of the tapes that make up the archive.

2 In the MediaHub, enable Browse for Archives.

3 In the Local Devices list, double-click the tape device to open it.

4 Click Open to open archive.

WARNING Do not select Format, as this would erase all data on the tape.

5 Click Archive Project. If the archive spans multiple tapes, Flame might prompt you to insert a specific tape.

Flame starts archiving the project. The actual duration of the archive process depends on the size of the project and on the options you enabled or disabled. You can cancel the process at any time by clicking anywhere on the screen; you cannot use the application for anything else while Flame archives your project.

6 Click Close Archive to finalize the archive.

Errors When Archiving

In some cases, Flame warns that it cannot archive some of the elements you had selected for your archive. You can quickly identify what was archived, or what was not archived and why, using the Media Panel.

To see the archived status of clips:

1 Open Timeline.

If the archive is still opened, you need to close it before changing tab.

2 Set Media Panel's View Mode box to Full Width, or type `Shift-~`.

3 Set the Media Panel to the List View.

4 Use the scroll bar tot display the Archived column.

If you cannot see the column, right-click a column header and select Archived from the list.

Restoring a Project from a Tape Device Archive

No two projects can use the same name on a given workstation. If you attempt to restore a project with a name already in use, you are prompted to change its name before restoring it. You can see a list of all the projects in **MediaHub ➤ Browse for Projects ➤ Local Projects**. You can delete projects from the menu **Flame ➤ Project and User Settings**. See Working with Projects (page 47).

To restore a project from a tape archive:

1. Load to the tape device one of the tapes that make up the archive.
2. In the MediaHub, enable Browse for Archives.
3. In the Local Devices list, double-click the tape device to open it and select Open.

   WARNING Do not select Format, as this would erase all data on the tape.

4. Navigate the archive, and select the project to restore. Projects are created at the root of an archive tape.
5. Click Restore Project. If the archive spans multiple tapes, Flame prompts you to insert the required tape.

   Flame starts restoring the project. The actual duration of the archive process depends on the size of the project. You can cancel the process at any time by clicking anywhere on the screen: you cannot use the application for anything else while Flame restores your project. Once restored, you can switch to the restored project using **Flame ➤ Project and User Settings**.

To restore project setups from a tape archive:

1. Load to the tape device one of the tapes that make up the archive.
2. In the MediaHub, enable Browse for Archives.
3. In the Local Devices list, double-click the tape device and click Open.

   WARNING Do not select Format, as this would erase all data on the tape.

4. Navigate the archive, and open the project that contains the setups to restore.
5. Locate and select the Project Setups folder.
6. Click Restore Setups. If the archive spans multiple tapes, Flame prompts you to insert the required tape.

   Flame starts restoring the project. The actual duration of the archive process depends on the size of the project. You can cancel the process at any time by clicking anywhere on the screen: you cannot use the application for anything else while Flame restores your project. Once restored, you can switch to the restored project using **Flame ➤ Project and User Settings**.

Archiving Clips to a Tape Device Archive

Archiving material to a file archive (as opposed to a VTR tape archive) consists of opening an existing archive file, or creating a new archive file, on a DLT tape device.

NOTE About *Archive and Close*. Previous versions of the application had an *Archive and Close* button. It was required because the archive header only updated when you explicitly closed the archive. But now, in the MediaHub, the archive header is systematically updated as the contents are written to the archive. This means that you no longer need to explicitly close an archive: as soon as the application is done writing to an archive, the whole archiving process is complete. The Close Archive button is there for you to dismiss an archive easily, and then open another one.

To archive a project to a blank tape:

1. Load the blank to the tape device.
2. In the MediaHub, enable Browse for Archives.
3. In the Local Devices list, double-click the tape device to open it.
4. Click Format to create a new archive.

   WARNING Formatting the tape erases all data already present on it. If you want to add to an existing archive, click Open, and follow the procedure described below.

5. In the New Archive Creation window, enter the required information and click Create.

   Flame creates and opens the archive in the MediaHub. You can now archive the project.
6. Drag and drop clips and folders to archive from the Media panel to the archive.

   What you add to the archive appears greyed out in the archive, indicating that it is yet to be committed to the archive: ready to be archived, but not yet copied to the archive. The Pending Archive folder can also be used as a summary of what is being archived.

   NOTE In the archive information panel, the Archive Size Pending field indicates the number of clips that will be archived, and the Archive Size Total field displays the size of the archive once those clips are archived.

7. Click Archive to start the archiving process.

   The actual duration of the archive process depends on the size of the material being archived. You cannot use the application for anything else while Flame archives the material. Cancel the process at any time by clicking anywhere on the screen.

To add clips to a tape already containing an archive:

1. Load to the tape device one of the tapes that make up the archive.
2. In the MediaHub, enable Browse for Archives.
3. In the Local Devices list, double-click the tape device to open it.
4. Click Open to open archive.

   WARNING Do not select Format, as this would erase all data on the tape.

5. Drag and drop clips and folders to archive from the Media panel to the archive.

   What you add to the archive appears greyed out in the archive, indicating that it is yet to be committed to the archive: ready to be archived, but not yet copied to the archive. The Pending Archive folder can also be used as a summary of what is being archived.

   NOTE In the archive information panel, the Archive Size Pending field indicates the number of clips that will be archived, and the Archive Size Total field displays the size of the archive once those clips are archived.

6. Click Archive to start the archiving process. If the archive spans multiple tapes, Flame might prompt you to insert a specific tape.

   The actual duration of the archive process depends on the size of the material being archived. You cannot use the application for anything else while Flame archives the material. Cancel the process at any time by clicking anywhere on the screen.

Errors When Archiving

In some cases, Flame warns that it cannot archive some of the elements you had selected for your archive. You can quickly identify what was archived, or what was not archived and why, using the Media Panel.

To see the archived status of clips:

1. Open Timeline.
   If the archive is still opened, you need to close it before changing tab.
2. Set Media Panel's View Mode box to Full Width, or type `Shift-~`.
3. Set the Media Panel to the List View.
4. Use the scroll bar tot display the Archived column.
   If you cannot see the column, right-click a column header and select Archived from the list.

Restoring Material from a Tape Device Archive

To restore material from an archive on a tape device:

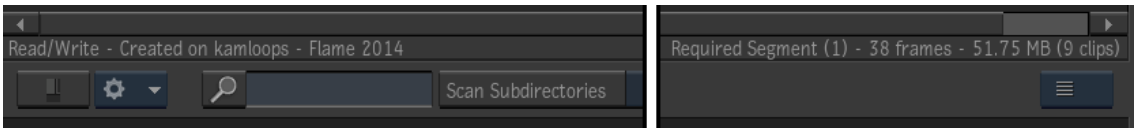
1. Load to the tape device one of the tapes that make up the archive.
2. In the MediaHub, enable Browse for Archives.
3. In the Local Devices list, double-click the tape device and click Open.

   WARNING Do not select Format, as this would erase all data on the tape.

4. Drag and drop clips and folders from the archive to the Media panel.
   Material you drag and drop appears greyed out in the Media panel, and is not restored until you click Restore.
5. Click Restore. If the archive spans multiple tapes, Flame prompts you to insert the required tape.
   The length of the restoration process depends on the size of the material being restored. You cannot use the application for anything else while Flame restores the material. Cancel the process at any time by clicking anywhere in the MediaHub.

To see how much space a selected item will take on your storage, as well as the archive segment where it is stored:

1. In the archive, select the items to restore.
2. Click the MediaHub Gear menu and select Update Selection Information.
3. The right of the MediaHub status bar displays the number of frames and the space required to restore the selected items.



   It also displays the archive segments that contain the selected items, which are required to restore the selection.

Restoring VTR Archives (Linux only)

Flame can only read VTR archives, it cannot create nor write to one.

When you change projects without exiting Flame, the video timing selected at start-up applies to the project you are switching to.

Configuring a VTR for Archive Restoration

Before you start using a VTR to restore an archive, configure both the hardware and the software.

Hardware Configuration

Verify that:

- Your VTR device is properly connected to video and audio sources and RS-422.
- The sync is properly set up.

Software Configuration

To enable Flame to read VTR archives:

1. With Flame closed, open the Flame Setup application.
2. Open the VTR tab.
3. Enable every VTR you plan on using with Flame.
4. Set VTR Archive Restore Device to VTR unless you plan on using an HDCAM VTR to restore archives, in which case you would select HDCAM. For HDCAM SR, you must select VTR.

   NOTE You can only restore an HDCAM-created archive by using an HDCAM VTR.

5. Click Apply.

   You can now restart Flame and read archives from the connected VTR.

Restoring Material from a VTR Archive

When you open a VTR archive in read/write mode, Flame performs an auto-test. The auto-test involves checking whether it can write, read, and verify the integrity of the archived material. This auto-test also scans binary and play/record delay information to verify the quality and accuracy of the video signal.

1. Insert the archive tape in the VTR.
2. In Flame, select **MediaHub ➤ Browse for Arcives**.
3. Select the VTR from the list of Local Devices.
4. Select the start timecode by doing one of the following:
   - Select Autostart if you selected this option when you created the archive (the archive rewinds the tape to find the correct start timecode).
   - Select Manual Start and enter the appropriate timecode if you selected this option when you created the archive.
5. Do one of the following:
   - Click Open Archive. Select Open Read/Write or Open Read Only from the Open Mode box.

     NOTE When you try to open a VTR archive in read/write mode, Flame checks for machine errors (for example, dirty heads). If a problem is detected, the archive does not open. If the tape is write-protected, the archive opens in read-only mode.

- Click the Open Using OTOC button and select the OTOC in the file browser. You are given the option to read slates from the tape. Answer 'Y' if you wish to see clip proxies.

The Flame cues the VTR. Once it's done reading the archive, displays its contents in the MediaHub.

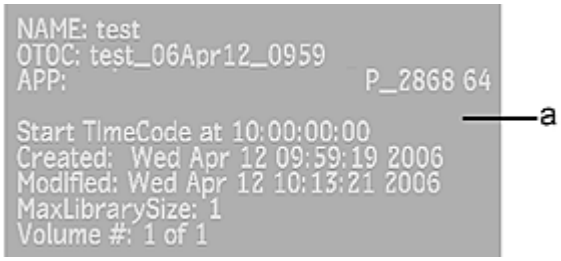
6 Restore clips one by one using drag and drop, or click Restore Project to restore a full project.

7 Click Close Archive once you are done.

Header Information Slate

All VTR archives contain a Header Info Slate, which is a single frame providing information about the archive. Located in the archive before the table of contents, the Header Info Slate provides information such as:

- Name of the archive
- Name of the Online Table of Contents (OTOC)
- Name of the workstation where the archive was created and name and version of the Autodesk application used for archive creation
- Start timecode
- Creation date and time
- Modification date and time
- Minimum size of the clip library buffer (in MB) required to restore the archive



(a) Header Info Slate example

Tips for Better VTR Archive Restores

Use the following tips to get better results when restoring from a VTR tape:

- Make sure the input from the computer to the VTR is connected properly. If it is not, yet the output from the VTR to the computer is correct, everything will appear normal when archiving and the monitor will show your material being saved. However, black or random noise is actually being saved to tape.
- Turn the video breakout box on *before* powering up the computer.
- On the video breakout box, make sure that your house sync is connected to the Sync Genlock and *not* to the Sync port in the GBRA/YUVA component. The sync should be connected to the Genlock input of the computer and be terminated with a 75-ohm terminator.
- Set the system reference of the VTR to External Analog.
- Track the VTR to get the best RF (Reference) value and error-rate level. If your VTR has an auto-tracking feature, disable it and track manually, or place it on “one shot.”
- Check the following Engineering menu settings:
  - Timecode type: Make sure the VITC & LTC timecodes are the same. If they are not, set timecode type to LTC.
  - Input format and sync: Make sure that input format and sync match the device.
  - Output format and sync: “House” is the recommended output sync.
  - The required Scan Mode option is F1.

Archiving to Multiple Volumes

If the items you select for archiving exceed the capacity of the archiving device, Flame splits the archive across multiple devices of the same type. The splitting of archives is seamless and virtually transparent to the archiving process.

The procedure for creating a multiple tape archive (an archive that is too large for a single tape) is almost the same as for creating an archive on a single tape. If the entries selected for archiving exceed the capacity of the tape, a message appears indicating that the archive requires multiple tapes. The archiving process begins for the first tape. When the first tape is full, you are prompted to insert a second tape. Insert and format this tape and continue with the archiving. This process continues for as many tapes as are required.

Opening Multi-Tape Archives

When creating large archives, you can use multiple volumes to store one archive. By splitting the contents of one archive onto separate tapes, you can keep similar types of material together in the same archive.

You can open material on any tape in a multiple tape archive but you need the last tape in the multiple tape archive sequence. The last tape contains necessary header information for locating the material you want to restore.

To open material from a multi-tape archive:

1. Insert the last tape in the tape device.
2. In the MediaHub, browse for Archives.
3. Select the tape device from the list of Local Devices.
4. Click Open Archive.
5. Follow the instruction.
   You can now restore a project or any part of the archived material.

Supplementing Archives with System Drive Backups

The recommended backup strategy for your Flame workstation is to archive project media and setups to an archive that can be saved to a remote file server or storage device. Do this as often as necessary to protect your media from storage or system failure.

The archiving approach, however, does not protect the data contained on your workstation's system drive. The system drive should not contain any media data, but it does include important project management data. In the event of hardware failure without a system drive backup, you will need to recreate this project data for each archive you restore. This can be a time-consuming and frustrating process, and is a significant issue in large SAN environments, where large quantities of media files are referenced from a shared standard FS volume. In such environments, consider using the data backup method of your choice to save key system drive information. This will facilitate restoration of multiple project archives.

Even with a system drive backup, you will still need to reinstall your software and restore archived projects in the event of a system drive failure.

There are several methods and commercial tools available for backing up system drives. Choose the method and tool that is right for you. Autodesk does not recommend any particular method or tool, and only presents

the directories of note that you should consider backing up. Depending on your requirements and available tools, you may decide to back up your full system drive or only the following critical files or directories.

File/directory	Purpose	Consequence of failure if system drive is not backed up
/opt/Autodesk/clip	Contains all clip metadata.	Metadata will have to be recreated for each valid archive that is restored.
/opt/Autodesk/project	Contains full project structures, including settings such as names, resolutions, and so on.	Project settings will have to be recreated for each valid archive that is restored.
/opt/Autodesk/user	Contains users and settings.	Users and user settings will have to be recreated.
/opt/Autodesk/sw/swdb	Contains the standard FS links database.	Media on a shared SAN volume will have to be re-imported.
/opt/Autodesk/sw	Contains all Stone® and Wire software and configuration files, including the preceding directory, swdb.	Storage and Wire settings will have to be reconfigured after installation.
/opt/Autodesk	Contains your product software, as well as all other directories described in this table.	See consequences for all previous items. Back up this directory if you want to be sure to restore all critical data, and do not mind including some superfluous files (such as old unused versions of the application.)

Browsing an Archive Offline

Each archive includes a header file, which is located at the beginning of the archive and contains the metadata necessary for properly restoring the archive. When you create an archive, two versions of the same table of contents are created: one as an ASCII file, and another as an HTML one. Created by default in `/opt/Autodesk/archive`, and displayed in the list of Archives in **MediaHub ➤ Browse for Archives**, they can be used in cases where the header information becomes corrupted.

You can view the table of contents with the ASCII or HTML TOC.

To view the contents of an archive using its HTML TOC:

1. Open **MediaHub ➤ Browse for Archives**.
2. Select the archive you want to view from the Archives list.
3. Click View Content.

   A Web browser opens the HTML TOC, allowing you to browse the contents of the archive without actually opening it.

To view contents of an archive using its ASCII TOC:

1. Open the ASCII version of the table of contents using a text editor to view the contents of an archive without opening it. ASCII TOC filenames have the following format:

<archive name>_<creation date>.atoc

Code	Meaning
P	Project
S	Project setups
W	Workspace or Shared folder
X	Desktop snapshots
L	Folder
R	Reel
C	Clip
E	Source clip
M	Media library
B	Batch snapshots
F	Snapshot folder
D	Desktop

Online Tables of Contents

The table of contents lists the contents of the archive, as well as information such as the order in which clips are assembled on the archive, clip IDs, transitions, and timecodes. When you restore an archive with the table of contents, Flame uses this information to restore the material.

A copy of the table of contents is saved in the filesystem. This copy is referred to as the Online Table of Contents (OTOC). You can open an archive in read-only mode using the OTOC. In read-only mode, you can load but not save or delete entries from the archive. With VTR archives, you can open an archive in read-write mode using the OTOC, and save and delete entries from the archive.

The OTOC is useful for:

- Recovering material in an archive when the table of contents on the medium is corrupted.
- Viewing the contents of an archive without opening it, which is often faster than actually opening the archive.

When an OTOC is created, Flame creates an ASCII text copy of it (ATOC) as well as an HTML and XML copy. Use the ASCII and HTML copies to view the contents of an archive without opening it. For example, use them to view the contents of a VTR archive without connecting to the VTR.

Use the XML TOC to easily populate a database with information about your archives.

The OTOC, ATOC, HTML, and XML table of contents are saved by default to `/opt/Autodesk/archive`. Or define the location by adding the following token to the `init.cfg` file:

ArchiveLibrary`<directory>`

where `<directory>` is the location for storing your archives.

The OTOC is updated each time you close the archive.

Deleting Tables of Contents

To prevent the loss of important data, Flame never overwrites the previously created tables of contents. If the archive becomes obsolete or if you are sure that older versions of an archive OTOCs are obsolete, you can delete tables of contents. The following name formats are used for the tables of contents.

Type	Format
Online (two files)	`<archive name>_<creation date>`.otoc `<archive name>_<creation date>`.otocx
ASCII	`<archive name>_<creation date>`.atoc
XML	`<archive name>_<creation_date>`.xml
HTML (two or more files)	`<archive name>_<creation_date>`.html `<archive name>_<creation_date>`/*

where `<archive name>` is the name that you type in the Name field when you create the archive.

Recovering a Corrupted Archive

If the table of contents of an archive on tape is corrupted, Flame may not be able to read it. If this happens, open the archive using the OTOC.

Once you open the archive, you can restore the entire contents of the archive and save it to a new tape. Do this, for example, if the tape has been damaged.

For VTR archives, you can also overwrite the corrupted table of contents by saving a single frame to the archive. If the table of contents is successfully overwritten, you can open the archive using the Open button.

To restore the archive:

1. Open the archive.
   If the table of contents is corrupted, the following message appears:
   `CLIP MGT: Warning! Cannot read archive header. Select online TOC?`
2. Click Confirm to open the OTOC.
   The file browser appears.
3. Select the OTOC for the archive you are trying to open and click Confirm.
4. If the archive is on a VTR, a prompt asks if you wish to retrieve the slates (proxies).
   After a moment, the archive appears.

NOTE When you open an archive with the OTOC, proxies only appear if the archive is on a VTR, and you answered Yes to the prompt. For all other types of archives, proxies appear as black images.

5 Select the entire contents of the archive. Make sure you have sufficient space on your storage for the contents of the archive.

6 Click Restore.

7 When the restore is complete, click Close.

The restored clips appear in the selected clip library.

8 Re-archive the material onto a new tape.

VTR Only: To overwrite a corrupted table of contents:

1 Open the archive.

If the table of contents is corrupted, the following message appears:

```
CLIP MGT: Warning! Cannot read archive header. Select online TOC?
```

2 Click Confirm to open the OTOC.

The file browser appears.

3 Select the OTOC for the archive you are trying to open and click Confirm.

A prompt asks if you wish to retrieve the slates (proxies).

4 Click Y to see proxies in the archive or N to have no proxy images.

After a moment, the archive appears.

5 From the Library Type box, select a clip library.

6 Select a reel entry by `Alt`-clicking the reel and then click Archive.

NOTE When you select a reel, no media is appended to the archive but the header is still forced to update.

7 Click Close to close the archive.

8 Click Open to open the archive.

If the archive still does not open using the table of contents on the tape, restore the entire archive.

Restoring Legacy Archives

In earlier versions of Flame and Smoke, archives did not include all the metadata that the current version needs to restore archives with the correct scan mode and aspect ratio. The Default Resolutions menu makes it possible to assign the scan mode and aspect ratio preferences to older archives. The information you need to assign depends on the software version in which the archive was created. Assign the information using the Default Resolutions menu. Use the following table to determine which information you need to assign to your old archive.

Archived clips prior to:	Information you must assign:
Smoke 6.0, Fire® 6.0, Inferno 5.5, Flame 8.5, Flint 8.5, and Backdraft 5.5	Scan mode
Smoke 5.2, Fire 5.2, Inferno 5.0, Flame 8.0, Flint 8.0, and Backdraft 5.0	Scan mode and aspect ratio

The parameters listed in the Default Resolutions menu act as default values for all listed resolutions. For example, if you want to change the default values for PAL clips, you must change the values for the PAL

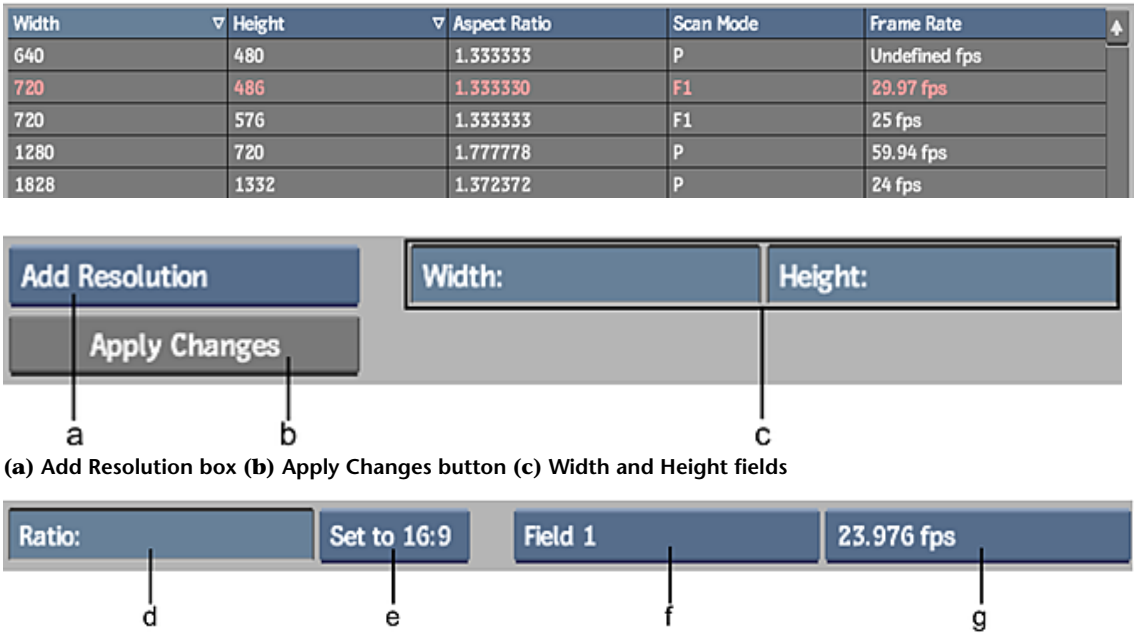
entry, which is the entry that has a width value of 720 and a height value of 576. If you are working with a PAL archive created in an earlier version, the aspect ratio would have automatically been set to 1.3333, but you may want to restore all your PAL archives with 16:9 aspect ratio. You can change this setting by changing the aspect ratio to 16:9 for the PAL entry.

NOTE You should only restore an old clip at its original resolution. If you restore versions of the same clip at different resolutions, you may encounter problems with the clip.

To add a default resolution:

1 In the MediaHub, browse for Archives and click Default Resolutions.

The Default Resolutions list and menu are displayed.



(a) Add Resolution box (b) Apply Changes button (c) Width and Height fields

(d) Aspect Ratio field (e) Aspect Ratio box (f) Scan Mode box (g) Frame rate box

2 Add new preferences to correspond to a specific clip resolution contained in the archive that you plan to open.

3 Click Add Resolution.

4 Click Apply Changes.

The new archive preference is added to the list. The next time you restore an archive missing either Scan mode or aspect ratio, Flame knows how to process that material.

TIP When you decide to open a VTR legacy archive, set the Open Mode box to Open Read Only to keep you from mixing new content with the old.

Pre-2017 Extension 1 Flame Projects

Starting with the 2017 Extension 1 version of the application, the Classic Engine rendering option is no longer available. Project Restore restores every project as a Flame Reactor-enabled project, even if that project did not use Flame Reactor originally (as is that case with every pre-2013 extension 1 projects). Due to differences in rendering engines, there might be differences between what was rendered then and what is rendered now, and adaptors might be added to a restored Batch Schematics to convert it to the supported 16-bit float bit depth.

Deleting Content from an Archive

To ensure data integrity and prevent archive corruption, you cannot delete entries from an archive.

MediaHub Reference: Browsing for Archives

Archives Options

With No Opened Archive

New Archive button Creates a new archive in the location displayed in the file browser. Opens the New Archive Creation dialogue box.

View Content button Displays basic information about the contents of the selected archive, in a web browser. This does not open the archive.

Open Archive button Opens the selected archive or device.

With an Opened Archive

Close Archive button Closes the archive and returns to the file browser.

Archive Project button Adds the current project to the archive; this includes every Workspace of the project, libraries, Shared libraries, and the Desktop; also includes all the setups.

Clear Pending Tasks button Clears the Pending Archive or Pending Restore folders. This does not delete material already archived or in the Media panel: it only clears out the queue.

Archive/Restore button Commits the Pending folders, either to archive or restore material clips. Nothing is archived or restored until you click this button.

The folder Pending Restore displays all that will be restored. The folder Pending Archive displays all that will be archived. Only one operation is permitted at any given time: you either restore or archive, you cannot do both simultaneously.

Archive/Restore Setups button Click to archive or restore all of the application's modules setups. You must be at the root of the archive to archive setups. You must select a Projects Setup folder to restore setups.

File-based Archive Options

Linked Archive Option box Select how archive paths are treated when restoring an archive. Use Archived Path is best used when you are restoring the archive on the workstation used to create the archive. Select Convert to Local Path in other cases.

Archive Verification box Verify Source Media checks the integrity of the source media before adding it to the archive. In case of missing source media, the archive proceeds without the clip. Verify Archived Data checks the integrity of the data as it is being written to the archive: archiving stops as soon as it detects an error.

Verify Source Media validates all the media, which can be time consuming.

The Archived column from the Details view of the Media Panel displays either the last time the clip was successfully archived, or why it failed to archive.

Cache Media on Archive button Enable to archive all source media as cached source media. This means that all media is cached in the archive in an uncompressed format. Use the Archive Renders box to decide whether or not to archive Timeline FX renders. Disable to archive already cached source media, but to archive

only links for source media that are not already cached. In this case, use Archive Renders to decide how to handle Timeline FX renders and already cached media.

Archive Renders box With Cache Media on Archive enabled, select whether or not to archive the Timeline FX and BFX renders. With Cache Media on Archive disabled, select how to handle Timeline FX and BFX renders, and already cached source media.

Cache Version box Controls how the media from multi-version clips will be archived. All Versions: caches the media of every versions in a multi-version clip. Used Version: caches only the media from the version currently used in the application. This option affects only the media, as the multi-version clips are always completely archived.

Default Resolutions button Legacy option. Opens the Default Resolutions List menu, used by the application to restore legacy archives from Smoke 6.0 and Flame 8.5 (or earlier).

These versions did not store resolutions and associated pixel aspect ratios, and the Default Resolutions table fills that gap.

Silent Mode button Enable to silently skip and ignore archiving errors such as missing media from the Cache On Archive option. Ignored errors are printed to the application log files and the shell. If errors were ignored during the archive, a message box is displayed at the end of the process.

Generate Proxies Enable to automatically generate proxies for restored clips, using the Proxy parameters defined in the restored Project, or the settings of the current project when restoring individual clips.

VTR Device Options

Auto/Manual Start box Select Auto Start to have the application rewind the tape and then go forward to try and discover the location of an archive. Select Manual to enter the start timecode and get to the archive faster.

Timecode field Displays the start timecode. Editable.

Rewind On Close button Rewinds the tape after when the archive is closed.

Open Mode box Select whether to open in read only mode or in read/write mode.

Archiving from the Command Line

You can archive and restore material from the command line, using `flame_archive`, found in`/opt/Autodesk/io/bin`.

NOTE In flame_archive, Wiretap paths are entered without a trailing "/".

A possible workflow you might try is to archive a project daily, and then restore a specific clip, as required.

Archive the project:

1. `cd /opt/Autodesk/io/bin`
2. Create and format the archive `daily_archive` with the default 1 GB segment. You can use -i to specify a segment capacity, in GB, where 1 GB = 1024 MB.

   `./flame_archive -f -F /usr/tmp/daily_archive`
3. List the project hosted on the workstation. This is useful to validate the name of the project to archive.

   `./flame_archive -l`
4. Append the `ProjectOne` project to `daily_archive`.

   `./flame_archive -a -P ProjectOne -F /usr/tmp/daily_archive`

Once the process completes, you have archived the whole project in the `daily_archive`.

Restore a single clip from the `daily_archive`:

1. `cd /opt/Autodesk/io/bin`
2. List the contents of `daily_archive`.

   `./flame_archive -l -F /usr/tmp/daily_archive`
3. Using the list from the above command, locate the clip to restore, and build its filepath. This filepath is used in the following command to restore the `newNoise` clip to the destination described using the Wiretap path.

   ```
   ./flame_archive -r -F /usr/tmp/daily_archive -S '/daily_archive/Archive Session - 2014/12/01
   14:54:24/projectOne/Workspace (projectOne)/Libraries/Default Library/newNoise' -E
   /projects/projectOne/988110ac_547cc27f_0005859a/988110ac_547cc27f_0005859c/988110ac_547cc283_000312c0
   ```

   NOTE Get the Wiretap path using `/opt/Autodesk/wiretap/tools/current/wiretap_print_tree`

You an also pass a text file containing the clips to restore to `flame_archive`. That text file lists one clip to restore on each line.

The above approach is fine for a manual restore, but for a programmatic approach, try using: `./flame_archive -l -x`, where -x outputs the archive contents in XML.

Some additional notes:

- Using the -N option forces the archiving of the media from clips, and of the renders. Leave out -N to minimize the size of the archive: only Timeline FX renders and already cached media are archived.
- -p is similar to setting Archive Verification to Verify Source Media in the Flame UI (**MediaHub ➤ Archive**). When using -g (on pre-verify error resume), consider enabling -v to know what failed.
- -y is similar to setting Archive Verification to Verify Archived Media in the Flame UI (**MediaHub ➤ Archive**).
- The @UUID method to locate entries within an archive is now deprecated and should be avoided.

About flame_archive Switches

USAGE: `flame_archive <operation> <modifiers> <options>`

```
flame_archive -h | --help
flame_archive -a [-N] [-k] [-p] [-y] [-g] -E <WIRETAP_PATH_SOURCE> -F <PATH_TO_FILE>
flame_archive -e [-E <WIRETAP_PATH_SOURCE>] [-F <PATH_TO_FILE] [-m
All|MetaData|Video|Audio]
flame_archive -f -F <PATH_TO_FILE> [-n <NAME>] [-t <COMMENT>] [-i <SIZE_IN_GB]
flame_archive -l [-F <PATH_TO_FILE> [-x] [-S <PATH_IN_ARCHIVE>]] [-E <WIRETAP_PATH>]
flame_archive -r -F <PATH_TO_FILE> -S <PATH_TO_ENTRY_IN_ARCHIVE> [-E
<TARGET_WIRETAP_PATH>]
```

Examples:

Format archive file

```
flame_archive -f -F /usr/tmp/archive1
```

Add Wiretap path to archive`arch1`

```
flame_archive -a -E
/projects/TestProject/a38310ac_53a82f6b_0006a1c1/a38310ac_53a82f6b_0006a1c3/a38310ac_53a82f77_000079d7/a38310ac_53a82ff6_000d6100
 -F /tmp/arch1
```

Archive project `prj1` **in archive** `arch1`

```
flame_archive -a -P pj1 -F /tmp/arch1
```

Restore project `prj1` **from archive** `arch1`

```
flame_archive -r -F /tmp/arch1 -S '/Archive Session - 2014/03/12 23:43:40/prj1'
```

Lists the contents of the framestore of the local host

```
flame_archive -l
```

List the contents of archive `arch1` **; output in XML**

```
flame_archive -l -F /usr/tmp/arch1 -x
```

List the contents of folder `MyFolder` **from project** `project1`

```
flame_archive -l -P project1 -E /wksp/Libraries/MyLib/MyFolder
```

Provide a size estimate for the result of archiving project `project1`

```
flame_archive -e -P project1
```

The different `flame_archive` operation modes:

- -a, --archive

  Archive clips to a file archive. This requires the Wiretap path to the material to archive, and the file path to the archive file. The archive file must already exist, either created from within Flame, or created using `flame_archive`. The archiving process is recursive: there are no provisions made for excluding material from the archive. If you need to archive specific clips, you must run `flame_archive` for each individual clip. -F specifies the file to use, -E the material to archive, as a Wiretap path (without a trailing '/').
- -e, --estimate

  Estimate the space required to store or restore material to and from an archive. The estimate is recursive.
- -f, --format

  Format an archive file, default size being 1 GB (1024 MB).
- -l, --list

  Lists contents of a project or of an archive, depending on the target.
- -r, --restore

  Restore clips from a file archive. Requires the source file archive from which to restore, and the path of the entry to restore. Do not specify a Wiretap path target when restoring a project: flame_archive restores a project so that it is accessible from Flame as a regular project.

Optional modifiers to use with the different `flame_archive` modes:

- With -l:
  - -x, --xml: Formats in XML the list of contents of the archive file.
- With -f:
  - -n, --name <NAME>: Uses NAME as the name of archive when formatting. Default is 'Archive'.
  - -t, --comment <TEXT>: Uses TEXT as the comment when formatting. Default is empty.

- -i, --capacity <SIZE>: Uses SIZE (in GB, so if SIZE=10 archive is 10240 MB) for the segment capacity when formatting an archive. Default is 1 GB.

- With -a:
  - -N, --normal: Enables 'Normal' mode, default is 'Compact'.
  - -k, --linked: Enables 'Archive Media as Uncached' mode; default is 'Cache Media on Archive'.
  - -p, --preVerify: Verifies source data before archiving, if missing, stop archiving.
  - -y, --postVerify: Verify archive data after archiving.
  - -g, --ignore: Ignores preVerify failure and continue archiving.

- With -e:
  - -m, --mode <MODE>: One of 'All', 'MetaData', 'Video', 'Audio', for size estimation.

- With any option:
  - -v, --verbose

Playing Back Media

9

Playback

Quick Access to the Players

You can quickly access the players by the following shortcuts:

- `Esc`: Player for selected clip. When re-applied, this shortcut returns to the previously selected view.
- `Ctrl-Esc`: Fullscreen player.
- `Ctrl-1`: Player view.
- `Ctrl-2`: Source-Sequence player.
- `Ctrl-3`: Triptych player.
- `Ctrl-4`: Trim View

Playing Back Clips in the Full Screen Player

To play back clips in the full screen player:

1. Click the Timeline tab.
2. Do one of the following to select the Player:
   - Click the Layout button and select Player.
   - If you've enabled Layout Selection Overlay in the preferences, swipe the right swipe bar, then select Player.
   - Double-click on the clip in the media library.

   The clip appears in the Player viewer.
3. Click the Full Screen button.
   The player displays as a full screen.

To return to the previous display size, click the Full Screen button.

TIP While in the full screen player, the Player Option Box lists Show Play Reel. This is the only viewer that provides this option. Selecting this option opens a scrolling clip list of available clips under the control bar. With the play reel displayed, you can re-order, duplicate, or delete clips. These actions are available gesturally, in the context menu, or with keyboard shortcuts.

Jogging and Panning a Clip

You can pan and zoom in the player, tryptich and source sequence viewport. By selecting the Viewing Mode button you can select the Zoom or Pan pointers.

- The Zoom pointer increases the zoom of the clip with a left-to-right motion, and will decrease the zoom in a right-to-left motion.
- The Pan pointer lets you grab the clip and move it around the viewer.

NOTE The pan and zoom values of the players are kept until a clip from a different resolution is loaded; you can change the clip selection without losing the pan and zoom values as long as the clip resolution is maintained. If changing to a clip of a different resolution, the clip will be resized into the viewer.

To jog through a clip:

1. You can jog through a clip by doing one of the following:
   - Drag the positioner along the timebar.

     When you jog through an edit sequence (a series of clips edited together), you can constrain the jogging to the current clip in the sequence by holding down Command while jogging.
   - Use the arrow keys on your keyboard.

     By default, pressing the left arrow and right arrow keys steps frame-by-frame.
   - Place the pointer in the viewer, above the control bar.

     The cursor will change to indicate it can jog/shuttle the clip. The further from the center of the clip the cursor is placed, the faster the shuttling.

Switching the Playback Focus between the Source and Sequence Clips

You can switch the focus between the source and sequence viewers by clicking on either of the Source-Sequence viewers, by selecting a track on the timeline, or by selecting the green (source) or red (sequence) tabs above the Player, Source-Sequence, or Triptych viewers.

Clicking the Lock icon that appears when the pointer is hovering over the source-or-sequence tab will lock the focus on that tab. Clicking again unlocks the viewer.

NOTE Changing the focus through the timeline will also switch the focus between the Source and Sequence viewers. Changing the focus through the Source-Sequence viewer, however, will not change the focus in the timeline. In other words, the player doesn't drive the timeline, but the timeline drives the player.

To switch the playback focus between the source and sequence clips in the Source-Sequence Player:

1. Click the Timeline tab.
2. Do one of the following to select the Source-Sequence Player:
   - Click the Layout button and select Source-Sequence Player.

- If you've enabled swipe in the preferences, swipe the right swipe bar, then select Source-Sequence Player.

3 Switch viewers by clicking in either the Source or Sequence viewer, or on the green (Source) or red (Sequence) tab.

The viewer with the focus has a white outline.

Comparing Three Shots in the Triptych Player

The triptych player provides three players, side-by-side, which you can use to simultaneously view the same clip in different contexts. For example, you can set the middle player to play the clip, the left player to show the clip's incoming frame of the current element in the timeline, and the right player to show the clip's outgoing frame.

To compare three shots in the triptych player:

1 Click the Timeline tab.

2 Select Triptych from the View Mode box.

3 Select a clip from the Media Library.

The clip appears in the triptych player in all three viewers. Coloured positioners help you identify clips in Triptych view (green, yellow, orange from left to right).

4 Use the Positioners Options box to set the behaviour of the viewports (such as All Synced or All Free).

TIP Triptych view is also available from all editors with multiview capabilities. You can set the positioners in the Playback Options box.

Play Back Speed Indicators

As you play clips, the positioner in the image window time bar changes colours if frames are dropped. The colours indicate the following:

Colour:	Indicates:
Yellow	Real-time playback with no dropped frames.
Orange	Real-time playback with no dropped frames of soft-imported clips.
Green/Blue	Alternating dropped frames. Green indicates the first and subsequent alternating dropped frame.
Red/White	Alternating occurrences of dropped audio. Red indicates the first occurrence.

Comparing Tracks in Split Screen

Use the split screen to compare two tracks, such as an online and offline version. Generally this is used to compare a rough cut to the final assembly. By merging both tracks into a single image, you can compare the two for editing synchronization, color matching, etc.

The split screen compares versions as well as video tracks.

To compare tracks in split screen:

1. Click the Timeline tab.
2. Set the Primary and Secondary tracks. (page 437)
3. In the View Area menu bar, select Options.
4. Select Show Viewing Settings.
5. Select Side By side or the split you will use to compare the two clips in the Compare Mode/Stereo section.

   Depending on the split, you will have different attributes available to change the partition between the two clips. Modify the values in these attributes to provide the best comparison.

Creating a Subclip in the Player

You can create subclips anywhere that you have access to the source clip (i.e. Thumbnail, Player, Triptych and Source-Sequence viewers).

NOTE Remember you will need handles for trimming and effects that you apply later. It is recommended to cue the tape a couple of seconds before the desired footage starts.

To create a subclip in the player:

1. Click the Timeline tab.
2. Set the positioner to the in point.
3. Click the In button.

   The start of the sub-clip is marked.
4. Set the positioner to the end point.
5. Click the Out button.

   The end of the sub-clip is marked.

   The section of the clip that is between the In and Out marks is the sub-clip.
6. From the Clip menu, select Create Subclip.

   The subclip is created as a new clip in the Media library. It has the original name and append ("_Subclip_") with an incremented number, starting with ("001").

Accessing Clip Information

Any viewer contains the following information located at the bottom right corner of the viewer:

- Name of the viewer OR name of the clip,
- Resolution and aspect ratio of the clip,
- LUT information of the clip (if LUT has been set).

Also, by alt-clicking on a clip proxy you will see additional clip information detail

Player options

To access the options below, select the Player Options box in the View menu bar.

Select between playing all frames and playing back media in real-time mode, with or without Timeline FX. When playing back all frames, cache frames from memory to increase performance. If the current project uses real-time deliverables, select the resolution at which to play back media.

Select This:	To Do This:
Play All Frames	Play back all frames as quickly as they are rendered, even if it results in lack of audio synchronization.
Play Real-Time	Play back frames synchronized with audio. May result in some frames being dropped.
Hide Effects	Play back without any rendered effects, except Resize and Timewarp. Toggle to render all Timeline FX on-the-fly.
Cache On Playback	Cache frames to media storage while playing back.
Deliverables	Play back using the current Real-Time Deliverable.

The Player Option box also accesses additional controls and view settings.

Select This:	To Do This:
Show Audio Desk	Display audio controls.
Show Overlays	Create, save and load overlays.
Show Vectorscope	Display the Vectorscope.
Show Viewing Settings	Specify image-related options.

NOTE The Audio Desk and the Vectorscope cannot be displayed when the player is in Source-Sequence, Triptych, or Trim View mode. Overlays and viewing settings are still accessible in these modes via pop-up dialog box by clicking Edit Overlays and Edit Viewing Settings in the Player Option box.

Cue Marks

Adding Cue Marks

Cue marks are displayed in the player timebar, as well as the timeline.

To add a cue mark:

1. In the timebar, do one of the following:
   - Drag the yellow positioner to the desired frame, or
   - Click on the desired frame on the time bar.
2. Click the Cue Mark button.

A cue mark is placed in the player timebar, as well as the timeline, to indicate the marked frame. You can change the colour of a cue mark by clicking the colour pot.

To delete a cue mark:

1. Click the Timeline tab.
2. Select the specific cue mark to be deleted.
3. Click Delete Mark, or select **Marks ➤ Delete Selected Markers** from the Timeline Gear menu.

Adding In and Out Marks

In and out marks are displayed in the player timebar, as well as the timeline.

To add in and out marks in the player:

1. In the timebar, do one of the following:
   - Drag the yellow positioner to the desired frame, or
   - Click on the desired frame on the timebar.
2. Click the In or Out button.

   TIP Press `Ctrl` and click the In or Out button to delete the mark.

Navigating between Marks in the Clip

To navigate between marks in the clip:

1. Click the Timeline tab.
2. From the Timeline Gear menu, select a Go To option:
   - In Mark or Out Mark
   - For a cue mark, select the mark's label from the Mark list.

   The positioner is set to the mark.

Changing the Color of Cue Marks in the Clip

You can set the color of the current Player cue mark to one of your preference.

NOTE To cancel the color selection, click on the Close (x) button in the Color Picker window.

The Color Picker window will close and the color swath will revert to the previous color.

To change the color of cue marks:

1. Click the Timeline tab.
2. On the View Area menu bar, click on the color swatch to the right of the Mark button.
   A Color Picker window appears.
3. Do one of the following:
   - Click the Pick button.

The cursor changes to a color picker, which you can use to select a color from one of the three color columns.

- Drag the colour sliders.

4 Click Ok to apply the color to this and any future cue marks.
The color swatch changes to reflect the color selected.

Playback Options

Adjusting the Viewing Transform in Batch

You can change the display of an image by selecting from the Viewing Rules that have been configured in Colour Management Preferences.

By modifying the Viewing Transforms in Viewing Settings, you can change the way the clip is displayed in the viewport.

You may press Shift+Alt+[0-9] to select individual Viewing Rules, or press Shift+Alt+ the plus or minus keys to cycle through them. You may press Shift+Alt+~ to completely bypass (or re-enable) viewport colour management.

To set up the image display viewer in the tools:

1 Click the View button.
2 Modify the display attributes.
The viewer updates the display with your changes.

Adjusting the Viewing Transform in the Player

You can change the display of an image by selecting from the Viewing Rules that have been configured in Colour Management Preferences.

By modifying the Viewing Transforms in Viewing Settings, you can change the way the clip is displayed in the player. Settings are persistent throughout different Player modes, depending on the timeline tab type (Source or Sequence).

NOTE In the case of the Triptych Player, the middle image window shares this persistency; while the left and right image windows each can have their own settings.

You may press Shift+Alt+[0-9] to select individual Viewing Rules, or press Shift+Alt+ the plus or minus keys to cycle through them. You may press Shift+Alt+~ to completely bypass (or re-enable) viewport colour management.

To select a Viewing Rule in the player:

1 Click the Timeline tab.
2 Select the Player.
3 In the View Area menu bar, select Options.
4 Select Show Viewing Settings.
5 Modify the Viewing Transform attributes.
The viewer updates the display with your changes.

Viewing Image Overlays in Batch

To view image overlays in Batch:

1. Click the Grid button.
2. Set your preference for overlay attributes.
   The new overlay attributes are set.

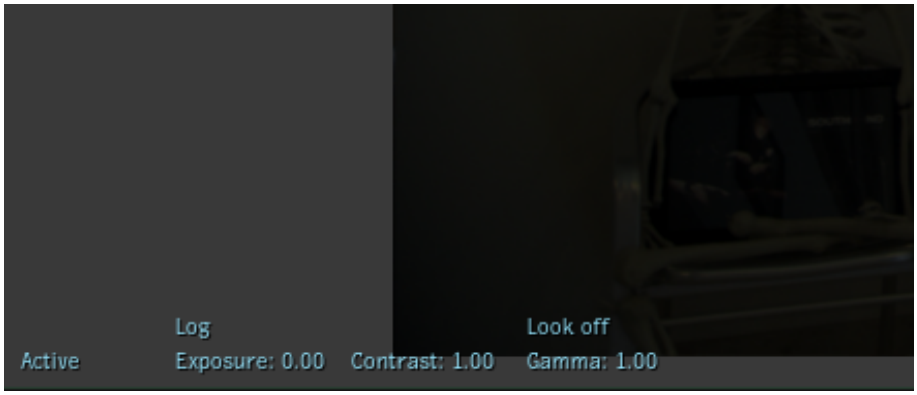
Viewing Image Overlays in the Player

To view image overlays in the player:

1. Click the Timeline tab.
2. Select the Player.
3. In the View Area menu bar, select Options.
4. Select Show Overlays.
5. Set your preference for overlay attributes.
   The new overlay attributes are set.

Modifying Exposure, Contrast, and Gamma in the Image Display

You can modify the overall brightness of images displayed on the workstation or broadcast monitor, as well as the contrast between light and dark colours. Note that these settings affect only the display, and do not modify the underlying colour values of the clips.



To modify exposure and contrast interactively in an image window

1. Click in an image window to make it active.
2. Do any of the following:
   - Press and hold Shift+E while clicking and dragging the mouse to modify the exposure offset (overall brightness). Dragging to the right increases values and dragging to the left decreases them.
   - Press and hold Shift+C while clicking and dragging the mouse to modify the contrast between light and dark colours.
   - Press and hold Shift+W while clicking and dragging the mouse to modify the gamma.
   - Click Reset to restore the default values.

- Press Ctrl+Shift+~ to temporarily bypass these adjustments.

To modify exposure, contrast, and gamma using the Image Display settings

1. Access the Image Display Viewer settings.
2. Do either of the following:
   - Edit the Exposure slider to modify overall brightness.
   - Edit the Contrast slider to modify the contrast between light and dark colours.
   - Edit the Gamma slider to modify highlight and shadow contrast.

Enabling Aspect Ratio Display in the Player

When you work with clips that have a resolution that does not use square pixels, such as NTSC or PAL, you can adjust the display so that it appears with the correct aspect ratio. By default, the adjusted image is sent to the broadcast monitor. You can switch the adjusted display from the broadcast monitor to the Player.

To enable the aspect ratio display in the Player:

1. Click the Timeline tab.
2. In the View Area menu bar, select Options.
3. Select Show Viewing Settings.
4. Enable the Use Ratio button.

   NOTE The Use Ratio button is only active if you are viewing a clip with a resolution that does not use square pixels.

   The original aspect ratio of the clip is applied to the clip.

Changing Playback Scan Mode Options in the Player

Use the playback scan mode option to define how you want the clip to play and display. By default, everything will be automatically set to the correct scan mode.

To set playback mode in the Player:

1. Click the Timeline tab.
2. Select the Player.
3. In the View Area menu bar, select Options.
4. Select Edit Viewing Settings.
5. Click on the Scan Mode button.

   NOTE The default choice is the clip's native scan mode.

6. Adjust the display to the desired scan mode format.
7. Click Done.
   The player displays the clip in that scan mode.

Displaying a Clip in the Broadcast Monitor

The broadcast monitor outputs a complete image or a selected viewport. When you select multiple clips, the broadcast monitor displays the first clip in a selection.

To set broadcast monitoring preferences:

1. Click **Flame ➤ Preferences ➤ Broadcast Monitor**.
2. Enter the preferences for the broadcast monitor in the Broadcast Monitor, Broadcast Multiview and Broadcast LUT sections.

   Your preferences will be set for broadcast monitoring.

Use the Show Selected Item option in the Broadcast Monitor box to display the clip you selected in the broadcast monitor. Use the Broadcast Monitor buttons to resize and set the proportions of the clip. If the clip is zoomed in on the application monitor, the clip is still displayed in its entirety on the broadcast monitor.

When the Show Selected Item option is selected, you can manage additional preferences.

TIP See Broadcast Monitor Preferences (page 1935) for details on all of the pertinent preferences.

Broadcast Monitor Supported Timings on Linux Workstations

The following broadcast monitor timings are supported on Flame Linux workstations.

AJA Kona 4

Format	Resolution	Supported Frame Rates	4:2:2 YCC 10-bit 1.5G	4:2:2 YCC 10-bit 3G HFR	4:4:4 RGB 10-bit 1.5G Dual Link	4:4:4 RGB 10-bit 3G	4:4:4 RGB 12-bit 1.5G Dual Link	4:4:4 RGB 12-bit 3G	Stereo 4:2:2	HDMI Out
SD	720 x 486	29i	1 x SDI		2 x SDI				2 x SDI	Yes
SD	720 x 576	25i	1 x SDI		2 x SDI				2 x SDI	Yes
HD	1280 x 720p	50p, 59p, 60p	1 x SDI		2 x SDI	1 x SDI	2 x SDI	1 x SDI	2 x SDI	Yes
HD	1920 x 1080	23p, 24p, 25p, 29p, 30p, 50i, 59i	1 x SDI		2 x SDI	1 x SDI	2 x SDI	1 x SDI	2 x SDI	Yes
HD	1920 x 1080	23psf, 24psf, 25psf	1 x SDI		2 x SDI	1 x SDI	2 x SDI	1 x SDI	2 x SDI	No
HD HFR	1920 x 1080	50p, 59p, 60p		1 x SDI						Yes

Format	Resolution	Supported Frame Rates	4:2:2 YCC 10-bit 1.5G	4:2:2 YCC 10-bit 3G HFR	4:4:4 RGB 10-bit 1.5G Dual Link	4:4:4 RGB 10-bit 3G	4:4:4 RGB 12-bit 1.5G Dual Link	4:4:4 RGB 12-bit 3G	Stereo 4:2:2	HDMI Out
2K DCI	2048 x 1080	23p, 23psf, 24p, 24psf, 25p, 25psf, 29p, 30p	1 x SDI		2 x SDI	1 x SDI	2 x SDI	1 x SDI	2 x SDI	Yes
2K DCI HFR	2048 x 1080	50p, 59p, 60p		1 x SDI						Yes
UHD	3840 x 2160	23p, 24p, 25p, 29p, 30p	4 x SDI			4 x SDI		4 x SDI		Yes
UHD HFR	3840 x 2160	50p, 59p, 60p		4 x SDI						Yes 4:2:0
4K	4096 x 2160	23p, 24p, 25p, 29p, 30p	4 x SDI			4 x SDI		4 x SDI		Yes
4K HFR	4096 x 2160	50p, 59p, 60p		4 x SDI						Yes 4:2:0

AJA Kona 3G Quad

Format	Resolution	Supported Frame Rates	4:2:2 YCC 10-bit 1.5G	4:2:2 YCC 10-bit 3G HFR	4:4:4 RGB 10-bit 1.5G Dual Link	4:4:4 RGB 10-bit 3G	4:4:4 RGB 12-bit 1.5G Dual Link	4:4:4 RGB 12-bit 3G	Stereo 4:2:2	HDMI Out
SD	720 x 486	29i	1 x SDI		2 x SDI				2 x SDI	Yes
SD	720 x 576	25i	1 x SDI		2 x SDI				2 x SDI	Yes
HD	1280 x 720p	50p, 59p, 60p	1 x SDI		2 x SDI	1 x SDI	2 x SDI	1 x SDI	2 x SDI	Yes
HD	1920 x 1080	23p, 24p, 25p, 29p, 30p, 50i, 59i	1 x SDI		2 x SDI	1 x SDI	2 x SDI	1 x SDI	2 x SDI	Yes
HD	1920 x 1080	23psf, 24psf, 25psf	1 x SDI		2 x SDI	1 x SDI	2 x SDI	1 x SDI	2 x SDI	No
HD HFR	1920 x 1080	50p, 59p, 60p		1 x SDI					2 x SDI	Yes

Format	Resolution	Supported Frame Rates	4:2:2 YCC 10-bit 1.5G	4:2:2 YCC 10-bit 3G HFR	4:4:4 RGB 10-bit 1.5G Dual Link	4:4:4 RGB 10-bit 3G	4:4:4 RGB 12-bit 1.5G Dual Link	4:4:4 RGB 12-bit 3G	Stereo 4:2:2	HDMI Out
2K DCI	2048 x 1080	23p, 23psf, 24p, 24psf, 25p, 25psf, 29p, 30p	1 x SDI		2 x SDI	1 x SDI	2 x SDI	1 x SDI	2 x SDI	Yes
2K DCI HFR	2048 x 1080	50p, 59p, 60p		1 x SDI					2 x SDI	Yes
UHD	3840 x 2160	23p, 24p, 25p, 29p, 30p	4 x SDI			4 x SDI		4 x SDI		No
4K	4096 x 2160	23p, 24p, 25p, 29p, 30p	4 x SDI			4 x SDI		4 x SDI		No

Broadcast Monitor Supported Timings on Mac Workstations

The following broadcast monitor timings are supported on Flame Mac workstations.

NOTE When using an AJA Io 4k or Io XT device, make sure to use the 4K firmware, and not the UFC firmware. You can check or change the firmware in the AJA Control Panel under the Firmware tab.

AJA Io 4K

Format	Resolution	Supported Frame Rates	Signal	422	444 Dual Link 3G	4:4:4 Single Link 3G	4:4:4 12-bit Dual Link 1.5G	4:4:4 12-bit 3G	Stereo 4:2:2	HDMI Out
SD	720 x 486	29i	1.5G	1 x SDI	2 x SDI	N/A	N/A	N/A	2 x SDI	Yes
SD	720 x 576	25i	1.5G	1 x SDI	2 x SDI	N/A	N/A	N/A	2 x SDI	Yes
HD	1280 x 720p	50p, 59p, 60p	1.5G	1 x SDI	2 x SDI	1 x SDI	2 x SDI	1 x SDI	2 x SDI	Yes
HD	1920 x 1080	23p, 24p, 25p, 29p, 30p, 50i, 59i	1.5G	1 x SDI	2 x SDI	1 x SDI	2 x SDI	1 x SDI	2 x SDI	Yes

Format	Resolution	Supported Frame Rates	Signal	4:2:2	4:4:4 Dual Link 3G	4:4:4 Single Link 3G	4:4:4 12-bit Dual Link 1.5G	4:4:4 12-bit 3G	Stereo 4:2:2	HDMI Out
HD	1920 x 1080	23psf, 24psf, 25psf	1.5 G	1 x SDI	2 x SDI	1 x SDI	2 x SDI	1 x SDI	2 x SDI	No
HD HFR	1920 x 1080	50p, 59p, 60p	3 G	1 x SDI	N/A	N/A	N/A	N/A	N/A	Yes
2K DCI	2048 x 1080	23p, 23psf, 24p, 24psf, 25p, 25psf, 29p, 30p	1.5 G	1 x SDI	2 x SDI	1 x SDI	2 x SDI	1 x SDI	2 x SDI	Yes
2K DCI HFR	2048 x 1080	50p, 59p, 60p	3 G	1 x SDI	N/A	N/A	N/A	N/A	N/A	Yes
UHD	3840 x 2160	23p, 24p, 25p, 29p, 30p	1.5 G	4 x SDI	N/A	N/A	N/A	4 x SDI	N/A	Yes
UHD	3840 x 2160	50p, 59p, 60p	3 G	4 x SDI	N/A	N/A	N/A	4 x SDI	N/A	Yes
4K	4096 x 2160	23p, 24p, 25p, 29p, 30p	1.5 G	4 x SDI	N/A	N/A	N/A	4 x SDI	N/A	Yes
4K	4096 x 2160	50p, 59p, 60p	3 G	4 x SDI	N/A	N/A	N/A	4 x SDI	N/A	Yes

AJA Io XT

Format	Resolution	Supported Frame Rates	Signal	4:2:2	4:4:4	Stereo 4:2:2	HDMI Out
SD	720 x 486	29i	1.5 G	1 x SDI	2 x SDI	2 x SDI	Yes
SD	720 x 576	25i	1.5 G	1 x SDI	2 x SDI	2 x SDI	Yes
HD	1280 x 720p	50p, 59p, 60p	1.5 G	1 x SDI	2 x SDI	2 x SDI	Yes

Format	Resolution	Supported Frame Rates	Signal	4:2:2	4:4:4	Stereo 4:2:2	HDMI Out
HD	1920 x 1080	23p, 24p, 25p, 29p, 30p, 50i, 59i	1.5 G	1 x SDI	2 x SDI	2 x SDI	Yes
HD	1920 x 1080	23psf, 24psf, 25psf	1.5 G	1 x SDI	2 x SDI	2 x SDI	No
HD HFR	1920 x 1080	50p, 59p, 60p	3 G	1 x SDI	N/A	N/A	Yes
2K DCI	2048 x 1080	23p, 23psf, 24p, 24psf, 25p, 25psf	1.5 G	1 x SDI	2 x SDI	2 x SDI	Yes

AJA Kona 3G

Format	Resolution	Supported Frame Rates	Signal	4:2:2	4:4:4	Stereo 4:2:2	HDMI Out
SD	720 x 486	29i	1.5 G	1 x SDI	2 x SDI	2 x SDI	Yes
SD	720 x 576	25i	1.5 G	1 x SDI	2 x SDI	2 x SDI	Yes
HD	1280 x 720p	50p, 59p, 60p	1.5 G	1 x SDI	2 x SDI	2 x SDI	Yes
HD	1920 x 1080	23p, 24p, 25p, 29p, 30p, 50i, 59i	1.5 G	1 x SDI	2 x SDI	2 x SDI	Yes
HD	1920 x 1080	23psf, 24psf, 25psf	1.5 G	1 x SDI	2 x SDI	2 x SDI	No
HD HFR	1920 x 1080	50p, 59p, 60p	3 G	1 x SDI	N/A	N/A	Yes
2K DCI	2048 x 1080	23p, 23psf, 24p, 24psf, 25p, 25psf, 29p, 30p	1.5 G	1 x SDI	2 x SDI	2 x SDI	Yes

Blackmagic Design

Format	Resolution	Supported Frame Rates	Signal	4:2:2	4:4:4 Dual Link 1.5G	4:4:4 Single Link 3G	Stereo 4:2:2	HDMI Out
SD	720 x 486	29i	1.5 G	1 x SDI	2 x SDI	N/A	2 x SDI	Yes
SD	720 x 576	25i	1.5 G	1 x SDI	2 x SDI	N/A	2 x SDI	Yes
HD	1280 x 720p	50p, 59p, 60p	1.5 G	1 x SDI	2 x SDI	1 x SDI	2 x SDI	Yes
HD	1920 x 1080	23p, 24p, 25p, 29p, 30p, 50i, 59i	1.5 G	1 x SDI	2 x SDI	1 x SDI	2 x SDI	Yes
HD	1920 x 1080	23psf, 24psf, 25psf	1.5 G	1 x SDI	2 x SDI	1 x SDI	2 x SDI	No
HD HFR	1080 x 1080	50p	3 G	1 x SDI	1 x SDI	N/A	2 x SDI	Yes
HD HFR	1920 x 1080	59p, 60p	3 G	1 x SDI	1 x SDI	N/A	2 x SDI	Yes
2K DCI	2048 x 1080	23p, 24p, 25p	1.5 G	1 x SDI	2 x SDI	1 x SDI	2 x SDI	Yes
2K DCI	2048 x 1080	23psf, 24psf, 25psf	1.5 G	1 x SDI	2 x SDI	1 x SDI	2 x SDI	No
UHD	3840 x 2160	23p, 24p, 25p, 29.97p, 30p,	3 G	2 x SDI	N/A	N/A	N/A	Yes
4K	4096 x 2160	23p, 24p, 25p	3 G	2 x SDI	N/A	N/A	N/A	Yes

Configuring a Dual Monitor Setup

Dual Monitor Support

The Flame family applications (Flame, Flame Assist, Flare) support dual monitor configurations. The second monitor can be used either as a Broadcast monitor, to display the Media Panel in Full Width or to display an additional viewport on the secondary screen while in Batch or Batch FX. When the Media Panel is not set to Full Width, the secondary screen defaults to displaying an extra viewport. You can set any Batch view to that viewport.

NOTE

- When working with a dual monitor setup, it is not possible to frame lock the two monitors to an external sync, nor is it possible to set the monitors' refresh rates to match the project frame rate.
- Dual monitor setups are not supported in pre 2015 Extension 1 versions of the application (i.e. 2015 and older). If you need to run a pre 2015 Extension 1 version of the software on a workstation configured with two monitors, the left monitor must be set as the primary monitor, using the Flame 2015 Extension 1 setup utility, for the application to run correctly (on a single monitor). The DVI/HDMI broadcast remains available for the right-side monitor.
- If working in a pre 2015 Extension 1 version of the application with on a workstation with a dual GPU dual monitor setup, twin view must be disabled, using the Flame 2015 Extension 1 setup utility.
- Though your Flame Premium workstation can be configured in a dual monitor setup, Lustre only uses a single monitor (primary).
- For both Flame and Lustre, SDI monitoring is accomplished through the AJA board.
- Certain features are not available when working in a dual monitor setup:
  - Flame: Real Time Deliverables is not available.
  - Lustre: Stereo3D preview and Screen Grab (CTRL+F7) are not available.

To setup a dual monitor configuration:

1. Plug in the second monitor.
2. Reboot the workstation.
3. Run the DKU or install the latest version of the application.
4. Log out and log back in.

NOTE

- The NVIDIA SDI card is not supported in a dual monitor setup.
- When two NVIDIA graphics cards are installed in the same workstation, KDE considers the two monitors as being a single wide monitor. This is normal and won't prevent Flame from using the two monitors properly.

When working with a dual monitor configuration, you must set one of the monitors as the Primary monitor.

To set a Primary monitor on a Linux workstation:

1. Launch the Flame Premium or the Flare Setup Utility.
2. From the Xorg.conf tab, next to Primary Monitor, select Left or Right.
3. Log out and log back in.

   NOTE If you connected your monitors in the wrong ports, you can toggle their relative position by clicking the Swap button.

To set a Primary monitor on an Mac OS X workstation:

1. Launch the System Preferences.
2. Double-click Display.
3. Select the Arrangement tab.
4. Click and drag the top menu bar onto the monitor you want to set as primary.

NOTE By default, dual monitor support is not enabled in Mac OS X Mavericks. To enable dual monitor support, do the following:

1. Launch the System Preferences.
2. Double-click Mission Control.
3. Disable "Displays Have Separate Spaces".
4. Log out and log back in.

Flame Assist can now use both displays.

You must also configure the Wacom tablet to work either only with the primary monitor or both monitors.

To configure the Wacom tablet in dual monitor setup on a Linux workstation:

1. Access the Input Devices tab of the Preferences.
2. In the Pointer section, under Pointer Span, select an option from the drop down menu. Options are:
   - Primary Monitor Only.
   - Both Monitors.

NOTE This can also be achieved using the `Win + Tab` keyboard shortcut.

To configure the Wacom tablet in dual monitor setup on Mac OS X, do the following:

1. Launch System Preferences.
2. Double-click Wacom Tablet.
3. Select the Display Toggle tab.
4. Select the Monitor 1, Monitor 2 or Both, based on your desired setup.

NOTE This can also be achieved using the `Alt + Shift + Tab` keyboard shortcut.

Playing Back with Proxies

The Proxy Quality options allow adjustments to the display quality in order to increase system responsiveness. This can be useful when a project progresses and the effects pipeline requires heavier computation. When proxy resolution is selected from the menu, the media is displayed at full frame size, but the quality is reduced by showing fewer pixels that are larger in size.

The Quality drop-down menu is available everywhere you have a viewer, in proxy enabled projects, including Batch. There are a few exceptions, such as the Text module, for example.

Using the Proxy Quality box, set an option for playing back clips across the system:

- Full Resolution: generate a full-resolution render for any player or viewer. A yellow border appears around the selected viewport.
- Proxy: generate a proxy-resolution render for any player or viewer. An orange border appears around the selected viewport.
- Proxy on Scrub: generate a proxy-resolution render when scrubbing and a full-resolution render while playing a clip normally. A yellow border appears around the selected viewport.

  Users can toggle between Full Resolution and Proxy by pressing the shortcut key `Ctrl+P`.

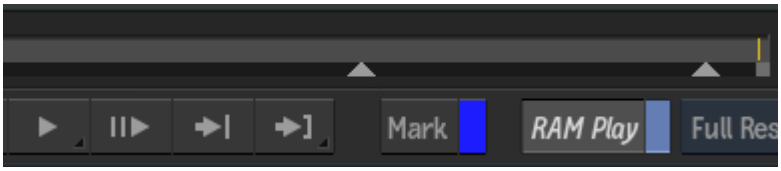
When playing back media in proxy quality mode, the viewer is bordered in orange, in the Timeline, as well as in Batch, so you can tell at a glance if you're in Proxy or Full Resolution mode.

Using the RAM Player to Cache Frames for Playback

By default, the Player plays clips from disk using the framerate of the project's default resolution, and the frame depth of the clip, in real time. Under some conditions (such as using a remote storage workflow), however, realtime playback cannot be delivered using the default Player settings. When this occurs, Flame plays back as many frames per second as your system can render on-the-fly. In these cases, you can use the RAM Player, allowing you to cache a range of frames into memory in order to improve playback.

To use the RAM Player:

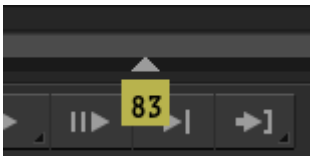
1 In the Player or full screen Player, enable the RAM Play button.



In and Out markers appear under the timebar. The markers give you the maximum range that can be cached into memory. If you try to drag a marker beyond the maximum caching range, the caching range shifts to ensure you can't cache more RAM than what you actually have on your system.

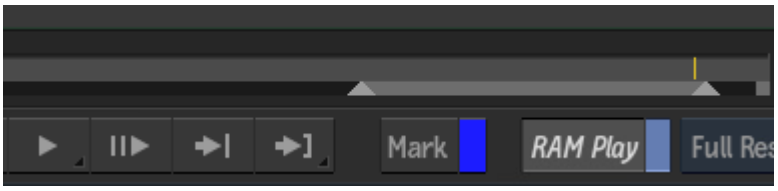
2 Click and drag the In and Out markers to the desired frames. The frames between the markers is the range that you intend to cache for playback.

As you drag the markers, you can see a visual cue of the frame number to help you.



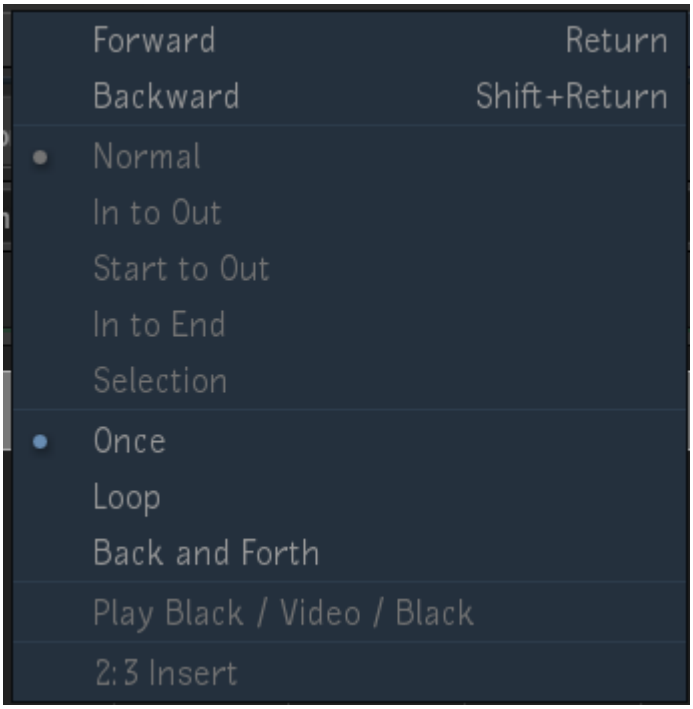
TIP Press `Shift` and drag either marker to move both In and Out markers proportionally.

3 Click Play. The frames between the markers are cached. You can see an indicator that the range between the markers is cached.



4 The range plays back.

If you click and hold the Play button, you'll notice some options are not available when RAM Play is enabled. For example, you can set the cached range to Loop, but you cannot enable 2:3 Insert.



5 Making adjustments, such as changing Timeline FX settings, or even moving away from the Player view flushes the RAM cache. In this case, you'll need to enable the RAM Play option again, and set the range to be cached.

Color and Light Levels

Accessing the Vectorscope

To access the Vectorscope:

1 Click the Timeline tab.
2 Select the Player.
3 In the View Area menu bar, select Options.
4 Select Show Vectorscope.
The Vectorscope is displayed.

NOTE The Vectorscope is unavailable in Source-Sequence, Triptych, or Trim View mode.

Changing the Settings in the Vectorscope

To change the settings in the Vectorscope:

1 Click the Timeline tab.
2 Select the Player.
3 In the View Area menu bar, select Options.

4 Select Show Vectorscope.
5 Click the Vectorscope Display button.
6 Select Vectorscope Settings.
The Vectorscope Settings window opens.

Scaling the Vectorscope

To scale the Vectorscope:

1 Click the Timeline tab.
2 Select the Player.
3 In the View Area menu bar, select Options.
4 Select Show Vectorscope.
5 Click the Vectorscope Display button.
6 Select Vectorscope > Colour 2D.
7 Shift + drag in the Vectorscope.
The scale in the Vectorscope scales to a new size.

Changing Playback Scan Mode Options

To set playback mode in the Player:

1 Click the View button.
2 Click on the Scan Mode button.

NOTE The default choice is the clip's native scan mode.

3 Select a scan mode.
4 Click Done.
The player displays the clip in that scan mode.

Enabling the Aspect Ratio Display

To enable the aspect ratio display:

1 Click the View button.
2 Enable the Use Ratio button.

NOTE The Use Ratio button is only active if you are viewing a clip with a resolution that does not use square pixels.

The original aspect ratio of the clip is applied to the clip.

Adjusting the Viewing Transform in Batch

You can change the display of an image by selecting from the Viewing Rules that have been configured in Colour Management Preferences.

By modifying the Viewing Transforms in Viewing Settings, you can change the way the clip is displayed in the viewport.

You may press Shift+Alt+[0-9] to select individual Viewing Rules, or press Shift+Alt+ the plus or minus keys to cycle through them. You may press Shift+Alt+~ to completely bypass (or re-enable) viewport colour management.

To set up the image display viewer in the tools:

1. Click the View button.
2. Modify the display attributes.
   The viewer updates the display with your changes.

Monitoring Dual Timecodes and Pulldown

When you load 24p clips into the Player, the dual timecode and 2:3 pulldown monitoring options are enabled.

To view dual timecodes and 2:3 pulldown, you must:

- View a 24p clip in a 29.97 fps frame rate project.
- Ensure that the Display Dual Timecodes button is enabled in the Preferences menu, in the Timecode section of the General tab.

In the Player, the following settings appear:

Pulldown Frame Type display Displays the corresponding pulldown frame type for the clip following pulldown insertion. This allows you to keep track of which transitions fall on jitter frames (video frames composed of two different film-based frames). 23.97 fps timecode uses a plus sign for the last punctuation mark, while 29.97 fps timecode uses a colon in non-drop frame mode, or a semi-colon in drop frame mode.

30 button Enable to view the 29.97 fps timecode that corresponds with each frame in the 23.97 fps clip following pulldown insertion

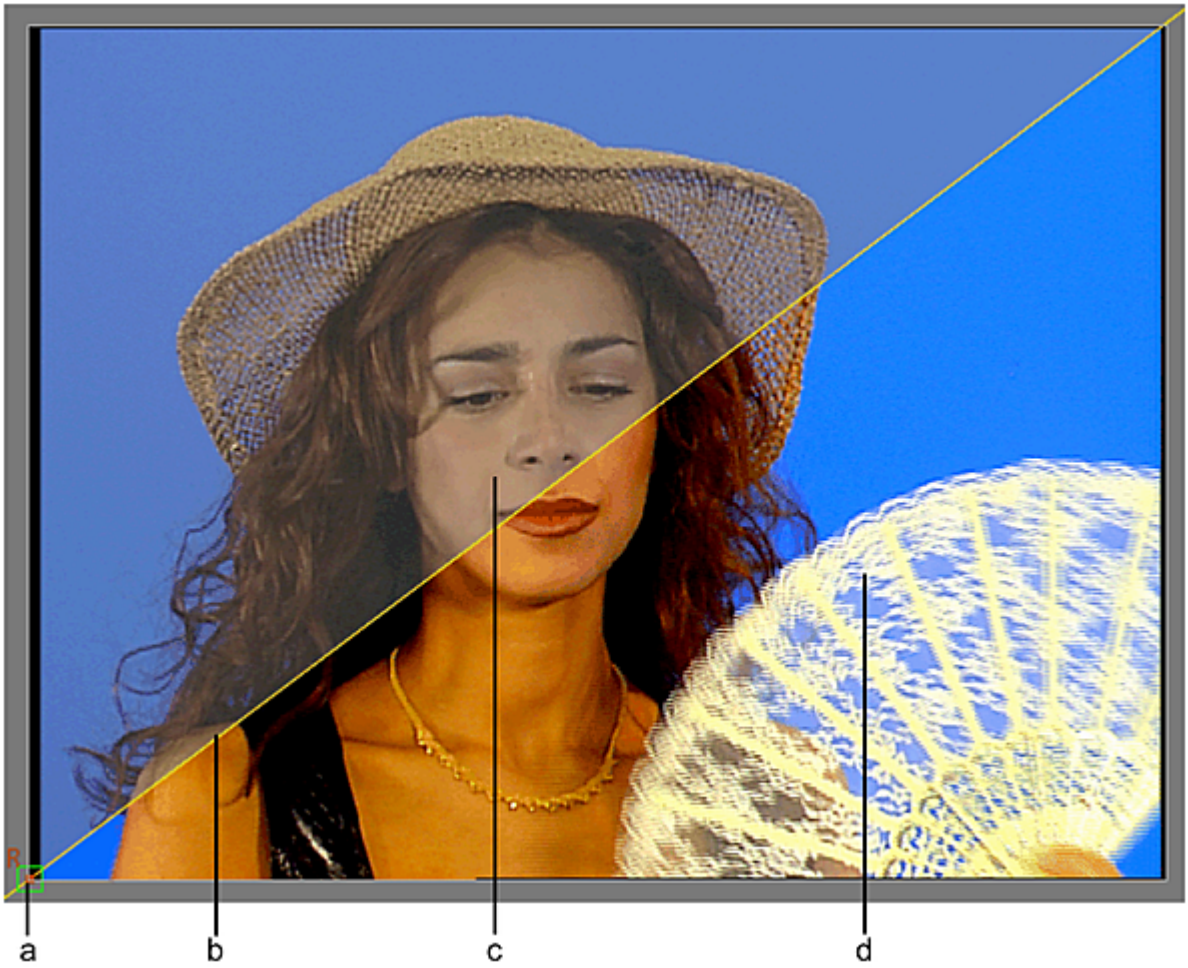
Displaying the Reference Area

In the image window of the tools that support multiple viewports, you can view the reference area with the currently displayed clip. A split bar divides the view between the current clip display (above the bar) and the reference area. The reference area can either display one of the current tool's working clips (for example, front, back, or matte), or one of the reference frames that you grab on-the-fly.

Using the Split Bar

To turn on the split bar, enable Compare in the Reference Buffer section of the View menu (also available below the viewports in Batch or Batch FX). When the split bar is on, by default it lies along the lower edge of the image window.

The split bar has a pivot point (indicated by a small box) around which the split bar can be rotated, or from which the split bar can be moved. The letter R next to the pivot box indicates the side of the bar where the reference clip is displayed.



(a) Pivot point **(b)** Split bar **(c)** Current display **(d)** Reference display
(a) Pivot point (b) Split bar (c) Current display (d) Reference display

Rotate and move the split bar using these techniques:

- To rotate the split bar around its pivot point, drag the split bar.
- To move the split bar, drag the pivot point. You can also `Ctrl+Alt`-click the image to place the pivot point under the cursor, and then drag over the image to move the split bar.
- To restore the split bar to its default position, `Ctrl`-click the pivot point.

Showing and Hiding the Split Bar

When the split bar is on, you can show or hide it in the image window.

To show or hide the split bar:

1 `Ctrl`-click the split bar.

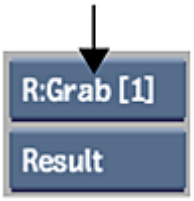
When the split bar is hidden, it is still active. The letter R is displayed indicating the location of the reference clip.

Loading a Clip into the Reference Area

You can display one of the tool's clip views in the reference area. For example, turn on the split bar to show the reference area, and then display the matte of a Colour Corrector clip while working on its result.

To load a clip into the reference area:

1 Select an option from the Reference box.



Storing Reference Frames in the Reference Buffer

You can grab any number of reference frames in the reference area, limited only by framestore space.

To store reference frames in the reference area:

1 Display the frame you want to grab in the viewport.

2 In the Reference Buffer section of the View menu of a tool, click Grab.

The current frame appears in the reference area.

3 To grab another frame, display it in the viewport (or make the changes to the current frame) and then do one of the following:

- Press `Ctrl+G` to replace the current reference frame.
- Press `Ctrl+Shift+G` to append the new frame to all existing reference frames.

Cycling Through Multiple Reference Frames

When you work with several frames stored in the reference buffer, you can cycle their display in the reference area.

To cycle through multiple reference frames:

1 With the split bar on and the reference area showing, click the Prev and Next buttons in the View menu.

To cycle through multiple reference frames using the reference overlay:

1 Press and hold `Ctrl+NUM5` to display the reference overlay.

2 While holding `Ctrl`, press `NUM4` and `NUM6` to cycle backwards and forwards through all stored reference frames.

Displaying Multiple Views

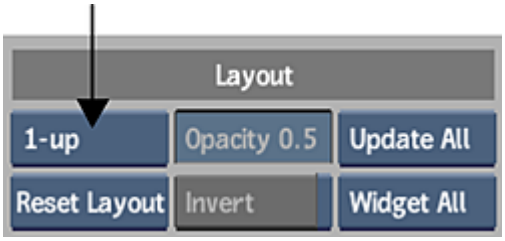
You can display up to four viewports at a time in the image window of most tools. Multiple viewports are convenient for setting channel values, working in Schematic view, and previewing your results all at the same time without having to switch views.

NOTE The Resize tool supports two viewports; other tools support up to four viewports.

You can apply a different 3D LUT to each viewport.

To view multiple viewports:

1. In a tool that supports multiple viewports, click View.
2. Select an option from the Viewport Layout box.



Select:	To view:
1-up	A single viewport (`Alt+1`).
2-up	Two viewports, side-by-side (`Alt+2`).
3-up split	Three viewports, two side-by-side, and one on top (`Alt+3`).
3-up	Three viewports, side-by-side (`Alt+3`).
4-up	Four viewports, two up and two down (`Alt+4`).

To display views in multiple viewports:

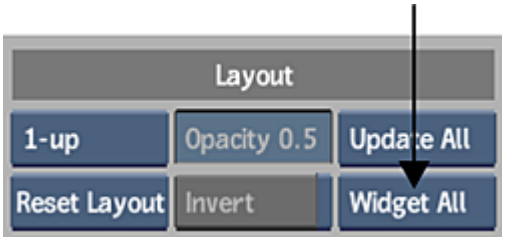
1. Do one of the following:
   - Place the cursor over the applicable viewport and press its associated keyboard shortcut.
   - Click a viewport to select it and then select an option from the View box.

Displaying Widgets in Selected Viewports

If you are working in multiple viewports, you can display widgets such as icons, crop boxes, and 3D scope in all viewports or in a selected viewport.

To display widgets in selected viewports:

1. In a tool that supports multiple views, click View.
2. From the Widget Display box, select Widget Sel to display widgets in the selected viewport, or select Widget All to display widgets in all viewports.

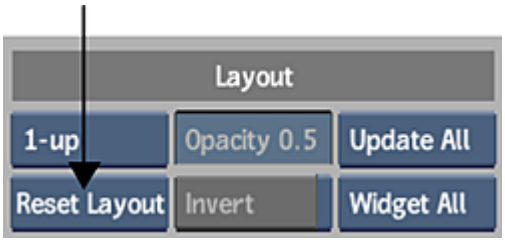


Changing the Multiview Layout

When you select a multiple viewport layout, a default layout appears, and each viewport is set to display a default view. You can change both the default layout and the views in each viewport to suit your needs.

To change the default viewport layout:

1. Do one or more of the following:
   - Hold the cursor over the lines dividing the image window and `Ctrl`-drag. The current zoom/pan settings automatically change so the frame matches its viewport's width.
   - Select the viewport to make it current (a yellow border indicates the current viewport) and then select a view option (for example, Front) or press a hotkey (for example, `F1`).
2. To restore the default settings, select an option from the Reset box.



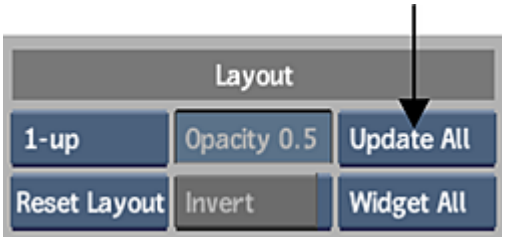
Select:	To:
Reset Layout	Restore the default layout for the current viewport layout option. You can also `Alt`-click a viewport border.
Reset All	Restore the default layout for all viewport layout options.

Synchronizing Current Frame Display Across All Viewports

By default, when you scrub or jog a clip, only the current viewport (a yellow border indicates the current viewport) is updated. The other viewports continue to display the last frame at which they were parked until you release the cursor. However, you can set all viewports to be updated in sync with the current viewport.

To synchronize the current frame display across all viewports:

1. Display the View menu.
2. Select an option from the Viewport Update box.

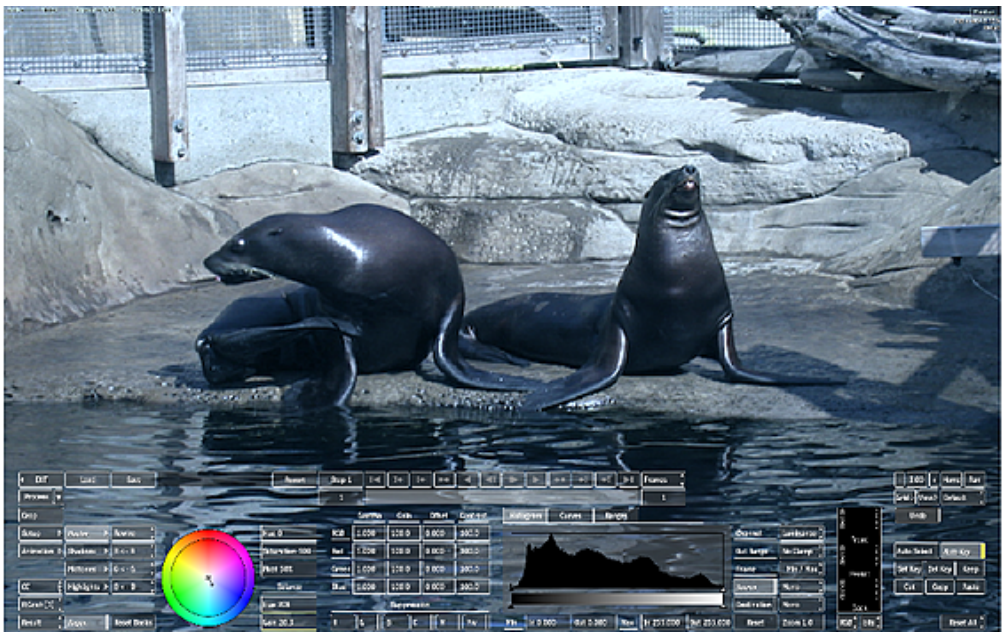


Select:	To:
Update All	Update all viewports to display the same frame as the current viewport.
Update Sel	Update only the current viewport. Other viewports continue to display the last frame at which they were parked only once you release the cursor.

Overlay User Interface

In a number of tool and effect menus (such as the Colour Corrector and 2D Transform), you can use the Overlay user interface. Its controls are identical to those on the regular user interface except that they are transparent and appear on top of the image. This allows you a greater viewing area when working with large or zoomed-in images.

To display the Overlay user interface, select Overlay UI from the Viewport Layout box.



NOTE The Overlay user interface is available when accessing tools and effects from Tools and as a timeline effect. It is not available when accessing nodes from Batch or Batch FX.

In addition to the increased area provided by the transparent controls, you can view even more of your image depending on the action you are performing. For example:

- Panning and zooming in the clip causes the Overlay user interface to disappear completely until the pan or zoom is complete.

- When you play the clip, the Overlay user interface disappears except for the timebar and the current frame number.
- When adjusting a menu parameter, all other user interface elements disappear.

The Overlay user interface does not support multiple views. When you switch to Overlay user interface while using multiple views, your viewport automatically goes to 1-Up until you toggle back to the regular user interface.

When you access a submenu that does not support Overlay user interface, such as Load or Save, the Overlay user interface is restored when you return to the main menu.

Changing the Look of the Overlay User Interface Buttons

You can adjust the opacity of the Overlay user interface buttons to suit your preference, and even invert the elements in the interface so that white outlines and text appear in a muted grey.

To set preferences for the Overlay user interface:

1 While in the Overlay user interface, click View.

In the Layout section, use the Opacity field to increase or decrease the opacity of the buttons. An opacity of 1.0 displays the buttons as they look in the regular user interface. You can also use the following keyboard shortcuts.

Press:	To:
`Ctrl+Alt+(num pad)+`	Increase the opacity of the Overlay user interface.
`Ctrl+Alt+(num pad)-`	Decrease the opacity of the Overlay user interface.

TIP You can repeatedly press the keyboard shortcuts to increase or decrease opacity, or simply hold down the keyboard shortcuts until the desired opacity is reached.

Enable Invert (or press `Ctrl+Alt+(num pad)*` to switch the Overlay user interface outlines and text from white to grey.

Viewing Settings tab

Compare Mode /Stereo

Primary Video Track field Displays the track and version assigned as the Primary track in the Player. Editable.

Secondary Video Track field Displays the track and version assigned as the Secondary track in the Player. Editable.

Preview Setup box Select which track or version to display in the Player. Options are available to display two tracks or versions.

Image

Use Ratio button Enable to simulate rectangular pixel display that corresponds to the delivery format.

Filtering button Enable to set the filtering.

Scan Mode button Adjusts the display to the desired scan mode format.

Playback

Hide Effects button Enable to hide effects during playback. Disable to display effects (may affect performance).

Preroll field Displays the number of preroll frames when playing a clip. Editable.

Postroll field Displays the number of postroll frames when playing a clip. Editable.

Proxy Quality box Select the display quality of your media. This setting is only available in proxy enabled projects.

Exposure, Contrast, and Gamma

Exposure field Displays the exposure that is used to transform image display in the image window. Editable.

Contrast field Displays the contrast that is used to transform image display in the image window. Editable.

Gamma field Displays the gamma that is used to transform image display in the image window. Editable.

Enable Look Transform button Toggles the dynamic look of a colour transform on and off. Available only when the selected viewing transform has dynamic look operators defined. Editable.

Channels

Red Channel button Enable to display the red channel of the clip. You may press Shift+R to enable and Shift+R again, or Shift+Z, to disable.

Green Channel button Enable to display the green channel of the clip. You may press Shift+G to enable and Shift+G again, or Shift+Z, to disable.

Blue Channel button Enable to display the blue channel of the clip. You may press Shift+B to enable and Shift+B again, or Shift+Z, to disable.

Isolation button Use Source Isolation to isolate the selected channel as a grey scale image, then apply the viewing transform. Use Display Isolation to isolate the selected channel(s) after the viewing transform (this is equivalent to turning off channels in the display itself). You may press Shift+X to toggle between the two modes.

Viewing Transform

Graphics Monitor Viewing Rule menu Select the Viewing Rule to use for the graphics monitor viewport.

Sync button Enable to apply the transformation for the current viewport to all viewports.

Broadcast Monitor Viewing Rule menu Select the Viewing Rule to use for the broadcast monitor.

Editing Clips and Sequences

10

The following procedures list a few methods to add clips quickly to a timeline after starting Flame for the first time. These methods are certainly not the only way to edit, but should give you a start in familiarizing yourself with editing in Flame.

When first starting Flame with a new project, you can select a workspace with a Source-Sequence player and then switch to the Timeline tab to see an empty sequence on the timeline. Once you have loaded some media into the Media Library using the MediaHub, you are ready to add clips to the timeline. You can also create new sequences from the Workspace Media panel, or right-click a clip and select Open as Sequence.

To insert a first clip on the timeline:

1. Select a clip in the Media panel to display it in the source player.
2. Drag the positioner in the timebar to the frame where you want your clip to start, then click In.
3. Drag the positioner in the timebar to the frame where you want your clip to end, then click Out.
4. Click the Insert button.

The clip is added to the timeline. Extra frames outside of your selected In and Out points remain as handles.

To insert a second clip on the timeline:

1. Select another clip in the Media panel to display it in the source player viewer.
2. Drag the positioner in the timebar to the frame where you want your clip to start, then click In.
3. Drag the positioner in the timebar to the frame where you want your clip to end, then click Out.
4. Click the Insert button.

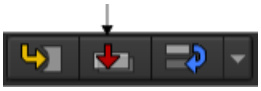
The clip is added to the timeline starting at the positioner location. If you hadn't moved the positioner after inserting the first clip, the positioner was located at the last frame.

To overwrite a clip on the timeline:

1. Move the timeline positioner to the first frame of the second clip that you inserted on the timeline.
2. Select another clip in the Media panel to display it in the source player.
3. Drag the positioner in the timebar to the frame where you want your clip to start, then click In.

4 Drag the positioner in the timebar to the frame where you want your clip to end, then click Out.

5 Click the Overwrite button.



Depending where the positioner was placed and the length of the second and third clips, existing material is overwritten, but the overall length of the timeline sequence does not change.

Continue adding clips to your sequence, as needed. At this point, you are ready to fine-tune your edit sequence by trimming clips, adding tracks for compositing, adding effects to your timeline clips, and using other Flame timeline and editing tools.

Editing with Keyboard Shortcuts

If you like to edit with keyboard shortcuts, a useful workflow to follow is to switch between the `F5`, `F6`, and `F7` shortcuts. It is good practice to open the sequence in which you plan to make edits, so that it shows up as a red tab on the timeline.

- `F5` forces the focus back to the timeline, and is useful if you explicitly click on a clip in the workspace, but want to switch back to editorial and drive timeline navigation.
- When the focus is on the timeline, `F6` switches to the source timeline tab, but also enables keyboard shortcut navigation within the media panel folder where the current source is located (using Select Next Clip or Select Previous Clip). You can use this to quickly navigate through your sources, yet not lose focus on the timeline.
- When the focus is on the timeline, `F7` sets the focus on the current active sequence, and allows you to navigate through its timeline using keyboard shortcuts (for example for Next or Previous Transition).

TIP If the focus is on the Player or the Source-Sequence Player, you can also use the `F6` and `F7` shortcuts to switch the focus between source and sequence players.

Inserting a Clip on the Timeline

To insert a clip to the timeline using the Insert button:

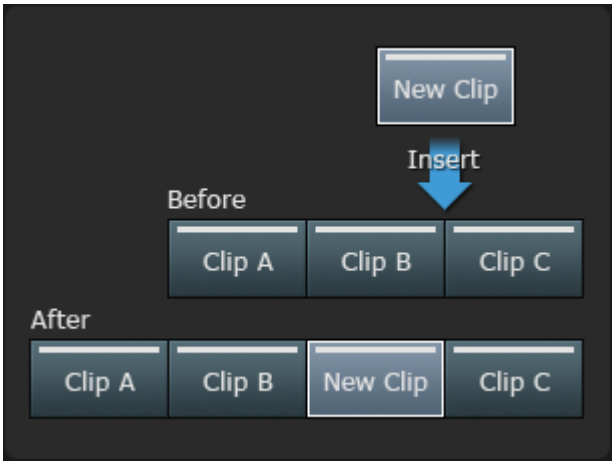
1 Move the timeline positioner to the frame that you want to use as the beginning of the insert.

2 From the Workspace, select the clip that you want to insert.

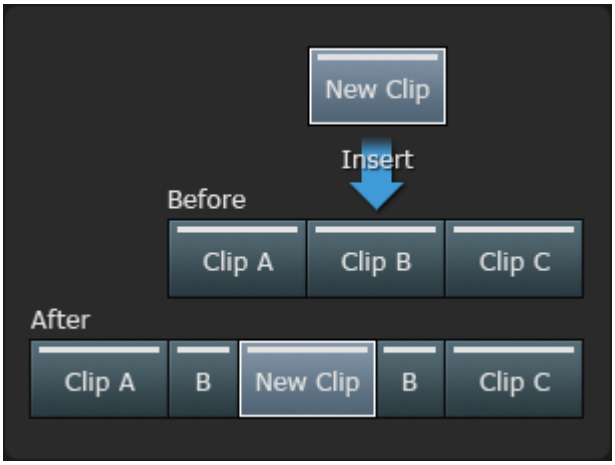
3 Add In and Out points, if needed.

4 Click the Insert button.

Depending where the positioner is placed, the clip is inserted, and the overall length of the timeline sequence is changed. The following example shows a New Clip inserted at the transition between Clip B and Clip C. Everything after the transition is moved ahead in order to make room for New Clip.



This example demonstrates the result of inserting New Clip at a frame in the middle of Clip B. Every frame after the edit point is moved ahead to accommodate New Clip.



NOTE You can also perform three-point or four-point insert edits with In and Out points on the source and timeline clips.

To insert a clip to the timeline gesturally:

1. Enable Ripple.
2. From the Workspace Media panel or Viewing panel, select the clip that you want to insert.
3. Add In and Out points, if needed.
4. Drag the clip towards the timeline.

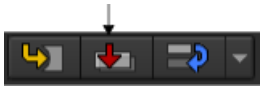
   As you hover over the timeline with your clip, you see a phantom visual guide as to the space on the track or tracks your clip will take when dropped.
5. Release the clip on the timeline.

TIP You can also drag and drop a clip to the timeline from the source tab of the Player.

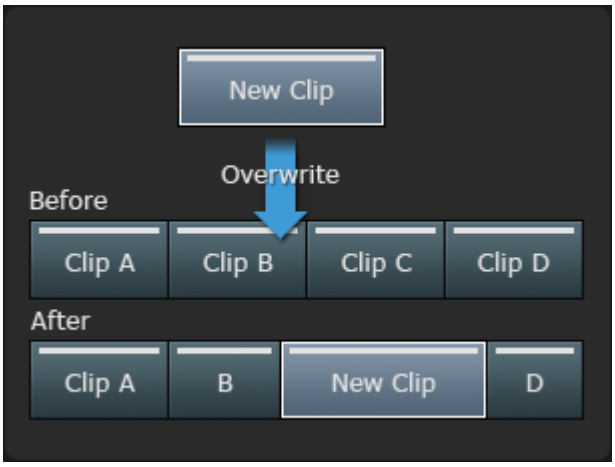
Overwriting a Clip on the Timeline

To overwrite a clip to the timeline using the Overwrite button:

1. Move the timeline positioner to the frame that you want to use as the beginning of the overwrite.
2. From the workspace, select the clip that you want to overwrite.
3. Add In and Out points, if needed.
4. Click the Overwrite button.



Depending where the positioner is placed, existing material is overwritten, and the overall length of the timeline sequence does not change. In the following illustration, New Clip is edited into the sequence at a frame in Clip B. Frames after the insertion point are overwritten by New Clip. All of Clip C and some of Clip B and D are overwritten.



NOTE You can also perform three-point or four-point overwrite edits with In and Out points on the source and timeline clips. In the case of a four-point edit, a Timewarp may be applied if the number of frames differs between the source and timeline clips. You can enable of disable Auto Timewarp in the Timeline section of the Preferences menu.

To overwrite a clip to the timeline gesturally:

1. Disable Ripple.
2. From the Workspace Media panel or Viewing panel, select the clip that you want to overwrite.
3. Add In and Out points, if needed.
4. Drag the clip to the timeline.

   As you hover over the timeline with your clip, you see a phantom visual guide as to the space on the track or tracks your clip will take when dropped.
5. Release the clip on the timeline.

TIP You can also drag and drop a clip to the timeline from the source tab of the Player.

Replacing a Clip on the Timeline

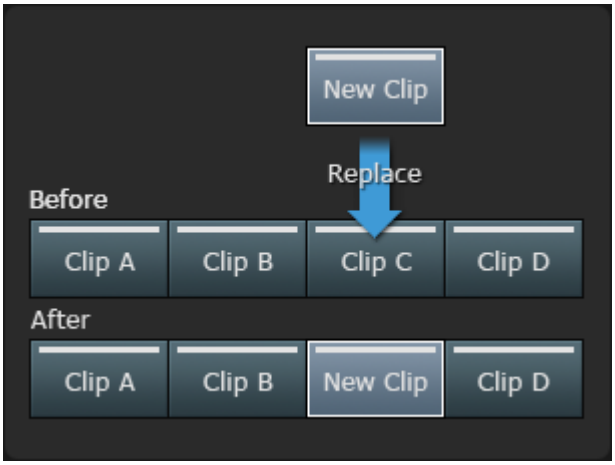
The Replace operation has two modes: Replace and Smart Replace.

Smart Replace uses the Source clip attributes to perform the operation. Content duration is compared first, then Source Timecode. If these do not match, you are asked if you want to perform a positioner to positioner Replace operation. This option is enabled by default. It can be disabled from the Secondary Edit box. This option can be especially useful in the Connected Conform workflow where you need to re-insert a rendered clip in the Shots Sequence. Smart Replace is enabled by default.

1. To replace a clip on the timeline using the Smart Replace operation:
2. Select a clip or multiple clips on the timeline that you want to replace.
3. From the workspace, select the clip that you want to use as the replacement clip.
4. Click the Replace button.

1. To replace a clip on the timeline using the Replace button:
2. Disable the Smart Replace option from the Secondary Edit box.
3. Do one of the following:
   - Move the timeline positioner over the clip that you want to replace.
   - Select a clip or multiple clips on the timeline that you want to replace.
4. From the workspace, select the clip that you want to use as the replacement clip.
5. Click the Replace button.

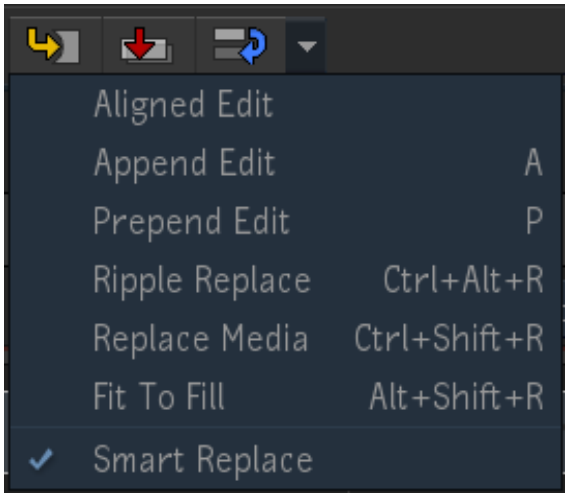
The result is the same, whether you performed a Replace or Smart Replace operation. The overall length of the sequence does not change. The replace clip must be of equal length or longer than each clip it is replacing. If longer, extra frames are loaded as tail frames.



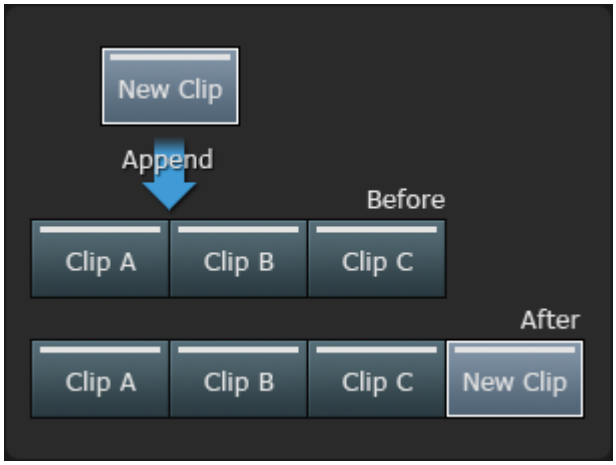
TIP See Other Editing Operations (page 418) for more Replace options.

Other Editing Operations

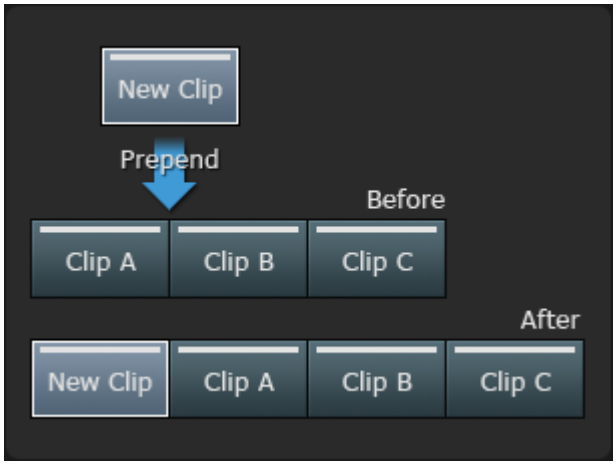
While insert, overwrite, and replace are the editing operations you use most often, there are other editing options available. For each of these operations, you select your source clip in the same manner as for an insert, overwrite, or replace edit. Then you select the appropriate editing operation from the Secondary Edit box.



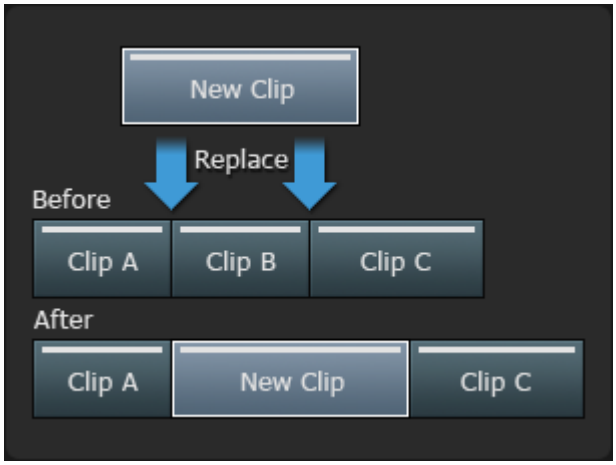
An Aligned Edit is usually performed on two tracks. You set the positioner on the timeline to an point you want to align to, then move the focus point of the positioner to a different track. Set the positioner of the source track to the point you want aligned. The source clip is added to the track, and both points are aligned.



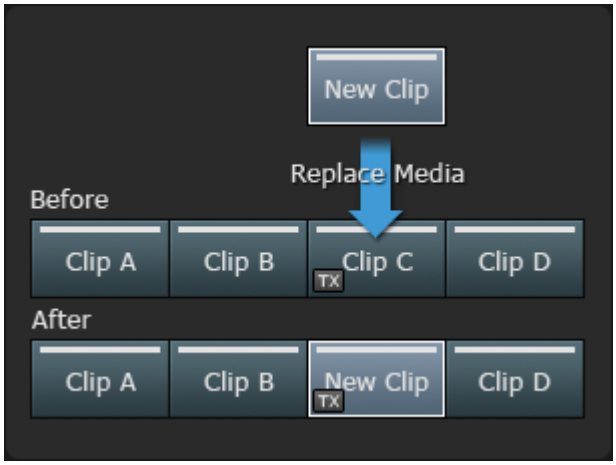
An Append Edit applies an edit that adds your source clip to the end of the edit sequence on the timeline.



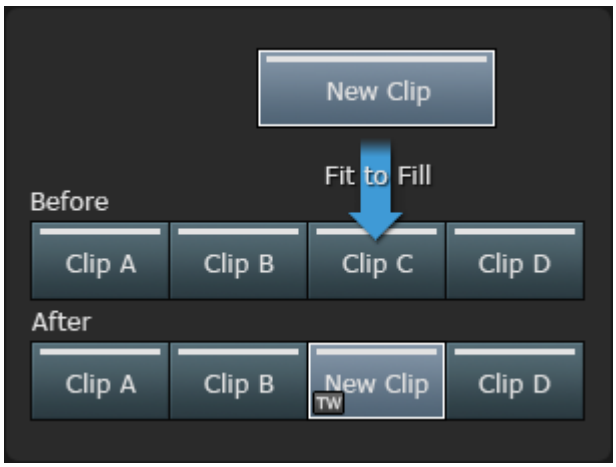
A Prepend Edit applies an edit that adds your source clip to the beginning of the edit sequence on the timeline.



Ripple Replace functions as a combination of an insert and replace edit. The source clip replaces the selected clip on the timeline, and the all of the clips to the right ripple to accommodate the length of the new clip.



Replace Media performs a replace edit but retains any effects applied to the replaced clip or clips on the timeline.



A Fit To Fill edit functions as a replace edit, but instead of adding tail frames to a longer source clip, a Timewarp is added to the replaced clip so that it fits the same space occupied by the clip it is replacing.

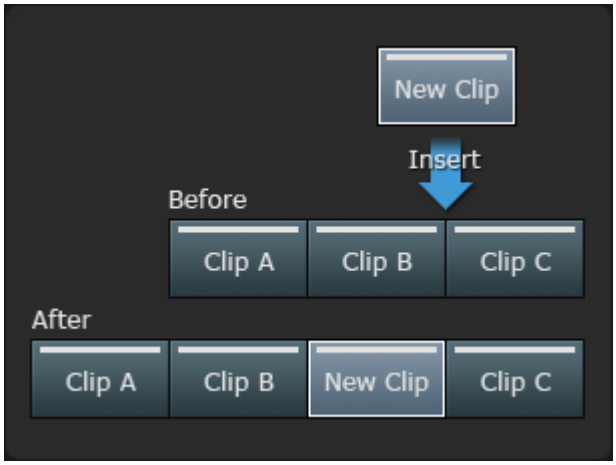
TIP When Smart Replace is enabled (this is the default), content duration is compared first, then Source Timecode. If these do not match, you are asked if you want to perform a positioner to positioner Replace operation. See: Replacing Clips on the Timeline (page 417) for more details.

Timeline Editing Settings

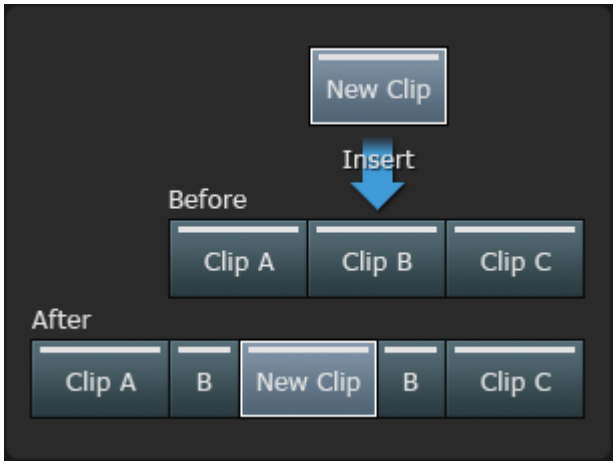
The settings you use for editing to and on the timeline can be found to the right and above the timeline. Most of these settings also have a keyboard shortcut associated with them.

Editing Operations

Insert Edit button Click to insert the selected source into the timeline at the positioner. The overall length of the timeline may change.

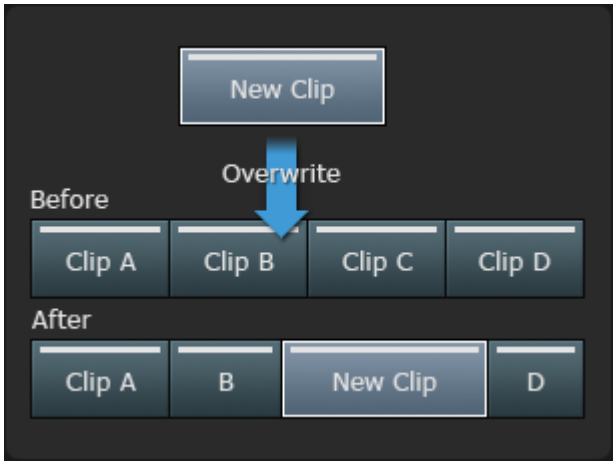


Insert Example 1



Insert Example 2

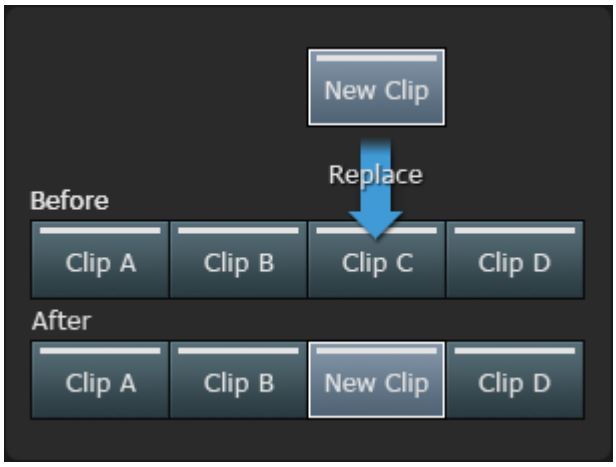
Overwrite Edit button Click to overwrite material in the timeline (at the positioner) with the selected source. The overall length of the timeline does not change.



Overwrite Example

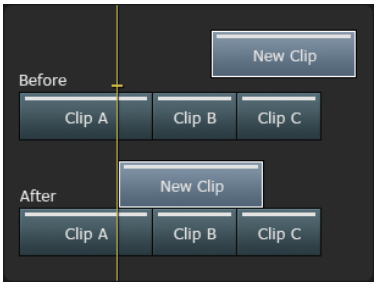
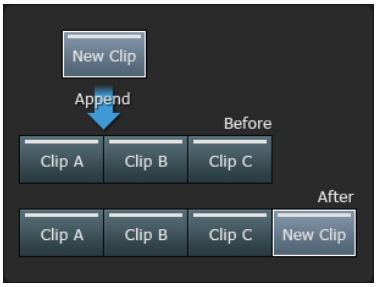
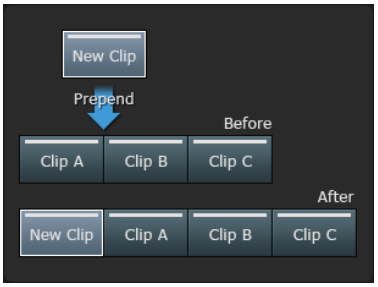
Replace Edit button Click to replace the selected timeline clip or clips with the selected source clip. The overall length of the timeline sequence does not change. The replacement clip must be of equal length or

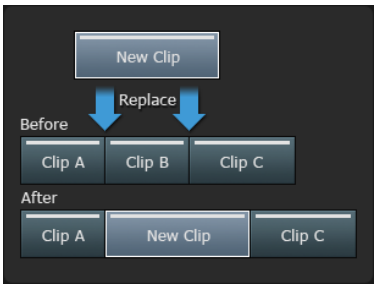
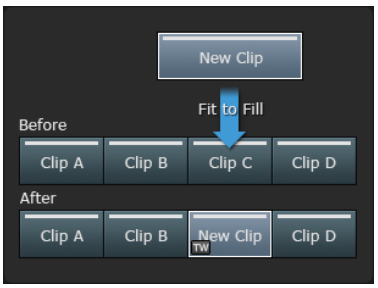
longer than each clip it is replacing. If longer, extra frames are added as tail frames.



Replace Example

Secondary Edit box Select an editing operation to apply to the selected source clip.

Select:	To apply:	Example
Aligned Edit	An edit aligning a point on the timeline with a point on the source clip.	
Append	An insert edit that adds your source clip to the end of the edit sequence on the timeline.	
Prepend	An insert edit that adds your source clip to the beginning of the edit sequence on the timeline.	

Select:	To apply:	Example
Ripple Replace	A combination of an insert and replace edit; where the source clip replaces the selected clip on the timeline, and the all of the clips to the right ripple to accommodate the length of the new clip.	
Replace Media	A replace edit that retains any effects applied to the replaced clip or clips on the timeline.	
Fit To Fill	A replace edit; but instead of adding tail frames to a longer source clip, a time-warp is added to the replacement clip so that it fits the same duration occupied by the clip it is replacing.	

Trimming Operations

Editorial Mode box Choose an editorial mode for working with clips on the timeline.

Select:		To:
Select		Select or move a timeline element, without performing any trim operations.
Select Backward / Forward		Select one of four options: Single Track Forward, Single Track Backward, All Tracks Forward, or All Tracks Backward. When selected, a single click on a Timeline segment selects one or all segments before/after the segment. The

Select:		To:
		state of the Link button affects the selection.
Trim		Set Trim mode to perform general trim (or roll) operations on a clip.
Slip		Set Slip mode, a trimming mode that offsets the frames in a clip without trimming the clips before or after it.
Slip Cuts		Set Slip Cuts mode, a trimming mode that considers multiple selected clips as a single segment.
Slide		Set Slide mode, a trimming mode that trims the clips before and after the clip without changing the frames used in it. The state of the Ripple button has an effect on this mode.
Slide Cuts		Set Slide Cuts mode, a trimming mode that trims the clips before and after it, while also changing the frames used in a clip. The state of the Ripple button has an effect on this mode.
Slide Keyframes		Set Slide Keyframes mode, a trimming mode that slides only animation keyframes on a clip.

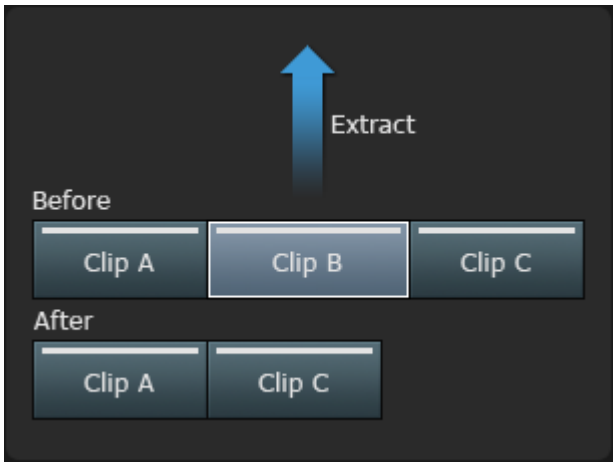
NOTE Trim, Slide, and Slide Cuts modes are affected by the state of the Ripple button. The icons display as yellow when ripple is enabled, and red when ripple is disabled.

Link button Enable to select and link all elements in an editorial group. This is useful if you want to trim all clips in a group by the same number of frames, for example. You can invert the Link mode on-the-fly

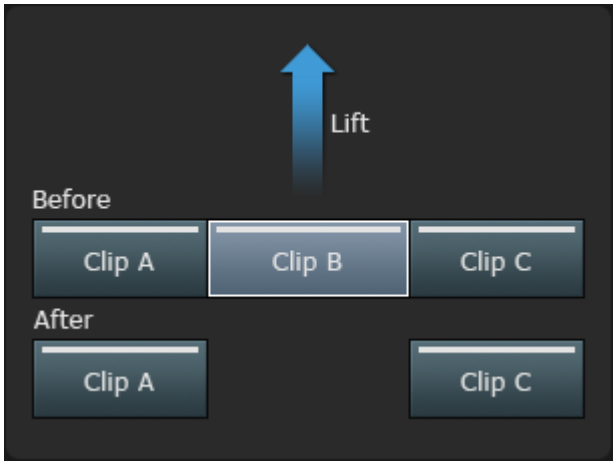
during manipulation, with the Invert Group Selection keyboard shortcut.

Link Example

Ripple button When gesturally adding material into an edit sequence, enable to perform an insert edit, disable to perform an overwrite edit. When moving (or trimming) clips on the timeline, enable to fill the gap left by the removed clip or frames, disable to leave the gap.



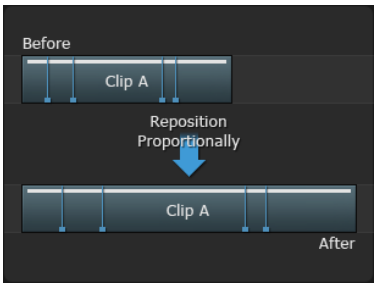
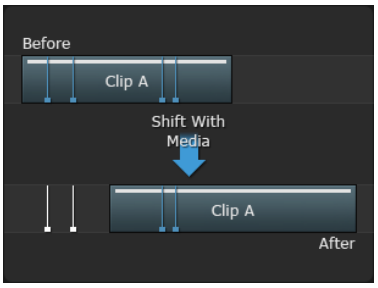
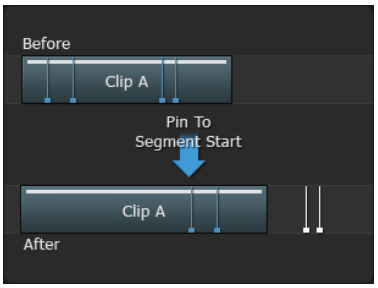
Ripple On



Ripple Off

Snap button Enable to snap to the closest transition, positioner, or mark to help gesturally align edits on the timeline. Press Shift to invert the snap mode during manipulation.

Keyframe Move Modes box Select how the animation channel is affected when you trim elements with animated effects.

Select:	To:	Example
Reposition Proportionally	Resize the channel as you trim. The animation channel is scaled to fit into the timeline element. This option has no effect when you slip or slide.	
Shift With Media	Link the keyframes to their original frame numbers. The animation channel moves to follow the original frames as you trim.	
Pin To Segment Start	Unlink the keyframes from their original frame number. The animation channel remains with the timeline element as you trim.	

Trimming Clips

Use the trimming tools to fine-tune your rough edits. You can trim video or audio clips (with or without effects), cuts, and transitions on the timeline. You can trim multiple tracks or segments at the same time. Flame keeps track of the source material that goes into trimmed shots, so you can go back and reintroduce material you had previously trimmed out.

You can trim using the Trim View, with keyboard shortcuts, or gesturally on the timeline. You can also trim by any combination of these methods, depending on your editing style. In Trim mode, the Trim View displays the last (outgoing) and first (incoming) frame above the timeline from the two clips you are trimming, allowing you to visualize your trimming operation. In Slip or Slide mode, the Trim View changes to a four frame display, which, in addition to the incoming and outgoing frames of the clip you are slipping or sliding, also displays the outgoing frame of the preceding clip and the incoming frame of the subsequent clip on the timeline.

The state of the Ripple button has an effect on *some* trimming operations. You can set the ripple state explicitly before trimming, or on-the-fly by pressing `Alt` while you are trimming.

Audio tracks are divided into sub-frames. There are 100 sub-frames of audio for every frame of video. When trimming audio tracks, you can trim on a sub-frame level by holding down the `Shift` key while you trim.

NOTE You cannot trim on a sub-frame level when simultaneously trimming audio and video tracks.

Trimming Quickstart

Use the following procedure to quickly start trimming. This method is certainly not the only way to trim, but should give you a start in familiarizing yourself with trimming in Flame. For more detailed explanations of the various trimming modes, see the related topics below.

1 Double-click a cut between two clips on the timeline. The positioner moves to this location, and a yellow or red highlight appears on either side of the cut (depending if Ripple is enabled or not).

The Trim View also opens, (provided that Auto-Toggle Trim View is enabled in **Preferences ➤ General ➤ Player**) displaying the outgoing and incoming frame of the two clips.

TIP You can also access Trim View from the View mode box. In this case, the closest cut on the timeline is automatically selected. You can use the Previous or Next Transition buttons to select a different cut.

2 From the Editorial Mode box, select Trim.

3 Drag the Outgoing or Incoming Trim Offset fields to your desired trim offset value (in frames). The corresponding view (incoming, outgoing, or both), and the timeline clips are updated.

If you are trimming a gap, the Outgoing and Incoming Trim Offset fields remain at 0.

Notice that the offset fields have the same colour borders as the highlight on the timeline cut (yellow when Ripple is enabled, or red when Ripple is disabled).

4 Use the loop option of the Play button to loop around your trim with a number of preroll and postroll frames to preview your trim.

Performing a Trim

Trim (or roll) a clip to add frames to, or remove frames from, a clip's head or tail.

To trim (or roll) a clip or transition on the timeline:

1. From the Editorial Mode box, select Trim.
2. Position the cursor near the head or tail of a clip or transition on the timeline.

   The cursor turns to a trim cursor (a yellow arrow and line if ripple is enabled, or a red arrow and line if ripple is disabled). If you want to trim a cut, position the cursor over the cut, and notice that the trim cursor becomes a double arrow with a line.
3. Drag left or right to remove the number of frames that you need.

   You can see the head or tail number on the clip change as you trim.

   TIP Enable Focus On Trim in the Timeline Options menu to snap the timeline positioner to the transition while trimming, allowing you to view the frames you are trimming in the Player, or switch to Trim View to see your incoming and outgoing frames.

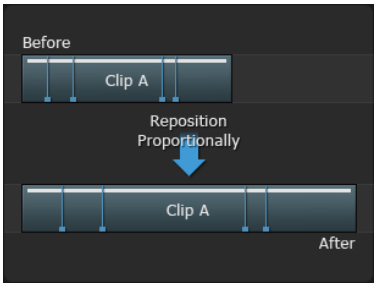
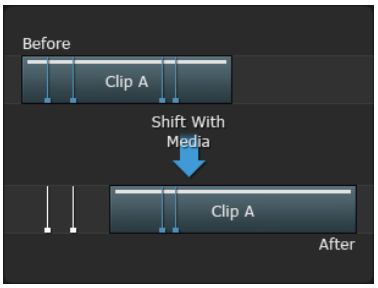
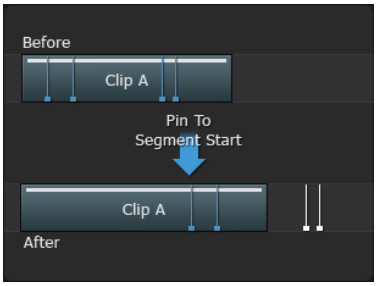
To trim (or roll) a clip or transition using Trim View:

1. Double-click a cut between two clips on the timeline, or select Trim View from the View Mode box.

   Trim View appears in the Viewing panel, displaying the last (outgoing) and first (incoming) frame from the two clips you are trimming.
2. Use any of the following techniques to trim your clips to the desired frames:
   - Click and drag directly in the left or right Trim View window. You can see the trim icon as you drag. The icon is displayed in yellow or red, depending on the state of the Ripple button.
   - Click and drag toward the middle of the two Trim View windows. Notice that the trim icon appears as a double-sided arrow, indicating that you are trimming the cut between the two clips.
   - Click any of the timecode or duration fields at the top of Trim View windows to enter specific values. If a timecode or duration field is not enabled, click the corresponding Trim View window or offset field to enable it.
   - Click and drag the outgoing or incoming Trim Offset field to specify the number of frames offset in the clip. You can also click the minus or plus buttons to offset by those amounts.
   - You can also trim on the timeline, and use the Trim View as a viewing aid.

Trimming Keyframes

If you set animation keyframes on timeline effects, you have different options as to their behaviour while trimming. Use the Keyframe Move Modes box to select how the animation channel is affected when you trim elements with animated effects.

Select:	To:	Example
Reposition Proportionally	Resize the channel as you trim. The animation channel is scaled to fit into the timeline element. This option has no effect when you slip or slide.	
Shift With Media	Link the keyframes to their original frame numbers. The animation channel moves to follow the original frames as you trim.	
Pin To Segment Start	Unlink the keyframes from their original frame number. The animation channel remains with the timeline element as you trim.	

In the Editorial Mode box, you can also select Slide Keyframes mode, which allows you to slide all of the keyframes of a specific timeline effect on a clip. For example, you can slide all of the Axis keyframes on a clip.

Performing a Slip

Slip a clip to offset the frames in a clip (by trimming head and tail frames of the clip) without trimming the clips before or after it. The state of the Ripple button has no effect on slip operations.

To slip a clip on the timeline:

1. From the Editorial Mode box, select Slip.
2. Position the cursor in the middle of a clip on the timeline.

The cursor turns to a slip cursor.

3 Drag left or right.

You can see the head and tail numbers on the clip change as you slip.

To slip multiple clips on the timeline at the same time:

1 Select multiple clips on the timeline.

2 Do one of the following:

- If you want to slip each of the clips that are part of the multi-selection, select Slip from the Editorial Mode box. In this case, each selected clip's head and tail frames are trimmed.
- If you want the multi-selection to be considered as one segment, select Slip Cuts from the Editorial Mode box. In this case, the head frame of the first clip, and the tail frame of the last clip are trimmed.

The cursor turns to a slip cursor.

3 Drag left or right on any of the selected timeline clips.

You can see the head and tail numbers on the clips change as you slip.

To slip a clip in Trim View:

1 Select the clip you want to slip on the timeline.

2 From the Editorial Mode box, select Slip.

3 Double-click the clip you want to slip, or select Trim View from the View Mode box.

In Slip mode, the Trim View displays four panels. In addition to the incoming and outgoing frames of the clip you are slipping, you can see the outgoing frame of the preceding clip and the incoming frame of the subsequent clip on the timeline.

4 Use any of the following techniques to slip your clip:

- Click and drag left or right directly in the Trim View window. You can see the slip icon as you drag.
- Click any of the timecode fields at the top of Trim View windows to enter specific values.
- Click the minus or plus buttons to offset by those amounts.
- You can also slip on the timeline, and use the Trim View as a viewing aid.

NOTE In Slip mode, the Duration and Offset fields of the Trim View update as you slip, but are non-editable.

Performing a Slide

Sliding changes a clip's position in the edit sequence. It simultaneously slides the clip under the cursor and trims the head and tail of the surrounding clips. The state of the Ripple button has an effect on slide operations.

NOTE Instead of using Slide mode, you can select Slide Cuts from the Editorial Mode box, a modified Slide mode that trims the clips before and after it, while also changing the frames used in a clip.

To slide a clip on the timeline:

1 From the Editorial Mode box, select Slide.

2 Position the cursor in the middle of a clip on the timeline.

The cursor turns to a slide cursor (a white square with yellow arrows if ripple is enabled, or a white square with red arrows if ripple is disabled).

3 Drag left or right.

The position of the clip changes, and surrounding clips' head and tails are trimmed.

To slide a clip in Trim View:

1. Select the clip you want to slide on the timeline.
2. From the Editorial Mode box, select Slide.
3. Double-click the clip you want to slide, or select Trim View from the View Mode box.

   In Slide mode, the Trim View displays four panels. In addition to the incoming and outgoing frames of the clip you are sliding, you can see the outgoing frame of the preceding clip and the incoming frame of the subsequent clip on the timeline.
4. Use any of the following techniques to slide your clip:
   - Click and drag left or right directly in the Trim View window. You can see the slide icon as you drag.
   - Click any of the timecode fields at the top of Trim View windows to enter specific values.
   - Click the minus or plus buttons to offset by those amounts.
   - You can also slide on the timeline, and use the Trim View as a viewing aid.

   NOTE In Slide mode, the Duration and Offset fields of the Trim View update as you slide, but are non-editable.

Trimming with Keyboard Shortcuts and the Calculator

You can use selection-based keyboard shortcuts to perform your trim, slip, or slide operations (depending on the mode selected in the Editorial Mode box).

1. On the timeline, select the segment that you want to trim.
2. From the Editorial Mode box, select the trim mode that you want to perform (for example, Slip).

   TIP Specific keyboard shortcuts are available to switch to each trim mode (Trim, Slip, Slide, Slide Cuts, and Slide Keyframes). See the Keyboard Shortcut editor for more information.
3. Use the keyboard shortcut appropriate to the action you want to perform (in the descriptions below, Trim refers to the selected trim mode).

Description	Smoke (FCP 7) Shortcut	Smoke Classic Shortcut	Flame Shortcut
Trim 1 Frame Forward	. (period)	N	. (period)
Trim 1 Frame Backward	, (comma)	B	, (comma)
Trim <n> Frames Forward	Shift+. (period)	Shift+N	Shift+. (period)
Trim <n> Frames Backward	Shift+, (comma)	Shift+B	Shift+, (comma)
Trim to In Mark	Ctrl+I	Ctrl+B	Shift+[
Trim to Out Mark	Ctrl+O	Ctrl+N	Shift+]
Trim to Positioner	E	Ctrl+P	Shift+P

Trimming with the Keypad and Calculator

If you know the amount of frames by which you want to trim, a quick way to perform the trim is by using the keypad to enter the amount into the calculator.

1. On the timeline, select the clip or cut you want to trim.
2. On the keyboard keypad, enter the number of frames by which you want to trim (positive or negative).
3. Press Enter.

   NOTE If there is no explicit selection on the timeline, trim operations are not performed using this method. Instead, the positioner is moved by the number of frames that you enter.

Dynamic Trimming

In Trim View, you can use key combinations to dynamically trim while your outgoing and incoming clips are playing.

To trim with the J-K-L keys:

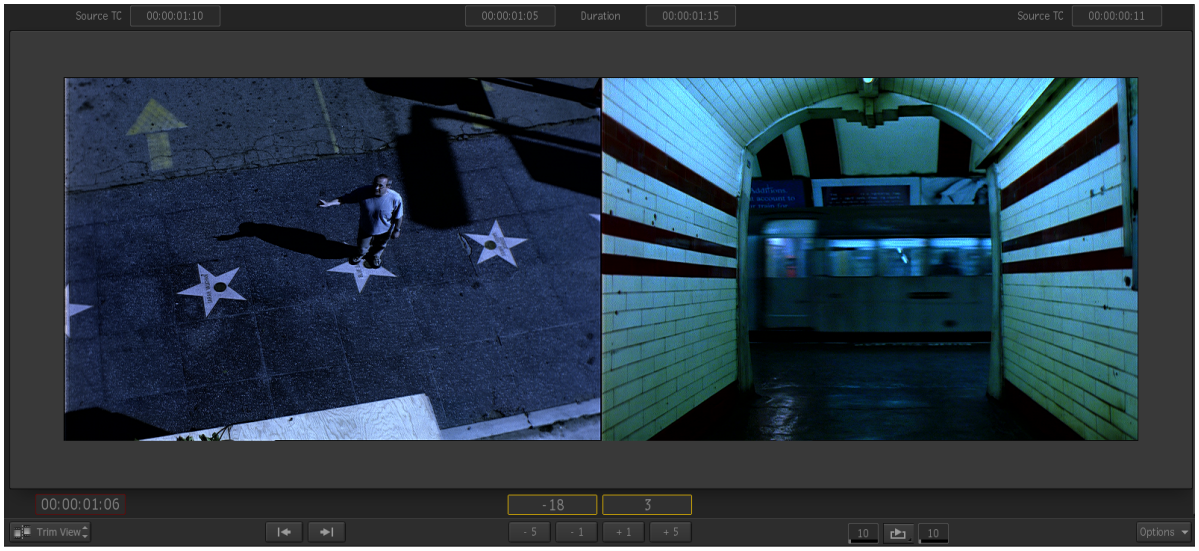
1. In Trim View, select either the incoming, outgoing, or both clips.
2. Use one of the following J-K-L key combinations to trim:
   - Press the `J` or `L` key once to play backward or forward, then press `K` to stop and perform the trim at the desired frame. You can also press `J` or `L` multiple times to play faster before pressing `K`. This method works for Trim mode only.
   - Press and hold `K`, then press `J` to trim one frame backward or `L` to trim one frame forward. This method works for Trim, Slip, and Slide modes.

   The trim viewports and the timeline update as you perform your trim. While pressing `J` or `L`, you can see a phantom positioner moving along the timeline to help you see where you want to trim.

To trim during playback:

1. In Trim View (with Trim, Slip, or Slide mode selected), select a playback option from the Play button (such as Loop), and click Play.
2. While your selection is playing (with the desired number of preroll and postroll frames), you can use the Trim 1 Frame Forward, Trim 1 Frame Backward, Trim <n> Frames Forward, and Trim <n> Frames Backward keyboard shortcuts.

Trim View Settings



In Trim mode, the Trim View displays the last (outgoing) and first (incoming) frame from the two clips you are trimming (as seen in the example, above). In Slip or Slide mode, the Trim View changes to a four frame display, which, in addition to the incoming and outgoing frames of the clip you are slipping or sliding, also displays the outgoing frame of the preceding clip and the incoming frame of the subsequent clip on the timeline.

Use these settings while working in Trim View. Most of the settings are available for all Trim View modes; exceptions are noted below.

Tail Source Timecode field Displays the tail source timecode of the outgoing segment. Non-editable if the outgoing trim offset is not selected.

Outgoing Duration field Displays the duration of the outgoing clip. Non-editable if the outgoing trim offset is not selected. Available in Trim mode only.

Incoming Duration field Displays the duration of the incoming clip. Non-editable if the incoming trim offset is not selected. Available in Trim mode only.

Head Source Timecode field Displays the head source timecode of the incoming segment. Non-editable if the incoming trim offset is not selected.

Duration field Displays the duration of the selected segment. Non-editable. Available in Slip and Slide mode. (Not shown)

Timecode field Displays the current timecode of the focus clip (green signifies a source clip, while red signifies a sequence clip). Editable.

View Mode box Select a view mode for the Viewing panel layout.

Previous Transition button Click to select the previous transition on the selected track.

Next Transition button Click to select the next transition on the selected track.

Outgoing Trim Offset field Displays the number of frames offset in the outgoing clip. Click to select the offset, and drag to change the offset value. A selected offset has a yellow border if Ripple is enabled, and a red border if Ripple is disabled. Available in Trim mode only.

Incoming Trim Offset field Displays the number of frames offset in the incoming clip. Click to select the offset, and drag to change the offset value. A selected offset has a yellow border if Ripple is enabled, and a red border if Ripple is disabled. Available in Trim mode only.

Trim Offset field Displays the number of frames offset as the result of the slip or slide operation performed on the selection. Non-editable. (Not shown)

-5 button Click to trim the selection five frames to the left.

-1 button Click to trim the selection one frame to the left.

+1 button Click to trim the selection one frame to the right.

+5 button Click to trim the selection five frames to the right.

Preroll field Displays the number of preroll frames when playing a clip. Editable.

Play button Click to activate the current play behaviour selected for this button. Click and hold to list a selection of playback types. The Play button will change to display the current play status; Once, Loop, or Back and Forth.

Postroll field Displays the number of postroll frames when playing a clip. Editable.

Player Options box Select an option for working in the player. Available options differ depending on the view selected.

About Timeline Tracks and Versions

Different areas on the timeline form a hierarchy that allows you the flexibility to composite on multiple tracks while maintaining different video streams or versions of your work.

Think of a version as a single stream of video on the timeline. Each version can have multiple tracks. While you can also use sequences on the timeline to accomplish similar results, use versions for the following:

- Compare two streams, such as an offline and online version. You can use the player in a split view to compare versions.
- Create matte containers with an Axis timeline effect, to help you key using a separate fill and matte, for example.
- Create different versions of a track or tracks, to try out different effects without affecting each other.

Tracks on the timeline are stacked vertically within a version to help you composite.

Track Indicators and Tools

On the left side of the timeline, in the Patch Panel area, you can find indicators and other tools for working with tracks and versions.

In the example above, the upper track (version 1 track 2) is highlighted, and therefore selected. To highlight a track and select all clips on the track, click in an empty space in the Patch Panel area for that track.

The track indicators and tools for this track are (from left to right):

Grab area Grab the handle area of the track to move and reorder a track or group of tracks. As you are dragging, a red message gives you an indication as to the operation being performed.

Track Collapse/Expand arrow Click to collapse or expand all tracks within a version. You can collapse tracks to remove clutter from your timeline.

Patch Identifier (Source track) Indicates which source track is patched to the track (in green).

Track Identifier (Destination or Sequence track) Displays the version and track number (in dark grey).

Track Visibility icon Displays or hides the track.

Track Lock icon Enable track lock to prevent editing operations from being performed on the track. A locked track displays grey diagonal lines.

TIP You can select clips on a locked track if you enable Selection Includes Locked Tracks in the Timeline Options menu.

Sync Lock icon Enable to maintain sequence sync on a track or tracks.

Video Track indicator Indicates if a track is the primary or secondary track. The primary track can also be set by moving the focus point on the positioner.

Track Name Drag the area between the Patch Panel area and the timeline to display the track name area. By default, tracks are named *. Right-click a track, and select Rename Track to add your unique track name.

Adding Tracks and Versions

To add a new video track:

1. On the timeline, select the version you want to add a track to.
2. On the bottom left of the timeline, click Track+, or from the Timeline Gear menu, select **New ➤ Video Track**.

   Depending on your selection in the Add Track box in **Preferences ➤ Timeline ➤ Editing**, a new track is either added to the version on top of the last existing track, or to the track above the focus point.

   If you want the track to be added below the last existing track, press **Ctrl** while clicking Track+. You can re-order your tracks, or even move a track to a different version by dragging the track from the handle at the left of the track.

To add a new audio track:

1. On the bottom left of the timeline, click Audio+ for a mono track or `Alt`+Audio+ for a stereo track, or from the Timeline Gear menu, select **New ➤ Mono Audio Track** or **New ➤ Stereo Audio Track**.

   A new audio track is added to the timeline below the last existing audio track.

To add a new version:

1. On the bottom left of the timeline, click Version+ for a mono version or `Alt`+Version+ for a stereo version, or from the Timeline Gear menu, select **New ➤ Mono Version** or **New ➤ Stereo Version**.

   A new version is added to the timeline above the last existing version. If you want the version to be added below the last existing version, press `Ctrl` while clicking Version+.

TIP To add multiple versions or tracks, you can use the keyboard to enter the number of versions or tracks you want to add, before using one of the above methods.

Moving, Resizing, and Deleting Tracks

To move a track:

1. Select the track or tracks that you want to move.
2. Grab the selected track from the left end of the Patch Panel area, and drag up or down to the desired location.

   A red message appears while dragging to help guide while dragging the track.
3. Let go of the track to perform the move.

   NOTE You cannot move a video track to the audio track area and vice-versa.

To resize a track:

1. Place the cursor at the bottom of the Patch Panel area for the track that you want to resize. Notice that the cursor becomes a line with a double-headed arrow.

2. Drag down to increase the height of the track (to see more information on the tracks clips, for example), or drag up to decrease the height of the track.

   TIP To increase or decrease the height of all tracks, drag left or right on the vertical scroll bar to the right of the timeline.

To delete a track:

1. Right-click on the Patch Panel area of the track you want to delete.
2. Click Delete Track.

About Timeline Patching

When you record a source clip to a sequence, you must decide what source clip channels you want to use, and to which tracks you want to record them. To connect the source channels to the destination timeline tracks, you use patching. When you select the source clip, green patch identifiers indicate what channels you can record from the source to the timeline.

In the above example, the top track (V1.3) is patched, but no source is assigned. The middle track (V1.2) is not patched and no source is assigned. The bottom track (V1.1) is not patched, but a source is assigned. If you click the source patch or destination identifier on the bottom track, the track becomes patched, identified by the green patch.

With the patch identifiers, patching information is kept with the sequence clip. If you patch a source clip and then add a new track to the timeline, the source clip follows the track to which it was originally patched. This allows for a natural workflow of setting up the patch for a source clip, adding a new track, then setting up the patching for another source clip.

Patching on the Timeline

To patch a source channel to a sequence track:

1. Select a source in the workspace.

   Source track patch identifiers appear in the Patch Panel area for each channel in the source.

2. Create the patch by right-clicking in the Patch Panel area, and using the Track Patching contextual menu to assign source tracks (from a list of available tracks). You can also select No Assignation to remove destination patch assignments. An assigned source can also be dragged up or down to change the assignment to a different track.

   The patch identifier turns green and its source channel number indicates what source you have recorded to the track.

While the Track Patching contextual menu can be used for all your patching needs, there are particular cases where you can use different patching methods:

- If the track is patched but no source is assigned, drag on the grey patch identifier until the source channel that you want to use appears.
- If the track is not patched and no source is assigned, drag on the grey patch identifier until the source channel that you want to use appears and then click the patch identifier.
- If the track is not patched but a source is assigned, click the grey source channel number on the patch identifier.

Navigating and Searching the Timeline

Although you can search for elements by scrolling the timeline and displaying information about each element, the Find and Select in Timeline options make it easier to find specific elements. You can also refine your search when searching specifically on segments. Once your search is complete, or if you have multiple segments selected on the timeline, you can easily navigate through them.

Click a segment anywhere on the timeline to explicitly select it. If no timeline segment is explicitly selected, the segment closest to the timeline positioner's focus point is implicitly selected (and outlined with a white border).

Navigating with the Positioner

The frame directly beneath the positioner's focus point is displayed in the Player or is the current location for an edit such as a dissolve or cut. Drag the positioner to the desired frame on your timeline.

In the timecode area below the tracks, a lighter yellow box attached to the positioner indicates the length of the current frame, at the timeline zoom level. Click and drag this yellow box to move the positioner, so as not to accidentally move clips on the timeline.

If you know the amount of frames by which you want to move, a quick way to navigate the timeline is by using the keypad to enter the amount into the calculator.

1. On the timeline, make sure that there is no explicit selection of a clip or cut (in this case, a trim may be performed).
2. On the keyboard keypad, do one of the following:
   - Enter the number of frames by which you want to move the positioner (for example, +10 or -10).
   - Enter a number (without a + or -) to navigate to the closest timecode value that finishes with the value entered (for example, enter 10 to move the positioner to the closest timecode value that finishes by :10)
3. Press Enter.

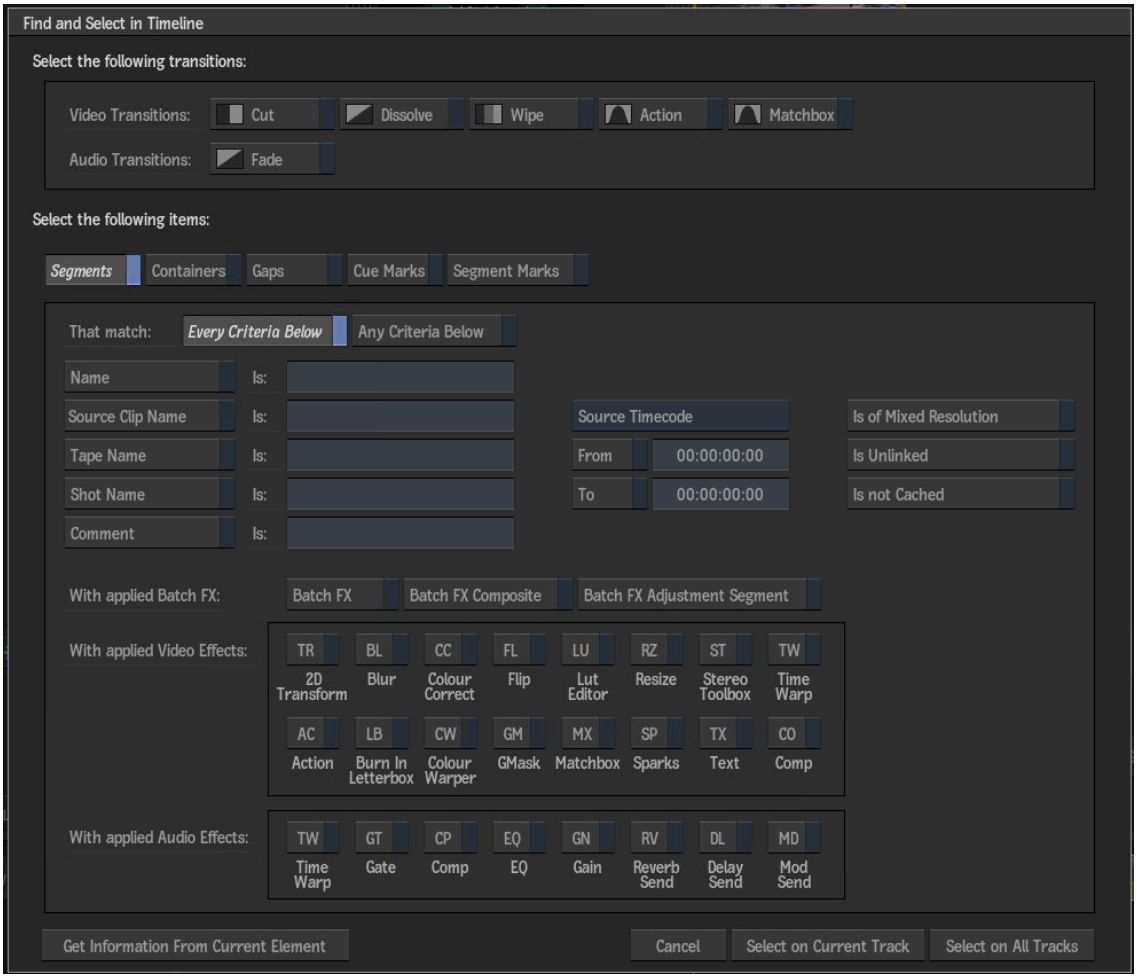
TIP Use the timeline scroll bars to zoom timeline, if needed. Click Home in the Timeline Layout combo box to reset the timeline view.

Searching the Timeline

1. Click the magnifying glass icon at the bottom right side of the timeline.

   The Find and Select in Timeline window opens.



2 Enable a button or multiple buttons to set your search criteria. For example, you can enable Dissolve in the Video Transitions area to search for all dissolves.

TIP You can enable Get Information From Current Element to automatically fill in information from the selected element into the search criteria.

3 Choose whether to search segments, containers, gaps, cue marks, or segment marks. You can select more than one or all of these options.

4 Set the strictness of the search by enabling Every Criteria Below or Any Criteria Below.

5 Click Select on Current Track or Select on All Tracks to perform the search.

The Find and Select in Timeline windows closes, and results are highlighted on the timeline.

Navigating Selected Elements on the Timeline

If you have multiple segments selected on the timeline (as the result of a search, or as a manual selection), you can easily navigate through them with keyboard shortcuts or with these procedures.

To navigate on the timeline in a linear manner:

1 In the Player, select and hold the Go To Next button to display more options.

2 Click Selected Element.

3 Use the Go To Previous and Go To Next buttons to scroll through the selected timeline elements.

To navigate to a specific selected timeline segment:

1 In the FX pipeline above the timeline, click the arrow beside the segment thumbnail.

You can see a list of all selected segments.

2 From the list, select the element you want to navigate to.

The timeline positioner moves to the selected segment.

Using Reveal to Locate Clips or Segments

To help you locate timeline clips or segments in the Workspace Media panel, you can use the reveal options. To locate a clip, you can double-click the timeline tab, or right-click the clip on the timeline, and select **Reveal ➤ Clip in Media Panel**. You can also locate a specific clip segment by right-clicking it on the timeline, and selecting **Reveal ➤ Segment in Media Panel**. The clip or segment is then highlighted in the Workspace Media panel, and the Thumbnail or Reels view, if displayed.

Using Markers on the Timeline

Use cue marks or segment marks to mark frames of interest on a clip. You can then quickly go to the marked frames. These marks are for reference purposes only; they do not affect the clip and are not used in any editing operations. You can add a mark to all tracks (*cue mark*) or to a single track or version (*segment mark*) on the timeline. Segment marks move on the timeline as you trim, slip, or slide a clip whereas cue marks stay at the same position on the timeline.

You can also add In and Out marks on timeline clips, as you can in the player.

To add marks on the timeline:

1 Go to the frame where you want to insert the mark.

2 From the Timeline Gear menu, in the Marks section, select an option to add an In, Out, Cue, or Segment mark.

3 If you added a cue or segment mark, you can change the name and colour of the mark to help you identify it more easily.

NOTE The Marks section of the Timeline Gear menu offers other options for clearing or deleting marks. The Go To section of the Timeline Gear menu offers navigation aids for existing marks.

About Timeline Containers

Containers are a convenient way to remove clutter from the timeline by grouping or nesting elements together. The elements, which can come from different tracks or versions, are treated as one unit yet remain individually editable. Containers behave like any other element but they appear in a separate timeline tab when you enter their editor. You can also add containers within a container.

Containers can also be used to take a clip and matte that are separate and add them to the timeline as a single element.

NOTE You cannot contain audio across tracks, but you can contain adjacent audio elements.

Creating a Timeline Container

To create a container:

1. Select all the elements on tracks or versions on the timeline that you want to collapse into the same container.

   NOTE The bottom track is always trimmed out so that you can dissolve from an element to a container.

2. From the Timeline Gear menu, select Contain Selection.

   All the selected elements are collapsed into a container and the element changes to a dark blue. You can add effects or perform other timeline operations to the contained element.

3. To edit the contents of the container individually, double-click the container, or click Open in the Container quick menu.

   The elements that make up the container appear in their own timeline tab (the tab has a blue line). Yellow marks indicate the duration of the container. From this view, you can add effects, tracks, versions, or any other timeline operation.

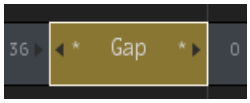
To remove a container:

1. Select the container in the sequence where it was created.
2. From the Container quick menu, click Uncontain.

   The container is removed and its contents are restored to the sequence.

About Timeline Gaps

Gaps are spaces on a track that do not contain media. You can use gaps to see through one layer to another or to apply effects to the tracks below the gap. If a gap exists on a track, the next track that contains video media is included in the edit as long as it is also below the focus track.



Here are some tips for working with timeline gaps:

- Since timeline gaps are independent of all media, they are useful for applying Timeline FX or Batch FX (as an Adjustment Segment) that affect all tracks under the gap.
- By default, empty gaps are transparent. However, you can make an empty gap opaque if you do not want to see through to the next track. To do so, select the gap on the timeline, then click Colour Source in the Gap quick menu, then select a colour for the gap. Click Remove Colour to return a gap to transparent.
- While editing on the timeline, you can also quickly remove a gap between elements by right-clicking the gap, and choosing Close Gap.
- When selecting elements on the timeline with a rectangle selection (that is, when drawing a bounding box around them), you can choose whether to include gaps by enabling or disabling Selection Includes Gaps in the Timeline Options menu.

Playing the Timeline

You can control how the timeline plays back. For example, frame-step through a shot to isolate a frame or jump to the start of an element to play a specific shot. The image window updates as the positioner moves on the timeline. While you can play your timeline in any of the Viewing panel's View modes, if the Player is displayed, you can also use any of the playback controls to move through the timeline. The Player also offers compare modes, where you can set up a side-by-side or split view to play primary and secondary versions or tracks at the same time.

To play the timeline:

1. Press `L` to play forward.
2. Press `J` to play backward.
3. Press `K` to stop playback.

TIP Press `J` or `L` multiple times to play faster. Press `Shift` along with `J` or `L` to play slower.

To scrub the timeline:

1. Drag the positioner, or the scrub area at the bottom of the timeline, left or right.
   When scrubbing in the Source tab, you cannot navigate past the first or last frame of the clip.

To jump to a particular location:

1. Do one of the following:
   - Click the scrub area at a particular location.
   - Enter a value in the Current Timecode field.

     TIP To enter a frame number instead of timecode, click the field to bring up the numeric keypad. Click TC to change to FRM, and then enter a frame number.

To frame-step the timeline:

1. Press the left or right arrow key.

   TIP To frame-step a specific number of frames, enter a number in the numeric keypad and then press the left or right arrow key.

Cutting Clips on the Timeline

When you splice clips together, the location at which one clip changes to another is called a cut. A cut is useful when you want to extract part of an element or end a shot at a given frame.

You can make a cut at any point on a segment. A cut creates a transition with a head equivalent to the duration of the segment before the cut and a tail equivalent to the duration after the cut. For example, if you make a cut after the 5th frame in a 10-frame clip, the cut has a tail of 5 frames and a head of 5 frames.

In a match frame cut, the outgoing and incoming shots are from the same source and the outgoing and incoming frames are consecutive. Match frame cuts can be applied to clips with no effects added, and are indicated by an “=” on the cut point.

To add a cut on the timeline:

1. Move the positioner and focus point over the frame where you want to add a cut.
2. Select Cut from the Timeline Gear menu.
   A cut is added at the specified location.

   TIP You can also add cuts while the clip is playing by pressing the cut keyboard shortcut.

To remove a match frame cut:

1. Select the cut on the timeline.
2. Right-click and choose Remove Match Cut.
   The cut is removed and the two elements are joined together.

Swapping Timeline Elements

You can quickly swap elements on the timeline. If segments contain timeline effects, they are swapped as well.

You can swap the following elements:

- Video segments or containers
- Audio segments or containers
- Tracks
- Versions
- Contiguous sequences of elements
- Video transitions (unless they start or end a contiguous sequence of elements)
- Audio transitions
- Gap effects

Elements must be compatible to be swapped. Compatible elements are:

- Video segments, video containers, gap effects
- Audio segments and audio containers
- Video transitions
- Audio transitions
- Tracks and versions

The following elements cannot be swapped:

- Gaps (unless they are between elements in a contiguous sequence)
- Cuts
- Cue marks
- Individual timeline effects

To swap timeline elements:

1. Select two elements belonging to the same family.
   A contiguous sequence of elements is treated as a single element as long as the transition between the elements is selected. A gap can be part of a contiguous element as long as it is not at the beginning or end of the sequence.

2 Press the keyboard shortcut for Swap Selected.

The elements are swapped according to the ripple setting. If the segments contain any timeline effects, they are swapped as well.

If Ripple is disabled, the segments are timewarped using a Constant Fit-to-Fill timewarp to fill each other's location. If there is already a timewarp on the segment, the timewarp is replaced by the Constant Fit-to-Fill timewarp. Gap effects, Matte containers, and contiguous sequences ripple regardless of Ripple status because they cannot be timewarped.

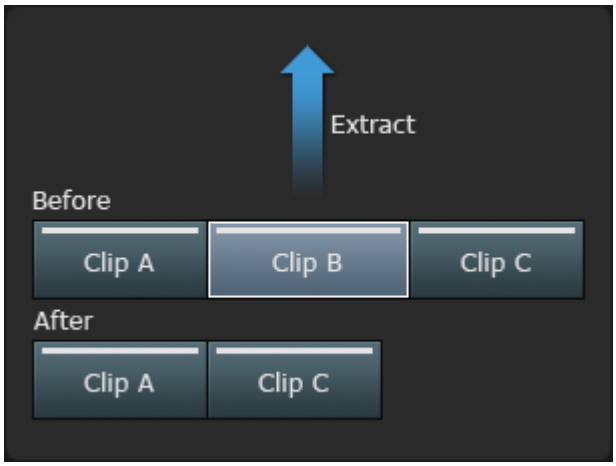
Transitions keep their alignment but their duration changes based on the head and tail frames of the destination segments.

Removing Elements From the Timeline

When you remove an element from an edit sequence gesturally, the effect depends on whether Ripple is on or off.

When Ripple is off, you *lift* an element, leaving a gap in place of the lifted element.

When Ripple is on, you *extract* an element, and the edit sequence collapses to fill the gap left by the extracted element. This is also called a *ripple delete*.



You can lift or extract elements from the timeline independently of the Ripple mode using the contextual menu or keyboard shortcuts.

To lift or extract an element from using the contextual menu:

1. Select the element (or gap) you want to lift or extract or mark in and out points around the material.
2. Right-click the element and select one of the following:
   - Select Lift to lift the element from the timeline, leaving a gap in place of the lifted element.
   - Select Ripple Delete to extract the element from the timeline and ripple the remainder of the timeline to fill the gap.

To gesturally remove an element from an edit sequence:

1. Do one of the following:
   - Turn off Ripple to lift.
   - Turn on Ripple to extract.
2. Select the element (or gap) that you want to lift or extract and drag it out of the timeline.

   NOTE Performing a Lift or Ripple Delete (Extract) operation on the timeline does not delete a segment between marks if another explicit selection is made.

Matching Clips with Their Sources

You can match a clip on the Desktop or timeline with its source clip. There are numerous options to match clips with their sources, depending on the result you are trying to achieve. You can select any or all of the following from the contextual Match options on a desktop or timeline clip:

- **Preserve Handles:** The Head / Tail of the timeline segment is preserved.
- **Use Sequence Info:** A new source with the timeline segment info is created.
- **Include Nested Segment:** The match is performed on the result BFX clip, or on the result clip and all the sources included in it.
- **Include Timeline FX:** The Timeline FX present on a clip are copied to the new Matched clip.

Once you have selected the match criteria from the list above, you can select Match or Match All.

The matched source is selected in the Media panel, and the cursor becomes a white arrow, prompting you to select a destination reel on which to display the source.

Grouping and Syncing Elements on the Timeline

You can create editorial groups to preserve the relationship between timeline elements in a vertical composition. For example, if you have a video clip with two tracks of audio overlapping on the timeline, you can create a group for them. When you edit one segment that is part of a group, all segments in the group are edited in the same way. If your group becomes out of sync, a visual cue is displayed on the timeline, and tools are available to help you resync the elements.

To create an editorial group:

1. On the timeline, select the overlapping video or audio elements to include in the group.
2. From the Timeline Gear menu, select **Group ➤ Group**.

   NOTE You can also right-click your selection, and choose Group from the contextual menu. If Group is unavailable in the contextual menu or the Timeline Gear menu, the selection is not able to be grouped.

Once a group is created, file names on the elements are underlined to indicate group status.

3 Enable Link to allow elements in the group to be edited together (you can use the keyboard shortcut for Invert Group Selection to temporarily change the status of the Link button during manipulation).

TIP If you cut a segment that is part of a linked group, all elements in the group are cut at the positioner location, and two separate groups are created from the cut segments.

Resetting Sync

If an editing operation removes the sync for a track in a group, a red plus (+) or minus (-) symbol appears in the element with the amount of frames that are out of sync to the left (-) or right (+).

You can attempt to manually fix the sync issue by editing the out-of-sync element. In this case, make sure that Link is disabled. Once you have resynced, you can enable Link again.

You can either resync the elements or reset the offsets, creating a new sync relationship based on the current position of the segments.

To regain sync:

1 Select the element that is out of sync.

2 Right-click the element, and select Resync.

The element is resynced, if possible, and the out-of-sync symbol disappears.

To reset sync offsets:

1 Select any element in the edit sync group.

2 Right-click the element, and select Reset.

The segments remain in their current positions. The sync offsets are removed creating a new edit sync group.

Deleting and Disabling Editorial Groups

When you delete an editorial group, the sync is removed from all elements that were part of the group. You can also temporarily disable an editorial group.

To delete an editorial group:

1 Select one of the elements that make up the group.

2 From the Timeline Gear menu, select **Group ➤ Ungroup**.

NOTE You can also right-click your selection, and choose Ungroup from the contextual menu.

To remove sync temporarily:

1 Select one of the elements that make up the group.

2 Disable Link.

3 Edit an element of the group.

No other elements in the edit sync group are affected. A red plus (+) or minus (-) symbol appears on every element that is out of sync.

TIP You can change the Link mode temporarily on-the-fly by using the keyboard shortcut for Invert Group Selection.

Syncing Tracks and Versions

You can use the track and version-based sync locks to decide the behaviour of tracks or versions when performing editing operations, such as trimming or an insert edit. In this case, you do not have to select individual elements on tracks.

The contents of a track set to Sync can ripple even when nothing is selected for the edit on it. This is to keep segments in sync throughout many tracks and versions. For example, performing an insert edit on a track can have the effect of a gap of the same length as the inserted material being created on another synced track.

To turn on or off syncing for tracks or versions, click the sync lock icon in the Patch Panel area. In the following example, sync is enabled for tracks V1.1 and V1.3, and disabled for track V1.2.

You can also right-click a sync lock icon and select to sync or remove sync from all tracks or versions.

Affected Editorial Operations

The state of the track or version sync lock is taken into account when performing the following operations:

- Insert
- Extract
- Delete*
- Trim*
- Move*
- Slide*

*Ripple must be enabled.

These editing operations ignore sync lock:

- Aligned Edit
- Overwrite
- Replace
- Append
- Prepend
- Lift

■ Slip

Working with Connected Segments

Connecting Segments Across Sequences

You can create a connected segment, use that segment across multiple timelines, and then sync the connected segments to propagate changes across timelines, without using the Shots Sequence and Connected Conform workflow.

1. Create a new Reel Group.
2. Move the sequences that will be using the same connected segment to that new Reel Group.
3. Open the sequences.
4. In one the sequences, select the segment you want to connect across your sequences.
5. Right-click and select **Create Segment Connection**.
6. Right-click and select **Copy**.
7. Paste the segment the other sequences.

You can now use the same tools available to you in Connected Conform: sync segments, navigate between connected segments, propagate Timeline FX across connected segments.

Shots Sequence is the best solution when working in a multi-sequence conform workflow, where you need to keep many segments in many sequences in sync.

Connected Segment Icon A segment connected across multiple sequences is identified with

.

Navigating Instances of Connected Segments `Right-click` a segment and select Jump to Connected Segment: the drop-down list displays the sequences with an instance of that segment. Selecting a sequence from the list opens that sequence, with the segment selected.

Syncing Connected Segments Modifications made to the Timeline FX, Batch FX, or History of a given segment are local to that segment until you sync it. For example, when working with the Shots Sequence.

1. In the Shots Sequence, apply a Timeline FX on a Connected Segment or on a selection of Connected Segments.
2. From the contextual menu, select Sync Connected Segments.
3. All corresponding Connected Segments located in the connected sequences in the Reel Group are updated.

   TIP Render all of the updated Timeline FX for all synced Connected Segments by rendering the original segments Timeline FX before syncing.

Notes:

- You can undo this sync.
- You can sync a selection of connected segments. You can sync a segment from any sequence part of the Reel group. When working with a Shots Sequence, sync from the Shots Sequence, since this sequence contains every connected segment.
- The number of connected segments that were synced appears in the Message Console.

Replacing Connected Segments To replace a connected segment across the connected sequences of a Reel group:

1. Open Timeline.
2. Select the segment to be replaced.
3. Select the clip that will replace the segment.
4. Do one of the following:
   - To replace the segment and overwrite the Timeline FX applied to the destination segment, use the Timeline Replace operation. Press `R`, or click the Replace button.

   - To replace the segment and preserve the Timeline FX applied to the destination segment, use the Timeline Replace Media operation. Press `Ctrl-Shift-R`, or click the Secondary Edit box and select Replace Media.

5. All instances of the connected segment are updated.

   NOTE The media used must be long enough to cover the whole of the segment it replaces, or this will fail.

The Replace operation can be performed on any clips, whether they have Timeline FX applied, a Batch FX, or include History.

Removing Segment Connections Sometimes you need to isolate a segment to perform a unique edit, such as applying language-specific titling to multi-language deliverables; the connection of segments' metadata becomes a hindrance. In this case, you want to remove the segment's connection to other segments.

1. Select the segment to isolate.
2. From the contextual menu, select Remove Segment Connection.

IMPORTANT While you can Undo this operation, there is no Restore Connection operation that can be performed later. Although you can create new connected segments.

Timeline Reference

Use these settings to help you navigate and perform tasks on the timeline. You can also right-click anywhere on the timeline to display a contextual menu of relevant operations for the type of element selected.

TIP There is also a Timeline tab (page 1943) in the Preference menu with numerous other timeline-related settings and preferences.

Navigation

Positioner The "playhead" for playing the clip, displayed as a vertical yellow bar. The frame directly beneath the positioner is displayed in the Player or is the current location for an edit such as a dissolve or cut.

In the timecode area below the tracks, a lighter yellow box attached to the positioner indicates the length of the current frame, at the timeline zoom level. Click and drag this yellow box to move the positioner, so as not to accidentally move clips on the timeline.

Focus point A horizontal yellow line on the positioner indicating the current track.

Back button Click to move back one page in the timeline window.

Horizontal scroll bar To pan, drag left or right. To zoom the timeline, drag up or down.

Horizontal Zoom

Forward button Click to move forward one page in the timeline window.

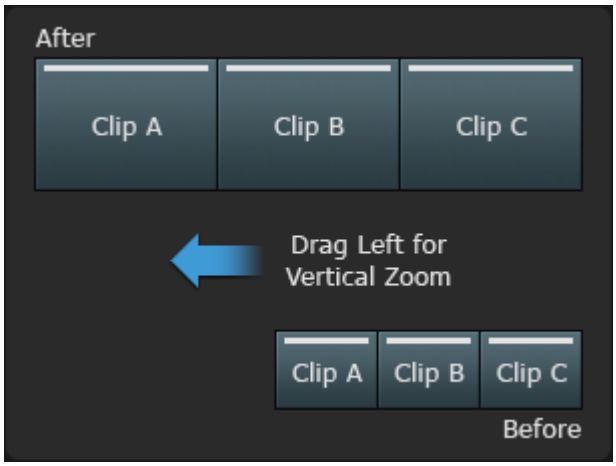
Current Timecode field Displays the timecode of the current timeline position. Editable. This field displays timecode or frame number depending on the selection in the Time Display Mode box located in User Interface preferences.

Current Frame field Displays the frame number of the current timeline position. Editable. This field displays timecode or frame number depending on the selection in the Time Display Mode box located in User Interface preferences.

- button Click to expand track height.

+ button Click to decrease track height.

Vertical scroll bar To pan, drag up or down. To zoom the timeline, drag left or right.



Vertical Zoom

Timeline Search button Click to open the Find and Select in Timeline window to search the timeline using many different criteria.

Timeline Layout combo box Select an option to display the timeline.

Select:	To:
Home	Reset the view of the timeline.
Reset Height	Reset the height of the tracks in the timeline.
Fit to Width	Reset the horizontal scale of the timeline.
Fit Selection	Centre the timeline on the selected element.

Colour Management

Tagged Colour Space menu Choose the colour space to tag your media with (this does not modify pixel values, but it does affect how viewports display your media).

Input Colour Space menu Select the source colour space of your media. "From Source" uses the the currently tagged colour space, which allows you to easily convert a set of clips in various colour spaces to a common working space.

Working Colour Space menu Select the destination colour space of your media. Set the Working Space first in order to filter the list of Input Colour Spaces to only show the compatible options. "From Project" takes the value from the Colour Management Preferences.

Tool Mode button Choose whether to convert your media into a different colour space or only tag them.

Get Colour Space From Selection button The tool starts with the colour space setting that was last used. Click this button to initialize with the colour space of your primary selection.

Rendering

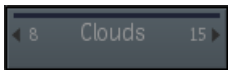
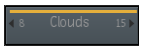
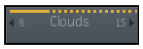
Rendering combo box Select an option to render timeline effects. You can edit effects after rendering.

Select:	To:
Render Selection	Render the selected elements.
Render	Render to topmost track in the timeline.
Full Resolution	Render at full resolution, regardless of the current working resolution.
Proxy	Render proxies only, not the corresponding high-resolution images. An amber outline appears on the timeline segment to indicate the state of the rendering. Available if proxies are selected during the setup of a project.
Full Resolution and Proxy	Render at bot full and proxy resolution.
Include Handles	Select to take the heads & tails into account when rendering.

Other options appear to set the rendering mode for the selected option if you are using Burn or Background Reactor.

TIP If you need to refresh (or force) a render on a selection, first select Flush Renders from the Media submenu in the Timeline Gear menu, then re-render the selection.

Depending on how a clip is rendered, you can see a visual representation on the render bar at the top of the clip on the timeline:

Render Bar:	State:
	Unrendered.
	Rendered.
	Partially rendered.
	Proxy rendered.
	Proxy partially rendered.
	Sent to Burn or Background Reactor.

Viewing

Timeline View Mode box Select a viewing mode for working in the timeline.

Select:	To Display:
Timeline	Video and audio tracks in a timeline format with a positioner at the location of the current frame (press `Shift+F5` to toggle between a larger timeline view and the normal timeline view).
Collapsed	Only the current frame of each edit sequence. The edit sequence appears in the form of a stack. Shuttle through the sequence by dragging the cursor across the bottom of the frame.
Head-Tail	The first and last frames of each clip. Shuttle through the sequence by dragging the cursor across the bottom of any frame.
Storyboard	Only the first frame of each clip in the edit sequence. Shuttle through the sequence by dragging the cursor across the bottom of any frame.

Select:	To Display:
Frames	Every frame in the edit sequence is displayed. Although you cannot edit in this mode, it is useful for viewing your entire edit.
History	A schematic or list representation of the majority of operations performed on a clip or timeline element.
Clip List	A list of objects and information about these objects that make up the timeline.

TIP In Collapsed or Storyboard view, you can drag sequences to reorder them. If you switch back to Timeline view, the new order of sequences is maintained.

Timeline Tasks

Timeline Gear menu Select a task to perform on the timeline. Some of the options have sub-menus with more options.

Select:	To:
New Tracks and Versions	Add new tracks or versions to the timeline.
Edit	Perform various element or timeline editing operations.
Rename	Rename the selected timeline element.
Rename Track	Rename the selected track.
Media	Options to flush renders, and to break the link between an element's metadata and media or high-resolution media.
Select All	Select all elements in the timeline.
Deselect All	Deselect all elements in the timeline.
Container	Work with containers on the timeline.
Group	Create an edit sync group for selected timeline elements.
Go To	Navigate to various areas of the sequence.
Marks	Work with In and Out, Cue, or Segment marks.
Stereo	Work with stereo tracks on the timeline.
Cut	Add a cut to the selected element at the positioner location.

Select:	To:
Cut All Tracks	Add a cut to all tracks at the positioner location.
Merge Tracks	Merge timeline tracks into a single track.
Swap Effects	Copy timeline effects between clips.
Lift	Remove the selected element from the edit sequence, and leave a gap in its place.
Ripple Delete	Remove the selected element from the edit sequence, and close the gap.
Close Gap	Remove a gap and collapse the elements to fill the gap.
Commit	Options to manage your timeline to simplify or save space. See Timeline Media Management Options (page 455)
Hard Commit	Commit an edit sequence to one continuous clip (also called a mixdown). Soft properties of the sequence such as transitions, head and tail frames, and timewarp data are removed.
Render	Manage render options for the timeline.

Other Options

Timeline Options menu Select an option for working on the timeline.

There are a number of audio options in this menu:

Select:	To:
Scrub Audio	Enable scrubbing audio while dragging the positioner.
Show Gain Animation	Display the animatable Segment Gain level.
Show Timeline Audio Meters	Display mini audio meters to the right of the timeline. Audio Meters are displayed only if the current clip or sequence has at least one audio track.
Show Waveforms With Effects	Display waveforms after an effect is applied to an audio segment.
Show Waveforms Without Effects	Display waveforms before an effect is applied to an audio segment.
Hide Waveforms	Not display waveforms.
Increase Waveform Range	Zoom in on the waveform.
Decrease Waveform Range	Zoom out on the waveform.

Use the Rectangle Selection options to select how timeline elements are included in a selection when drawing a bounding box around them.

Choose:	To select:
Inclusive	All elements that are partially or entirely included in the bounding box.
Partial	Only the portions of the elements that fall within the bounding box. Audio is selected on a sub-frame basis if Sub-frame Positioner is selected in the Timeline section of the Preferences menu.
Partial A/V	Only the portions of the elements that fall within the bounding box. Audio is selected on a frame basis even if Sub-frame Positioner is selected in the Timeline section of the Preferences menu.
Bounded	Only elements that are fully in the bounding box.

Enable Selection Includes Gaps to include empty gaps when performing a timeline selection.

Enable Selection Includes Locked Tracks to allow you select a Timeline segment even if is resides on a locked track.

Enable one or both snap options to include in the timeline snap criteria:

- Snap To Positioner on Gestural Insert
- Snap Includes Marks

Enable Focus on Trim if you want the timeline positioner to snap to the transition while trimming, allowing you to view frames in the player.

Timeline Media Management Options

The Commit area of the Timeline Gear menu offers a number of options to help you clean up your timeline or save space when archiving a project.

Commit Options

When you commit an edit sequence, it becomes one continuous clip. The soft properties of the sequence such as transitions, head and tail frames, and timewarp data are removed. The visual effects of the transitions and timewarps remain but you cannot edit them.

You can commit any selection of elements or tracks, or the entire timeline. Commit a portion of an edited sequence to simplify a timeline that is too complex or to recoup disk space.

You cannot recapture or reimport committed clips.

The following rules apply to hard commits:

- Hard commits are permanent; however, they can be undone using the Undo button (as long as undos remain in the undo buffer). If you want to consolidate elements but have the ability to make changes at a later time, use containers instead.
- Hard commits cannot be restored—the commit is permanent.
- Hard commits force a render on the selected elements.

You can also choose to only commit Timeline FX or Batch FX segments on the timeline.

TIP Commit options are also available from the contextual menu when right-clicking clips on the timeline. When right-clicking a timeline cut, you can also choose Hard Commit to commit the incoming and outgoing clips, as well as the transition (this option is unavailable if the cut is adjacent to a timeline gap).

Consolidate Handles

If you have video or audio elements on the timeline that have excessive handles, use the Consolidate option to remove head and tail frames. Consolidate your clip before archiving a project to avoid archiving a large number of unused frames. You can keep a specified number of head and tail frames when you consolidate (the calculator appears when you select Consolidate for you to enter the number of handles you want to keep). You can consolidate a single element or a selection of elements.

History Options

You can commit a clip with a history, or a clip segment with a history just as you would any other clip. When committing a clip, you delete the clip history without removing the clip or element's soft properties. Delete History deletes history frames for all selected segments.

Editing on the Desktop 11

About Editing on the Desktop

You can apply many types of edits to clips on the Desktop - splitting clips in two, splicing clips together, and adding dissolves.

Editing functions such as splicing clips create soft edits, which you can modify any time until you commit them. Soft edits created on the Desktop can be brought into the timeline for further manipulation.

All Desktop editing operations are applied to the audio and video by default. To edit the video only, select Edit Video Only from the Reel Editing box.

Some Desktop editing operations can be accomplished in two ways - by using menu commands and by using drag and drop gestures with the mouse or stylus. For these cases, this chapter describes both methods.

You can disable broadcast monitor support option in the User Interface tab of the Preferences. Disabling this option enhances interactivity when manipulating clips and performing gestural edits on the Desktop Reels.

NOTE When this option is disabled, Timeline FX are not automatically rendered and display Unrendered Frame on the Reels. However, the frame on which the Timeline positioner is parked, is rendered.

Editing Multiple-Resolution Clips

Editing operations that are performed on more than one clip, such as splicing two clips together, can only be done on clips of the same resolution, aspect ratio, and bit depth. For most editing operations involving two clips, if you attempt to use clips of differing resolutions, a pop-up menu appears stating that the source and destination clips are mismatched. If you click confirm, the source clip will be resized to match the destination clip.

Gestural Editing on the Desktop

Gestural editing is a technique in which you use your mouse or stylus to move clips among reels on the Desktop, and to perform cut, copy, insert, splice, and replace edits.

Gestural Editing Cursors

When you drag a source clip over a target clip, the cursor changes as the source clip moves over different areas of the target clip. The type of cursor indicates the type of edit that occurs when you drop the source clip on that spot.

All gestural splices, insert edits, and replace edits ripple the edit sequence; that is, they change the duration of the target clip unless you replace a clip with another clip of the same duration.

Cursor:	Appears when:	Use to:
 Insert	The source clip is positioned between frames on the target clip.	Perform an insert edit.
 Replace	The source clip is grabbed at any frame other than its first or last frame, and is positioned over a frame on the target clip.	Perform a replace edit. You cannot gesturally replace frames with a two-frame clip. However, you can perform this type of edit with a one-frame clip.
 Replace ahead	The source clip is grabbed at its first frame and is positioned directly over a frame on the target clip.	Perform a replace edit starting at the frame under the cursor and moving in the direction of the arrow.
 Replace back	The source clip is grabbed at its last frame and is positioned directly over a frame on the target clip.	Perform a replace edit starting at the frame under the cursor and moving in the direction of the arrow.

To perform a gestural insert or replace edit:

1. Drag the source clip over the frame in the target clip where you want to make the edit until you see the appropriate edit cursor.

2. Drop the source clip.

   The clip is updated with the edit. The source clip replaces the frames of the target clip

Ganging Clips on the Desktop

You can gang clips and sequences together on the Desktop so that when you jog one, they are all jogged. This is useful for multicam setups. When you play a clip that is part of a gang, the current frame of all other ganged clips is updated once playback has stopped.

To gang two clips on the Desktop:

1. Make sure the clips are on different reels.
2. Align each clip to be ganged to the timecode you want to lock.
3. Select the clips while holding the `Ctrl` key, to make a multi selection.
4. From the contextual menu, select Gang.
   The selected clips are ganged and the clip information turns green.

To select all clips in a gang:

1. Select any clip and/or sequence that is part of a gang.
2. From the contextual menu, select **Gang ➤ Select Gang**.
   All ganged clips/sequences are selected.

To remove a clip from a gang:

1. Select the clip(s) and/or sequence(s) you want to remove from the group.
2. From the contextual menu, select **Gang ➤ Ungang**
   All selected clips and/or sequences are unganged.

Cutting Clips

You can split a clip into two or more clips using the Cut command. When you cut a clip, you can retain the cut frames as head and tail frames (handles) on each resulting clip, creating a soft clip, or you can discard the frames.

The Cut options are described as follows.

Select:	To cut the clip:
After Selected Frame	After the frame that you select when in Cut mode.
At Splices	At every soft splice that occurs on the clip.
Every Nth Frame	At the intervals that you specify in the numeric field that appears when you select this option.

To cut a clip after the selected frame:

1. Select the frame after which you want to apply a cut.
2. From the contextual menu, select **Cut ➤ After Selected Frame**.

   The clip is divided into two cuts after the selected frame. The frames in the second clip are renumbered starting at frame number 00001.

To cut a clip at its splices:

1. Select the clip on which you want to perform the cut operation.
2. From the contextual menu, select **Cut ➤ At Each Splice**.

   The clip is cut into segments at every splice point. If the clip contained no splices, no cuts are made.

To cut a clip at every Nth frame:

1. Select the clip on which you want to perform the cut operation.
2. From the contextual menu, select **Cut ➤ At (n) Frame**.
3. Enter the number of frames to be contained in each cut in the Frame Number field.

   The clip is cut into segments of the selected number of frames.

Cutting Dissolves and Timewarps

Performing a cut on an uncommitted dissolve or timewarp divides the soft edit into two soft cuts.

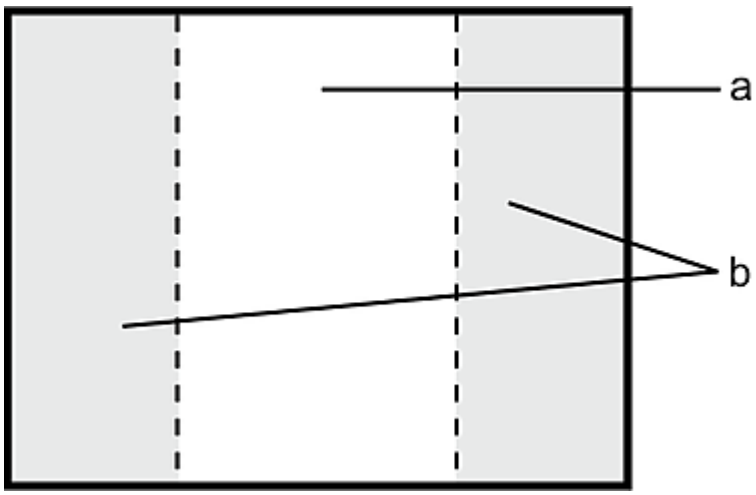
If the cut is performed on a frame within a dissolve, the dissolve is committed before the cut is made. The two resulting cuts are soft edits linked to the original source clips.

If the cut is performed on a frame within a timewarp, two clips with unrendered timewarps are created.

NOTE You should commit a soft cutout prior to archiving to avoid saving the soft cutout's source clip. See Hard Committing from the Desktop (page 466).

Cutting Clips Gesturally

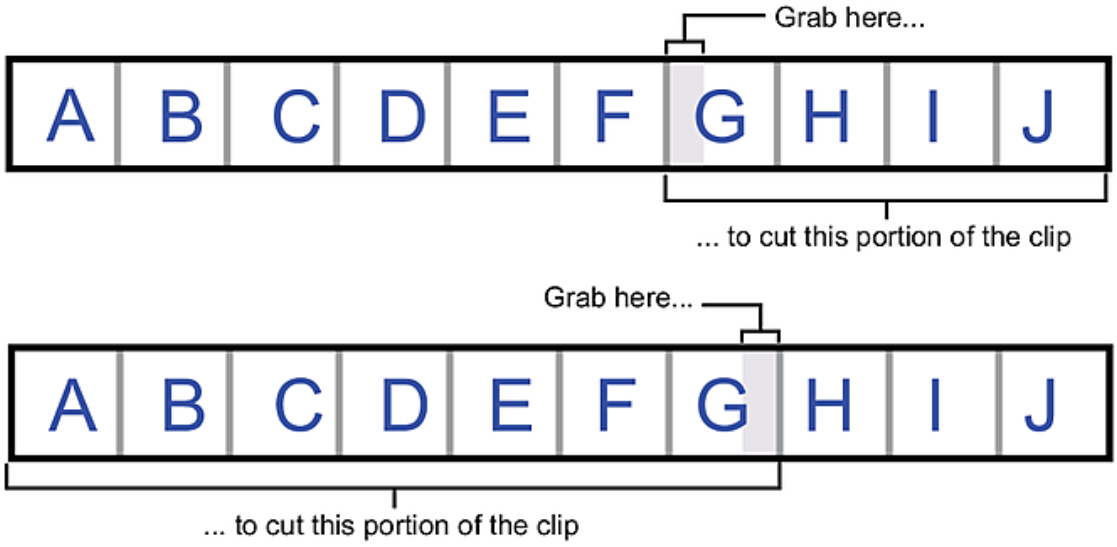
Each frame in a clip contains two types of “hot spots”. The Move Clip hot spot is the part of the clip that is used to move clips around on the Desktop. The Cut Clip hot spots are the parts of the clips that are used to cut the clip.



(a) Move Clip hot spot (b) Cut Clip hot spots

To cut a clip gesturally:

1. Position the cursor over the frame where you want to make the cut. Place the cursor in the Cut Clip hot spot on the frame edge closest to the cutting point.
2. Grab the frame and drag up or down to cut it from the clip.
3. Drop it in another location on a reel.



Splicing Clips

You can join together either two or more clips on a reel. This creates a single clip with a soft edit (a splice) at the location where the clips were joined.

To splice two or more clips together:

1. Select the clips you want to splice together, by Ctrl+clicking them.
2. From the contextual menu, select Splice Selected.

The two clips are spliced together to form a single clip. A yellow transition handle indicates where the two clips were spliced together. To remove the yellow handle, hard commit the clip. See Hard Committing from the Desktop (page 466).

To splice together all clips on a reel:

1. Place the cursor in an empty area to the right or left of the reel.
2. From the contextual menu, select Splice Reel.

   All clips on the reel are spliced together into one clip.

   Yellow transition handles indicate where the clips were spliced together. To remove the yellow handle, hard commit the clip. See Hard Committing from the Desktop (page 466).

Splicing Clips Gesturally

To splice two clips together gesturally:

1. Select the source clip you want to splice into a target clip, using the Move Clip hot spot.
2. Move the selected clip so that it overlaps either the first or last frame of the target clip. The Insert cursor is displayed.
3. Release the clip.

The two clips are spliced together to form a single clip. A yellow transition handle indicates where the two clips were spliced together. To remove the yellow handle, hard commit the clip. See Hard Committing from the Desktop (page 466).

Inserting Frames and Clips

You can insert a source clip or a portion of a source clip after a frame in the target clip.

Inserting Frames Gesturally

You can insert a portion of a source clip after a selected frame in the target clip.

To insert frames:

1. Select the source clip from which you want to insert frames into a target clip.
2. In Collapsed mode, scrub the source clip to the first frame you want to insert and drop a marker, using the Open Bracket key ([ ).
3. Scrub the source clip to the last frame you want to insert and drop a marker, using the Close Bracket key ( ]).
4. While holding Ctrl+Alt+Shift, drag up from the source clip to extract the selected frames.
5. Move the selected frames so that they overlap the target clip in between the two frames within which you want to insert the source frames. The Insert cursor is displayed.
6. Release the frames.

The source frames are inserted in between the selected frames in the target clip. The remaining frames of the target clip are placed at the end of the inserted clip.

Inserting Clips Gesturally

You can insert an entire clip between two frames in the target clip.

To insert a clip:

1. Select the source clip you want to insert into a target clip, using the Move Clip hot spot.
2. Move the selected clip so that it overlaps the target clip in between the two frames within which you want to insert the source clip. The Insert cursor is displayed.
3. Release the clip.

The source clip is inserted in between the selected frames in the target clip. The remaining frames of the target clip are placed at the end of the inserted clip.

Replacing Frames and Clips

You can replace frames in the target clip with a source clip or with frames from a source clip.

Replacing Frames Gesturally

You can replace frames in a clip with an entire source clip. There are two ways to replace clips gesturally:

- **Replace** will replace the frame on which you drop the source with the current frame of the source and the source frames before and after the current source frame will replace the corresponding frames in the target clip.
- **Replace Ahead** will replace the frames after the selected frame in the target clip (including the frame on which the source clip was dropped).
- **Replace Back** will replace all the frames before the selected frame in the target clip (including the frame on which the source clip was dropped).

To perform a Replace frames operation:

1. Make sure the source clip used is not at the first or last frame.
2. Move the selected clip over the first frame that you want to replace in the target clip. The Replace cursor is displayed.
3. Release the clip.

The current frame in the source clip is matched to and replaces the frame on which you dropped the source clip and the source frames before and after the current source frame replace the corresponding frames in the target clip.

To perform a Replace Ahead operation:

1. Make sure the source clip used is at frame 1.
2. Move the selected clip over the first frame that you want to replace in the target clip.

   The Replace Ahead cursor is displayed.
3. Release the clip.

   The current frame in the source clip is matched to the frame on which you dropped the source clip and the frames after the selected frame in the target clip are replaced with the frames of the source clip, including the frame on which the source clip was dropped.

To perform a Replace Back operation:

1. Make sure the source clip used is at the last frame.
2. Move the selected clip over the first frame that you want to replace in the target clip.

   The Replace Back cursor is displayed.
3. Release the clip.

   The current frame in the source clip is matched to the frame on which you dropped the source clip and the frames before the selected frame in the target clip are replaced with the frames of the source clip, including the frame on which the source clip was dropped.

Replacing Clips Gesturally

You can replace frames in a clip with an entire source clip. There are two ways to replace clips gesturally:

- **Replace Ahead** will replace the frames after the selected frame in the target clip (including the frame on which the source clip was dropped).
- **Replace Back** will replace all the frames before the selected frame in the target clip (including the frame on which the source clip was dropped).

To replace frames with a clip using the Replace Ahead operation:

1. Make sure the source clip used is at frame 1.
2. Move the selected clip over the first frame that you want to replace in the target clip.

   The Replace Ahead cursor is displayed.
3. Release the clip.

   The source clip replaces the frames after the selected frame in the target clip, including the frame on which the source clip was dropped.

To replace frames with a clip using the Replace Back operation:

1. Make sure the source clip used is at the last frame.
2. Move the selected clip over the first frame that you want to replace in the target clip.

   The Replace Back cursor is displayed.
3. Release the clip.

   The source clip replaces the frames before the selected frame in the target clip, including the frame on which the source clip was dropped.

Matching Clips with Their Sources

You can match a clip on the Desktop or timeline with its source clip. There are numerous options to match clips with their sources, depending on the result you are trying to achieve. You can select any or all of the following from the contextual Match options on a desktop or timeline clip:

- **Preserve Handles:** The Head / Tail of the timeline segment is preserved.
- **Use Sequence Info:** A new source with the timeline segment info is created.
- **Include Nested Segment:** The match is performed on the result BFX clip, or on the result clip and all the sources included in it.
- **Include Timeline FX:** The Timeline FX present on a clip are copied to the new Matched clip.

Once you have selected the match criteria from the list above, you can select Match or Match All.

The matched source is selected in the Media panel, and the cursor becomes a white arrow, prompting you to select a destination reel on which to display the source.

Swapping Segments

You can swap a segment within a sequence with a source clip, directly from the Desktop, while preserving any Timeline FX applied to the original segment. When you swap segments, you select a frame in the target clip and a frame in the source clip. When the swap operation is performed, the selected frame from the source clip takes the place of the selected frame in the target segment and the remaining frames are aligned relative to this position.

To swap segments:

1 In the target clip, select a frame in the segment to be swapped.

NOTE If you are in collapsed view, the soft positioner at the bottom of the clip determines the selected frame. If you are in storyboard view, the first frame of the segment is selected by default.

2 From the contextual menu, select Swap Segment.

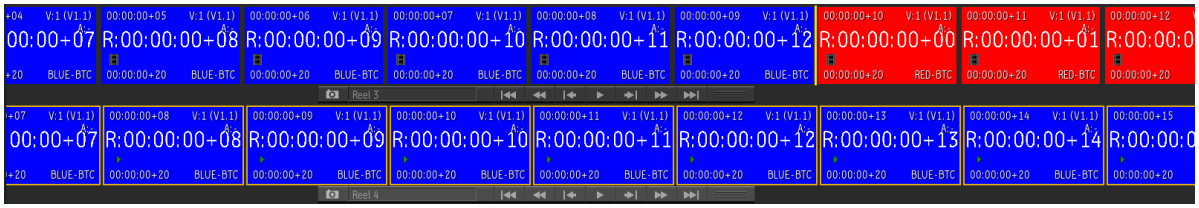
NOTE If you are swapping a shot in a dissolve, from the contextual menu, select **Swap Segment ➤ Incoming** or **Swap Segment ➤ Outgoing** to determine whether you are swapping the outgoing or incoming segment.

3 Click a frame in the source clip.

The selected segment in the target clip is replaced by the source clip and the selected frame from the source clip takes the place of the selected frame in the target segment and the remaining frames are aligned relative to this position. The selected frames are aligned in the swap operation.

NOTE If there are not enough frames before or after the align frame in the source clip to fit before or after the align frame in the selected shot an error message appears in the message bar.

Two clips on the Desktop prior to the Swap Segment operation. In this example, the operation will swap the green segment for the blue clip while aligning frame 8 from the green clip to frame 11 from the blue clip.



After the Swap Segment operation, the green segment has been swapped with the blue clip, while aligning frame 8 from the green clip and frame 11 from the blue clip.

Hard Committing from the Desktop

You can hard commit sequences and segments from the Desktop. When you commit a sequence or segment, the soft properties of the sequence such as cuts, transitions, head and tail frames, and timewarp data are removed. The visual effects of the transitions and timewarps remain but you cannot edit them.

To hard commit a segment:

1. Make sure the clip is at a frame within the segment to be committed.
2. From the contextual menu, select **Hard Commit ➤ Segment**.
   The segment is committed.

To hard commit a sequence:

1. Select a sequence on a Desktop reel.
2. From the contextual menu, select **Hard Commit ➤ Sequence**.
   The sequence is committed.

NOTE You can hard commit a sequence via Burn, by selecting **Hard Commit ➤ Burn**, from the contextual menu.

Rendering from the Desktop

You can render segments or sequences directly from the Desktop.

To render a segment:

1. Make sure the clip is at a frame within the segment to be rendered.
2. From the contextual menu, select **Render ➤ Segment**.
   The segment is rendered.

To render a sequence:

1. Select a sequence on a Desktop reel.
2. From the contextual menu, select **Render ➤ Sequence**.
   The sequence is rendered.

NOTE You can render a sequence via Burn, by selecting **Render ➤ Burn**, from the contextual menu.

Audio 12

You can perform many of the same editing operations on audio that you use to edit video, such as cutting, trimming, adding fades, and creating stereo tracks. A comprehensive set of audio effects tools is available in the timeline and Audio Desk for mixing and mastering your audio.

When working with audio you can:

- In the MediaHub, import and export audio files using a variety of formats.
- In the timeline, synchronize audio with video, and apply audio fades. You can edit these on the fly while the clip is playing. You can also assign input strips to output strips. You can also assign audio tracks to Audio Desk input strips.
- In the Audio Desk, adjust Gain, Pan, Mute, Phase Shift, and Solo settings for up to 32 individual input strips. You can apply these on the fly while the clip is playing. You can also assign input strips to output strips.
- In the EQ Desk, apply and adjust High Shelf, Mid Notch, Mid Presence, and Low Shelf filters to input strips.
- In the Auxiliary Effects Desk, adjust global settings for the Modulation, Delay, and Reverb Auxiliary Effects.
- In the Audio Desk, adjust the output Gain and Limiting for output strips.

NOTE You can only see the Audio Desk in the single player.

Accessing the Audio Desk

The Audio Desk components are split into two panels; the patch panel will display to the left of the viewer, while the EQ Desk and the Auxiliary Effects Desk will display to the right of the viewer.

NOTE The Audio Desk is accessible from the Player.

To access the Audio Desk:

1. From the Player view, select the Options drop down.
2. Click Show Audio Desk.
   The Audio Desk panels are split to either side of the viewer.
3. Click the Audio Desk Panel button to switch between the Auxiliary Effects Desk and the Eq Desk panels.

Importing Audio Files

To import an audio file:

1. Click the MediaHub tab.
2. Enable Browse for: Files.
3. Browse to the location of the audio file.
   The audio file thumbnail appears in the browser display.
4. Drag and drop the audio file from the browser into a media panel folder.

Monitoring Audio in Batch orBatch FX

You can select different audio sources to monitor in **Prefs ➤ Preferences ➤ Audio Monitoring**, you can choose your audio source from the Audio Source box.

Options are:

- **No Audio**: No audio is monitored.
- **Main Level**: Monitors the audio of the Timeline containing the Batch FX. This option is only available in Batch FX, not Batch.
- **Context 2**: Monitors the audio of the Batch orBatch FX clip set to Context 2 in the flow graph.
- **Context 1**: Monitors the audio of the Batch orBatch FX clip set to Context 1 in the flow graph.
- **Current**: Monitors the audio of the current selection in the flow graph.

Once you've selected an audio source, there are several ways you can monitor it.

- You can scrub your audio tracks from the Batch orBatch FX Timebar:
  - To analog scrub your audio, hold down `Ctrl+Shift` and scrub the Timebar.
  - To digital scrub your audio, hold down `Ctrl` and scrub the Timebar.
- From the Animation menu, under the Audio tab, you can playback your audio in real time, using the playback controls. See also Generating Keyframes Based on Audio Analysis (page 1705).
- Within Action, you can select a clip from the Action schematic and press F2 to playback the audio of the selected clip in real time, using the playback controls above the Timebar.

Adding an Audio Effect on a Segment

To add an audio effect on a segment:

1. Select the audio segment.
2. Right-click on the segment and select Add Effects.
3. Enable one of the audio effects.
4. Adjust the parameters for the audio effect on the Effects Ribbon.

Adjusting the Audio Levels on Part of an Audio Segment

To adjust the audio level on part of an audio segment:

1. Select an audio segment.
2. Set the positioner to the relevant part of the segment.
3. Right-click on the segment and select Add Effect.
   The Effects Ribbon for audio effects is displayed.
4. Select Gain Audio effects.
   The toolbar displays the Gain features.
5. Enable Edit and Auto Key.
6. Right-click on the Gain slider and set Key frame.
7. Move the positioner down the sequence.
8. Adjust the Gain slider.
   Another key frame is automatically created at the gain level selected.
   The audio level is adjusted to the new gain.

You can click and drag the keyframes in the segment to adjust the audio level. Holding Shift + dragging constrains the adjustment vertically or horizontally. Setting the gain slider to zero brings the audio level back to the segment default.

Cross-fading Two Audio Segments Together

To cross-fade two audio segments together:

1. Move the vertical focus point inline with the relevant audio track.
2. Navigate to the cut point of the two audio segments with the positioner.
3. Click the Transition button.
4. In the Audio Transitions tab, enable Fade.
   An audio transition is applied to the cut point of the two audio segments.
5. Adjust the length of the fade.
   The audio cross-fades between the two audio segments.

Fading Audio Into or Out of a Clip

To fade audio into or out of a clip:

1. Move the vertical focus point inline with the relevant audio track.
2. Navigate to the beginning (or end) of the segment.
3. Click the Transition button.
4. In the Audio Transitions tab, enable Fade.
   An audio transition is applied to the audio segment.
5. Adjust the length of the fade.
   The audio is now fading into (or out of) the clip.

Scaling the Size of the Waveform

To scale the size of the waveform:

1. Do one of the following:
   - Drag the audio track to increase or decrease the track size, or
   - From the Audio pull down menu in the Timeline Menu Bar, choose Increase Wave Form (or to decrease it, Decrease Wave Form).

Displaying Audio Waveforms

To turn audio waveforms on:

1. From the Timeline Menu Bar at the bottom of the screen, select Audio > Show Waveforms With Effects.
   The Waveform display for all audio segments is turned on.

To turn audio waveforms off:

1. From the Timeline Menu Bar, select Audio > Hide Waveforms.
   The Waveform display for all audio segments is turned off.

To show audio waveforms without effects:

1. From the Timeline Menu Bar, select Audio > Show Waveforms Without Effects.
   All Waveforms from the source are displayed and any audio timeline effects modifications are not reflected in the Waveforms.

To refresh the audio waveforms:

1. From the contextual menu, select Refresh Audio.
   The audio cache of the selected media and its waveforms are flushed and automatically regenerated.

NOTE Unlinking media or flushing local media of a given clip achieves the same result.

Muting Audio Tracks

To mute an audio track in the timeline:

1. Locate the track you want to mute.
2. Click the speaker icon in the track identifier strip.
   The speaker icon displays as a crossed-out icon to indicate the track is muted.

To mute an input strip:

1. In the Audio Desk, enable the Mute button for the input strip that you want to mute.
   The strip is muted. In the case of a stereo audio track, both strips are muted.

Mapping Audio Tracks to Audio Desk Input Strips

In the patch panel each audio track has a small square box with a number. That number is the audio input that is matched to the numbers at the bottom of each audio strip at the bottom of the audio desk.

To map audio tracks:

1. Click on the audio input channel box in the patch panel.
2. Drag right or left and choose the audio input strip into the audio desk you would like this particular audio track to be patched too.

   An audio track input is selected.

If the input channel box is empty, the audio track is not patched to an audio desk input. You will not hear audio segments for that track.

Normally all these mappings are automated. It is important to understand these outputs are either the audio channel outputs for your hardware or discrete channel outputs for files exported to the application.

Converting Audio from Stereo to Mono or Mono to Stereo

To convert a stereo audio track to two mono audio tracks:

1. Select the stereo track by clicking on the audio channel strip.
2. From the Timeline Gear menu, select **Stereo ➤ Split Stereo Track**.

To convert mono audio to stereo audio:

1. Select two mono tracks by control-clicking on the audio channel strips.
2. From the Timeline Gear menu, select **Stereo ➤ Merge Into Stereo Track**.

Rendering

13

When working in Batch or Batch FX and the timeline, you can get instant feedback when viewing and playing back clips without the need to render. To output your clips, however, you do need to render. In many areas of Flame, rendering is straightforward by using the Render box, or contextual rendering options. In most cases, rendering clips and sequences from the Workspace only renders the required effects and transitions (and not all effects and transitions located on multiple video tracks).

See the following sections for some tips on rendering in Flame, and follow the links in the Related Information section for specific rendering-related topics.

Static Frame Pipeline Optimization

Rendering media can be time-consuming, and depends on a number of factors, including the type of media and the effects applied to it. Static frame pipeline optimization can significantly decrease rendering times for sequences or Batch or Batch FX schematics that use static source generators (such as the Coloured Frame Node, for example), static media (stills), and static effects. A static effect is one that is not animated over time, such as static Text or a static GMask. When rendering static media, the result of a static source generator or a static effect, Flame renders and caches a single static frame, and uses that frame for the duration of the static element, rather than rendering each individual frame. This enables noticeably shorter rendering times.

Flame Reactor

Flame Reactor is the name given to the hybrid CPU/GPU processing pipeline in Batch and Batch FX. Batch and Batch FX have been optimized to make better use of GPU and CPU rendering capabilities in this pipeline.

Some Flame Reactor notes:

- Most Batch and Batch FX nodes work in 8-bit or 16-bit floating point. 10-bit and 12-bit media are automatically converted to 16-bit float; exceptions being I/O nodes (Clip, Gateway Import, Render, etc.), as well as nodes that are meant to handle bit-depth operations (LUT Editor, Resize, etc.). These nodes support 10-bit and 12-bit media.
- When working with content created in previous releases, final renders may look different.
- The Flame Reactor option may not be available on some hardware configurations (such as configurations using NVidia Quadro FX4500 and FX5500 graphics cards).
- It is recommended that you use Burn nodes with GPU processing capabilities when sending remote rendering jobs from a project that has the Render option set to Flame Reactor, to avoid discrepancies.

NOTE The Classic Engine is now retired, after years of honorable service. Starting with version 2017 Extension 1, every project uses Flame Reactor. Older projects which were set up to use the Classic Engine are automatically converted to Flame Reactor when opened in a recent version of the application. Due to differences in rendering engines (Classic was CPU-only), there might be differences between what was rendered then and what is rendered now, and adaptors might be added to a restored Batch Schematic to convert it to the supported 16-bit float bit depth.

Burn

Burn allows you to render images in the background using low-cost Linux workstations and frees your workstation for more creative tasks. With Burn, facilities can leverage a larger pool of rendering power so that complex visual effects requiring intensive processing can be created more quickly. See the Rendering on Networked Hardware section of the Installation and Configuration Guide for more information of how to install, license, and configure Burn to work with Flame.

Background Reactor

Adding a second GPU on HP Z800, Z820, and Z840 Linux workstations (running Flame Premium, Flame, or Flare software) allows you to use Background Reactor, a background rendering service. Rendering options for Media Library, Timeline, BFX, Batch, and Tools are available to select between foreground or Background Reactor rendering. Using Background Reactor allows you to continue working while effects are being generated. See the Rendering on Networked Hardware section of the Installation and Configuration Guide for more information of how to install, license, and configure Background Reactor.

About Automatically Disabled HWAA

In order to increase performances when in interactive mode, the hardware anti-aliasing (HWAA) level in *interactive* mode gets automatically disabled whenever the graphics card doesn't have enough memory. This can happen in Text, GMask, and Action modules.

Whenever the graphics card has to swap some of the textures to disk in order to allocate memory for a large HWAA buffer, the responsiveness of the system is affected. To increase the reactivity of the interactive mode, Flame disables the HWAA whenever this happens. The availability of HWAA in interactive mode depends two factors: the amount of graphics memory available when the HWAA level is set, and on the resolution of the monitor where the user interface is displayed.

IMPORTANT The HWAA that you set is always applied in Preview and when rendering; it is only disabled to improve the responsiveness of the user interface.

Rendering Preferences

In the Preferences menu, there are a few options to help you set default rendering settings. See **Preferences ➤ General ➤ Default Rendering Options** and **Preferences ➤ Timeline ➤ Rendering**.

NOTE Rendering differences may occur when using different generations of graphics hardware on different systems.

Timeline FX and Transitions

14

Timeline FX are effects you add directly to clips on the timeline, and timeline transitions are effects you apply directly to the cuts between clips. As such, Timeline FX and transitions accelerate the process of creating and modifying effects because you don't have to enter the editor to apply them. Some adjustments can be applied using a selected effect's quick menu.

There are separate video and audio effects, depending on what type of clip you want to apply effects to. Timeline FX can be accessed by selecting a Timeline FX in the timeline, the workspace or the Timeline FX pipeline. In order to access the timeline transitions, you must explicitly select a cut or transition in the timeline or in the Timeline FX pipeline.

The FX pipeline and quick menu appear above the timeline:

TIP If you need extra space, you can temporarily hide the FX pipeline and quick menu by clicking the FX Pipeline Display button at the top right of the timeline area (or use the shortcut `Spacebar+Tab`).

Working with Timeline FX

Adding Timeline FX

To add Timeline FX to a clip:

1. Select a clip from the timeline.
2. Do one of the following:
   - Click the FX button.
   - Right-click on a segment in the timeline and choose Add Effect.

   The Effects ribbon appears.

   NOTE Effects that are enabled have already been added to the current Timeline FX pipeline.
3. Select the effect you want to apply.

   The effect appears in the Timeline FX pipeline and on the timeline segment.

   NOTE You may have to resize the video or audio track so that Timeline FX are visible on the timeline.

4 You can adjust the effect's settings using the quick menu underneath the Timeline FX pipeline. Click the Editor button if you want to enter the full editor of the effect. Once in a Timeline FX editor, you can choose which range to display on the timebar (Segment, Media, or Timeline) from the Playback Options box. The Playback Options box also has options to display audio waveforms and cue marks.

When adding Timeline FX to stereo clips on the timeline, in some cases (such as with the Colour Warper effect), you are not able to use the quick menu to adjust settings. In this case, simply click the Editor button to enter the full editor.

Reordering Timeline FX

You can reorder most applied Timeline FX directly from the Timeline FX pipeline.

NOTE In the Timeline FX pipeline, a BFX clip or pre-processing options always appear at the beginning of the pipeline, and the Comp effect always appear at the end of the ribbon. Therefore, these effects can not be reordered.

1 From the Timeline FX pipeline with multiple Timeline FX applied, select an effect icon (for example, a Flip effect that occurs after a 2D Transform effect).
2 Click and drag the effect to its new location.
3 Release the effect.

Copying Timeline FX

You can copy Timeline FX from the timeline to the Workspace, from the Workspace back to the timeline, or from one segment on the timeline to another. Additionally, you can copy multiple Timeline FX simultaneously, as long as the Timeline FX you want to copy are from the same segment. You can also copy to and from the Media Panel or the Desktop reels.

A copy can be made gesturally or using contextual menus. Contextual menu operations can simplify copying operations when segments are far apart or in a different timeline, or if you plan to do other editing operations between copying and pasting.)

When you copy Timeline FX of the same type over another Timeline FX, the copied effect overwrites the pre-existing one.

NOTE You may have to resize the video or audio track vertically, so that Timeline FX are visible on the timeline.

To copy Timeline FX to the workspace:

1 Do one of the following:
- Select a Timeline FX in the Timeline FX pipeline.
- Select a Timeline FX in a segment on the timeline.
- Select a Timeline FX in the Media Panel.
- Select a Timeline FX in the Desktop reels.

2 Do one of the following:
- Drag the Timeline FX to the workspace. If you want to copy multiple Timeline FX, `Ctrl`-click the ones you wish to copy and drag them to the workspace.
- In the Timeline FX pipeline, Media Panel or Desktop reels, right-click the Timeline FX, and select Copy. In the workspace, right-click and select Paste.

The Timeline FX appears in the workspace. It displays a thumbnail with the front view of the effect, and the shortened name of the effect(s) you copied.

To copy Timeline FX to the timeline:

1. Do one of the following:
   - Select a Timeline FX in the workspace.
   - Select a Timeline FX in the Timeline FX pipeline.
   - Select a Timeline FX in a segment on the timeline.
   - Select a Timeline FX in the Media Panel.
   - Select a Timeline FX in a Desktop reel.
2. Do one of the following:
   - Drag the Timeline FX to the Timeline FX pipeline.
   - Drag the Timeline FX to a segment on the timeline.

The Timeline FX appears in the Timeline FX pipeline and in the timeline segment.

To copy Timeline FX from one segment in the timeline to another:

1. Do one of the following:
   - Select a Timeline FX in the Timeline FX pipeline.
   - Select a Timeline FX in a segment on the timeline.
2. Do one of the following:
   - Drag the Timeline FX to a different segment on the timeline.
   - Right-click the Timeline FX in the Timeline FX pipeline, and select Copy. In the the current timeline or another timeline, right-click a segment and select Paste.

To copy Timeline FX to the workspace:

1. Do one of the following:
   - Select a Timeline FX in the Timeline FX pipeline.
   - Select a Timeline FX in a segment on the timeline.
   - Select a Timeline FX in the Media Panel.
   - Select a Timeline FX in a Desktop reel.
2. Do one of the following:
   - Drag the Timeline FX to the workspace. If you want to copy multiple Timeline FX, `Ctrl`-click the ones you wish to copy and drag them to the workspace.
   - In the Timeline FX pipeline, Media Panel, right-click the Timeline FX, and select Copy. In the workspace, right-click and select Paste.

To copy Timeline FX to the Desktop reels:

1. Do one of the following:
   - Select a Timeline FX in the workspace.
   - Select a Timeline FX in the Timeline FX pipeline.
   - Select a Timeline FX in a segment on the timeline.
   - Select a Timeline FX in the Media Panel.
   - Select a Timeline FX in a Desktop reel.

2 Do one of the following:
   - Drag the Timeline FX to the Desktop reel. If you want to copy multiple Timeline FX, `Ctrl`-click the ones you wish to copy and drag them to the workspace.
   - In the workspace, Timeline FX pipeline, or Media Panel, right-click the Timeline FX, and select Copy. Right-click a Desktop reel and select Paste.

Muting and Deleting Timeline FX

If you want to temporarily remove Timeline FX from a clip, you can mute it. You can quickly assess if a Timeline FX has been muted, because its LED is turned off.

To mute Timeline FX:

1 On the timeline, select the segment with the Timeline FX you want to mute.
2 In the Timeline FX pipeline, click the blue LED on the Timeline FX.
   The LED darkens to indicate that the Timeline FX is turned off.
3 To unmute an effect, click the LED on the Timeline FX in the Timeline FX pipeline. The LED will turn blue, to indicate an unmuted Timeline FX.

To delete Timeline FX:

1 On the timeline, select the segment with the Timeline FX you want to delete.
2 Do one of the following:
   - From the Timeline FX pipeline or the timeline segment, drag the Timeline FX to the bottom of the screen.
   - Right-click the Timeline FX and select Delete.

Modifying Multiple Timeline FX

You can modify Timeline FX from different clips at the same time by multi-selecting clip segments. Selecting multiple segments allows you to apply changes to effects in one segment, and have those changes applied to the same effects in the other segments.

To add multiple Timeline FX to a segment:

1 Select a clip from the timeline.
2 Click the FX ribbon.
3 `Ctrl`-click the effects you want to apply to the segment.
4 Do any of the following:
   - Click a final effect button to add it to the selection.
   - Click outside of the FX ribbon.
5 The FX ribbon closes. The selected effects are applied to the segment.

To copy and paste multiple Timeline FX from the timeline:

1 Do one of the following:
   - `Ctrl`-click to select Timeline FX in the Timeline FX pipeline.
   - `Ctrl`-click to select Timeline FX in the segment.
2 Do one of the following:
   - `Ctrl`-drag the selection to the Workspace.

- Ctrl-drag the selection to another segment in the timeline.
- Right-click the another segment and select Paste Effects.

To delete multiple effects:

1. Do one of the following:
   - Ctrl-click to select Timeline FX in the Timeline FX pipeline.
   - Ctrl-click to select Timeline FX in the segment.
2. Do one of the following:
   - From the Timeline FX pipeline or the timeline segment, Ctrl-drag the selection to the bottom of the screen.
   - Right-click the effects and select Delete.

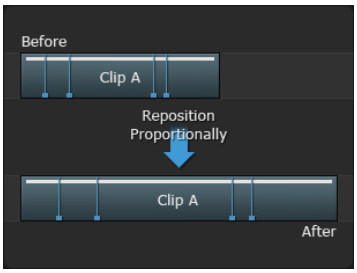
To modify a Timeline FX on multiple segments:

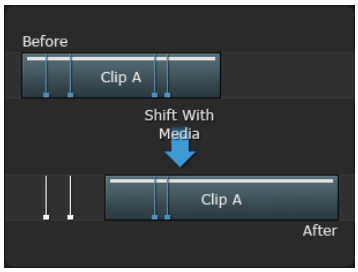
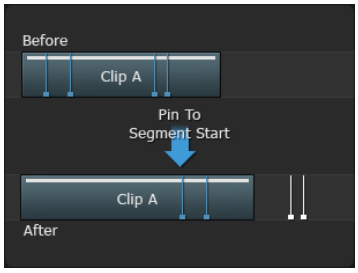
1. Ctrl-click to select multiple segments in the timeline.
2. Do one of the following:
   - Select the FX button. The FX ribbon appears and you can add one more effects.
   - Mute, unmute or delete Timeline FX in the Timeline FX pipeline. The effect state is updated in selected segments.
   - Right-click the segment and select Paste Effects to paste to all selected segments.

Sliding Timeline FX Keyframes

To slide Timeline FX within a clip:

1. In the Timeline FX pipeline or in the timeline segment, select the Timeline FX that you want to slide.
2. If the clip does not already contain animation keyframes, set keyframes on the clip to animate the FX.
3. Select how the animation channel is affected when you slide the animated effects by choosing an option from the Keyframe Move Modes box.

Select:	To:	Example
Reposition Proportionally	Resize the channel as you trim. The animation channel is scaled to fit into the timeline element. This option has no effect when you slip or slide.	

Select:	To:	Example
Shift With Media	Link the keyframes to their original frame numbers. The animation channel moves to follow the original frames as you trim.	
Pin To Segment Start	Unlink the keyframes from their original frame number. The animation channel remains with the timeline element as you trim.	

4 From the Editorial Mode box, select Slide Keyframes, which allows you to slide all of the keyframes of a specific Timeline FX on a clip.

NOTE You can only slide keyframes from one Timeline FX at a time.

Working with Timeline Transitions

Adding Timeline Transitions

NOTE Due to the nature of Matchbox on the timeline, adding and working with Matchbox transitions is slightly different than the procedure below. See Using Matchbox as a Transition (page 1363) for more information.

To add a transition to the timeline:

1 Select a cut or transition in the timeline to enable the Transition button.

A proxy of the last frame of the left-hand segment, and a proxy of the first frame of the right-hand segment appears in the Timeline FX pipeline, with a cut or existing transition between the segments.

2 Do one of the following:

- Click the Transition button to display the Transitions ribbon.
- Right-click the transition on the timeline or in the Timeline FX pipeline.

3 Select the transition that you want to apply.

4 You can adjust the transition's settings using the quick menu underneath the Timeline FX pipeline. Click the Editor button if you want to enter the full editor of the effect.

NOTE Only one timeline transition can be applied at a time. Each time you apply a new transition, the latest transition that you applied appears between the segments, and you are only able to modify the settings of the current transition.

Copying, Moving and Removing Timeline Transitions

A transition that is copied or moved over another transition overwrites the prior transition and replaces any prior settings.

To copy and paste a timeline transition:

1. Select a timeline transition.
2. Right-click the transition on the timeline or in the Timeline FX pipeline and select Copy or press `Ctrl+C`.
3. Select a cut or another transition in the timeline or in the Timeline FX pipeline and either right-click and select Paste or press `Ctrl+V`.

To gesturally move a transition:

1. Select a transition on the timeline.
2. Drag the timeline transition over another cut or transition and release the cursor.

To remove a transition:

1. Select a timeline transition on the timeline or in the Timeline FX pipeline.
2. Do one of the following:
   - Right-click the transition on the timeline or in the Timeline FX pipeline and select Delete.
   - `Alt+click` the selected transition.
   - Drag the transition to the bottom of the screen and release the cursor.

Working with Timeline FX and Transition Presets

A library of presets is available for certain Timeline FX and transitions. A preset can be applied as is to achieve a particular effect, or it can be tweaked as needed, also speeding up the process compared to customizing a Timeline FX from its default settings in the Editor.

All Timeline FX quick menus display a Presets box to the right of the Editor button. If the Presets button is enabled, then presets can be applied from the quick menu or in the Editor. If the Presets box is greyed out there are currently none available for that type of Timeline FX.

To load a preset to a Timeline FX:

1. Display a Timeline FX quick menu (for example, a GMask Timeline FX).
2. Do one of the following:
   - Click the Presets drop-down list to select a preset by a filename in the list.
   - Click the Presets button to open the file browser. Navigate to the preset and select it to return to the quick menu.

   Menu settings are changed to reflect the chosen preset.

Navigating Timeline FX and Transitions

If you have multiple Timeline FX or transitions in a segment, or use the same type of Timeline FX in a track, you can easily navigate them with keyboard shortcuts.

Navigating timeline transitions updates the display of the timeline and the Timeline FX pipeline with the current selection. Navigating Timeline FX will update the timeline, and Timeline FX, and the player. If you are in an effects editor, Timeline FX navigation shortcuts are still active and will update the image display window.

Timeline Navigation

Function	Flame Shortcut	Smoke Classic Shortcut
Go to Clip Start	Ctrl - Left Arrow	Ctrl - A
Go to Clip End	Ctrl - Right Arrow	Ctrl - S
Go to In Mark	Alt - [	A
Go to Out Mark	Alt -]	S
Next Cue Mark	Ctrl - \	Alt - S
Prev Cue Mark	Alt - \	Alt - A
Zoom In	Ctrl - Space-Up Arrow	Smoke + PAD+
Zoom Out	Ctrl - Space - Down Arrow	Smoke + PAD-
Fit Timeline to Contents	Ctrl - Home	=
Fit Timeline to Selection	Shift - =	Shift - =
Next Transition (Current Track)	Down Arrow	X
Next Transition (All Versions/Tracks)	Ctrl - Down Arrow	Shift - X
Previous Transition (Current Track)	Up Arrow	Z
Previous Transition (All Versions/Tracks)	Ctrl - Up Arrow	Shift - Z
Select Outgoing Side of a Transition	Win (Smoke) - Left Arrow	Space - B
Select Incoming Side of a Transition	Win (Smoke) - Right Arrow	Space - N
Select Both Sides of a Transition	Win (Smoke) - Down Arrow	Space - H
Next Timeline FX (Current Segment)	Shift - Ctrl - Right Arrow	Alt - Smoke - X

Function	Flame Shortcut	Smoke Classic Shortcut
Previous Timeline FX (Current Segment)	Shift - Ctrl - Left Arrow	Alt - Smoke - Z
Next Instance of Current Timeline FX (Current Track)	Shift - Down Arrow	Alt - Z
Previous Instance of Current Timeline FX(Current Track)	Shift - Up Arrow	Alt - X

Adding Timeline FX to Gaps

Gaps are empty spaces between elements in an edit sequence. When you apply Timeline FX to a gap, you create a gap effect.

The effects applied to gaps affect the media under them.

Gaps usually appear in the following places:

- The empty space between media elements on a video track.
- An empty video track.
- The space between cut points on an empty video track. Cut points are useful for containing a gap effect applied to an empty video track. The areas to either side of the cut points are also gaps.

Gap effects are not restricted by in points, out points, cuts and transitions, so they are easily trimmed, moved and duplicated. You can trim a gap effect over the entire duration of the video track regardless of the underlying cut points.

You can also freely edit cuts and transitions between elements on video tracks under the gap effect. You can cut an element, add a transition at the cut point, and then replace the incoming element without affecting the gap effect. In addition, gap effects can be copied and placed over different media in the timeline.

To add Timeline FX to a gap:

1. Do one of the following:
   - Move the positioner and its focus point over the gap.
   - Select a gap between two segments or two cuts.
   - Select a gap between a segment and a cut.
   - Select an empty video track.
2. Click the FX button to display the Effects ribbon.
3. Select the Timeline FX you wish to apply to the gap.
4. Edit the Timeline FX using the quick menu or click the Enter Editor button to access the full editor.

TIP You can also convert a gap into a colour source by clicking Colour Source in the quick menu when a gap is selected. Colour source settings appear in the quick menu. To remove a colour source and revert to a gap, click Remove Colour.

Working With Clip Format and Import Options on the Timeline

You can access format and import settings of an imported clip directly on the timeline. In the Editing panel, these options are represented in the Timeline FX pipeline, prior to the application of any Timeline FX. These options can be edited for the selected clip, and copied and pasted to other segments that use the same clip format.

For example, after importing R3D footage and adding it to the timeline, you can edit the colour curves of a clip in the Timeline FX pipeline, and also copy the option from the Timeline FX pipeline directly to other R3D clips that are used in the same timeline.

To access the Format and Pre-processing Options Editor:

1. Click the Timeline tab.
2. Select the clip in the timeline to display its Timeline FX pipeline.
3. Do one of the following:

   NOTE Format and import options are accessible in the same editor. You can open the editor once, then switch between menus in the editor to edit different option types.

   - Select the Format Options button in the Timeline FX pipeline, and click Editor in the quick menu. The Basic menu opens. Format options will be saved to this menu.
   - Select the Pre-Processing button in the Timeline FX pipeline, and click Editor in the quick menu. The Resize menu opens. Import options will be saved to this menu and the RGB LUT menu.

To copy format and import options to another segment on the timeline:

1. Select the clip you wish to copy in the timeline.
2. In the Timeline FX pipeline, do one of the following:
   - To copy basic clip format settings, Click and drag the Format Options button to another segment on the timeline.
   - To copy resize and RGB LUT settings, click and drag the Pre-processing button to another segment on the timeline.

2D Compositing on the Timeline

Use the Comp Timeline FX to control the composition of a segment with the input on the track beneath it. By default, the currently selected segment is used as the foreground of the composition, and the layer underneath is used as the background; however these inputs can be swapped. The type of composition is determined by the blending mode and the level of transparency of the foreground input.

By default, the Comp Timeline FX button is unmuted: compositing is active and, by default, the foreground and background mattes are on. When muted, only the RGB clip is displayed.

You can use Comp Timeline FX to control the composition and display of the output matte used in the segment's Timeline FX pipeline. In the editor, you can view the composited matte, and edit how it is blended and premultiplied.

TIP You can set the default premultiplication setting for the segment, background, and output of timeline Comp effects in the Timeline FX / BFX section of the Preference menu.

Adding Matte Containers

A matte container is a container with the RGB portion of an image on one track and its matte on another. If the matte for a clip exists separately from it, you can add the clip and its matte to the timeline as a single element by creating a matte container for them. You can also create matte containers on import, or from the Tools tab.

To create a matte container on the timeline:

1. In the timeline, select the clip you want to add the matte container to.
2. In the Timeline FX pipeline, select the proxy for the clip.

   Underneath the pipeline, a quick menu appears.
3. Click the Add Matte button.
4. Select a matte from the workspace. The Matte Container button appears in the Timeline FX pipeline.

   If the matte media length is equal to the front length, then the matte segment is positioned at the same start as the front segment. If the matte media length is different to the front length, then the matte segment is positioned at the Container segment start (yellow marks).
5. To open the Matte Container, click once on the Matte Container button and then click Open in the quick menu, or double-click on the Matte Container button.

   Once the Matte container is open, you can adjust the settings for the matte or add other effects to it.
6. If you want to mute the matte, right-click the Matte Container button and select Mute.
7. If you want to remove the matte, click once on the Matte Container button and then click on the Uncontain button.

NOTE If you use the matte container clip in Batch or Batch FX, the front and matte clips are present in the clip node.

Working with Mattes and Masks on the Timeline

Masks and mattes increase the versatility of the effects you apply on the timeline, allowing you to delimit the regions used for processing and effect, with a mask, and to create regions of total or partial transparency in the clip, with a matte.

Using Mattes in 2D Transform Timeline FX

In the timeline, a matte container on a track is stacked on a track with a clip of a cloud image.

The 2D Transform Timeline FX is applied, with the Matte Offset controls disabled. The front clip and matte transformations are linked.

The 2D Transform Timeline FX is applied, with the Matte Offset controls active. The matte position is offset independently of the RGB clip in the matte container.

The matte can be offset from its original position by changing its position along the X and Y axes, or by rotating or scaling it. Edit the offset in the 2D Transform quick menu or editor. In the 2D Transform quick menu, use the 2D Transform Tab selector to display the Matte Offset menu and its controls.

Using Mattes in Blur Timeline FX

In the timeline, The matte container of the talent's head is stacked on a cloud image background.

Image courtesy of The House

The Blur Timeline FX is applied, and the same blur is applied to the clip and the matte (Lock to Front is enabled).

The Blur Timeline FX is applied, and a blur is applied to the matte instead of the clip (Lock to Front is disabled).

The matte values can be blurred separately or with the same values as the front clip.

In the Blur quick menu or the editor, you can apply blur values to the matte. Click the Lock to Front button to use the same blur values that are applied to the front clip. In the Blur editor, you can also invert the matte or use the matte to apply blending in the regions defined by the matte.

Using Masks in Colour Correct and Colour Warper Timeline FX

The RGB clip will be added to a matte container with a matte surrounding the talent.

Image courtesy of Das Werk

A Colour Warper Timeline FX is applied to the matte container. By default, it creates a transparency.

When the matte clip is used as a mask, it defines where to apply the Colour Warper Timeline FX on the RGB clip.

You can change the default functionality of the matte clip and instead use it as a mask to isolate the area in which you want to add colour effects and corrections.

In the Colour Corrector or Colour Warper quick menu, enable the Use Mask button, which uses the matte to delimit the region that will be processed with the colour effects.

NOTE When Use Mask is on, all mattes generated before the Colour Correct or Colour Warper Timeline FX are ingested for use in these effects as masks, and matte functionality is no longer available for Timeline FX used after it in the pipeline.

Using Mattes in Comp Timeline FX

In a matte container containing a logo, The Comp Timeline FX is active by default. An image of a gladiator was stacked on a track underneath, so it is composited in the result.

Image courtesy of Buzz Image Group, Inc.

When the Comp Timeline FX is disabled or the matte is turned off, only the RGB clip is displayed.

All mattes used in the Timeline FX pipeline for the segment are composited using this effect. The composited mattes for each segment can be turned on or off, or be inverted.

A clip can be in a premultiplied or unpremultiplied state. An unpremultiplied clip has its pixels divided by the pixels in the matte. A premultiplied clip has its pixels multiplied by the pixels in the matte. Use the Comp Timeline FX quick menu or the editor to set the Segment and Background Premultiplication boxes to the appropriate states.

By default, a segment on a track is blended with the background, and the matte is turned on by default. In the Comp quick menu and editor, use the Matte box to also turn the matte off or invert it.

See 2D Compositing on the Timeline (page 484).

Using the GMask Timeline FX

On an RGB clip, using GMask Timeline FX will apply the matte to create a garbage mask that defines which areas of the image appear in the result clip. With a matte container, the matte clip and the GMask matte are composited in the result matte.

Use the GMask quick menu to select a shape from a library of presets, or enter the editor to create your own.

See Masking and Rotoscoping (page 1102).

Using Masks in Text Timeline FX

In RGB rendering mode, the Text Timeline FX is overlaid on the clip.

In RGBA rendering mode, only the text appears on the clip and the image is masked out. In this example, a cloud image was stacked on a track underneath, so it appears as a background.

In A Only rendering mode, the text appears with the RGB clip used as a fill. In this example, a cloud image was stacked on a track underneath, so it appears as a background.

There are three states in which you can manage text in Timeline FX, (two of which allow masking functionality).

In the Text Timeline FX quick menu, use the Alpha Rendering Mode box to select a state:

- RGB: If you are using an RGB clip (or matte container), to generate a clip (or RGB portion of the clip) with the text overlaid.
- RGBA: If you are using an RGBA clip/matte container, to generate a text overlay and use the text outline as a mask to display it on background tracks.
- A Only: To generate a source based on the text as a mask and segment as a fill. For best results, use unpremultiplied input. In the segment's Comp Timeline FX, verify that the Segment Premultiplication box is set to Unpremultiplied.

Compositing in 3D Space on the Timeline Using Action

Use Action as a Timeline FX to work in 3D space directly on the timeline. Think of Action as a super tool on the timeline, unlocking many tools in a 3D environment, allowing you to:

- Import Alembic or FBX geometries (as well as other 3D formats).
- Create 3D text that you can extrude and manipulate as a geometry.
- Add lights and shadows, as well as other lighting effects, such as lens flares and rays.
- Look at, and work with your scene, through an Action camera.
- Access the Modular Keyer, where you can key out green screen footage.

TIP While Action can perform simple operations, such as picture-in-picture, there are other Timeline FX tools that you can use that may be more lightweight, such as 2D Transform or Flip.

Using Action on the timeline offers you a way to create complex compositions as an alternative to working in Batch or Batch FX flow-graph representations. But this workflow allows you to start compositing on the timeline with Action, then promote and explode your comp in Batch or Batch FX to continue working.

Action is also available as a timeline transition, and has all the same options available as Action Timeline FX in the quick menu. In addition, you can access specific transition settings by choosing Transition from the Action Quick Menu selector. When you enter the full Action menu after applying an Action timeline transition, notice that the outgoing clip is listed in the Media list as the back media, while the incoming clip is listed as the front/matte media.

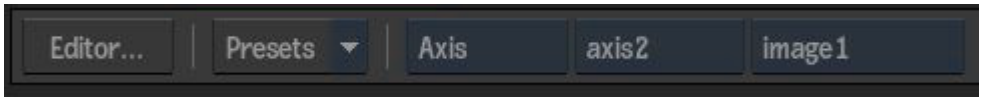
NOTE If you access Action as a Timeline FX or transition, you are limited to one front/matte media. Therefore, some Media menu settings and nodes are not available. For example, if you need to work with stereo clips, you should access Action from Batch or Batch FX, or from the Tools menu.

Using the Action Quick Menu

As with other Timeline FX, you can use the quick menu to apply settings without having to enter the full editor. Since Action is comprised of many nodes, each with their own menu settings, there are additional quick menu controls to switch between settings for nodes, while allowing you to still view parent/child relationships.

When applying an Action Timeline FX, a default composition is created with an axis, surface, shadow, light, and camera (you can delete any of these objects, except for the default camera). This allows you to start working directly from the quick menu (use the Action Menu Selection box to switch between Action menus). To access the full Action menu, click the Enter Editor button. Once you have added and connected more nodes to your Action scene, you can return to the timeline and still access settings for most nodes from the quick menu, if desired.

Use the three navigation boxes to select the settings that you want to see:



Menu Selection box Select which quick menu to display.

Object Selection box Select which object's menu to display. Objects appear in this list if there are more than one of the type selected in the Menu Selection box.

Parent or Child Selection box Select a parent or child menu to display. Objects appear in this list if there are any parents or children in the scene of the object selected in the Object Selection box.

For some menus, such as Lights, there is another sub menu box to further refine which settings are displayed in the quick menu.

NOTE When selecting multiple clips with Action Timeline FX applied, you can use the quick menu to apply settings to all the selected clips (except for camera settings).

Compositing Examples

Depending on the effect you are trying to achieve, you can decide how your Action Timeline FX is composited (either by the Comp node, or by Action itself). The following two general examples show how you can comp your effect in different ways.

Example 1: Comping with the Comp node.

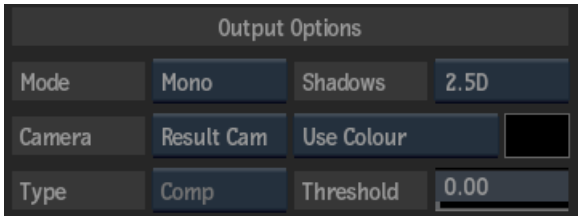
1. On the timeline, add an Action Timeline FX to a gap on a track above another clip.
2. In the FX ribbon, make sure that the Comp node is enabled.
3. In the Action quick menu, disable Use Back.

   TIP You can set the default behaviour of this button in **Preferences ➤ Timeline FX / BFX ➤ Timeline Action**.

4. Double-click the Action entry in the FX ribbon to enter the Action editor.
   You can see that the lower clip from the timeline is the front/matte and background in Action.
5. In Action, add a 3D Text node, change and animate the text, as needed.
6. In the View box, select **Context ➤ Primary Track** to see your effect in the context of the timeline.

   TIP You can also toggle the `F4` keyboard shortcut between the Comp output and Matte output.

Notice that in the Action Output menu's Comp box for the Comp output, Use Colour is selected, since you entered Action with Use Back disabled.



Since Action as a Timeline FX is limited to Comp and Matte outputs, in this case you are creating a fill and alpha to then allow the Comp node to blend the Action result with the background. When you return to the timeline, you can change any blending settings in the Comp node menu.

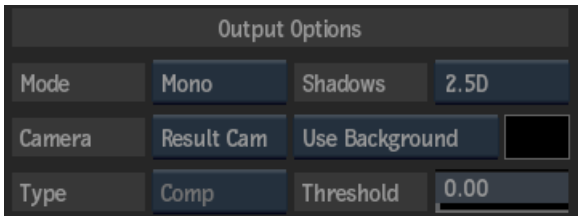
Example 2: Comping in Action.

1. On the timeline, add a green screen clip on a track above a moving background.
2. Add an Action Timeline FX to the green screen clip.
3. In the Action quick menu, enable Use Back, then enter the Action editor.

   TIP You can set the default behaviour of this button in **Preferences ➤ Timeline FX / BFX ➤ Timeline Action**.

4. Use the Modular Keyer within Action to key the green screen footage onto the background.
5. Add lights, lens flares, and any other Action nodes, as needed.

Notice that in the Action Output menu's Comp box for the Comp output, Use Background is selected, since you entered Action with Use Back enabled.



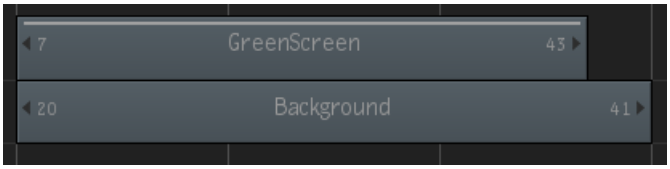
In this case, when you return to the timeline, you are sending the complete result, so there is no need to use the Comp node at the timeline level.

Keying on the Timeline

You can create chroma keys directly on the timeline using the Modular Keyer in Action. When you enter the Modular Keyer, the clip loaded as the Front and Matte (or Key In) is the clip with the Action Timeline FX, and the clip loaded as the Back is the next available track on the timeline.

To key on the timeline:

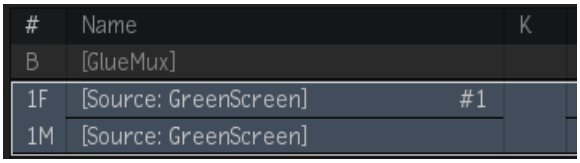
1. Build a multitrack timeline with a front layer and a background.



2. Select the track you want to key, and add an Action Timeline FX (in the example above, select the GreenScreen clip).

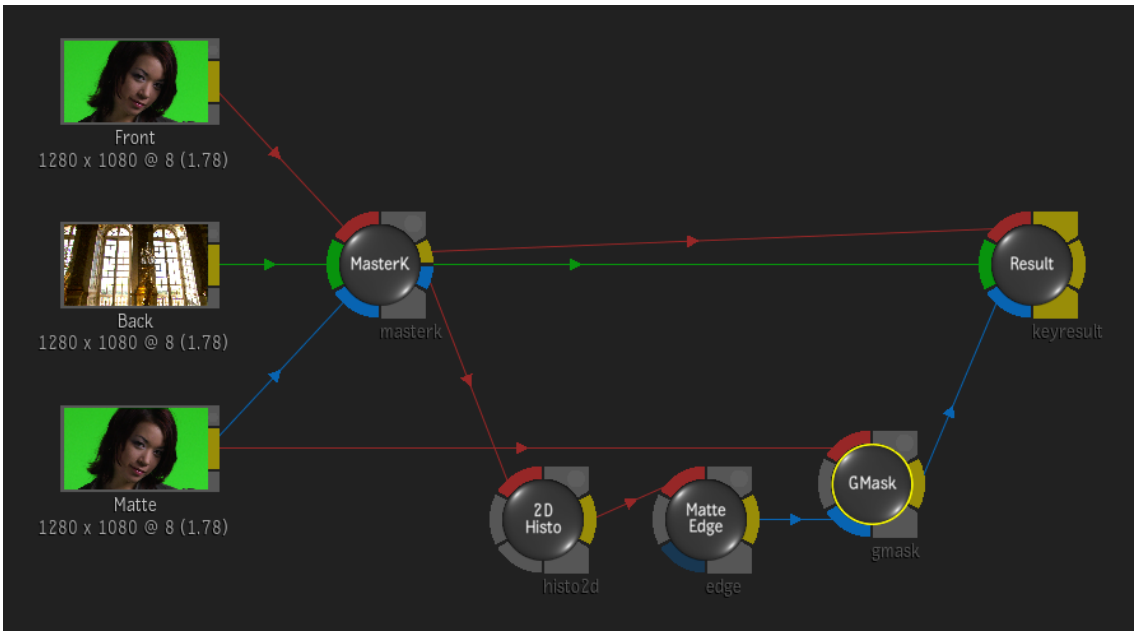
3 Double-click to Action entry in the FX ribbon, or click the Editor button to enter Action.

4 In Action, click the Media button to display the Media menu.



5 Double-click the K field in the front/matte row to enter the Modular Keyer.

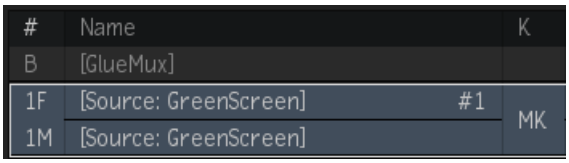
Use the desired Start Mode, and create your key. The following example uses the Master Keyer start mode:



TIP You can set up the viewports in 2-up mode using the Viewport Layout box. In this case, you can use one View box to set one viewport to the modular keyer schematic view, and the other viewport to Result or Action Context view.

6 When you are finished creating the key, click Return to return to Action.

Notice that the K field in the front/matte row of the Media menu now has an MK displayed.



7 Perform any other Action tasks, as needed, then click Exit to return to the timeline.

Creating New Sources Using Tools from the Tools Tab

15

The Tools tab provides access to numerous effects and tools allowing you to create modified clips that you can then use as sources on the timeline or in Batch or Batch FX. The tools use input clips (or in some cases are generated from scratch) that are then modified through parameters to produce a result directly in the workspace.

Some tools allow you to set parameters directly in the Tools quick menu, and instantly see a rendered result in the workspace. Other tools have full editors that you enter, allowing you to tweak numerous settings before rendering your result back to the workspace. In either case, new modified clips are created that you can use in your project.

Saving and Loading Tool Setups

For each tool that has a full editor, you are able to save and load setups. A setup is a file that contains a record of all changes you make to a clip in a particular tool. This record includes references to clips used—not the clips themselves. Setups let you save your work separately from clips, so you can load and work on the setup at anytime or apply the setup to other clips. Setups can be shared between instances of the same effect, whether accessed from the Tools tab, as a Timeline FX, or from the Batch or Batch FX node bin.

In the tool editor, use the Save and Load buttons to enter the file browser (pointing to the default directory location for each tool type). From the Load browser, you can also delete previously saved setups. For most tools, you can load or save setups or preferences/defaults. Preferences are settings that let you customize the display or functioning of some tool elements, keyboard shortcuts, pen and tablet, and audio (defaults are the default preferences). Some tools, such as Paint, have more saving and loading options.

Accessing Tools

To access specific tools

1. From the Tools tab, find and click the button for the tool you want to use.
2. If needed, use the Input Mode box to select the number of source clips you want to modify (for example, front only or combinations of front, back, and matte).
3. Set any tool parameters, as needed.
4. In the workspace, select the source clip(s) needed.

   The colour and text of the cursor indicates which clip you should select next. After selecting the source clips, the cursor changes to white and the text Render Here appears.
5. Select the area to place your modified clip.

If the result requires no other settings (other than tool parameters), then the rendered result appears immediately. If more settings are required, a full editor appears.

TIP Some tools, such as Paint, allow you to enter the full editor without inputs. In this case, you can double-click the tool name, and directly apply the Render Here location.

To access a tool with the same clips as previously used:

1. From the Tools tab, find and click the button for the tool you want to use.
2. Click the "S" button to the right of the tool name, or click Use Current Setup.
3. Select the area to place your modified clip.

   The result is rendered or the editor opens with the clips from the previous session. The most recent settings are also restored.

Procedural Compositing with Batch and Batch FX

16

Batch provides a flow graph environment for procedural compositing with integrated access to almost all Flame effects and image-processing commands. Click the Batch tab to enter the Batch view. Switch between tabs such as Timeline and Tools without losing any current work in Batch.

A Batch FX is a setup applied directly to one or more segments on the timeline. Creating a Batch FX takes a selection of timeline segments into a Batch flow graph environment for procedural compositing. A Batch FX node can be edited and reordered without affecting anything else in the pipeline.

Use the Batch processing environment to assemble a process tree of clips and nodes, where the result of each operation serves as the source for the next one. A Batch pipeline is never fixed, giving the flexibility to reorder and edit nodes as necessary to achieve an effect.

Use the bottom tabs to switch between the Desktop, Timeline, and Batch without losing work. Add clips to Batch from the Media panel or use the Read File node within Batch to bring in clips from anywhere on the network. Clips containing stereo tracks can be brought in as well as clips of any resolution and bit depth. Note that some nodes require all source clips to be of the same resolution and bit depth. Clip settings can be accessed directly from Batch.

- The default viewing layout for Batch and Batch FX is a 1-Up view displaying the schematic containing the clips in the schematic reel and the process tree. Use the Layout settings under the viewport to change the layout, or return to this default view. In Batch FX, the default layout can be changed to 2-Up in the Timeline FX / BFX section of the Preferences menu, with the schematic displayed on the left, and the result on the right.
- To change the mode of the selected viewport, using the View Mode box in the toolbar below. If the selected node has its own context-specific views (such as the Master Keyer and Action nodes), these options will also appear.
- The bottom portion of the Batch or Batch FX view displays the nodes that can be added to the process tree. This bottom area can also display the menu for the selected node in the schematic view. To switch between the node view and selected menu view, use the FX Nodes button located at the bottom left.

Accessing Batch

The Batch compositing environment can be quickly accessed at any time directly from the Editing panel.

To access Batch:

1. Click the Batch tab.

   The icon of the active Batch group is shown in yellow in the Media panel. If working multiple Batch groups, double-click the desired Batch icon in the Media panel.

2 Add clips and nodes to the Batch schematic in order to build process trees and create effects.

Switching between tabs such as to Timeline or Tools does not discard any work in Batch. Batch clips can be opened as timeline sequences by right-clicking the clip node in the Batch schematic and selecting Open As Sequence.

NOTE Switching between tabs such as to Timeline or Tools does not discard any work in Batch. Batch clips can be opened as timeline sequences by right-clicking the clip node in the Batch schematic and selecting Open As Sequence.

Imported media is stored in the Media panel in one or more of the following folders:

- Schematic Reels
- Batch Shelf
- Libraries

In each Batch, there are three Schematic reels, one Batch Shelf, and one Default Library created by default. This can be changed by editing the Default Reels Number fields in the User Interface Preferences (page 1947). A Batch Renders reel is also provided under Batch Shelf as the initial destination for any rendered clips.

TIP Use drag and drop to quickly move media: Drag media from anywhere in the Media panel or library to another folder to create a copy. Drag from the Media panel to the schematic to add a clip to the top reel. Drag a clip onto the Batch icon to create a new reel with the clip's name.

To clear a Batch setup:

1 Do one of the following:
   - From the Media panel, right-click the Batch group, and select Clear Batch.
   - From within Batch, select the "Clear Batch Setup" option, located in the "New" button's dropdown menu.

   All nodes and sources are cleared, and any unsaved changes to the current Batch setup are lost.

Creating Batch FX

To apply Batch FX on a timeline segment:

1 Select one or more video segments on the sequence on which to apply Batch FX.
2 Click the FX button.
3 Select one of the following options and click Create Batch FX to enter Batch FX view:

Select:	To:
No Option	Enter Batch FX with a single timeline segment. Any existing Timeline FX will be discarded.
Selection As Clip	Enter Batch FX with any existing Timeline FX converted into a single Batch FX clip. If required, click Explode FX in the Clip Settings menu to explode all Timeline FX in the clip into a process tree.
Selection As Flow-graph	Enter Batch FX with existing Timeline FX converted to a Batch FX pipeline. The Batch FX view recreates the timeline as a flowgraph with effects nodes representing each Timeline FX (and Comp node). In some cases, MUX nodes can be used to help connect multiple RGBA inputs to the outputs.

Select:	To:
Selection As 3D Comp	Enter Batch FX with existing Timeline FX converted into a 3D composition Batch FX pipeline with an Action node. The Batch FX view recreates the timeline as a flowgraph with effects nodes representing each Timeline FX, and outputting to an Action node. In some cases, MUX nodes can be used to help connect multiple RGBA inputs to the outputs.
Add Adjustment Segment	Create a gap on a track above the selected segment, and enter Batch FX with the segment clip designated as a Back clip. The adjustment segment uses the top media of the tracks below the timeline gap as the input. Inside Batch FX, the input is called the Back Clip. For example, if three tracks are composited together and an adjustment segment is added on top, the image available through the Back Clip is only the output of the third track.

After exiting Batch FX, click Enter Editor in the FX pipeline to re-enter it or alternatively double-click the BFX icon on the segment or pipeline.

Tips for working in Batch FX:

- The default naming convention for Batch FX is: <clip_name>_<segment_name>, which is equivalent to <batch_name>.
- When returning to the timeline, only the BFX icon appears on the modified clip, as any previously-created Timeline FX are now nested within the BFX.
- When working with multiple video sources in Batch FX, Batch FX, only one source will remain upon exiting, comprising the composite of all previous sources. To restore the previous vertical edit on the timeline, click Remove BFX and Recover Stack. The previous configuration of the video sources is restored; any pre-existing Timeline FX are lost.
- The Adjustment Segment option is useful for editing on the timeline in a gap above multiple clips, without losing any of their applied effects. After creating an adjustment segment, add new clips or edit existing clips on the timeline under the adjustment segment gap, and the Batch FX will be maintained above. Edits can be performed on an adjustment segment like any other timeline segment, such as trimming or even adding other Timeline FX.
- In addition when using the Adjustment Segment option, the Back Clip node is available in the Batch FX I/O bin. The Back Clip node does not contain any media; instead, it offers a link to the Timeline, giving access to the topmost media located below the BFX segment currently being worked on. Since the Back Clip is only a link to media, common clip settings within Batch FX cannot be accessed. The only settings available when double-clicking the Back Clip node is the media available for the head and tail of the adjustment segment.
- Create as many schematic reels as necessary to organize clips in Batch FX, however note that these reels are not shown Desktop Reels view.
- You can choose which range to display on the timebar (Segment, Media, or Timeline) from the Playback Options box. The Playback Options box also has options to display audio waveforms and cue marks.

About Recursive Batch FX

You can create recursive BFX (a BFX clip inside a BFX clip), and have the ability to spawn an infinite number of BFX levels. Using BFX levels, you can more easily manage your effects shots, by containing similar effects within a BFX level.

Creating a Nested BFX

To create a nested BFX:

1. In the BFX schematic, select the clip within which you want to embed a BFX level.
2. Select the Timeline tab.
3. Create a BFX on the Timeline segment.

- The new BFX is created and contained in the parent BFX.
- The number of BFX levels is displayed between brackets on the BFX tab.
- The BFX levels are dynamically displayed, from child to parent (from the bottom up), in the Media panel.

NOTE The Media panel always displays the BFX branch from the selected clip to its top-level parent. But clips on the same level, that aren't branched to the same parent BFX as the current selection, will not be shown.

Editing a Nested BFX

At any point, you can go into any BFX level and make changes, without disrupting the BFX structure.

In this example, we assume that we have four BFX levels and that we want to edit the BFX on level 4.

To edit a nested BFX:

1. Select the BFX clip from the Top BFX level.
2. Double-click the BFX clip.
3. Click the Edit BFX button. You are taken to BFX level 2.
4. Repeat these steps until you reach the desired BFX level (in our case, level 4).

TIP You can display your BFX clip in the BFX Timeline tab and make timing adjustments, add TimelineFX, etc.

Closing a BFX Level

You can close a BFX level at any point, by clicking the "X", to the right of the BFX level, in the Media panel.

About the Batch and Batch FX Process Tree

Use Batch or Batch FX to assemble a series of creative finishing tasks using nodes. Each node represents a specific Flame function. Connect clips with nodes and use the result of one node as the source for the next. The assembled result Batch or Batch FX in the schematic builds a process tree from left to right, processing as many output clips as needed.

A process tree begins with a clip, contains at least one node, and ends with a Render, Write File, or BFX Output node (a BFX tree invariably ends with a BFX Output node).

While working in Batch or Batch FX, the Timeline tab can be accessed to edit the clip and add Timeline FX.

Adding additional clips and nodes to achieve the desired result. As the schematic pipelines become larger, group nodes or add notes to nodes to manage any clutter.

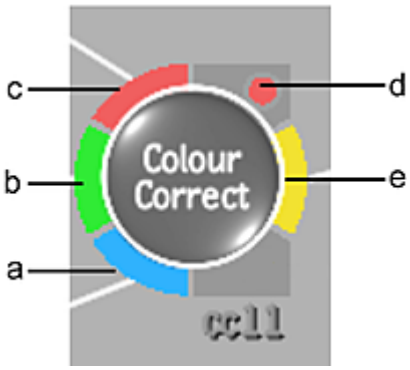
About Nodes

Nodes have one or more colour-coded input and output points (also called tabs) used for connecting to clips and to other nodes. All nodes have an output tab. For example, a Colour Correct node can accept a Front, Matte, and Back connection, whereas an Auto Matte node accepts a Front input.

When adding nodes, connect them to the process tree by linking the result from one node and using it as a source (front, matte, or back) for the next node in the process tree. Nodes can also be connected by linking backward from the source of one node to the output of another (to reuse a node's output).

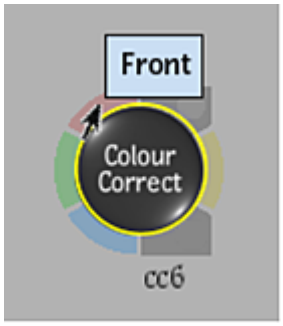
To connect nodes, use the coloured tabs on the node's left side known as *source tabs*. The colours of the source tabs represent different input types. The yellow tab on the right side of the node is called the Result tab (for some nodes, a blue tab is also present on the right side, this is the OutMatte tab). Use the Result tab of a node to connect its result to the input tab of another node.

This is a typical node tab configuration. Some nodes have specific tabs related to their function (such as the forward vector tab on the Pixel Spread node).



(a) Matte tab (b) Back tab (c) Front tab (d) Warning tab (e) Result tab

TIP If Auto Display of tooltips is enabled in the Preferences menu (**Preferences ➤ User Interface ➤ Tooltips**), hover over a tab to see the name of the tab.



Colour	Tab	Description
Red	Front	Connects a front clip to a node.
Green	Back	Connects a back clip to a node.
Blue	Matte	Connects a matte clip to a node.

Colour	Tab	Description
Light Blue	Misc	Miscellaneous tab specific to certain nodes, such as a Z-Depth or Forward Vector input.
Turquoise	Audio	Connects a clip with audio to a Render node.
Yellow	Result	Connects the result of a node to other nodes.
Blue	Output Matte	Connects the output matte of a node to other nodes.
Red circle	Warning	Warns that clip input to this node is unconnected or is missing media, or clips parented to this node do not share the same resolution or a compatible bit-depth. See Adding Clips to the Process Tree (page 513).

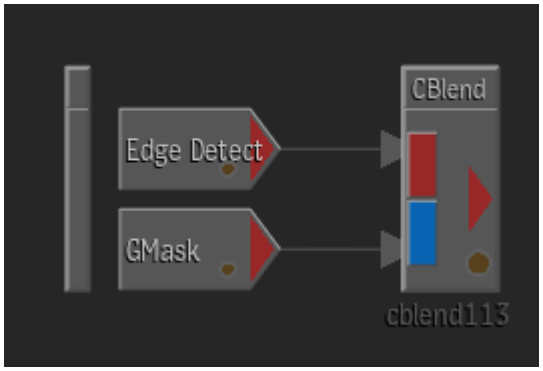
The available source tabs depend on the node. If the node accepts a front, back, and matte clip, all coloured source tabs are available. If the node only accepts a front clip, the red source tab is available and the other source tabs are grey. When a source tab or Result tab is not connected to a clip or to another node, the coloured tabs are dimmed.

TIP Right-click any node in the schematic to reveal a contextual menu of operations available for that type of node.

Blend Nodes

Colour Blend (CBlend) and Matte Blend (MBlend) nodes look and behave differently from other Batch or Batch FX nodes.

A blend node is essentially a container that you can use to create a "mini-composite" at different parts of the pipeline. CBlend and MBlend nodes each comprise one or more layers that include the components needed for a composite: a front, matte, and back component. These components are referred to as pipes, and they may contain as many nodes as needed to create the desired image.



NOTE When used in Batch or Batch FX, bypassing pipes is not supported.

Blend nodes are used extensively in the Modular Keyer. See Using Blend Nodes (page 1005) for more information.

Batch Setup Start Frame

You can modify the Start Frame of a Batch setup, using the numeric field located above the Current Frame field. Modifying the Start Frame automatically slips the existing keyframes and frames the Animation channel, by the desired amount.

You can toggle between the Duration and End Frame display from the Options drop down button. This setting is persistent in the User profile.

If appending a setup that starts at a different start frame than the current Batch Group, the application prompts you to select whether to align the appended setup to the current Batch Group start frame, to preserve the keyframe location of the appended setup or to define a custom location.

The Batch Start Frame is automatically set to the Background segment Source Start value when the Create Batch Group function is used from the Timeline / Conform.

The Batch Start Frame is automatically set when the Create Batch function is used in Lustre. If a segment has heads in Lustre and the Render > Local > Head&Tails option is disabled in Lustre, then the Batch Group will set the start frame to the head value instead of the media start.

It is possible to create a new Batch Group at the software launch using the following arguments:

- **--batchGroupName:** Lets you define the Batch Group name.
- **--batchStart:** Lets you define the Batch start frame.
- **--batchEnd:** Lets you define the Batch end frame.
- **--batchDuration:** Lets you define the Batch setup duration.

NOTE --batchDuration has precedence over --batchEnd if both are defined (They should not be).

Adding and Connecting Nodes to the Process Tree

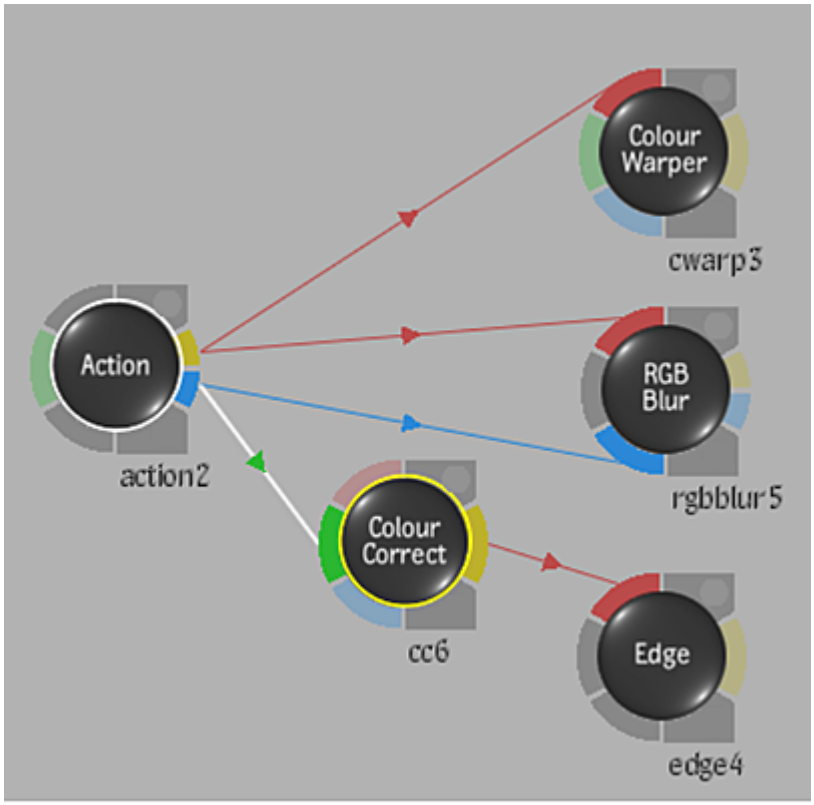
To add a node from the node bin to the schematic, either double-click a node or drag it directly to the schematic. There are several ways of connecting clips or nodes together, either manually or automatically.

TIP When adding nodes to the schematic in Batch or Batch FX, the resolution of a clip can be applied to a specific node using a keyboard shortcut. To do this, select the node to resize and `T+Click` the clip whose resolution is to be applied. With Render nodes, the Tape/Reel/Source name is also copied.

To insert a node between connected nodes:

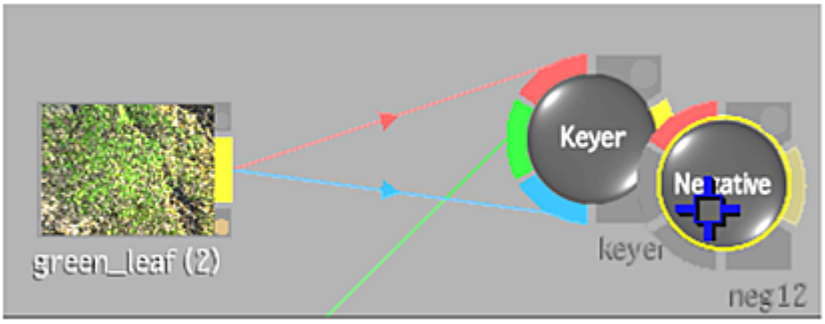
1. Do one of the following:
   - If Auto Insert is enabled in the Prefs menu Schematic settings, drag a node and navigate to the link between two connected nodes. The link is highlighted in orange.
   - If Auto Insert is disabled in the Prefs menu Schematic settings, press and hold `Shift`, then drag a node and navigate to the link between two connected nodes. The link is highlighted in orange.
2. Release the node.

The node is inserted, while retaining the input and output connections.

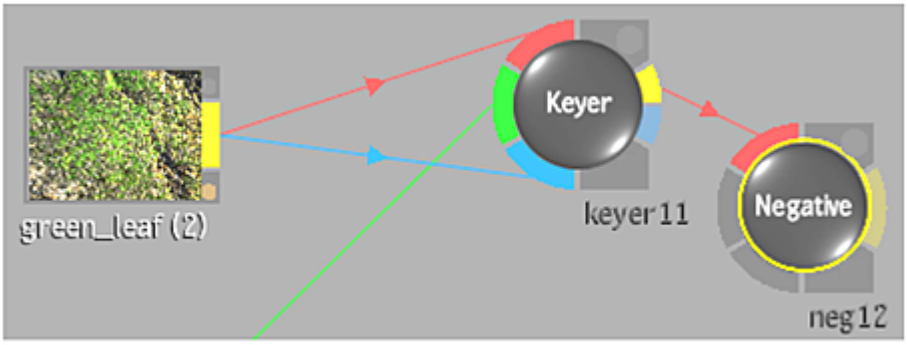


To connect nodes automatically with autolink:

1 Use the Autolink feature by pressing `Shift` and dragging a node to another node so their tabs touch.
Action: Press and hold Shift while dragging the Negative node to the Keyer node



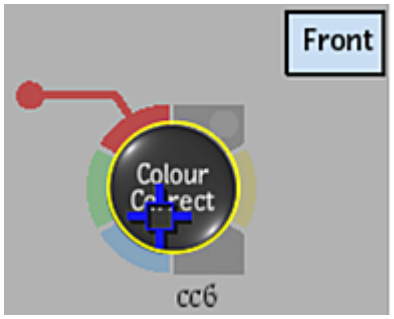
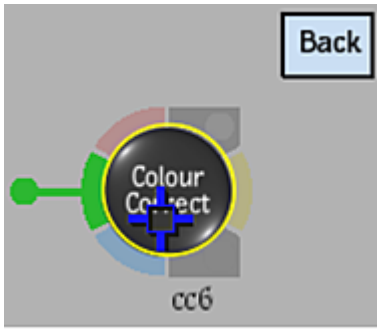
Result: The Keyer output is the front for the Negative node



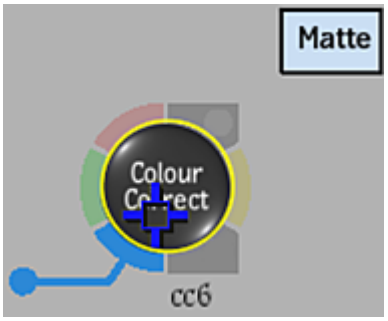
To connect nodes using the advanced autolink:

1 To distinguish between the various source tabs, use the Advanced Autolink feature by pressing `Alt` repeatedly while still holding `Shift` to extend a similarly coloured arm from each source tab (starting with the topmost source tab, and cycling counter-clockwise with each press of `Alt`). The name of the tab is also displayed above the node. Touch the extended arm to the tab to be connected.

Approaching a node with multiple outputs, such as a multi-channel clip imported with a Read File node, automatically expands that node.

Hold `Shift` and press `Alt` to extend the Front tab.	
While still holding `Shift`, press `Alt` again to extend the Back tab.	

While still holding `Shift`, press `Alt` a third time to extend the Matte tab.



Any connections made using this method will be retained by releasing the node over an existing connection as it turns orange.

To create a node connection using tap-tap linking:

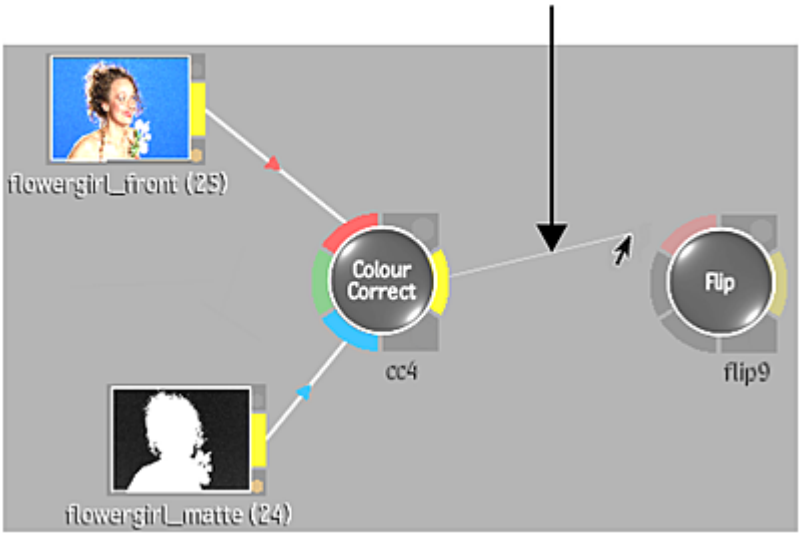
1. Click the tab of the first node or clip to be connected.
2. Click the tab of the node to be connected to the first node.

   The two nodes are connected.

To connect nodes manually:

1. Click the Result tab of a node whose result to be used and drag the cursor to one of the source tabs of the next node in the process tree.

   An arrowed line is drawn from the node to the source tab. For example, click the Result tab of the Colour Correct node and drag the cursor to the front tab of the Flip node to flip the result of the colour-corrected clip.



TIP When working with a large Batch or Batch FX flow graph, you can switch to a 2-Up view with the schematic displayed in both viewports, then drag and connect nodes from one viewport to the other.

To extract a linked node while maintaining connections:

1. Press `Alt+Win` and drag a linked node away from the link.

   The node is disconnected and connections between existing nodes are reformed.

To disconnect nodes or clips:

1. Drag the cursor across the connecting line between a clip and a node or between two nodes.
   The arrowed line is cut and the source tab is dimmed.
2. Cut multiple connections in a single stroke by clicking and dragging over several connecting lines in the schematic.

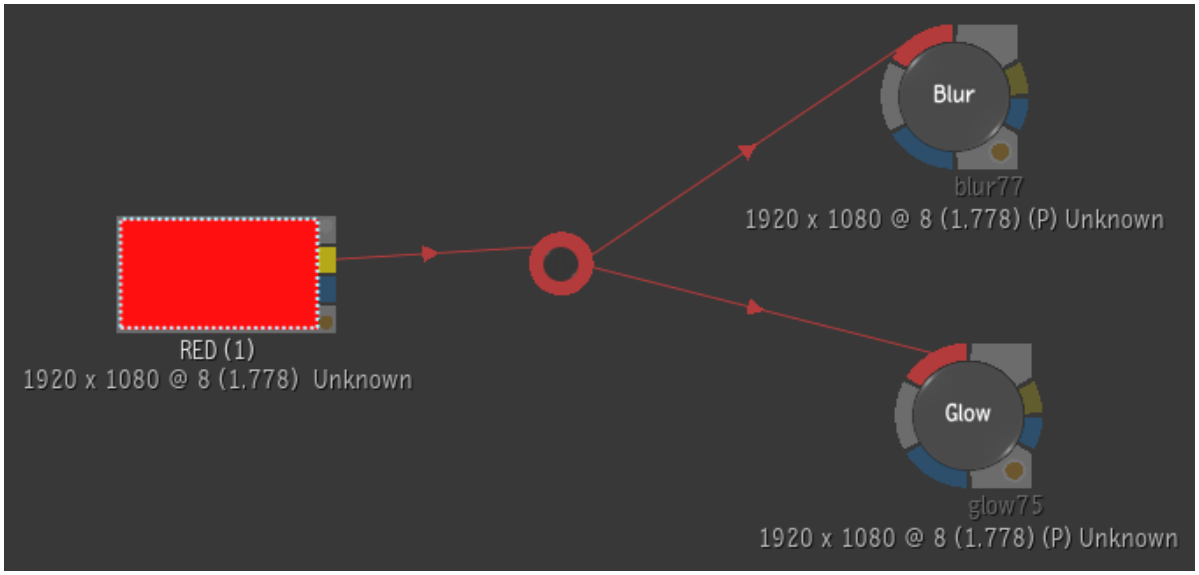
To delete a node:

1. Do one of the following:
   - Right-click a node and choose Delete.
   - Drag a node toward the bottom of the screen, releasing when the garbage can icon appears.
   - From the Tools box, select the Delete tool and click any nodes to delete.

     TIP Press the keyboard shortcut `D` to quickly access the Delete tool.

Creating Break Points in Node Connections

Instead of using the default straight line connections between nodes, you can add Elbow nodes to reshape the node connections. Elbow node connections can be useful to clean up the schematic view, and avoid links crossing each other.



There are many benefits to using Elbow nodes in your schematic:

- You can create multiple outgoing links from an Elbow node to other nodes in your schematic.
- The Elbow node inherits the colour of its link. It turns red, if connected to a Front input, blue, if connected to a Matte input and grey when connected to Back input or when connected to multiple inputs.
- An Elbow can be explicitly selected. This means that it is now possible to move Elbow nodes along with the other nodes within a selection. And that you can move Elbow nodes independently of its parent and child nodes.

  NOTE Break Link points are automatically converted into Elbow nodes when an older setup is loaded.

Adding an Elbow Node to the Schematic

An Elbow node can be added to the Schematic in the following manners:

- By enabling the Add Points Edit mode and clicking on a link to add an Elbow node.
- By holding down the Add Points shortcut (A) to add elbow nodes and then letting go of the shortcut to return to the previously selected Edit mode.
- By clicking on a node output to insert an Elbow node between the output socket as well as all of its outgoing links.

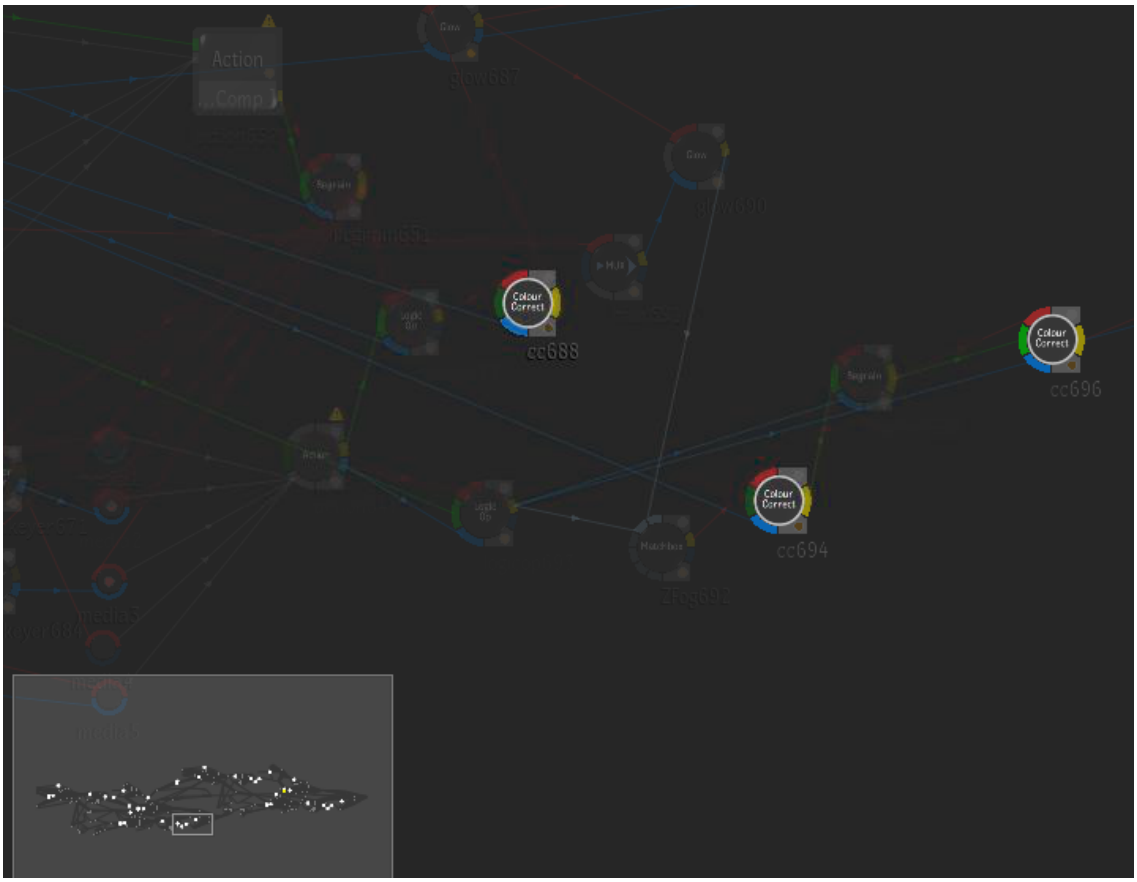
Searching for Nodes in the Schematic

Since schematics can get large very quickly, you can search and highlight nodes of the same type in the schematic, and easily move to a particular node.

To search for a node type in the schematic:

1. In the Node bin or schematic, hover over the node type that you want to search for, and press `Win+F` (this is mapped to Find Node(s) in Batch / MK Schematic in the keyboard shortcut editor).

   All nodes of the selected type are highlighted in the schematic.



TIP Enable the Display Navigator from the View menu to help you see the complete schematic. This also displays all of the highlighted search results.

2 Press `Tab` to cycle forward through the selected nodes, or `Shift+Tab` to cycle backwards. You can also explicitly select a node in the schematic and use `Tab` and `Shift+Tab` to cycle through all nodes of the same type.

You can also switch to Result view (`F4`), and use the `Tab` and `Shift+Tab` shortcuts to cycle through the search results.

Selecting Nodes in the Schematic

Use the following keyboard shortcuts to help you select and work with nodes in the Batch and Batch FX schematic.

NOTE These following keyboard shortcuts are based on the default User Profile for the application, and may change if you select a different profile at startup or during a session, in the Project and User Settings window.

To:	Do this:
Select a node.	Click node.
Select multiple nodes.	Press Ctrl, then click and drag a rectangular selection around the nodes.
Deselect everything.	Click on an empty space in the schematic, or press Ctrl and click-drag on an empty space in the schematic.
Add or remove nodes from a selection.	Press Ctrl and click the nodes you want to add or remove.
Select all the nodes before the current node (ascendants).	Press Alt and click the node.
Select all the nodes after the current node (descendants).	Press Alt+Spacebar and click the node.
Remove a branch from a selection.	Press Ctrl+Alt and click the branch.
Add or remove the branch after a node (descendants) to the selection.	Press Ctrl+Alt+spacebar and click the node.
Select the entire tree.	Press Alt+Win+Spacebar and click any node in the tree.
Add or remove a tree to or from the selection.	Press Ctrl+Alt+Win+Spacebar and click a node in the tree.
Select the entire setup.	Press Ctrl+A.
Move only one node in a selection.	Press Shift and move the node in the selection.

Adding Notes to the Schematic

You can add notes to the Batch or Batch FX schematic, or to a specific clip, node, or group. Notes are useful when collaborating on an effect or project with other users. Notes are saved with the setup so they are visible to all. You can copy and paste content between notes.

To create a freeform schematic note:

1. Drag the note icon from the All Nodes bin to the schematic.

2. Double-click the note icon.

   The Note text editor appears.
3. Click the editor window to activate it, and type your note text.
4. Click the upper-left corner of the text editor to close the editor.

To create a note on a clip, node, or group:

1. In the schematic, select the clip, node, or group to which you want to add the note.
2. Press `Shift+V`.

   The Note text editor appears.
3. Type text into the Note field.
4. Click the upper-left corner of the text editor to close the editor. The note icon appears to the right of the clip, node, or group.

This note is attached to the clip, node, or group.

Viewing Schematic Notes

You can modify how notes are displayed in the schematic.

To display an existing note:

1. Use one of the following commands with the cursor over the note (for freeform notes), or over a clip, node, or group that has a note.

Press:	To:
V	Temporarily display a note.
Ctrl+Shift+V	Expand a note for display.
Shift+V	Expand a note for editing.

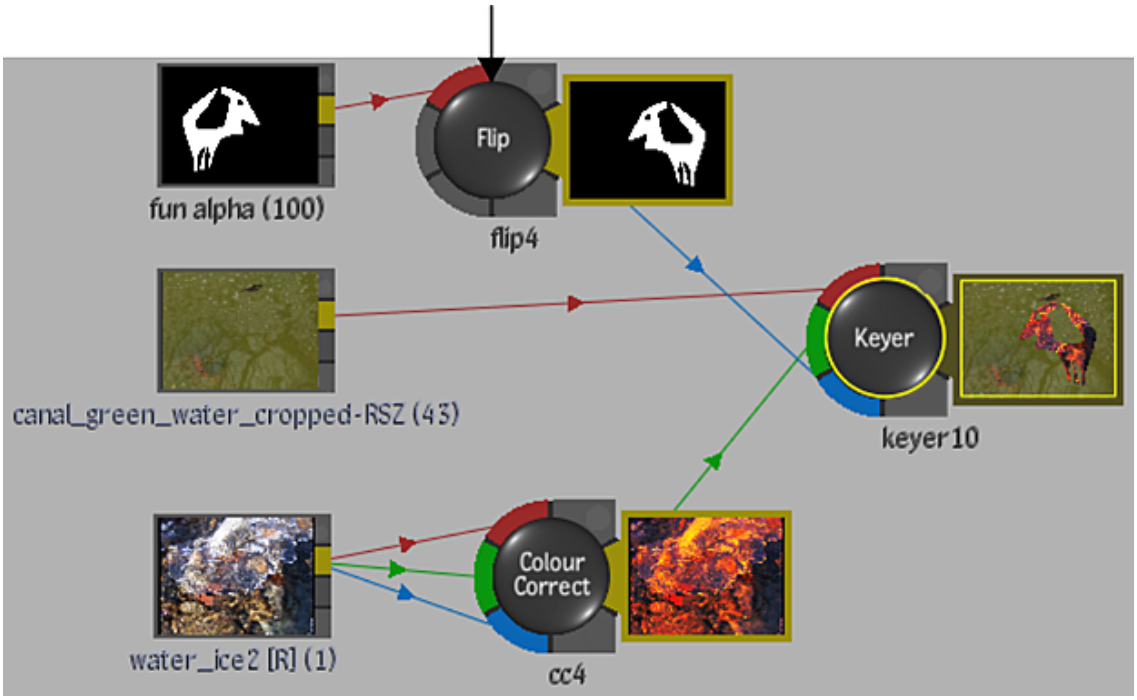
Bypassing Node Inputs

You can improve your workflow in Batch or Batch FX by deactivating certain nodes and rendering only the nodes that you want to process right away.

For any node, you can pass the Front, Back, Matte, or Key-in clip—depending on the type of node you select and its source tab inputs—as the input to the next node in a branch. If you bypass a node with multiple outputs (such as Action and Modular Keyer), the matte output will be the same as the result output (depending on the current selection in the Bypass box).

To bypass a node input:

1 In the schematic, select the node whose input you want to bypass.



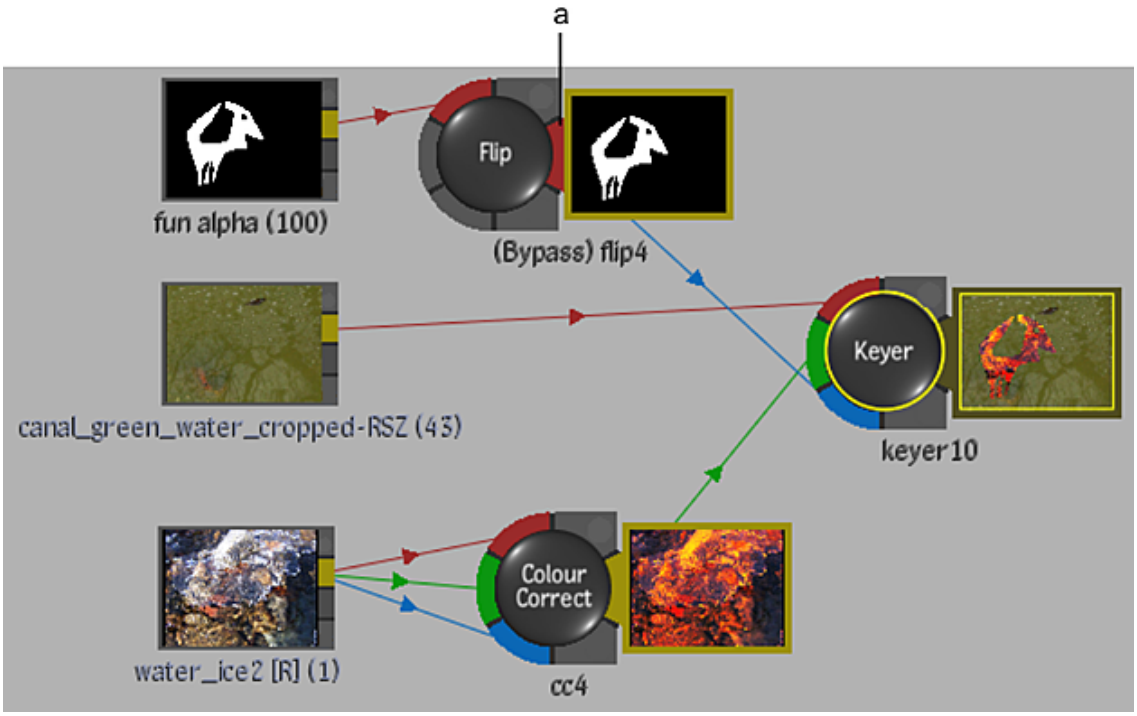
2 Enable Bypass.

The Bypass box becomes active.

3 Select the clip that you want to pass as the input to the next node from the Bypass box.

NOTE The Flip node, which was selected in step 1 of this example, has input tabs for a front source. Therefore Front is the only option in the Bypass box.

The output tab of the bypass node changes colour. In the following example, the tab turns red indicating that the Front input is passed to the next node in the tree. As well "(Bypass)" is added to the node name.



(a) Output tab of bypassed Front input

To unbypass nodes:

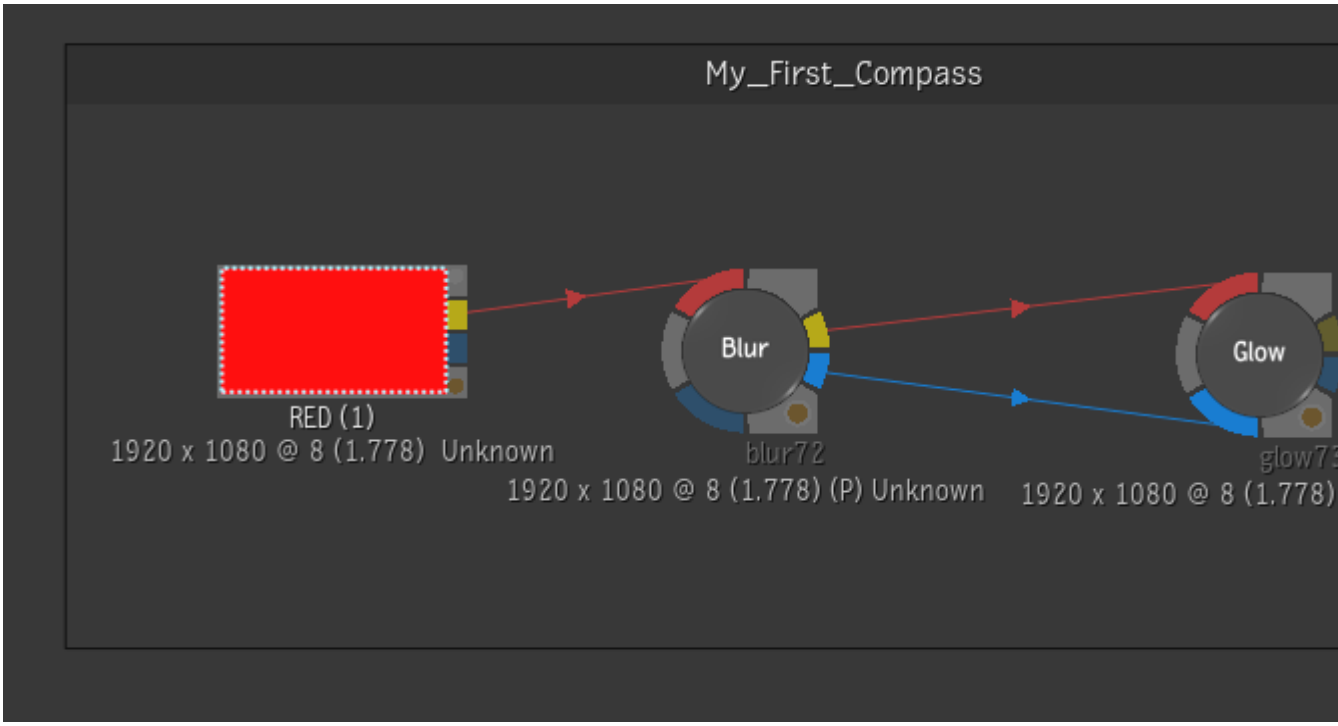
1 Do any of the following:

- Select the node with the bypassed input and disable the Bypass button.
- If the bypassed node is part of a multi-selection in the schematic, click Disable Bypass On Selection.

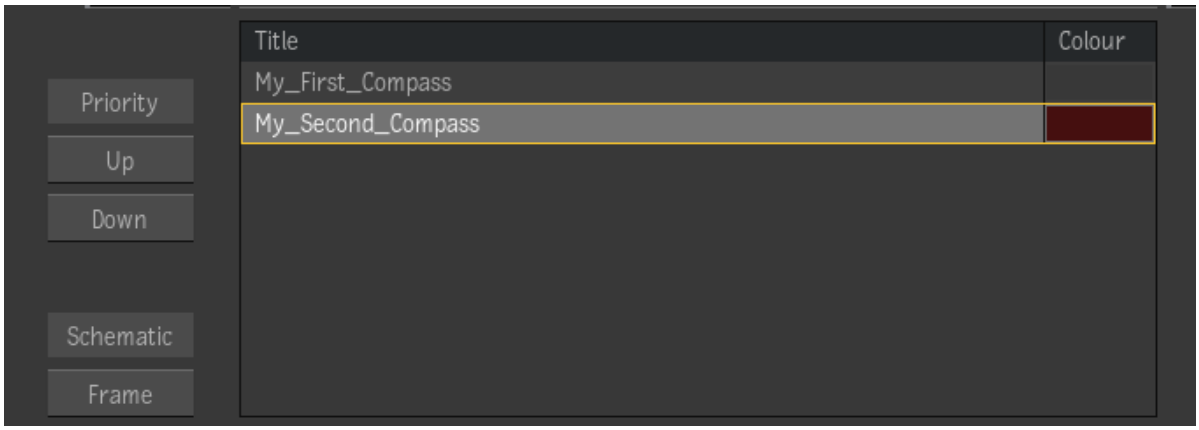
NOTE Click Restore Bypass On Selection to redo the bypass.

Using the Compass Node

The Compass node is a node that can be used to define and identify an area in the Batch, Batch FX or Modular Keyer schematic, by encompassing the nodes of your choice in your process tree. Adding the Compass node to the schematic creates a box that you can resize and place over the nodes you want it to encompass. All the nodes that are included in the Compass are automatically selected when you click on the Compass header and all the nodes included in the Compass will follow when moving the Compass. The Compass node makes it easier to manage your process tree, particularly if it is very large.



The Compass menu, which appears when a Compass node is selected in the schematic, contains a table that enables you to sort the different compasses in Z space, using the Up and Down buttons. You can also assign a colour to each Compass object in the schematic. The default colour is grey.



Adding a Compass Node to the Schematic

There are different ways to add a Compass node to the schematic:

- By dragging the Compass node from the Node bin. This creates an object of fixed width and height. The width and height can be modified afterwards.
- By double-clicking the Compass object from the Node bin and drawing the Compass, enabling you to immediately determine its size.
- By using the "Draw Compass" Edit mode. As above, this enables you to determine the Compass size as you draw the object.
- The Draw Compass Edit mode is assigned to the following shortcuts by default:
  - **Flame:** C

- **Smoke Classic:** Space-V
- **Smoke FCP:** X
- **Lustre:** C

Naming / Renaming a Compass Node

Once a Compass node is added to the schematic, the naming field, in its header, is immediately activated.

You can rename a Compass node by clicking on the selected item name within the Compass menu table.

Resizing a Compass Node

Compass nodes can be resized by selecting a corner or an edge of the box and dragging to the desired size.

Assigning a Colour to a Compass Node

There are different ways to assign a colour to a Compass node:

- By clicking the colour pot on the right side of the Compass title header. This displays the Colour Picker for colour selection.
- By clicking the colour pot to the right of each item in the Compass menu table. This displays the Colour Picker for colour selection.
- You can also define a default colour for Compass nodes from the Preferences tab in the Batch Prefs menu, in the Compass section of the Schematic header.

Framing a Compass Node

You can frame a Compass to the center of the schematic by selecting it in the Compass menu table and clicking the Frame button, on the bottom left. This is also achieved by double-clicking a Compass node entry in the Compass menu table.

Manipulating Compass Objects

You can Copy, Cut, Paste, Duplicate or Delete a Compass, from the contextual menu. However, it is not possible to save or load a Compass setup.

Keyboard Shortcuts

- You can move the Compass away from the nodes included in it using the "Enable Smart Extract of Nodes and Tabs" keyboard shortcut.
- You can navigate between Compasses using the Find in Schematic function (Meta + F) and the Select Previous / Next Node keyboard shortcuts.

Cleaning Up the Schematic

You can clean up a Batch, BFX or Modular Keyer Schematic using the "Clean Up Branch Upstream" and "Clean Up Schematic" functions.

- Clean Up Branch Upstream is available on all nodes and is accessible from the Contextual menu. All the nodes upstream are re-organized when this option is selected.
- Clean Up Schematic is available from the Contextual Menu when you click in the Schematic background. All nodes in the Schematic are re-organized.
- These operations are undoable.

Working with Clips in the Process Tree

The quickest way to add clips to a Batch process tree is through the Media panel. Even though Batch FX process trees are populated with clips from the timeline, more clips can be added from the Media panel.

NOTE The Import node is one method to add media to Batch or Batch FX. Using the Desk and Media Panel nodes, as well as dragging a clip from the Media Panel: these are all different methods that result in the same type of clip node.

To add clips from the Media panel:

1. Select one or more clips from a reel or library in the Media panel.
2. Drag the clips into the Batch or Batch FX schematic.
   The clips are now available as a clip node to use in the process tree.

Once a clip is present in the Batch or Batch FX schematic, connect its output tab to other nodes in the same manner as connecting nodes to nodes. The same clip can be connected to multiple nodes in the process tree.

Using the Media Panel, Desktop, and Import nodes from the I/O node bin are other methods to add a clip node to the process tree.

The right-click contextual menu of a schematic clip also offers many options, such as Open or Open As Sequence.

To replace a clip:

1. Drag a clip from the Media panel into the Batch or Batch FX schematic.
2. Release the clip on top of an existing schematic clip, once you see a replace icon.
   The clip is replaced.

   TIP If the clip proxy is black after replacing, right-click the clip and select Reset to reset the timing offset of the clip.

Alternatively, press `Shift` and double-click a selected clip in the schematic to open the Desktop in Clip Select mode, where a new clip can be selected to replace the Batch or Batch FX clip.

To display a clip's channels:

1. Select the clip in the schematic.
2. Press `Shift-C`.

To find a clip on the reels:

1. In a schematic, right-click the clip to be located.
2. Select Find in Reels.
   The view switches to Reels and the specific clip is highlighted.

To find a clip in a schematic:

1. In the Media panel, right-click the clip to be located.
2. Select Find in Schematic.
   The view switches to the correct schematic and the specific clip is highlighted.

To switch between versions of an Open Clip:

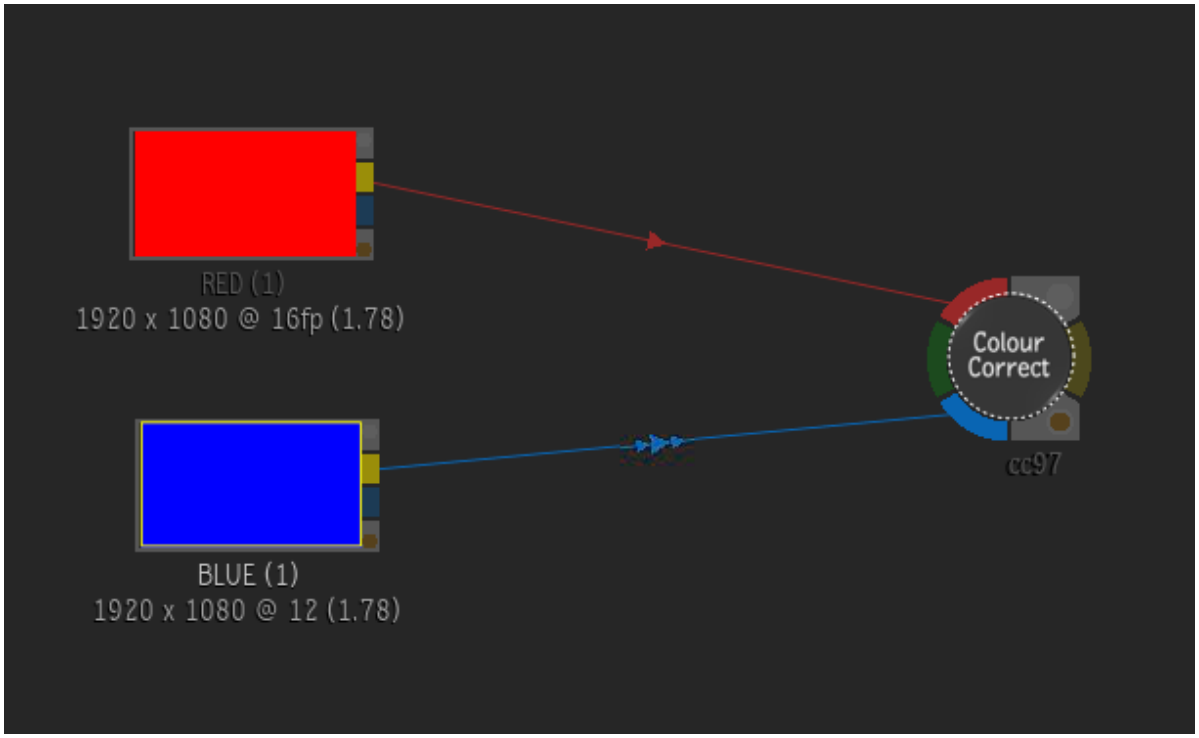
1. Double-click the clip in the schematic to display its properties.
2. Open the **Extended** menu.
3. From the Clip Versions box, select a version.

   The clip changes to selected version, updating the schematic. Existing channels are matched using a strategy that attempts to preserve existing links.

About Adaptors

Because the Batch or Batch FX environment exclusively supports 8-bit or 16-bit fp media, 10 and 12-bit media needs to be converted to 16-bit fp. While this can be done manually through the Resize or the LUT Editor, you can apply Adaptors, which automatically apply the proper bit depth conversion(s), when you bring media into Batch or Batch FX. This greatly reduces the number of mismatches in Batch or Batch FX (red dots on nodes), without the need to manually convert anything.

NOTE Manually applying a LUT or Resize makes the adaptor(s) disappear.



Enable Show Adaptors in the Batch or Batch FX preferences menu. You can see that an adaptor has been applied by an arrow inside two smaller arrows on the link between nodes in the schematic. By hovering over the adaptor, the applied conversion is displayed in a text box, much in the same way as tooltips (i.e. [12-bit u -> 16-bit fp]).

Editing a Batch Setup Included with an Open Clip

1. Double-click the clip in the schematic to display its properties.
2. Open the **Extended** menu.
3. Click Append to current Batch.

   The Batch included with the Open Clip is added to the schematic as a Group Node.

The appended Batch includes the original Write File node. This allows you to export and add a new version of the media to the Open Clip .

Clip Information Reference

Detailed information about clips such as resolution, frame rate, and size can be displayed directly in the schematic. Visual cues such as symbols and letters identify clips that have had specific types of operations applied to them.

The colour of the clip information and the letters and symbols can also provide useful information at a glance.

Clip Information

Specify the clip information to be displayed under clips in the schematic by selecting an option from the Thumbnails area of the User Interface Preferences (page 1947):

Select:	To display:
Clip Name Only button	Enable to display only the name of clips on thumbnails.
Secondary Information box	Select the secondary clip information displayed on thumbnails. The options are: No Secondary Information, Record Timecode, and Source Timecode.
Resolution Information box	Select the clip resolution information displayed on thumbnails. The options are: No Resolution Information, Resolution, Frame rate, and Resolution & Frame rate.
Display Keycode button	Enable to display keycode information on thumbnails.
Render Effects button	Enable to display rendered frames in thumbnails of clips with applied but unrendered effects. Disable to display "Unrendered" in the thumbnails of such clips.

Clip Name Colouring

The colour of the clip name in the schematic shows where the particular clip originated.

Colour:	Origin:
White	Media panel
Blue	Batch iteration
Pink	Read File node
Red	No media found (clip is also displayed as a black proxy)
Green	BFX Back Clip

Clip Status Symbols

The following visual cues show the type of clip and its status:

(A) symbol and Audio Context outline Indicates that an audio context is set for the clip or node. In addition to the A, the clip has a dotted purple outline. The letter A is followed by the number of the context, for example, A2 is the second Audio context.

(C) symbol and Context outline Indicates that a context is set for this clip or node. In addition to the C, the clip has a dotted green outline. The letter C is followed by the number of the context, for example, C1 is the first video context.

LUT symbol Indicates that a LUT was applied to the clip. The first and second number represent the source and destination bit depths, for example [8 -> 16f].

[R] symbol Indicates that a Resize was applied to the clip.

[Multi] symbol Indicates a multi-channel clip.

(BFX) symbol Indicates that a Batch FX was applied to a timeline made up of one segment, or the clip was converted as a Create BFX. The clip can be expanded.

(bfx) symbol Indicates that a Batch FX was applied to a timeline made up of multiple segments. Setups of clips containing more than one segment cannot be expanded.

Clip Settings

- Pressing `Shift` and double-clicking a clip in the schematic opens the workspace in Clip Select mode, where you can select a new clip to replace the Batch or Batch FX clip.
- In Batch, you can also double-click a clip in the schematic to switch to the Timeline with the selected clip available as a source clip. You can apply any timeline editing operation on your clip, and when you return to Batch, your clip is updated.
- With a clip selected in the schematic, press `Shift+C` to expand the node and display its channels in the schematic.

The following settings are available when selecting a clip in the schematic. Depending on the type of clip selected, some of the settings may differ. You can also right-click a clip node in the schematic to reveal a contextual menu of operations, such as opening the clip as a timeline sequence, or render options.

Basic Settings

Basic button Opens the Basic menu where you set timeline and timewarp options, control clip locking and slipping, and define how missing media is displayed and rendered.

Head Media box Select to substitute missing media at the beginning of a clip with black frames, the first frame of media, or leave as is.

Gap Media box Select to substitute missing media in gaps with black frames or leave as is.

Tail Media box Select to substitute missing media at the end of a clip with black frames, the last frame of media, or leave as is.

Offset field Displays the number of frames by which selected clips and/or segments are offset. This same setting can be found in the Timing View as well as the Clip Settings. Editable.

Lock Frame button Enable to display the current frame for the duration of the clip.

Edit BFX button Click to access the next level of nested BFX, making it available for editing.

Explode FX button Click to explode all Timeline FX in the clip into a process tree. If a clip proxy is black after exploding, right-click the clip and select Reset to reset the timing offset of the clip.

Explode History button Click to convert the source into a clip node to access the pre-processing settings of the clip.

Explode One button Click to explode one level of the clip's BFX setup.

Explode All button Click to explode all levels of the clip's BFX setup.

Convert to 3D Comp button Click to convert an existing clip to a 3D composition pipeline with an Action node.

Split Tracks button Click to split a multitrack or stereo clip into individual clips.

Alpha Active button Enable to generate an alpha or matte from a clip.

Alpha Colour box Select whether to output a white or black alpha of your clip.

Extended Settings

Extended button Opens the Extended menu where you set the channel options, from the channel displayed in the node proxy to the available outputs.

Version Selection box Select the version of the selected clip. Versions are only available to Open Clip-based clips. If Original displayed for an Open Clip: only one version is loaded, click to check for new versions. For any other type of clip: Original is always displayed.

Creation Date field Displays the creation date of the version selected in the Clip Version box. Non-editable.

Source Setup Name field Displays the name of the Batch setup saved with the open clip. Available to Open Clips that include a Batch setup, such as output with a Write file node with Include Setup enabled.

Append to current Batch button Click to add to the current Batch schematic a group made of the Batch setup stored with the open clip. Available to Open Clips that include a Batch setup, such output with a Write file node with Include Setup enabled.

Channels list Displays the clip's channels, allowing you to select its proxy (Proxy), hide or display its channels (Icon) and its outputs (Outputs).

Type field Displays the type of channel. Non-editable.

Name field Displays the name of the channel. Non-editable.

Colour Space field Displays the colour space of the channel. Non-editable.

Icon field Enable to display the channel in the schematic; disable to hide the channel. The clip must be expanded (press `Shift+C`) to view the channels in the schematic. Hiding a channel does not break its link to a node.

Proxy field Enable to use a particular channel as the clip's proxy in the schematic.

Outputs field Click to display or hide the outputs available for each channel from the clip in the schematic. Hiding an output does not break its link to a node. There is one output each for RGB and its alpha (if any).

Clip Information

The Clip Information tabs display non-editable clip information.

Clip Info tab The Clip Info tab displays the name of the original media, file format, resolution and FCM, as well as the number of tracks in the clip.

Metadata tab The Metadata tab displays metadata information about the imported media, including but not limited to: path to the sources, camera settings (RED and ARRI), file creator.

Versions tab The Versions tab lists all the versions available for the selected clip. Available to Open clips imports with versions.

Resize Settings

Resize settings are not available to multi-channel clips. Use Resize nodes to resize the different channels. Or display the clip in the Timeline, and open the Pre-Processing options.

Most of the Resize menu is the same as when using the Resize node in Batch or Batch FX. The following settings are specific to the Resize tab for a clip:

Resize button Opens the Resize menu where you change the size of a clip, as well as its aspect ratio.

Active button Enable to activate the Resize settings.

Load button Loads a Resize setup from the library.

Save button Saves the current resize setup.

Colour Management Settings

Colour Management settings are not available to multi-channel clips. Use Colour Management nodes to colour manage the different channels. Or display the clip in the Timeline, and open the Pre-Processing options.

Most of the RGB LUT menu is the same as when using the LUT Editor node in Batch or Batch FX. The following settings are specific to the RGB LUT tab for a clip:

RGB LUT button Opens the LUT Editor for the selected clip.

Active button Enable to activate the LUT settings.

Back Clip Settings

These settings are available for a Batch FX Back Clip created for a timeline adjustment segment.

Head Media box Select what media is available before the In Point of the adjustment segment. Select Timeline Level to have the Back Clip read the head media from the segments preceding (and below) the adjustment segment.

Tail Media box Select what media is available after the Out Point of the adjustment segment. Select Timeline Level to have the Back Clip read the tail media from the segments following (and below) the adjustment segment.

Adding Media to an Action Node

To add indirect media to an Action node:

1. In the Batch or Batch FX schematic, double-click the Action node and then click Media in the Action menu.
2. From the Media List box, select New Input.

   A Media node is added to the Action node.

   Notice that the Media list is empty. Although you have added a Media node to the Action node, you have not yet added any media.
3. Connect front and matte clips (image clips, or the output of another node in the Batch or Batch FX process tree) to the red and blue input tabs of the Media node, respectively.

   The Media list is updated with the indirect media. Brackets (“[ ]”) around the clip names indicate the media is indirect.

When you select a Media node in the schematic, the corresponding media in the Media list is highlighted. Conversely, selecting media in the Media list highlights the corresponding Media node and the link to the Action node.

NOTE A red dot in the Media node indicates an input resolution mismatch between the front and matte clips.

To add direct media to Action using Clip Select:

1. In the Batch or Batch FX schematic, double-click the Action node and then click Media in the Action menu.
2. From the Media List box, select New Media.
3. From the workspace, select a front and matte clip to load as media.

   NOTE You can select any number of front and matte clips by holding the Ctrl key while selecting clips. Each front/matte selection is added to its own line in the Media list, and image nodes are automatically added to the schematic if Auto Image is selected in the Node Prefs menu.

To add direct media to Action gesturally:

1. From the Workspace Media panel, drag a clip to the Action schematic or Media list.

   An image node and axis are added to the Action schematic (if Auto Image is selected in the Node Prefs menu), and the Media list is updated with the front media.
2. To replace the media in an image node, drag a new clip from the workspace Media panel onto an existing image node in the Action schematic. Release the clip when you see the red replace arrow.

   The image is replaced in the schematic, and the Media list is updated with a new entry for the media. The replaced media remains as a Media list entry in case you are using it elsewhere in your Action scene.

To convert direct media to indirect media:

1. Select the media that you want to convert from the Media list.
2. Click Extract.

   A Media node is added to the Batch or Batch FX Action node and the selected media is automatically connected to the respective input tabs. In the Media list, brackets appear around the media name, indicating the media is indirect.

About Action in a Batch or Batch FX Workflow

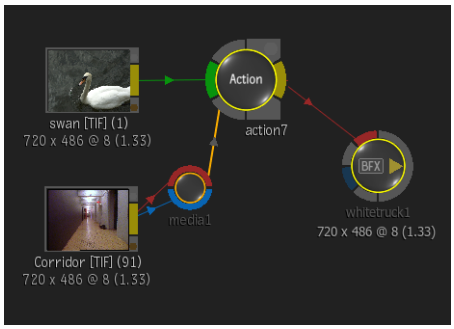
Action is a multilayer compositing tool for creating complex visual effects, with its own distinct schematic and node bin. The Action node supports direct media and indirect media.

Indirect media is connected directly to an Action Media node and appears in the Batch or Batch FX schematic. With indirect media, you can connect any source (such as images, or the output of another node in the Batch or Batch FX process tree) to an Action node. Media nodes are permanently parented to the Action node (you cannot sever the process lines). You can add multiple Media nodes to an Action node.

Direct media appears directly inside an Action node, and in the Batch or Batch FX Sources folder in the Workspace Media panel, as well as in the Batch or Batch FX Timing View. Although direct media does not appear in the Batch FX schematic, media and all related settings are saved with the Batch or Batch FX setup.

TIP You can also parent a back clip to the Action node. Although doing so is not necessary, a parented back clip node provides a good visual reference for identifying the Action composite in the process tree.

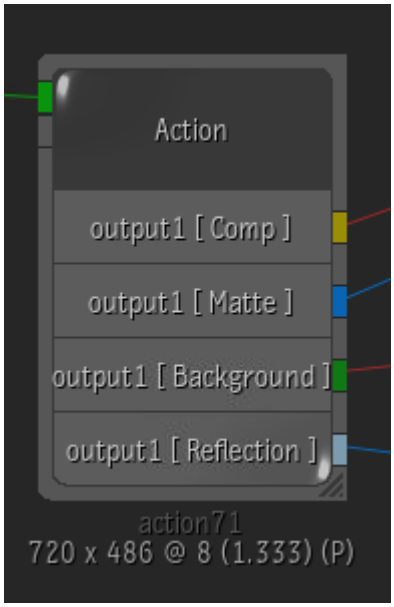
The following example displays indirect media in Action. The *swan* clip is attached to the Action node as the back clip, whereas the *Corridor* clip is attached to an Action media node as the front and matte media.



The Action node supports multiple outputs. For example, you can set up your scene to output your entire composition or just the mattes and then select the type of output you want to process. You can also output stereo results of your scene.

About Action Node Output Tabs

The Action node's output tab changes depending on the number and type of outputs it has, as shown in the following example.



Expanded Action node

Collapsed Action node

The tab colours of a multi-output Action node are based on the type of output (for example, yellow for composite, blue for matte).

TIP To be able to see and connect other nodes to a multiple-output Action node, use the `Shift+C` keyboard shortcut to expand or collapse the node.

The number of outputs is determined by the outputs set up in the Output list. See About Rendering Outputs from Action (page 598).

Setting Stereo Startup Mode for Action

When dragging an Action node to the Batch or Batch FX schematic, you can automatically set the node to stereo startup mode. If you change any of the stereo startup settings, you can revert back to the default stereo startup settings. You can also change the settings of an existing Action mono node to the stereo startup settings.

For information on the settings that are affected by the stereo startup mode, see Rendering Tab (page 588).

To set the stereo startup mode for a new Action node:

1. Enable the Stereo Mode button below the node bin.

   The node is renamed to Action_STEREO, the output tabs display the left eye and right eye outputs, and all stereo settings are automatically set.

To apply the stereo startup setting to an existing Action node:

1. Select the Action node.
2. In the Node Prefs menu for the Action node, click Reset To Stereo Mode in the Rendering section.

   Once confirmed, the current setup is replaced by the stereo setup and all media is deleted.

Viewing Nodes in Context

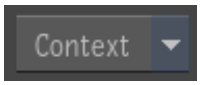
You can view a node in context with another node to compare intermediate results throughout the process tree. By working with Context views, you can modify nodes in the process tree and immediately view the impact those changes have on the nodes further along in the process tree. You can set up to 10 Context views in a process tree. Context views can be set and used in Batch,Batch FX and in the Modular Keyer and, once set, can also be accessed from the Timeline.

Setting a Context View

Contexts can be set:

- **From the Contextual menu.** Options are:
  - **Set As Context:** Select the node you want to set as a Context view. From the contextual menu, select Set As Context, and select the specific Context view you want to assign it to.
  - **Set As Next Available Context:** Select the node you want to set as a Context view. From the contextual menu, select "Set As Next Available Context". The selected node is set as the first available Context view in the list. When all ten Context view slots are used, this option is no longer available.

- **Using the Context button:**



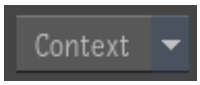
  - Pressing the Context button sets the selected node to the first available Context view in the list.
  - Clicking the drop-down arrow displays the Context list and you can select the specific Context view you want to set.

- **Using Keyboard Shortcuts:**

  Several keyboard shortcuts are available for setting Context views.

  NOTE The keyboard shortcuts described below apply to the Flame profile. The names of the shortcuts are displayed in parenthesis to allow users of other profiles to quickly search for the corresponding keyboard shortcuts.

  Pressing `= + click` (Set Context) on a selected node displays the Context list menu, with the first available slot highlighted.



  This keyboard shortcut also enables you to select a node output as a Context view, by clicking the output of the node.

  There are three ways of setting a Context view from here:

  - Pressing `Enter` automatically sets the highlighted slot.
  - Pressing the number key `(1 to 0, where 0 represents 10)` corresponding to the desired slot.
  - Clicking the desired slot.

    NOTE When all the slots are used, this menu appears, regardless of the input method used, so that you can select the specific Context you want to overwrite.

Pressing `Meta + = + click` (Set As Next Available Context) on a selected node assigns it to the first available Context view in the list.

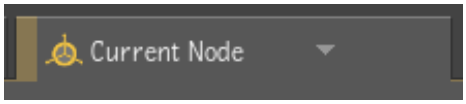
Displaying Context Views in the Viewport

You can display the context view of your choice, by pressing `Space + [the number corresponding to that Context view : 1 to 0, where 0 represents 10]`.

You can navigate between the next and previous Context view, using `Tab` (Previous Context) and `Shift + Tab` (Next Context).

In the Batch, Batch FX and Modular Keyer schematic, you can use the `Alt + =` (Select All Context Nodes) keyboard shortcut to highlight all the nodes that are set as Context views. This is similar to the Find function `Meta + F`.

You can display any of the set Context views in the Player, from the Timeline. You can select the desired Context view, by clicking the drop-down arrow on its tab, at the top of the screen. This is possible as long as the media in the Timeline is present in Batch or Batch FX.



You can also display any of the set Context views in the Timeline FX Editors, by clicking the drop-down arrow of the Context button. Again, the selected media segment must be present in Batch or Batch FX.

You can also set node outputs as a Context view, for example the Matte Output.

About Implicit Selection in Batch or Batch FX

Aside from Contexts 1 to 10, you have two other context views: Current Clip Node and Current Node.

- **Current Clip Node:** When a clip is actively selected, it displays a yellow border. If you then actively select a node, the node displays the yellow border and the last selected clip displays a dotted border - it remains the Current Clip Node. You always have the ability to display the last selected clip node, whether it is actively selected or not.
- **Current Node:** When a node is actively selected, it displays a yellow border. If you then actively select a Clip Node, the clip displays the yellow border and the last selected node displays a dotted border - it remains the Current Node. You always have the ability to display the last selected node, whether it is actively selected or not.

NOTE To display the Current Clip Node or the Current Node in the Viewer, select the view you want to display from the View box in Batch or Batch FX, on the Timeline Viewer tab, or from the Timeline FX user interface.

About Previewing Results in Batch or Batch FX

While creating your process tree, you can preview results at any time. If you do not like the result of one operation, you can modify or delete it without affecting the other operations in the sequence. You can preview your results in Batch or Batch FX by:

- Using proxies in the schematic.

  Double-click the result tab on the node to display or hide the proxy. If Auto Update is enabled in the Schematic section of the Preferences menu, the proxy updates automatically as changes are made.
- Selecting one of the options in the View box to view results at the current pipeline level in the image window.

  Select the node whose results you want to preview, then select an output or result view from the View box (use a 2-Up view to view the schematic and result at the same time). In Batch FX, you can select BFX Timeline Result to see your composite in the context of the timeline.
- Setting up to context nodes in Schematic view and viewing the results in context in the image window.
- Using implicit selection to view results in context in the image window.
- Display the BFX view to display all of the BFX levels as a hierarchy graph. To display the BFX View, select the view from the Timeline tab's View Mode box.
- You can open a rendered Batch or Batch FX clip in the full screen player directly in Batch or Batch FX. The Player button is available in Batch if there are any clips in the Batch Renders folder. If Show Play

Reel is enabled in the player options, you can play all clips in the Batch Renders folder. In Batch FX, the Player button appears after you render, and displays the current BFX output.

Working with Audio Contexts

While working on one clip or branch of the process tree, you can listen to the audio of another clip. Audio contexts work in much the same way as video contexts. You can set up to two audio contexts in Batch or Batch FX.

To set an audio context:

1. In the schematic, select a clip that contains audio.
2. Click the Au Context button, or right-click the clip and select Set as Audio Context.
   Clips with an audio context are labelled A1 or A2 and are outlined by a pink dotted line.
3. To hear an audio context, select Context1 or Context2 from the Audio Source box in the Batch or Batch FX Preferences menu.

NOTE Playback of audio is only available when playing a clip. However, you can scrub audio anytime in Batch or Batch FX. You can scrub your audio context by using `Ctrl` (quick scrub) or `Shift+Ctrl` (real-time scrub) on any node.

About Batch Rendering

When ready to generate final clips or intermediate results, define the settings for each Render and Write File node to be processed, then use the Render box to make a rendering selection. By default, rendered clips appear in the provided Batch Renders reel under the Batch Shelf. Use the Renders Destination box of Render Node settings menu to specify another location.

Use the Render node to output clips to the Desktop or Media Library. Use the Write File node to export open clip files or image sequences in any supported format. With the Render node, metadata is kept; with the Write File node, metadata is not kept, unless Create Clip Data is enabled.

Render and Write File nodes support RGB, RGB-A, and Stereo processing modes.

When working with multiple resolutions, the resolution of the Render or Write File node is determined by the clip or node feeding it. However, the output can be resized to any resolution.

Use the Render List to manage multiple Render and Write File processes.

Proxy Quality and Rendering

The proxy quality workflow affects the Batch rendering options. There is a Resolution button, in the Basic menu of the Batch interface, which offers two options:

- **Working Resolution:** This tells Batch to render the media at the current working resolution. If the current working resolution is set to Proxy, Batch renders at proxy resolution. If the current working resolution is set to Full, Batch renders at full resolution. This is the default option.
- **Full Resolution:** This option forces Batch to render at full resolution, regardless of the current working resolution.

Naming Output Nodes

To help organize outputs, Output node can be created using the name of any output socket in the process tree. With the Output node selected in the schematic, press `N` and click a node output—the name of the

Output node is updated. To change the Output node name, or if to change the name of the original output socket, redo the naming process.

Tokens are also available to set default Render node and Write File node names:

- To define a name for one user, use the Default Render Node Name and Default Write File Node Name fields located in the Timeline FX / Batch / Batch FX tab of the Preferences panel.
- To define a name based on specific criteria, use the *batchDefaultRenderNodeName* and *batchDefaultWriteFileNodeName*python hooks, located in */opt/Autodesk/<installed_application>/python/batchHook.py*.

Exporting With Open Clip Data

Open clip files can be created using a Write File node with Create Clip Data enabled. This enables the ability to save a copy of the active setup with the exported material, and package exports with the setups that created them or version the exports.

NOTE An open clip file is an XML file which describes channels, media, and versions. Create open clip files to be used inside Batch with the Read File node, such as within a context of render passes between Flame and a 3D application.

Advanced: Executing Batch from the Command Prompt

Batch can be executed directly from a command shell in order to render complex process trees while the workstation is not in use, such as overnight. The following command starts Flame for a specified user, and renders a specified Batch setup:

flame -b batchsetup -U user - J project

NOTE The specified user and project name must be valid.

Render Node Settings

Use node settings to define how individual Render nodes are processed and to specify a destination for each rendered result.

The Render Node is available in Batch and Batch FX and perform the same function in both views. However when working in Batch FX, the Render Node can only render a portion of the schematic; to render the entire schematic, use the Batch FX Output Node.

To access Render Node settings, double-click a Render node in the schematic. If there are multiple Render nodes and Write File nodes in the Batch pipeline, use the Render List to manage them.

A default automatically-incremented name for all Render nodes can be set in the Timeline FX / Batch / Batch FX Preferences (page 1944).

Basic Settings

Basic button Opens the Basic menu to set Render Node formatting and destination settings.

Export Format box Select the type of clip rendered by the node. RGB-A sets one red RGB input tab and one blue matte tab. Stereo sets two RGB input tabs (one for each eye) as well as two blue matte tabs (one for each eye). Stereo requires a <polarity> token in the Naming field; if there is no <polarity> token, the application automatically adds it to the name of the exported files.

Export Resolution box Select to render the clip at either full or working resolution. Available only when proxies are enabled for the project.

These options are also accessible in the Render List tab, in the Resolution column.

Frame rate box Select the frame rate of the output clip.

Output Depth box Select the bit-depth of the output clip.

Storage box Select a 12-bit storage option. Select Packed to save image data so that it requires less space on the framestore. Select Unpacked to save an image that can be retrieved without an intermediate step. Active when the Output Depth option is set to 12-bit.

Offset Timecode button Enable to apply any timecode offsets made in the Render List. For example, a timecode of 00:00:00:00 is specified and rendered from frame 10, the first frame of the render is TC 00:00:00:09.

Source Timecode field Displays the source timecode for the output. Editable.

Record Timecode field Displays the record timecode for the output. Editable.

Tape Name field Displays the tape name for the output. Editable.

Renders Destination box Select whether to render the clip to Batch reels, reel groups, or libraries.

Selected Batch Reel box Select an existing Batch reel for the rendered clip or create a new schematic or shelf reel as the destination.

Create Reel Group button Select an existing reel group for the rendered clip or create a new group as the destination.

Range From field Displays the first impacted frame. Editable.

Range To field Displays the last impacted frame. Editable.

Render Range button Click to output based on the ranges in the To and From fields.

Rendering Option box Select either Foreground or Background Reactor as the rendering engine. This option is hardware dependent.

For supported systems, select Background Reactor to continue working while effects are being generated.

Include History button Enable to track and update operations applied to the rendered node.

Burn Range button Click to burn based on the ranges in the To and From fields.

Resize Settings

The Resize settings are the same as when using the Resize node in Batch. Click the Active button to activate the Resize settings.

RGB LUT Settings

The RGB LUT settings are the same as when using the LUT Editor node in Batch. Click the Active button to activate the RGB LUT settings.

Write File Node Settings

Use these settings to define how individual Write File nodes are processed and to specify a destination for each processed result.

The Write File node is available in Batch and Batch FX and work the same way in both views. However, in Batch FX, the Write File node can only render a portion of your schematic, not the entire schematic. That is always handled by the Batch FX Output Node.

To access Write File Node settings, double-click a Write File node in the schematic. If you have multiple Render nodes and Write File nodes in your Batch pipeline, you can manage them in the Render List.

A default and automatically-incremented name for all Write File nodes can be set in the Timeline FX / Batch / Batch FX Preferences (page 1944).

NOTE When rendering to an Open Clip with Clip Version enabled, make sure the frame rate remains the same between versions: having differing frame rates within one Open Clip is not supported by the format.

Basic Settings

Basic button Opens the Basic menu where you set Write File formatting, rendering, and naming settings.

Export Format box Select the type of clip rendered by the node. RGB-A sets one red RGB input tab and one blue matte tab. Stereo sets two RGB input tabs (one for each eye) as well as two blue matte tabs (one for each eye). Stereo requires a <polarity> token in the Naming field; if there is no <polarity> token, the application automatically adds it to the name of the exported files.

Frame rate box Select the frame rate of the output clip.

File Format box Select the file format of the images to export.

File Extension field Select the extension for the file type selected in the File Format box. Editable.

Output Depth box Select the bit-depth of the output clip.

Source Timecode field Displays the source timecode for the output. Editable.

Record Timecode field Displays the record timecode for the output. Editable.

Frame Pad field Displays the padding of the frame ID appended to the file name. "4" indicates that up to four zeros are prefixed to the file name for each frame; frame 1 is written as 0001, frame 22 as 0022, frame 555 as 0555.

Offset field Displays any timecode offsets that you made in the Render List. For example, if you specify a timecode of 00:00:00:00 and render from frame 10, the first frame of the render has TC 00:00:00:09.

Transfer Characteristics box Select an option to identify the attributes associated with a particular film or video format, such as resolution, frame rate, or colour space. Active when File Format is set to DPX.

Compress button Enable to apply compression to the exported file. This results in smaller file size but lower image quality. Available for Tiff, Sgi, and OpenEXR format types.

JPEG Quality field Displays the degree of quality versus compression for the Jpeg format type. 0 gives the lowest quality (and highest compression). 100 gives the best quality (but applies no compression). Editable.

Create Clip Data button Enable to create an Open Clip file, which includes extended clip data with the exported files. Required to create versioned clips.

Include Setup button Enable to include the Batch setup information with the rendered media. This allows someone accessing the Open Clip file to view and modify the original Batch setup, similar to how you can edit Clip History.

Clip Versioning box Select Custom Version or Follow Iteration to version the rendered exports with an Open Clip. The resulting Open Clip has versions accessible through a clip node in Batch, or from the Timeline. Follow Iteration keeps the version number in sync with Batch's Iteration number. Consider using a <version> token in the Clip Naming field or the next rendered version will overwrite previously rendered media.

- No Versioning: No versioning in the created Open Clip.
- Custom Version: Versioning in the Open Clip, with a version number set by the Version Field. This is the standard behavior.

- Follow Iteration: Displays the value of the current Batch Group's Iteration in the Version field. With Follow Iteration, the Version number always matches the value of the Iteration: iterating the Batch Group automatically updates the Write File node's version.

Version field Displays the version identifier, which automatically increases at each subsequently rendered export if Clip Versioning is set to Custom Version. Or set the version value manually. Ctrl+click to display the next available value. Re-using a previous value overwrites that version with the new export. Editable.

Version Pad field Displays the number of zeroes to left-pad the version number with. Editable.

Version Name field Displays the Version Name. Editable.

Range From field Displays the first impacted frame. Editable.

Range To field Displays the last impacted frame. Editable.

Render Range button Click to output based on the ranges in the To and From fields.

Burn Range button Click to burn based on the ranges in the To and From fields.

Name field Displays the name of the exported files. Build a dynamic naming scheme using the Add Token menu, as well as use any of the characters allowed in a Linux file name. Create folders using the slash (/) character. The application always appends the file ID and the extension of the file format selected for the export. Editable.

Destination Path field Displays where the exported files are created. Also used for Open Clip (Create Clip Data enabled) and Batch setup (Include Setup enabled). Polarity and Version are automatically added unless specified in the Name field. Editable with Change Path.

Add Token menu Inserts in the adjacent field the selected token.

Clip Name field Displays the name of the exported Open Clip. Build a dynamic naming scheme using the Add Token menu, as well as use any of the characters allowed in a Linux file name. Create folders using the slash (/) character. Defaults to the name of the Write File node. Editable.

Setup Name field Displays the name of the exported Batch setup. Build a dynamic naming scheme using the Add Token menu, as well as use any of the characters allowed in a Linux file name. Create folders using the slash (/) character. Defaults to the name of the Write File node. Editable.

Resize Settings

The Resize settings are the same as when using the Resize node in Batch. Click the Active button to activate the Resize settings.

RGB LUT Settings

The RGB LUT settings are the same as when using the LUT Editor node in Batch. Click the Active button to activate the RGB LUT settings.

Matte LUT Settings

The Matte LUT settings are the same as when you are using the LUT Editor node in Batch. Click the Active button to activate the Matte LUT settings.

Advanced: OpenEXR Compression

It is possible to define the OpenEXR compression used in the Batch Write File output (when using compressed OpenEXR target), using the DL_OPENEXR_COMPRESSION environment variable.

OpenEXR available compressions are:

- none
- RLE
- ZIP-multiscanline
- ZIP-scanline
- PIZ
- PXR24
- B44
- B44A
- DWAA
- DWAB

To set the Write File node to generate PIZ OpenEXR files:

1. Exit the application.
2. In the shell used to launch the application, type:

   ```
   setenv DL_OPENEXR_COMPRESSION piz
   ```
3. Start the application from that same shell.
4. Render from Batch to a Write File node, making sure OpenEXR is the selected File Format and Compress is enabled.

NOTE To avoid having to manually define this variable, add this to the `.cshrc` file located in the home of the application. To be able to use the modification, you can either open a new shell or source the `.cshrc` with the following command: `source .cshrc`

Render List Settings

Use the Render List to manage output and export processes. You can set up several output or export processes, specify the frames at which to start and end a process, as well as set job priorities.

You can execute jobs immediately or at a later time using a Batch script. You can attach scripts that convert processed images to a movie file (*.mpg*) or that send messages about the status of Batch processes. The jobs that appear in the Render List are saved in the Batch setup file.

The Render List also displays some settings you defined for a specific Render or Write File node. For example, it lists the destination you specified for a processed output as well as the type of output.

To access the Render List, click the Render List button. You can switch between the Render Node or Write File node menu and the Render List as needed. A horizontal scroll bar is available when the Render List table exceeds the screen width.

List Filtering Settings

Nodes box Select whether to activate or deactivate all Render and Write File nodes in the Render List.

List box Select whether to render only visible Render and Write File nodes in the current Render List view, or include Render and Write File nodes nested in groups.

Render List Columns

Process Indicator column Activate individual Render and Write File nodes for processing. Click an indicator to activate or disable one job at a time.

The arrow in the P column is yellow when a job is active, grey when disabled, and red when there is a processing error.

Node Name column Displays the name for the processed clip. Click a Node Name field to rename the clip.

Start and End frames columns Specify the range of frames you want rendered in a processed result. Corresponds to the Range From and To fields in the Render or Write File node settings.

For example, one node could be set to process the first 30 frames of a 60-frame clip, and another node could process the remaining 30 frames.

NOTE The Start and End fields override the total number of frames in the timebar. For example, if the timebar shows a 60-frame clip and a node is set to start at frame 20 and end at frame 100, then 80 frames are processed.

Destination column Displays the destination for the output or exported result. Use the Output Destination box in the Render or Write File node settings to change the destination.

Priority column Displays the priority of the jobs to be processed. If two processing nodes are connected to the same process tree, they have the same priority.

If there is a processing error, for example, if the Render or Write File node is not connected to a process tree, the job is deactivated and the priority is automatically set to 0.

Script column Attach a script file whose content is executed when the Render or Write File node is finished rendering. Scripts must have the file extension *.bscript*.

Click this field to load an existing script from the file browser. See the Script Reference section below for more information.

Source column Generate your output with a source timecode. Corresponds to the Source Timecode field in the Render or Write File node settings. To edit the timecode, double-click the field and enter a new value.

Record column Generate your output with a record timecode. Corresponds to the Record Timecode field in the Render or Write File node settings. To edit the timecode, double-click the field and enter a new value.

FCM column Displays the framecode mode. Change this value using the Frame rate box in the Render or Write File node settings.

Tape Name column Displays the tape name specified in the Render Node settings. Editable.

Type column Displays the output type of clip to be processed: RGB, RGB-A, or Stereo. Corresponds to the Export Format box in the Render or Write File node settings.

Resolution column Displays the resolution of the render: working (the current proxy setting) or full. Click to toggle.

Script Reference

Batch scripts are ASCII files that contain instructions related to a Batch job. They must have the file extension *.bscript* for Batch to recognize them. A Batch script is executed after the Render or Write File node to which the script is associated has finished rendering.

NOTE Use the *imgview* and *movie* scripts with the Read File node, and use the *listVariables* script with the Render or Write File nodes.

You should consult with your system administrator when creating scripts. Scripts are created using either C shell, Korn shell, or Perl. Also, Batch scripts must have the proper file access, directory access, and execution permissions for the files, directories, and applications in the script.

When the script is executed, Flame continues with its own processes and does not wait for the script to terminate. Flame imposes no restrictions with executing scripts; however, make sure that no one else is using the system when a script is executed and that the script is approved by your system administrator.

Script variables that you can set are defined in the following table.

Variable	Description
BATCH_NODE_NAME	Corresponds to the name of the Render or Write File node. This is the same name that you provide in the Render List Node Name field.
BATCH_FIRST_FRAME	Corresponds to the first frame displayed in the Render List for the selected node name.
BATCH_LAST_FRAME	Corresponds to the last frame displayed in the Render List for the selected node name.
BATCH_ASPECT_RATIO	Corresponds to the framestore aspect ratio.
BATCH_FRAME_RATE	Applies only to scripts associated with Write File nodes. This is the frame rate of the final clip result.
BATCH_FRAME_WIDTH	Applies only to scripts associated with Write File nodes. Corresponds to the width of the final clip result.
BATCH_FRAME_HEIGHT	Applies only to scripts associated with Write File nodes. Corresponds to the height of the final clip result.
BATCH_OUTPUT_DST	Applies only to scripts associated with Render nodes. Indicates the destination of the final clip result as either the Desktop or Media Library.
BATCH_BIT_DEPTH	Applies only to scripts associated with Write File nodes. Specify the bit depth of the final clip result.
BATCH_CROP_WIDTH	Applies only to scripts associated with Write File nodes. Specify the crop width of the final clip result.
BATCH_CROP_HEIGHT	Applies only to scripts associated with Write File nodes. Specify the crop height of the final clip result.
BATCH_FILE_FORMAT	Applies only to scripts associated with Write File nodes. Indicates the numbering format of a sequence of exported images in the final clip result. This numbering sequence ensures that the order of frames in the clip is intact when you import the clip.

Variable	Description
BATCH_FILE_EXT	Applies only to scripts associated with Write File nodes. Specifies the format to which the final clip result is being exported: Alias *(.als)*, Cineon, Dpx (Spirit), Jpeg *(.jpg)*, Pict *(.pict)*, Pixar *(.picio)*, Sgi *(.sgi)*, SoftImage *(.pic)*, Targa *(.tga)*, Tdi/Maya *(.iff)*, Tiff *(.tif)*, or Wavefront *(.rla)*.
BATCH_FILE_DIR	Applies only to scripts associated with Write File nodes. Indicates the file location of the final clip result.

About BFX Output Nodes

A Batch FX pipeline always terminates with a BFX Output node. This node is created automatically when entering Batch FX from the timeline and cannot be deleted.

Both RGB and alpha results can be generated from a BFX Output node to the timeline.

Since an effect created in Batch FX is a setup applied directly to one or more timeline segments, BFX can be rendered directly in Batch FX or upon returning to the timeline. If rendered in Batch FX, a clip may be previewed by clicking the Player button.

Naming Output Nodes

The BFX Output node and Batch FX inherit the timeline segment name by default. To help organize outputs, an Output node can be automatically named as any output socket in the process tree. To do this, select the Output node in the schematic, then press N and click any node output: the name of the Output node is updated accordingly. To change the Output node name or to change the name of the original output socket, simply redo this naming process.

Renaming the BFX Output node automatically renames the segment at the timeline level as well as the Batch FX in the Media panel. Renaming the timeline segment similarly updates the BFX Output node and the Batch FX.

NOTE Once the Batch FX name is explicitly modified, the timeline segment and BFX Output Node names are no longer automatically updated.

Render and Write File Nodes in BFX

Render and Write File Nodes are also available in BFX. Use the Render or Write File nodes to render portions of BFX trees (intermediate renders) and generate new media.

NOTE While a Write File or Render node may be connected anywhere in the BFX tree, it is ultimately the BFX Output node that is patched back into the timeline. This behaviour cannot be bypassed.

Outputting an Alpha to the Timeline

Alpha results may be outputted in addition to the RGB result from a BFX Output node. Connect the RGB result to the front input of the BFX Output node, and the alpha result to the matte input of the BFX Output node. The output node now displays BFXa. Both results are updated on the timeline.

A Comp Timeline FX is automatically added on output, provided that the Add Comp On Matte Output button in the Batch FX section of the Preferences menu is enabled (default).

Output Node Settings

Double-click the BFX Output node in the schematic to see its settings.

Basic Settings

The settings in the Basic menu are non-editable, and are available so you can see the output resolution and timecode of the timeline segment you entered Batch FX with.

Width field Displays the width resolution of the output clip. Non-editable.

Height field Displays the height resolution of the output clip. Non-editable.

Frame Depth field Displays the output frame depth. Non-editable.

Aspect Ratio field Displays the output aspect ratio. Non-editable.

In Timecode field Displays the in point timecode of the output clip. Non-editable.

Out Timecode field Displays the out point timecode of the output clip. Non-editable.

Duration Timecode field Displays the timecode duration of the output clip. Non-editable.

Resize Settings

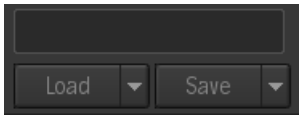
The Resize settings for the BFX Output node are only available if you attach a clip to the pipeline that is a different resolution than the original BFX output. In this case, the Resize settings are active, but you are not able to change the Destination settings (these must remain the same as when you entered BFX from the timeline).

RGB LUT Settings

Similar to the Resize settings, The RGB LUT settings for the BFX Output node are only available if you attach a clip to the pipeline that is a different resolution than the original BFX output.

Saving and Loading Batch or Batch FX Setups

Batch or Batch FX setups can be saved and loaded. When saving a setup, there is the option of saving the complete Batch or Batch FX setup, or only the selected items in the schematic. Use the drop-down list beside the Save button to make a selection before clicking Save.



When loading a Batch setup, there is a choice of loading (replacing any existing nodes), or load and append.

NOTE A preference exists (in the TL FX / Batch / Batch FX tab) to also save a setup file when saving a Batch iteration. This preference is enabled by default.

In Batch or Batch FX, specific node setups can be saved and loaded. The node load and save buttons appear next to the node name field when a node is selected in the schematic.

Batch or Batch FX setups can be renamed directly above the Load and Save buttons.

Clips, Media, and Batch Setups

A Batch or Batch FX setup contains information for the clips used, and the path to the media files each clip references.

This means that when you restore setup, Flame uses this information to first try and locate each clip in the Media Panel. If a clip is not available in the Media Panel, then Flame uses the clip's included media path to locate the media and relink it to the clip.

If either method works, the clip node in the Batch schematic displays the expected media.

If both methods fail, the clip is replaced by a black slate in the schematic.

NOTE This black slate also replaces imported and generated media.

Limitations

- Cache and proxies are not saved with the Batch or Batch FX setup. Manually cache or proxy the clips in the restored setup; use the contextual menu on each clip node.
- For the clips to properly relink to their media, the same media path must be available to the workstation loading the Batch or Batch FX setup.
- Batch or Batch FX setups from versions prior to 2017 Extension 1 only contain references to the clips, not their media. The setups will only restore clips that are present in the Media Panel, and cannot try to relink using the media path since that information is not present.

Saving Work using Batch or Batch FX Iterations

Batch and Batch FX offers the option of toggling between previously saved states known as iterations. Iterations are dynamic snapshots of the Batch environment that are used to save and share specific stages or versions of ongoing work. The advantage of creating Batch iterations instead of saving and loading setups is that all related sources are included. In addition, each new iteration can be automatically incremented using user-defined naming conventions.

The visibility of the Iterations folder can be turned on and off. In the Media panel, right-click Batch and select either Show Batch Iterations or Hide Batch Iterations to toggle, or use the Show Batch Iterations button in General Preferences (page 1936). Another iteration-related preference exists (in the TL FX / Batch / Batch FX tab) to also save a setup file when saving a Batch iteration. This preference is enabled by default.

Creating Iterations

To create a Batch iteration:

1. Click the Batch tab.
2. Select the relevant Batch Group in the Media panel.
3. In the Batch Iteration Name field, add any other desired naming conventions by selecting tokens from the Add Token box.

   Additional text may be entered directly and existing tokens can be manually deleted. To access the Rename Batch dialog where the resulting name may be previewed, `Shift`-click the Batch Iteration Name field.

   NOTE <batch name> is a mandatory token and cannot be deleted.

   Default incremental naming conventions can also be set for the entire project. For more information, see Default Batch Iteration Name in Timeline FX / Batch / Batch FX Preferences (page 1944).
4. Click the Iterate button.

In Batch, a new iteration is created in the Iterations folder of the current Batch Group, with all associated reels and sources duplicated. The Batch Group in the Media panel shows the corresponding name.

In Batch FX, the iterations are not directly attached to the Batch FX structure. A Batch Iterations library is created in the Libraries section of the Media panel, and a Batch group containing the iterations is created for every Batch FX name. Batch FX iterations are also shown when the BFX tab of the Media panel is selected.

To save a Batch iteration with an associated setup file, first enable Save Setup File in the Timeline FX / BFX tab of the Preferences menu. The setup file is saved in the default Batch folder *~/batch/flame* in a subfolder called Iterations. Note that these setup files are not managed by Flame. If the iteration is later deleted, the corresponding setup file will not be deleted automatically.

IMPORTANT Batch iterations are saved in the parent Batch. Clearing Batch also clears existing Batch iterations, just like clearing reels. Batch iterations and reels are not saved until the Desktop is saved from the menu at the bottom-right of the screen. *Always make sure to save the Desktop in order to save all work.*

To create additional Batch iterations:

1. Click the Batch tab and select the Batch to be iterated.
2. Click the Iterate button.

   The Create Batch Iteration dialog box appears.
3. Select Replace to overwrite the current iteration or Increment to create a new iteration with an incremented name.

   TIP Alt-click the Iterate button to replace the current iteration without opening a confirmation dialog box.

Restoring an Iteration

Whenever there are multiple iterations available in Batch, the Iterate button displays a blue arrow, allowing users to toggle between them.

To toggle between Batch iterations:

1. View the Batch Schematic.
2. Click the blue arrow on the Iterate button and select the desired iteration from the list.

   The Replace Batch Setup dialog box appears.
3. Select whether to Append or Replace the existing setup. The iteration is restored, appearing as the currently-active Batch group.

   NOTE The Replace option overwrites the current schematic, causing the loss of any unsaved changes in the current setup.

Batch iterations can also be appended or replaced in a number of other ways:

- While in Batch or Batch FX, drag and drop an iteration to the schematic, and choose whether to replace or append.
- From anywhere in Flame where the Media panel is available, drag and drop an iteration over the Batch group. A Delete All dialog box appears and, upon confirming, the Batch tab is selected automatically.
- From anywhere in Flame where the Media panel is available, right-click a Batch iteration, and select Replace Current Batch Setup or Append to Current Batch Setup. A Delete All dialog box appears and, upon confirming, the Batch tab is selected automatically.

- You can set a custom value for an Iteration using the "Iterate As..." option located in the "Iterate" button's drop down list. This option enables you to:
  - Overwrite a previous Iteration.
  - Set a custom value so that the first iteration in your group can have a value other than 1.

NOTE Using any of the above methods while in Batch FX only allows you to append to your existing Batch FX setup.

Keyboard shortcuts also exist to help you save iterations:

- Use the shortcut Save Current Batch/BFX Iteration to save the current Batch Iteration to the destination folder in either Batch or Batch FX.
- Use the Save Desktop/Reel/Clip/Batch Under Cursor shortcut to save the current Batch Group to the destination folder when used in Batch, and to save the current Iteration to the destination folder when used in Batch FX.

Using a Create BFX Clip for Caching or Versioning

Use Create BFX within Batch or Batch FX to create a BFX clip that can be useful for disk caching and offer the flexibility of re-using or versioning complex pipelines.

A BFX clip is a nested group or container that has benefits over Group nodes in Batch or Batch FX.

Creating a BFX Clip

To create a BFX clip, right-click a node in the schematic, and select Create BFX. A new clip node is created that includes the selected node and all downstream inputs node to the left. Old nodes can be deleted and the new BFX clip connect to the pipeline.

Batch FX clips preserve the Batch schematic reel structure in their Batch FX setups.

Disk Caching Capabilities

To improve performance, switch to the Batch or Batch FX Timing view, select a BFX clip in the timeline, and click Render. Batch or Batch FX Once the complete Batch or Batch FX pipeline is ready to be rendered, the BFX clip does not need to be re-rendered if unmodified.

Flexibility

Another benefit of using a BFX clip is the flexibility it offers for versioning or re-using a clip. Use the offset feature in the Timing view to offset the start time of the BFX clip, or even slide or trim the media and animation keyframes contained within the clip. Created BFX clips reside in the Batch or BFX Sources folder in the Media panel. The clip can be copied into another folder in the Media Library, and re-used in another timeline segment or project.

Exploding BFX Clips

To return a BFX clip to its original separate nodes, right-click the clip, and select Explode BFX. This action is non-destructive, but does discard any rendering.

Because a BFX clip can be nested inside another BFX clip, the explode operation only explodes to one level. In the Basic menu for the BFX clip, there is an option to Explode All levels of the clip back to its original nodes.

Using the Timing View to Offset Clips

When you offset a clip, you simultaneously offset all its segments on each track. You can also offset multiple clips simultaneously, including all clips in a group.

To offset clips:

1. To access the Batch or Batch FX Timing view, click the Timing button.
2. Select the clips and/or segments to offset. If you are offsetting all clips in a group, select one clip or segment in the group and then click Select All Clips in Group. To select all clips on a track, click the clip proxy to the left of the track.
3. Drag left or right in the Offset field, or enter the number of frames by which to offset the clip.

All selections are offset by the same amount.

NOTE If you offset multiple clips simultaneously, the value in the Offset field resets to 0 when the offset is complete.

About the Timing View

The Timing View displays the timeline of all Batch or Batch FX clips in one view. This view is especially useful for edits where you want to see the relative position of all clips in time. For example, offset multiple clips or edit one clip in relation to the others.

You can adjust the timing of any number of clips as well as perform basic editing operations such as slipping, sliding, or trimming. You cannot perform editing operations that involve combining clips or inserting frames from one clip into another.

Edit a clip gesturally, or use same buttons and keyboard shortcuts as you would on the timeline.

To access the Timing View, click the Timing button. All clips in the current schematic, including clips that are part of a group node, are represented by a clip proxy and primary video track. Gaps are displayed based on Media settings, which are set in the clip's Basic menu.

You can select multiple clips on different tracks to perform timing operations at the same time. Selected clip proxies and tracks are highlighted by an orange bounding box.

Timing View Settings

Some Timing View settings relating to trimming, slipping, or sliding clips (such as the Ripple and Snap buttons), or to the actual timeline itself (such as the Timeline Options menu) are exactly the same as the main Flame timeline.

The following settings are specific to the Timing View:

Use Start Frame button Click to offset the selected Batch clip so that its source frame matches the timeline's timecode. This allows you to also offset based on header timecode. Not available in Batch FX.

Offset field Displays the number of frames by which selected clips and/or segments are offset. This same setting can be found in the Timing View as well as the Clip Settings. Editable.

Filter box Select the type of clips to display in the Timing View.

Select:	To display:
Show All	All clips in the schematic.
Show Tree	All clips in the selected processing tree.
Show Branch	All clips in the current node branch.
Show Selected	All clips and nodes selected in the schematic.

Select All Clips in Group button Click to select all clips that are part of a group. First select one clip or segment in the group.

Timing View Render box Select an option and click to render selected Timing View clips. Also available as a contextual option when right-clicking a clip node.

Caching Frames

You can avoid reprocessing the same nodes and clips as you navigate in the timebar by specifying which nodes and clips should be cached. Cached frames are stored on the system framestore, increasing system performance and efficiency. Stored cache clips are contained in a _Cache_ library and correspond to the clip IDs in the setups. These cache library files are named according to setup and project name.

Node and clip caching is enabled in the Cache section of the Batch or Batch FX Preferences menu. The Cache menu also displays the number of rendered frames in the cache for the selected nodes or clips, and the remaining space on the framestore.

Any clip node can be cached as long as it is edited, resized, or has had a LUT conversion applied. Batch setup files include the location of any cached frames, which are stored with the clip ID until removed. The history associated with a clip can also be cached, but not if the clip was rendered in Proxy mode.

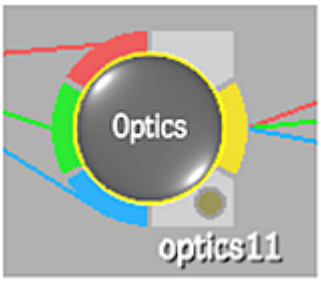
Each node and clip has a cache tab on the lower-right corner.

The following steps are not required to actually cache frames. Typically, you use intelligent caching, which automatically caches frames as you move through the timebar. Caching occurs as the frame is displayed and the node operation is applied. A cached node automatically saves frames as you display—or visit—them and then reloads them each time you visit the frame again. Your workflow speed is increased.

(a) Cache tab

Read and write mode Cache enabled

Read-only mode Frames already cached are accessible

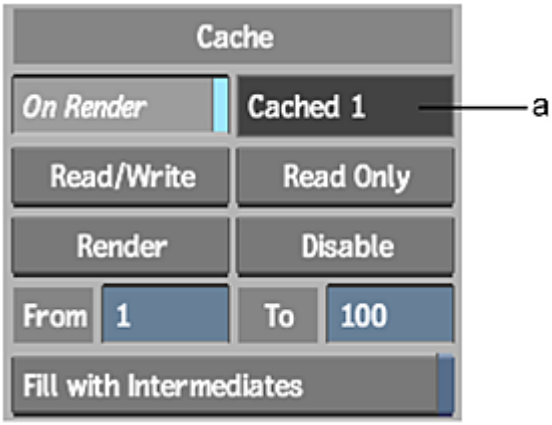


Cache disabled

Click the cache tab to activate any of the following modes.

Mode	Colour	Description
Disabled	Grey	Disables the cache. You can also `Alt`-click to disable and remove cached frames for the selected node.
Read and write	Yellow	Saves and loads data to and from the cache.
Read-only	Amber	Is shown with a yellow outline. No data can be saved to the cache, but rendered frames remain stored. Use this mode to prevent the framestore from getting full.

Changing or cutting an input or any node further along the process tree that has cached frames clears the cache buffer. The Cache section of the Batch or Batch FX Preferences menu (Rendering tab) shows how many frames are cached for the selected node and the number of frames available on the framestore. This allows you to clear the cache of the selected node or all cached frames.



(a) Cached frame counter

To cache frames:

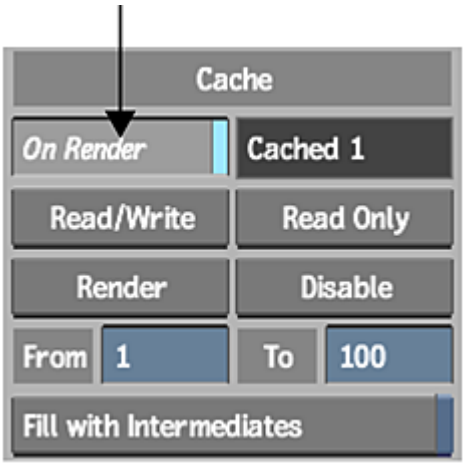
1. Specify the frames you want to render and store in cache using the From and To fields.
2. Click Render.

   The cached frames are stored in the framestore and added to the cached frame counter.

To turn off the cache and remove cached frames, `Alt`-click the cache indicator for the selected node.

To render node caches while rendering:

1. Enable the cache for any nodes or clips whose frames you want to cache.
2. Enable On Render.



3. When you are satisfied with current settings in your process tree, click Render.

   The clip is rendered, and all node caches are rendered. While rendering, the message bar displays the number of frames in the node and the time remaining until the render is complete.

Grouping Nodes

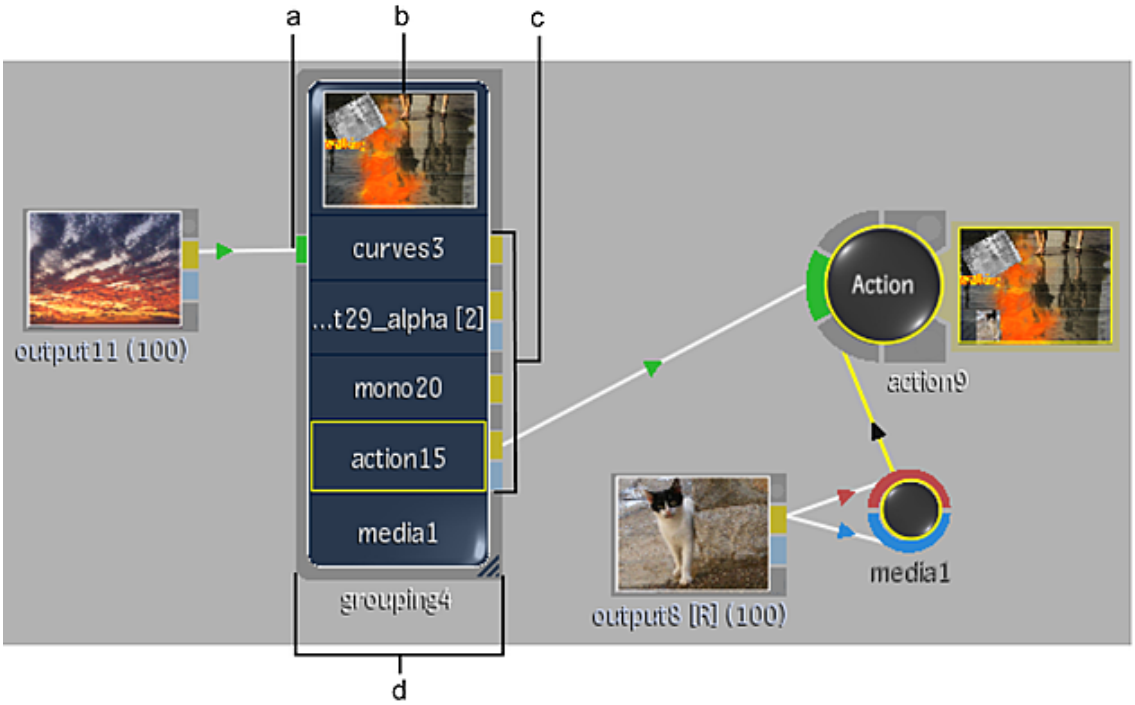
To group nodes:

1. `Ctrl`-drag to select the nodes you want to group.

   Selected nodes are outlined in white.
2. Click Group, or right-click the selection, and select Group.

   The selected nodes collapse into a Group node. The Group node lists the nodes contained in the group, as well as the non-hidden input tabs and output tabs.

   TIP You can also create a group inside a group.



(a) Input connection (b) Proxy window (c) Output connections (d) Group icon

3 To rename the group, enter a name in the Node Name field.

4 Click the group node to display group display and editing settings.

To display a proxy of an output in the proxy window in the group node:

1 Do any of the following:

- Click the field in the Proxy list that corresponds to the node whose output you want to display.
- Press the `Shift+up or down arrow` as you navigate through the node's Group List.
- If a node has multiple outputs, press `Ctrl+Shift+up or down arrow` as you navigate through the node's Group List to display all the outputs of a node.

To change the size of the group icon:

1 Drag the lower-right corner of the icon to make it wider or narrower.

Editing a Group

Use the Edit Group controls to edit the contents of a group.

To edit a group:

1 Select the group you want to edit.

2 In the Group settings, click Edit.

The nodes in the group appear in the schematic ungrouped, with other schematic nodes grayed out.

TIP Instead of editing the group, you can access a single node's menu by selecting the node's name from the Node List box. Click the Return to Group button to return to the Group List menu.

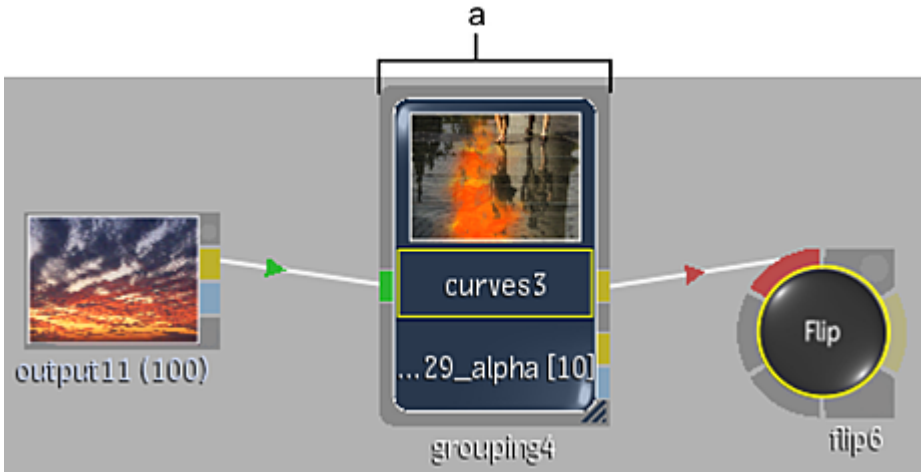
3 Modify the nodes as required.

4 Click Exit Group to return to the previous schematic.

NOTE Click Ungroup to expand the group of nodes to their pre-grouped positions in the schematic.

About Group Nodes

A group node is a selection of nodes and clips that are collapsed into one node icon. Each node icon displays the group's contents, inputs, and outputs. You can create groups for different branches of your process tree and work on each separately. Groups are useful for simplifying cluttered schematics.



(a) Group node

You can create several groups and work on each group separately. For example, group a Keying and Colour Correction branch separately from an Action and a filtering branch, and then work on each branch independently. If you need to edit the nodes in a group, you can expand the group and make the necessary modifications.

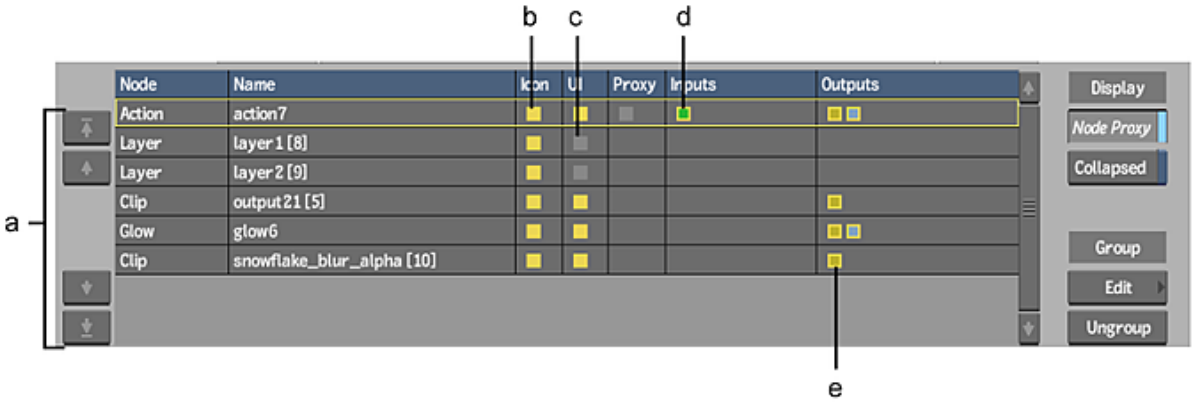
A clip with multiple outputs, (a stereo clip with left and right matte outputs, for example) can be displayed as a group node, allowing all output tabs to be displayed for connection.

Stereo nodes are displayed as group nodes, with inputs and outputs for the left eye and right eye.

Group Settings

Customize the group node using the Group List menu. The Group List menu lists all the nodes and connections contained in a group. You can rename and hide the contents of the group, as well as define which input and output connection sockets are visible and available for connection in the schematic.

You can select a node in the Group List menu to display the group at the selected node's stage. The View box must be set to Result, Front, or Back to use this display option. You can also display a preview proxy of a node in the group.



(a) Sort Order buttons (b) Enabled icon visibility (c) Disabled UI visibility (d) Enabled input visibility (e) Enabled output visibility

Use the Group List menu as follows:

Click:	**To:**
Sort Order buttons	Move the selected node one position up or down (click the single-arrow buttons). To move the selected node to the first or last position in a group, click the arrow-line buttons.
Name	Change the node's name.
Icon	Toggle the node's visibility in the group. Icons in the Group List are yellow when the node is visible and grey when hidden.
UI	Toggle the node's availability in the Node List box. Icons in the Group List are yellow when the node is displayed in the box and grey when hidden.
Proxy	Display a proxy of the corresponding node in the proxy window. Icons in the Proxy list are white when locked and grey when unlocked. Lock an icon when you do not want its proxy in the window to change as you navigate the Group List.
Inputs	Hide or unhide the selected node's input sockets. Sockets are colour-coded with the same scheme as nodes that are not part of a group. Icons in the Group List have a yellow border when the input socket is visible.
Outputs	Hide or unhide the selected node's output sockets. Sockets are colour-coded with the same scheme as nodes that are not part of a group. Icons in the Group List have a yellow border when the output socket is visible.

Node Proxy button Enable to display a preview proxy of a node in the group icon. To change the proxy, press Shift + up arrow or down arrow as you navigate through the Group List.

Collapsed button Enable to collapse the group icon so that only the preview proxy is displayed.

Edit button Click to display the schematic of the group contents, with all other nodes greyed out. To return to the Group List menu, click EXIT Group.

EXIT Group button Click to return to the Group List menu after editing.

Ungroup button Click to expand the group of nodes to their original positions in the schematic.

Node List box Displays the menu of the selected option. Select List or click the Return to Group button to return to the Group List menu.

Return to Group button Displays the Group List menu.

Mimicking, Copying, and Duplicating Nodes

When working in the Batch or Batch FX schematic, you have a few options to help you quickly get settings, or even complete nodes, to and from similar nodes. See the procedures below for information on how to access each of these options, but as a quick summary, here are the reasons for using each option:

- Mimic Link: You can keep two nodes in the schematic in sync by creating a Mimic Link between them. When you change the settings of one node, they are automatically mimicked in the other node.

  NOTE To mimic nodes, the nodes must be of the same type. For example, you can link between Flip nodes but not between a Flip node and a Colour Correct node. Some nodes, such as Action or Modular Keyer do not support the Mimic Link option. In this case, you can see an error message in the message bar.

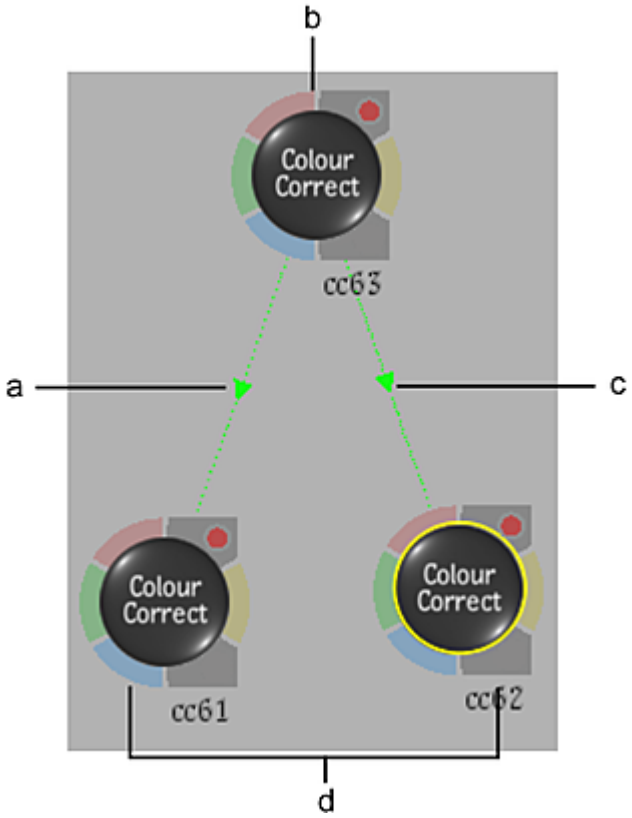
- Duplicate: Create another instance of a node or group of nodes within the same Batch or Batch FX schematic. All node information is duplicated as well.
- Copy: Copy is similar to Duplicate, except that the selected nodes or group of nodes is copied into the clipboard, so you can paste exactly where you want in the schematic. An added bonus of using Copy and Paste instead of Duplicate, is that you can paste a copied node or nodes into a different Batch or Batch FX schematic. You can even copy a Batch node and paste it into a Batch FX schematic or vice-versa.

Using Mimic Link

To create a Mimic link between nodes connected to mono clips:

1. From the Tools box, select Mimic Link.
2. In the schematic, drag between nodes of the same type.

   A green-arrowed dotted line indicates that the nodes are linked as duplicates. The direction of the arrow indicates which node is the master.



(a) Duplicate link (b) Master node (b) Duplicate link (d) Duplicated nodes

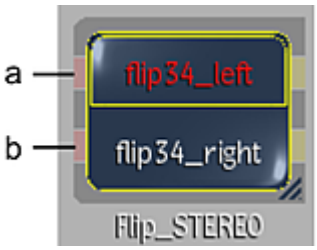
3 Connect each node to a mono clip.

4 Change the settings of any node.

To work with Mimic link using a stereo group node:

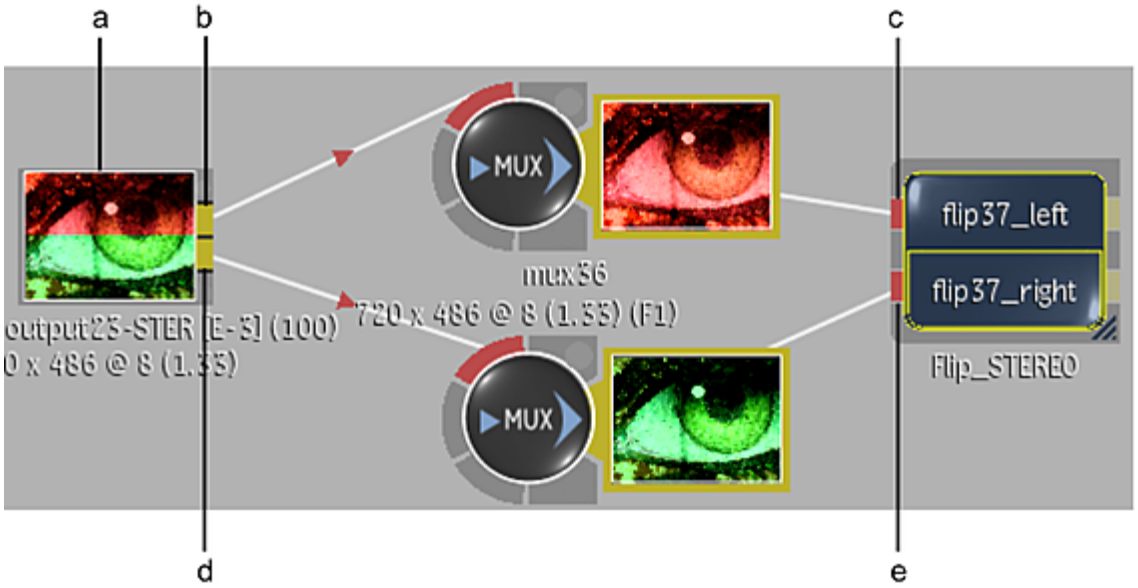
1 With Stereo Mode selected, drag a node from the node bin to the schematic.

A stereo group node containing left eye and right eye inputs appears in the schematic.



(a) Left eye input (b) Right eye input

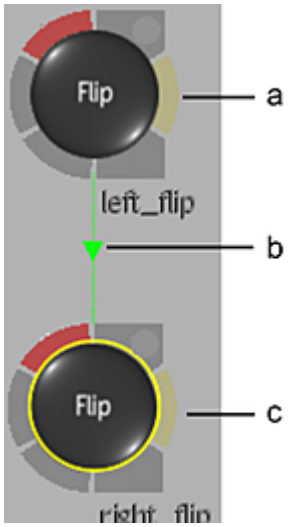
2 Connect the left and right eye inputs of the stereo group node to the clip node's left and right eye outputs, as shown in the following example.



(a) Clip containing stereo track (b) Left eye output tab (c) Left eye input tab (d) Right eye input tab (e) Right eye output tab

3 Double-click the stereo group node or click Edit in the Group List menu.

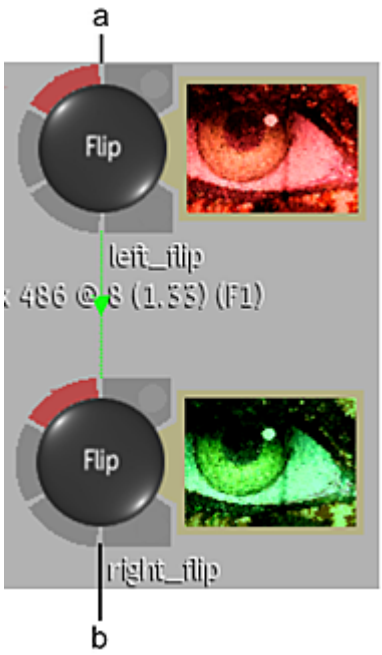
The schematic is dimmed except for the two nodes of the same type as the one you dragged from the node bin. A green-arrowed dotted line indicates that the nodes are linked as mimics.



(a) Master node (b) Mimic link (c) Mimicked node

4 Change the settings of either node.

Notice that the changes made to one node are mimicked on the other node so that both the left and right eye clips are affected simultaneously.



(a) Flip node connected to left eye is edited (b) Settings are automatically mimicked on Flip node connected to right eye

5 Click Exit Group.

Duplicating Nodes

To duplicate nodes:

1 In the schematic, select a node, or group of nodes.

2 Click Duplicate, or right-click the selection, and choose Duplicate.

A duplicate of the node or nodes (with all node settings) appears in the schematic.

Copying Nodes

To copy nodes:

1 In the schematic, select a node, or group of nodes.

2 Right-click the selection, and choose Copy.

A copy of the node or nodes (with all node settings) is added to the clipboard.

3 Navigate to the location in the schematic (or another Batch or Batch FX schematic) where you want the copied node or nodes to reside, right-click and choose Paste.

A copy of the node or nodes (with all node settings) appears in the schematic.

Customizing the Batch or Batch FX Tools Bin

Create custom bins and populate them with your most commonly used nodes to optimize your workflow. As well, change the order of the tabs along the top of the bin and rename them to reflect the contents of a bin.

NOTE The All Nodes and Matchbox tabs cannot be customized or deleted.

To create a tab:

1 Click the plus sign tab.

2 Name the tab in the keyboard that appears.

NOTE You can create a maximum of 6 new tabs.

To copy a node to another bin:

1 Drag the node on top of the destination tab.

2 Release the cursor.

3 Click the destination tab.
The copied node appears in the bin. Nodes are added to the end of a bin in the order copied.

NOTE Nodes cannot be duplicated within the same bin.

To move a node to another bin:

1 Press `Alt+Win` and drag the node on top of the destination tab.

2 Release the cursor.

3 Click the destination tab.

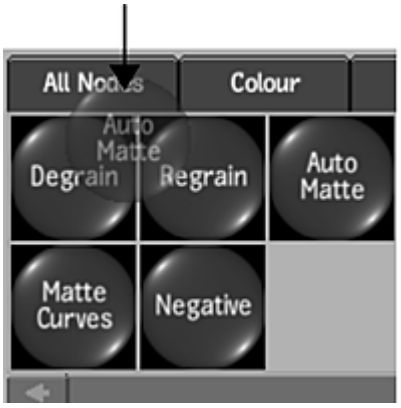
The node is moved from its original location to the destination bin. Nodes are placed at the end of a bin in the order moved.

NOTE Nodes cannot be duplicated within the same bin.

To reorder a node within a bin:

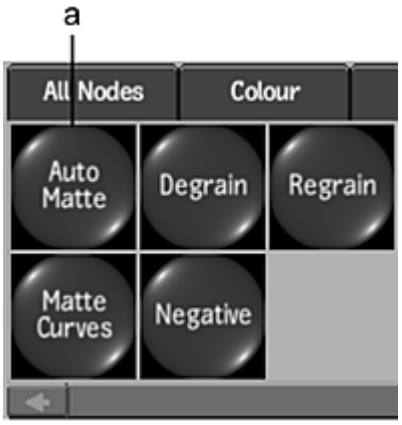
1 Press `Alt+Win` and drag the node to a new location. You can move nodes from one row to another as well as reorganize nodes within a row.

In the following example, the AutoMatte node is dragged on top of the Degrain node.



2 Release the cursor when it appears where you want the node moved.

If you dragged the node on top of an existing node, the existing node shifts to the right and the moved node is inserted in its place. In the following example, the AutoMatte node is inserted in the place of the Degrain node, and the Degrain and Regrain node shift to the right.



(a) Reordered node

NOTE Click Sort to reset a tab to alphabetical layout

To delete a bin:

1 Press `Alt+Win` and drag the tab to the bottom of the screen.

2 Release the cursor when it changes to a delete cursor.

WARNING There is no undo capability when deleting a bin.

The entire contents of the bin, including the tab, are deleted.

To delete a node from a bin:

1. Press `Alt+Win` and drag the node to the bottom of the screen.
2. Release the cursor when it changes to a delete cursor.

   WARNING There is no undo capability when deleting a node.

To reorder a tab:

1. Press `Alt+Win` and slide the tab to its new location.
2. Release the cursor when it changes to a green crosshair at the new location for the tab.

   If you dragged the tab on top of an existing tab, the existing tab shifts to the right and the moved tab is inserted in its place.

Matchbox Node Bin Tab

To help you quickly access Matchbox effects, a dynamic node bin is available that behaves slightly differently from the other node bin tabs. Keep in mind the following:

- The contents of the Matchbox node bin tab is dynamically generated to display all Matchbox shaders that are housed in a folder of your choice. You can set the folder location in the Shader Paths tab of the Preference menu. The Default folder selection is the folder where the presets are housed: *opt/Autodesk/presets/<product_version>/matchbox/shaders* (*.mx* or *.glsl* shaders appear in the node bin). Click the Browse button to select a different folder location.

  TIP You can also set the folder location in the FlameSetup utility, under the Shader Paths tab. By setting the folder location here, the default path is available for all projects.

- If new shaders are added to the selected file location, click the Refresh button under the node bin to refresh the dynamic display.
- Similar to the All Nodes tab, the Matchbox tab cannot be customized or deleted. Also, you cannot move a node from the Matchbox tab to a custom node bin or tab.

About Batch and Batch FX Node Bins

In Batch and Batch FX, a node is a graphical representation of a function or effect that affects a clip or another node in the process tree. The available nodes are organized in bins that contain all nodes required for building a process tree. The nodes are divided into the following three groups, classified by tabs.

To access the Batch or Batch FX node bins, click the FX Nodes button. To add a node from the node bin to the schematic, double-click a node, or drag a node directly to the schematic. Enable Stereo Mode to add stereo nodes to the schematic.

I/O Bin

Use the nodes in the I/O bin to load clips into Batch or Batch FX or to output rendered clips. Render and Write File Nodes are available in Batch FX. You can use the Render or Write File nodes to render portions of your Batch FX trees (intermediate renders) and generate new media.

NOTE While you are free to connect a Write File or Render node anywhere in your Batch FX tree, it is ultimately the BFX Output node that is patched back into the Timeline. This cannot be bypassed.

Node	Description
Import	Import clips in any supported formats from any available external location .
Render	Output clips to the Desktop; metadata such as clip history and timecode is kept.
Write File	Export image sequences in any supported format.
Desk	Select and load a clip from the Desktop area. Especially useful when you are working with a hidden Media Panel.
Media Panel	Select and load a clip from the Media Panel. The display settings for this Media Panel are independent from those of the regular Media Panel. Especially useful when you are working with a hidden Media Panel.
MUX	The MUX (multiplexer) node is a tool that helps you create cleaner schematics by allowing you to have multiple RGBA inputs feeding your outputs. The selection of the active input can be changed over time, therefore MUX also acts as an animated switcher.
Back Clip	The Back clip node enables you to create composites and effects in Batch FX, when using the Add Adjustment Segment option from the timeline.

You cannot customize the I/O bin.

NOTE Use the Import node to import multi-channel clips in Batch or Batch FX. Avoid using the Read File node: it is there to ensure backward compatibility with previous setups. Also, clip imported through the Read File node cannot be archived, and do not appear in the Media Panel.

Tools Bin

The Tools bin contains effects and formatting nodes classified by tabs. The All Nodes tab contains all nodes except the nodes found in the I/O bin. The other tabs in the Tools bin allow you to create and customize bins.

Some of the nodes in the All Nodes bin are duplicated in other bins according to a preset tab classification. For example, the Gradient node is found in both the Colour and Comp bins.

The nodes in the Tools bin are listed in alphabetical order from top to bottom of each row. You can customize any of the bins and preset tabs in the Tools bin except the All Nodes bin.

Enable the Stereo Mode button below the Tools bin to access only stereo nodes (nodes not available as stereo nodes are greyed out).

TIP Use the scroll bar under the applicable bin to scroll through all available nodes. You can also frame and highlight all the nodes starting with the same letter by pressing the key corresponding to this first letter on your keyboard while the cursor is sitting over a node bin. For example: pressing "K" while hovering the cursor above the All Nodes bin automatically highlights all of the Keyer nodes.

User/Project Bin

The User/Project bin contains custom nodes classified by a User tab and a Project tab. Use this bin to save custom setups per user or project. See Creating Custom Nodes (page 552).

Creating Custom Nodes

Create custom nodes of specific setups that you use often. A custom node can consist of a single node with specific settings or multiple nodes that create a particular effect.

You create custom nodes by dragging individual nodes, groups, branches, or entire trees into the User/Project bin.

To create a custom node:

1. Click the User or Project tab to open the bin.
2. In the schematic, select a node or group of nodes.
3. Drag the selection to the User or Project tab.
4. Enter a name for your custom node.

   The selection is copied to the bin. The original selection remains in the schematic.

To use a custom node:

1. Select a custom node from the User or Project bin. If necessary, scroll through the bin to find the node.
2. Drag the node to the schematic, or double-click the node.

   The same configuration of nodes and clips that was used to create the custom node appears in the schematic.

   You can use custom nodes as often as you like. Each time you drag a custom node to the schematic, a new number is appended to its name.

To manage the custom node bins:

1. Select any of the following from the buttons below the bins.

Select:	To:
Load Project Bin or Load User Bin	Load custom nodes from another project or user. **NOTE** If you load unsupported nodes, the unsupported nodes appear greyed out when dragged to the schematic.
Save Project Bin or Save User Bin	Save the current custom nodes so they can be loaded by another project or user.
Reset Project Bin or Reset User Bin	Delete all custom nodes in the Project or User bin.

Select:	To:
Sort	Place the nodes in the bin in alphabetical order.

To delete a custom node from the User/Project bin:

1. Press `Alt+Win` and drag the node to the bottom of the screen.
2. Release the cursor when it changes to a delete cursor.

   WARNING There is no undo capability when deleting a custom node.

Batch and Batch FX Preferences

Click the Batch Prefs or BFX Prefs button to access settings to customize rendering, preferences, and adaptive degradation in Batch or Batch FX.

NOTE Some Batch or Batch FX nodes also have specific preferences related to the functionality of the node. Click Node Prefs to access the preferences for a selected node.

Rendering Tab

Free Frames field Displays the space remaining on the framestore. Non-editable.

Hardware Anti-aliasing Sample box Select a hardware anti-aliasing sampling level to accelerate edge anti-aliasing with no performance penalty. Available values are dependent on your graphics card.

On Render button Enable to allow node and clip caching.

Current Frame Cache field Displays the current number of cached frames. Non-editable.

Read/Write button Click to set cache to read and write status.

Read Only button Click to set cache to read-only status.

Render Cache button Click to render the cache of selected nodes for the frames defined in the From and To fields.

Disable button Click to turn the caching of selected nodes on or off.

From field Displays the lower limit in the range of frames that can be cached. Editable.

To field Displays the upper limit in the range of frames that can be cached. Editable.

Fill with Intermediates button Enable to fill each node's cache with the intermediate clips contained in the history. If a clip was rendered in proxy mode, the history is not cached.

Preferences Tab

Audio Source box Select the audio context you want to hear when working with another clip. This same setting appears in the Timing View menu.

Text Log button Enable to save rendering status information in a text file, located in /opt/Autodesk/project/<project name>/batch/log.

Status Webpage button Enable to create three status HTML files, located in /opt/Autodesk/html.

Update Numerics button Enable to update numerics in fields as the timebar is scrubbed.

Show Negative Frame button Enable to displays negative frame numbers in the Current Frame field when a segment is offset before frame 1. Disable to display only frames from frame 1 onward.

Clip Info box Select what information to display under clip nodes in the schematic. (P) is also displayed on proxy versions of clips.

Transparency field Displays the transparency of currently unselected nodes in the schematic. Editable.

Auto Parent button Enable to allow Parent and Cut options in the schematic while using the Select edit mode.

Auto Insert button Enable to automatically insert a node when dragged between two connected nodes. When disabled, press Shift to auto insert.

Highlight Path button Enable to highlight the path of the currently selected node in the schematic.

Show Indicators button Enable to display certain setting values in the schematic view (under the node) for some nodes.

Available indicators are:

- MUX: Timing Offset and Range.
- Clamp: Clamp Mode.
- Comp: Blending Mode.
- Coloured Frame: Mode.
- Mono: Channel.
- Render: Range.
- Write File: Range.

Show Adaptors button Enable to display an adaptor icon on the connection between two nodes in the schematic when an automatic bit-depth conversion is applied.

Update Clip box Choose whether to automatically update all clips or only selected clips.

Clear Buffer button Click to clear the Undo buffer of all previous undo operations.

Check for Updated Versions button Checks each imported Read File .clip file to see if it is the latest version, and updates it if that is not the case. Asks for confirmation before updating the version. Only available when a Read File node is used to import .clip files in the schematic.

Adaptive Degradation Tab

Use these settings to temporarily deactivate taxing operations during interactive manipulations. These settings do not affect the final renders.

Degrade Mode box Select how to propagate display degradation in the image, according to settings for supported nodes. Nodes in the pipeline respect this degraded image. In this case, press Preview to see your results with full settings on.

Change Node Degradation box Select whether to enable or disable adaptive degradation before clicking Apply To Selected.

Apply To Selected button Click to set the selected nodes to the chosen option in the Change Node Degradation box.

The following two settings appear in the Node Prefs menu for specific nodes that support adaptive degradation:

Active button Enable to activate adaptive degradation display settings. Use to prevent slowdowns in displaying your results when changing supported settings.

Synchronize All button Enable to synchronize all nodes of the same type with the Adaptive Degradation settings of the current node.

Compositing in 3D Space with Action 17

About Action

Action is a multilayer compositing tool for creating complex visual effects. Use Action to animate clips in 3D and add camera, lighting, and shadow effects.

You create effects and animations by manipulating objects in the scene. Objects you work with include surfaces, light sources, axes, particles, shadows, and the camera.

Action Concepts

The following concepts are used throughout the Action topics to describe the workflow and user interface.

Scene The scene is Action's representation of 3D world space. It is where objects are placed and animated. World space has three directions: X (left/right), Y (up/down), and Z (in/out).

Object An object is any element in a scene. An axis, a camera, a surface, a light, or a model can also be referred to as an object.

Media Media consists of a front and matte clip only.

Surface A surface is a special type of 3D geometry onto which media is mapped.

Geometric Object (Models) A 3D geometric object can be something as simple as a cube, sphere, or cone, or as complex as a character. Although some basic geometric objects are included with Flame, most geometry is imported from a 3D application such as Maya or 3ds Max. A model is another way to refer to a geometric object.

3D Models and Text You can import 3ds Max files, FBX format files, Alembic format files, Wavefront files, Inventor files, and Paint geometry into Action. Also, you can create 3D text with custom beveling.

Camera The camera represents the 'eye' you are using to see the scene. The camera is used to “record” the scene in Action. By default, the camera consists of two objects: the camera eye and the point of interest (the coordinates the camera is 'looking' at).

Axis An axis is the element of an object or media that can be manipulated to determine the object's 2D or 3D space, position, and movement.

Schematic The Action schematic is an icon representation of the scene. It shows all the objects in the scene and their relationship with each other. The Action schematic is different from the Batch or Batch FX schematic (which is more of a process tree view).

Particles Action contains a particle system that can simulate environmental effects such as rain, snow, and many other user-defined effects.

Accessing Action from the Timeline

To access Action from the timeline:

1. Select the Timeline tab.
2. Do one of the following:
   - Click the FX button.
   - Right-click on a segment in the timeline and choose Add Effect.

   The Effects ribbon appears.

   NOTE Effects that are enabled have already been added to the current Timeline FX pipeline.
3. Enable AC (Action).
4. You can adjust the effect's settings using the quick menu underneath the Timeline FX pipeline. Click the Editor button if you want to enter the full editor of the effect. Once in a Timeline FX editor, you can choose which range to display on the timebar (Segment, Media, or Timeline) from the Playback Options box. The Playback Options box also has options to display audio waveforms and cue marks.

   When adding Timeline FX to stereo clips on the timeline, in some cases (such as with the Colour Warper effect), you are not able to use the quick menu to adjust settings. In this case, simply click the Editor button to enter the full editor.

Accessing Action From the Tools Tab

You can access Action from Batch or Batch FX, allowing you to be able to use the Workspace Media panel to add or organize clips and setups; or from the timeline, allowing you to apply the Action effect directly to your ongoing edit. But you can also access Action through the Tools tab, allowing you to create modified clips that you can use as a rendered result in the workspace.

To access Action from the Tools tab:

1. From the Tools tab, click Action.
2. Select an option from the Input Mode box.

Select:	To:
Front Back Matte	Select source clips from the workspace. The clips are selected in the order front, back, and then matte.
None	Enter Action with no media or back clip. From the Resolution menu that appears, choose a resolution, width, height, pixel aspect ratio, bit depth, scan mode, and frame depth. If Action already contains a setup, entering with the None option keeps the objects present in the scene and creates empty media placeholders in the Media list for each object. Press the `Alt` key when you click the Action button to automatically select the None option. Press `Alt` when you select your destination to open Action with all media and nodes deleted and all parameters (except for resolution) set to default.

Select:	To:
Clear All	To reset all parameters (except for resolution), delete all nodes and media, and prompt you to select front, back, and matte clips. Press the `Ctrl+Alt` key when you click the Action button to automatically select the Clear All option.

Although you can use differing resolutions per media, you cannot specify clips of different resolutions to act as the front and matte clip of media (except for 10-bit, 12-bit, and 12-bit unpacked clips, which are interchangeable). An error appears in the message bar if the specified clips are not able to be loaded to the same media.

TIP To standardize some of your clip resolutions, you can resize them before entering .

3 Select the Action front, back, and matte clip. If you selected None from the Input Mode box, proceed to the next step.

You can press `Ctrl` to load multiple fronts and mattes. The `Ctrl` key changes the order of clips to front and matte of media 1, front and matte of media 2, and so on. The back clip and destination are selected after the last media is filled or when the `Ctrl` key is released.

TIP Pay attention to the cursor colours to know which clips you are selecting (front = red cursor, matte = blue cursor, background = green cursor, and destination = white cursor).

4 Select the destination.

Action appears and the selected clips are loaded into the appropriate media. If Auto Image is enabled in the Action Setup menu, objects are automatically added to the schematic.

The composite of the front, matte, and back clips appear in the image window and their names are listed in the Media list. The Total Frames field defaults to the length of the longest clip loaded.

Action Interface Overview

Action is divided into a number of menus and sections designed to help you quickly create your composites.

Accessing Action Menus

The left side of the Action menu houses a number of buttons that allow you to switch between the various Action sections and menus. Clicking any of these buttons opens the specific Action section, such as the Action Bin or Output menu.

Changing Views

You can change the orientation to view a scene from many angles. This can help you set up motion paths, light sources, and camera angles more easily.

From the View box (on the viewing/layout toolbar under the viewports), select an option to set the view in the image window. Most of the View box options have keyboard shortcuts displayed in the expanded list. Some of these keyboard shortcut options are also cycle options. For example, pressing `F5` repeatedly cycles through the animation channels, tracks, and info views.

View	Comments
Schematic	You can toggle ` (on the Tilde key) to alternate between the Schematic view and the previous view.

View	Comments
Priority	Displays the Priority view; you can also display the full Priority Editor menu by clicking the Priority button. See Changing the Drawing Order with the Priority Editor (page 565).
Media List	Displays the Media List view; you can also display the full Media List menu by clicking the Media button. See Action Media (page 606).
Sources Working Analyzer Working	Working views are useful, in that they provide you with the ability to view your scene using a working camera, through which you can experiment without saving anything while preserving your camera setups. When in a Working view, a Reset View button appears at the bottom of the Action menu list so that you can reset any camera movements.
Object Views	An Action Object Solo view (F8 view) is available for a number of objects (such as Lights, Projectors, and GMasks). The Object Solo view allows you to see the scene through the point of view of the selected object, as if it was a camera. When in Object Solo view, you can use camera manipulation tools, such as Dolly, Roll, Tilt, and Orbit (available from the Tools box). **NOTE** You can even use the Object Solo view on a Camera node, allowing you to see the scene through the point of view of the Camera, without actually making the camera active. If you create a Group node, you can use the same F8 shortcut, or select Group Schematic, to display the nodes inside a Group.
Animation	Pressing the Animation keyboard shortcut repeatedly allows you to cycle through the animation Channels, Tracks, and Info views.
3D Views	There are a number of options included in 3D Views: ■ Working ■ Default Camera ■ Front ■ Side ■ Top
Sources	Use the keyboard shortcut to cycle through the Sources options of Result, Front, and Matte. See Action Source Nodes (page 894).
Analyzers	Use the keyboard shortcut to switch between the Analyzer 2D and Analyzer 3D views. See Action Analyzer (page 871).
Inputs	This grouping includes Media Front, Media Back, and Media Matte views.
Outputs	You can switch to any Render Layer output that you set up in the Output menu. If you have more that one output, you can use the keyboard shortcut to cycle between the Render Layers outputs. You can also cycle through selected Render Passes associated with a Render Layer (press 1 to navigate down the list, or press 2 to navigate up the list). See About Rendering Outputs from Action (page 598). See the next section for information about different viewing modes for outputs.

As with other areas of Flame with viewports, you can use the UI items under the image window on the right side to zoom and fit viewports. You can also set defaults in **Preferences ➤ User Interface ➤ Player &**

Viewports. Note that some Action views (such as the schematic and working view) have their own zoom and pan settings that are not altered by the views that use the common zoom and pan settings.

Framing Views

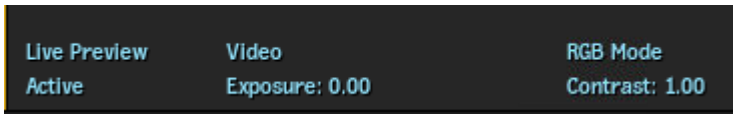
You can easily frame the schematic or 3D Views to zoom and center on a selected object, or frame the scene in its entirety.

- Press `Space+F` to center the schematic or the 3D View on the selected object.
- Press `Space+A` to frame the schematic or the 3D View so it displays every object or node in the scene.

NOTE Enable the **Action Setup ➤ Preferences ➤ Working Cam** button to automatically switch to the Working Camera when framing a view. Framing using the Working Camera prevents you from accidentally editing the rendering camera. Disabling this options ensures you are framing with the active camera.

Live Preview and 3D View

When viewing the output of your Action scene, you have the option of displaying a 3D View or a Live Preview. The current view is displayed at the bottom of the viewing panel, when in a result view. Click the view to switch it, or use the `3` keyboard shortcut to cycle between the two views.



3D View This is the full viewport in screen resolution, and includes items outside of the render area. 3D View does **not** display hardware anti-aliasing or any post-processing effects, such as Camera FX, Lens Flares, Rays, Blooming, and Ambient Occlusion.

TIP You'll see a warning icon

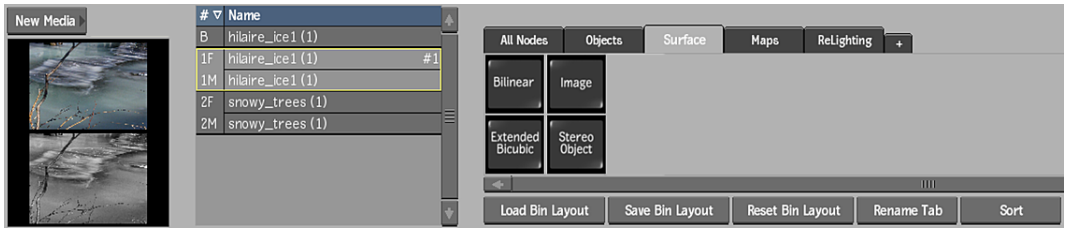
displayed at the bottom right corner of the image window as a visual reminder that you are in 3D View.

Live Preview This is the default view. Live Preview displays a WYSIWYG pixel-based view of the render area. Interactive performance may be better in Live Preview mode, especially when combined with proxy settings. The Adaptive Degradation (page 594) section of the Action Setup menu also includes a number of options that may help interactive viewing in Live Preview mode (such as for Motion Blur and Anti-Aliasing).

Using the Action Node Bin

The Action node bin contains nodes classified by tabs. The All Nodes tab contains all Action nodes. The other tabs in the node bin allow you to create and customize bins. The node bin also contains an image proxy that displays a proxy of the currently selected media, as well as a mini Media list.

To access the node bin, click the Action Bin button.



Some nodes are duplicated in other bins according to a preset tab classification. For example, the Light node is found in both the Objects and ReLighting tabs.

Nodes are listed in alphabetical order from top to bottom of each row. You can customize any of the bins and preset tabs except the All Nodes bin.

To add a node from the node bin:

1. If applicable, select media from the mini Media list.
2. Do one of the following:
   - Drag the selected node (or image proxy) from the node bin and place it in the schematic. If the node is an image type node, an axis and an image are created and linked together.
   - Drag the node (or image proxy) from the node bin and place it where you want it in Result view. For example, as you drag a Light node into Result view, it becomes active, so you can see its effect on the scene before placing it exactly where you want.
   - Double-click a node (or image proxy). Depending on the node, it appears next to the last added object, or is attached to a selected node in the schematic. You do not need to be in Schematic view to add a node in this manner.
   - In Front, Top, or Side views, also known as orthographic views, double-click a 3D Text, Light, or Surface node. The node appears between the last selected camera and the center of the orthographic view.

TIP You can frame and highlight all the nodes starting with the same letter by pressing the key corresponding to this first letter on your keyboard, while the cursor is sitting over a node bin. For example, pressing "L" while hovering the cursor above the All Nodes bin automatically highlights Lens Flare, Lens Texture, and Light nodes.

Customizing the Node Bin

Create custom tabs and populate them with your most commonly used nodes to optimize your workflow. As well, change the order of the tabs along the top of the bin and rename them to reflect the contents of a tab.

NOTE The All Nodes, Lightbox, and Matchbox tabs cannot be customized or deleted.

To create a tab:

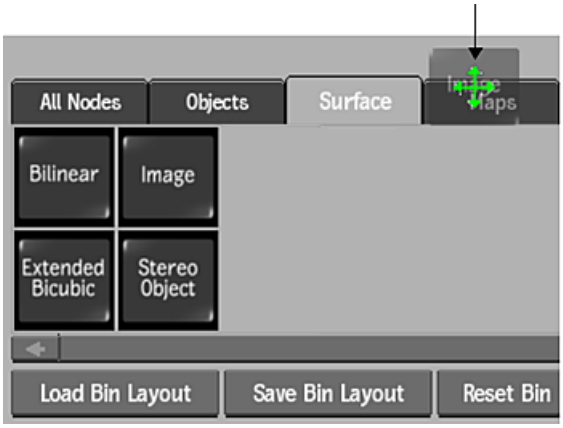
1. Click the plus sign tab.

2. Name the tab in the keyboard that appears.

   NOTE You can create a maximum of 6 new tabs.

To copy a node to another tab:

1. Drag the node on top of the destination tab.
2. Release the cursor.



3 Click the destination tab.

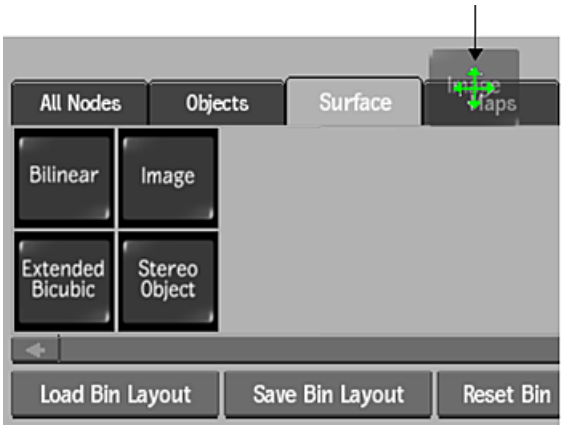
The copied node appears in the new tab. Nodes are added to the end of a tab in the order copied (following the same alphabetical node order of the rows, from top to bottom of each row).

NOTE Nodes cannot be duplicated within the same tab.

To move a node to another tab:

1 Press `Alt+Win` and drag the node on top of the destination tab.

2 Release the cursor.



3 Click the destination tab.

The node is moved from its original location to the destination tab. Nodes are placed at the end of a tab in the order moved (following the same alphabetical node order of the rows, from top to bottom of each row).

NOTE Nodes cannot be duplicated within the same tab.

To reorder a node within a tab:

1 Press `Alt+Win` and drag the node to a new location. You can move nodes from one row to another as well as reorganize nodes within a row.

2 Release the cursor when it changes to a green crosshair at the location where you want the node moved.

If you dragged the node on top of an existing node, the existing node shifts to the right and the moved node is inserted in its place.

To reset a bin to alphabetical layout:

1. With the applicable tab active, click Sort.
   The nodes in the bin are reset to their alphabetical layout.

To delete a tab:

1. Press `Alt+Win` and drag the tab to the bottom of the screen.
2. Release the cursor when it changes to a delete cursor.

   WARNING There is no undo capability when deleting a tab.

   The entire contents of the tab are deleted.

To delete a node from a tab:

1. Press `Alt+Win` and drag the node to the bottom of the screen.
2. Release the cursor when it changes to a delete cursor.

   WARNING There is no undo capability when deleting a node.

To rename a tab:

1. Click the Rename Tab button.
2. Enter a new tab name in the keyboard that appears.

To reorder a tab:

1. Press `Alt+Win` and slide the tab to its new location.
2. Release the cursor at the new location for the tab.
   If you dragged the tab on top of an existing tab, the existing tab shifts to the right and the moved tab is inserted in its place.

To save a bin layout:

1. Click Save Bin Layout.
2. Name the layout.
   The layout of the entire node bin is saved, including all new and customized tabs. You cannot save only select tabs.
   Layouts are saved per user, not by project.

To load a bin layout:

1. Click Load Bin Layout.
2. Select the layout you want to load.

   NOTE If you load a bin layout containing unsupported nodes, the unsupported nodes do not appear.

Matchbox and Lightbox Node Bin Tabs

To help you quickly access Matchbox and Lightbox effects, dynamic node bins are available for each, that behave slightly differently from the other node bin tabs. Keep in mind the following:

- The contents of the Matchbox and Lightbox node bin tabs are dynamically generated to display all shaders that are housed in a folder of your choice. You can set the folder locations in the Shader Paths tab of the Preference menu. The Default folder selections are the folders where the presets are housed (click the Browse button to select a different folder location):
  - For Matchbox: *opt/Autodesk/presets/<product version>/matchbox/shaders* (*.mx* or *.glsl* shaders appear in the node bin).
  - For Lightbox: *opt/Autodesk/presets/<product version>/action/lightbox* (*.lx* shaders appear in the node bin).

  TIP You can also set the folder locations in the FlameSetup utility, under the Shader Paths tab. By setting the folder locations here, the default paths are available for all projects.

- If new shaders are added to the selected file location, click the Refresh button under the node bin to refresh the dynamic display.
- Similar to the All Nodes tab, the Lightbox and Matchbox tabs cannot be customized or deleted. Also, you cannot move a node from the Matchbox or Lightbox tab to another tab.

Changing the Drawing Order with the Priority Editor

When you add an object in Action (such as a surface, GMask, Matchbox, Light, or geometry), it appears in front of all other objects in the scene. These overlapping objects create a stack and an order of priority as one object is drawn in front of the other.

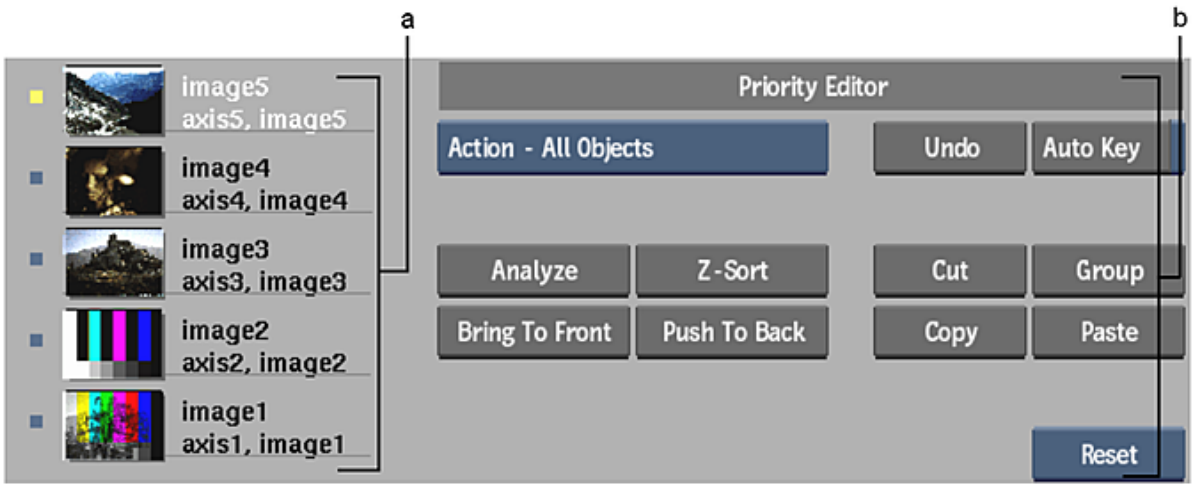
You can change the drawing order of these objects using the Priority Editor.

Accessing the Priority Editor

To access the Priority Editor, click the Priority button, or swipe the bar at the bottom of the Media or Object menu. You can also display a larger version of the Priority Editor as a selection from the View box.

You can specify to view the Priority Editor for a type of object, such as Lights Only, or all objects. Each entry in the Priority Editor indicates the object's name, the order of the object in the stack, and the location of the object in its branch. There is also a proxy showing what the object looks like in Schematic view.

NOTE If a source node is selected in the schematic, the Priority Editor that is displayed is that of the source node (the source node name is prepended to the branch name).



(a) Drawing order of objects in the scene (b) Priority Editor controls

Use the Priority Editor controls to analyse the scene for changes in rendering priority, to change the rendering priority, and to cut, copy, and paste priority information from one frame to another.

Changing the Drawing Order

The objects in the scene are shown according to z-depth, while still being drawn in the order of their priority: highest priority at the top of the list to the lowest priority at the bottom. The lowest level object is drawn first, the second to lowest level object is drawn next, and so on, until the top of the list.

To change the drawing order of objects:

1. From the Priority Editor box, select which type of object to view in the list.
2. In the Priority Editor list, select the object (or multiple objects).

   The selected object name is highlighted in yellow, and the corresponding object is selected in the Action schematic.
3. Do one of the following:
   - Drag the object to the desired position in the stack.
   - Click the Push To Back button to move the selected object one position lower in the stack, or click the Bring To Front button to move the selected object one position higher in the stack.

   NOTE If the Priority Auto Key button is enabled when you move an object in the stack, a keyframe is added to the timebar.
4. Click Preview to ensure that the drawing order is correct.

   Occasionally, a matte may obscure another object if the drawing order is unordered. If this happens, move the object up and down in the stack until the drawing order is corrected, or click Z-Sort.

Grouping Media

Select objects in the Priority Editor and group them to edit or affect several media at once. You can analyse or Z-Sort groups as you would single media.

Groups are named according to their contents. For example, if a group is created from Image 3 and Image 32, the Group's label will be “Image 2 Image 32”.

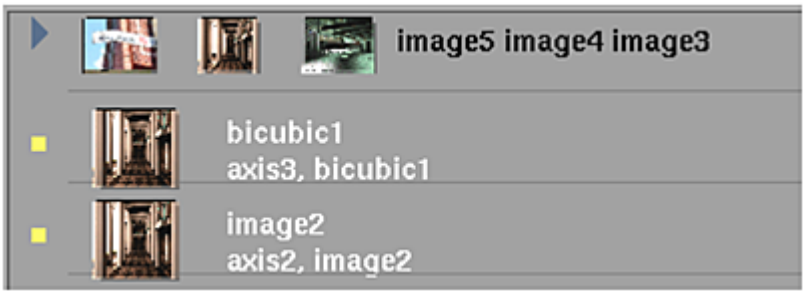
It is not possible to add a scene object to an existing group. To do this, you must ungroup the objects, reselect those you want to include in the group, and click Group.

NOTE Single element groups cannot be created.

To create groups in the Priority Editor:

1. Select multiple scene objects in the Priority Editor using either `Alt`-click for individual objects or `Shift`-click to select a range.
2. Click Group in the Priority Editor menu.

   Media is minimized to a single line in the Priority Editor. To expand the group and view its contents, click the arrow to the left of the group. Group contents can be reordered within the group.

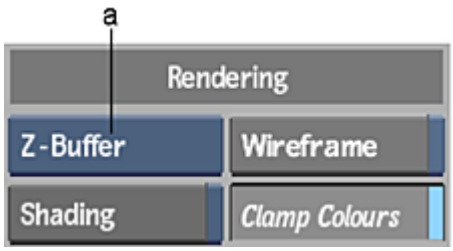


3. Scene objects can be ungrouped by clicking Group once again.

Analyze, Z-Sort, and the Z-Buffer

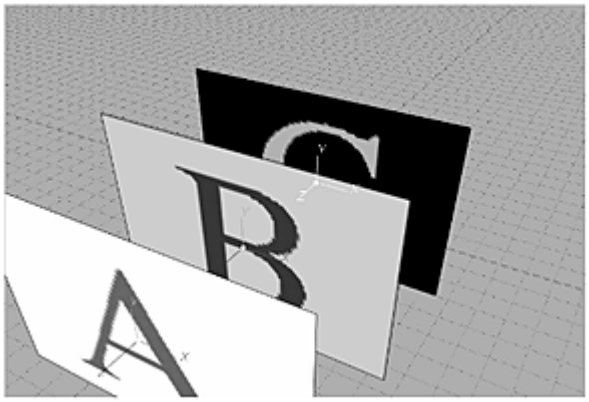
When you use Analyze or Z-Sort, the objects in the scene are compared using their position in the scene and not the individual pixels of a shape or model. The Priority Editor uses distance between the camera and the nearest and farthest points on the objects in the scene to determine the sorting order.

Since the Priority Editor is geometry based and not polygon based, you cannot properly order objects or groups that intersect due to their rotation, scale, or shear values. The Z-buffer uses the Z-value of each pixel for the sorting order. Use the Priority Editor in conjunction with the Z-Buffer box in the Setup menu Rendering section.



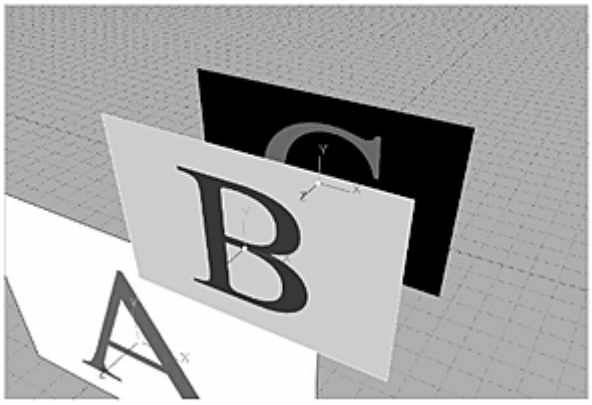
(a) Z-buffer box

When Z-buffer is on, objects and groups are arranged according to their distance from the camera eye. Since the camera is pointed towards the Z-axis by default, objects and groups are arranged according to their location on the Z-axis. In the following example, Image A is at Z position 200, Image B is at Z position 100, and Image C is at -50. When Z-buffer is on, Action draws these images as shown.



If you move the camera, objects are sorted according to the axis the camera is pointed towards.

When Z-buffer is off, the distance of objects from the camera eye is not considered. Objects are drawn in the order shown in the Priority Editor. To follow the previous example, you can turn off Z-buffer and change the priority of Image B so that it is drawn on top of both Image A and Image C. The Z position of these images is therefore ignored.



Shadow Mix, like Z-buffer, arranges objects according to their distance from the camera eye. Use Shadow Mix so that each shadow is rendered in the correct Z order with its corresponding surface.

Priority Editor Settings

Priority Editor box Select which Priority Editor to display.

Analyze button Click to analyze the entire scene for changes in rendering priority.

Z-Sort button Click to analyze the current frame for changes in rendering priority.

See Analyze, Z-Sort, and the Z-Buffer (page 567).

Bring To Front button Click to move the selected object one position higher in the stack, in front of the next higher object.

Push To Back button Click to move the selected object one position lower in the stack, behind the next lower object.

Undo button Undoes the last Priority Editor operation, except for Z-Sort or Analyze.

Use Action's Undo List to view a list of recent operations and revert to a prior state. Select an item in the list to return to that state. All actions that occurred after the selected item are undone.

Priority Auto Key button Enable to create keyframes when reordering media.

Group button Click to create groups of selected media (use this to analyze, Z-Sort, or manage several lines of media at once). See Grouping Media (page 566).

Cut button Click to cut priority information between frames.

Copy button Click to copy priority information between frames.

Paste button Click to paste priority information between frames.

NOTE A mark indicates when a change in priority occurs. If you copy and paste a mark that does not change the priority, no mark appears on the timebar.

Reset box Select whether to reset priority information for the current frame or for all frames.

Animating with the Channel Editor

Use the Channel Editor to animate the media, axis, surface, light, camera, and other properties of every object in the scene.

To open the Channel Editor and display the Action channels:

1. In Action, click Animation.
2. In the Animation menu, select Channel from the View box.

   The top-level folder in the hierarchy is the Scene, which provides the overall view of the animation. The Scene folder contains the Result Camera and folders of objects in the Action scene. Initially these are the media, motion blur, camera, axis, and image folders.

   If you add an object to the scene, the Channel Editor adds a folder to the channel hierarchy for the new object. For example, if you add a light, a Light folder is added in the channel hierarchy.

Channel Editor Action folders

Camera Animates the camera position and point of interest. It also contains channels for animating camera roll, field of view, and the near and far clipping planes.

When Free Camera is selected, the point of interest channels are replaced by rotation channels in the Channel Editor.

Media Animates media properties such as blur, crop, shadow softness. A Media folder is listed for each media in the scene.

Axis Animates axis properties such as position, rotation, scaling, and shearing.

Image, Bilinear, Perspective, or Extended bicubic Animates surface properties such as material, offset, and displacement. The Material folder contains a shininess channel and folders for the specular highlight, diffuse, and ambient lighting.

Shadow Animates the shadow colour and shadow transparency.

Light Animates light properties such as intensity, falloff, spread, position, rotation, and colour.

TIP Selecting nodes in the schematic will automatically select the associated channels in the Channel Editor.

Most of the settings in the Animation menu are standard for all tools that support animation, but there are a few animation settings specific to Action, available from any Action menu (on the right side).

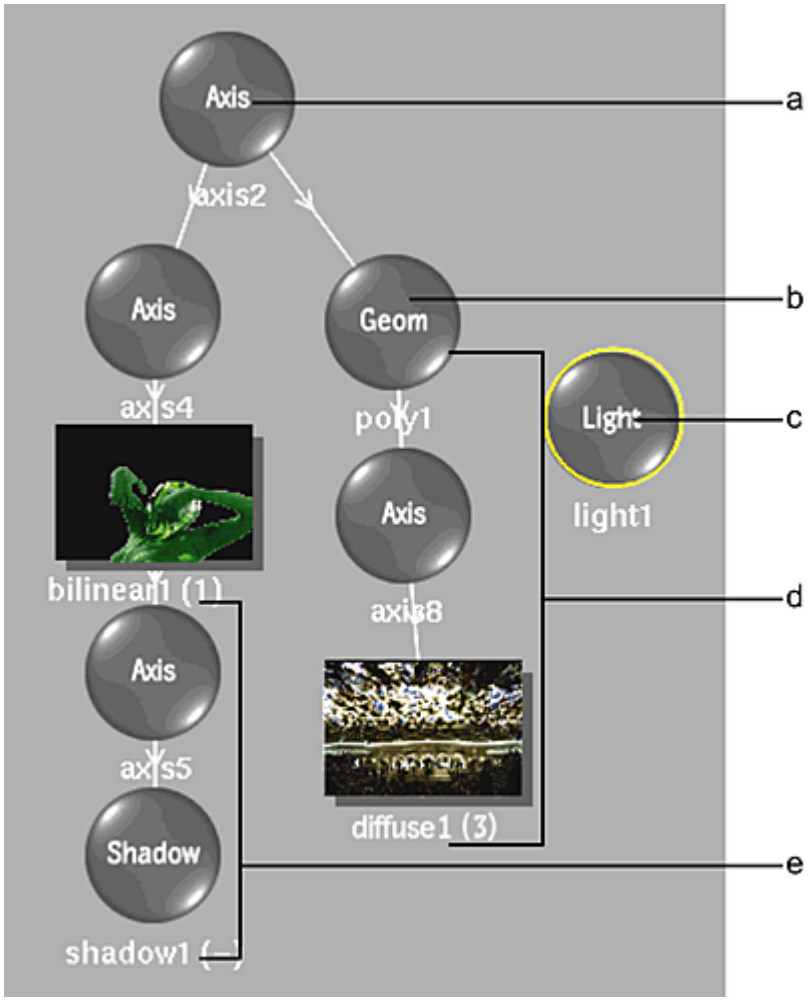
Auto Key button Enable to set a keyframe automatically each time you change a value at any frame.

Set Key button Click to set the current values for the selected channels in the current frame (when Auto Key is disabled).

Delete Key button Click to delete the selected keyframe.

About the Schematic and Menu Tabs

In Action Schematic view, a node exists for all objects in the scene, for example, shadows, lights, and texture maps.



(a) Axis object is the parent of axis 4 and poly1 (b) 3D model (c) Light source (d) Diffuse texture applied to the 3D model (e) Drop shadow of bilinear1

Here are some hints when working in the schematic with some of the various node types.

Surface Nodes When you add a surface node to the schematic, the node name is displayed with a number in parentheses. The number indicates the media applied to the surface. For example, a bilinear object labeled (1) shows that the bilinear uses the clips from Media 1.

For more information on the relationship between media and surfaces, see Adding Surfaces (page 623).

Camera Node The camera node appears in Schematic view by default and you can link it to any image. Use the camera node to rotate the camera about its own axis, and parent other nodes including shadow, texture, and geometry nodes.

Shadow and Texture Nodes Shadow and texture nodes each display a single number in parentheses beside the name that indicates the media used for the shadow or texture. For example, a shadow labeled (2) shows that the shadow uses the matte from Media 2.

Source Nodes Source nodes are used as part of an advanced schematic structure that separates the media's matte and front so that each clip can be animated individually. You can also use sources to create complex compositing effects such as nesting.

TIP Adding many nodes and connections can quickly make for a disorganized schematic. To solve this, press `Alt+T` to clean up the schematic.

Node Settings

Use these settings when working with Action nodes in the schematic.

Object Node Name field Displays the selected object node name. Editable.

Previous Node button Scrolls to the previous similar node.

Next Node button Scrolls to the next similar node.

The right side of Action also includes other settings to help you work with nodes in the schematic.

Solo button Enable to solo an object or a branch (object and its children), depending on what is selected in the Selection Mode box. See Soloing Objects (page 580).

Hide button Click to hide an object, a branch (object and its children), or all, depending on what is selected in the Selection Mode box.

Group button Click to collapse the selected nodes into a group.

Duplicate button Click to duplicate an object, a branch (object and its children), or all, depending on what is selected in the Selection Mode box.

Delete button Click to delete an object, a branch (object and its children), or all, depending on what is selected in the Selection Mode box.

Selection Mode box Make a selection to copy, delete, hide, or reset.

Clear All button Available in the Action Setup menu. Resets all parameters, and deletes nodes and media as well. Press Alt to bypass confirmation

Reset All button Available in the Action Setup menu. Resets all parameters but does not delete nodes or media. Press Alt to bypass confirmation.

Populating Menu Tabs of Selected Objects

Depending on what type of object is selected in the schematic, the tabs in the Object menu are populated based on different rules, as illustrated in the following examples.

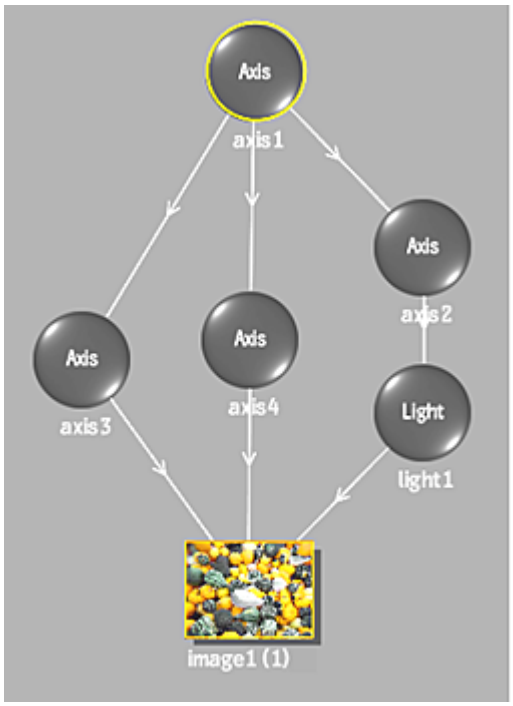
An object's name appears on the Object menu tab as well as beneath its node in the schematic and as a folder in the Channel Editor.

Axis selected

The Axis menu of the selected axis appears on the left side of the Object menu, and a limited number of the children objects' menus appear on the right side of the Object menu. The children objects are identified by scanning the hierarchy of the schematic from top-to-bottom (starting at the selected axis). The hierarchical scanning stops for any given branch when a non-axis object is encountered.

The order of tabs is determined on a per branch basis; that is, all of the tabs of one branch are listed before moving to another branch, starting with the highest levels in the parenting hierarchy (lowest index levels).

For example, in the following schematic, Axis 1 is selected.



The tabs in the Object menu appear as follows. The image1 tab appears in light blue to signify that multiple objects are connected down to it. Use the tabs to switch between menus within the Object menu. The Camera tab appears in orange as the first tab on the right side of the Object menu, and is exempt from the tab population rules.

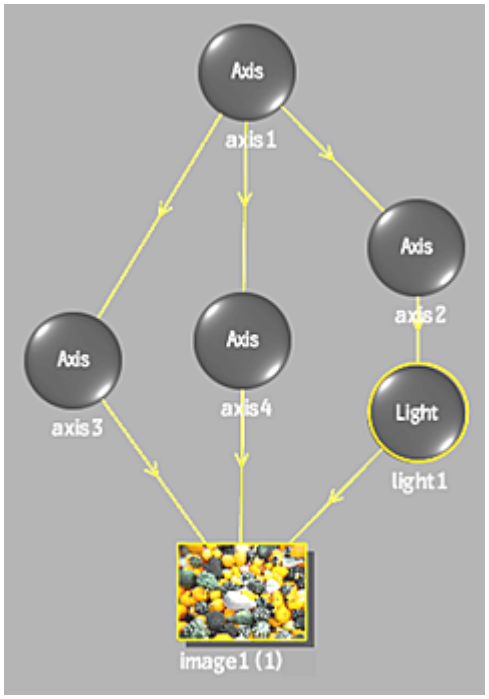
NOTE If there are more than five tabs on either side of the Object menu, use the arrows beside the tab names to navigate to the desired tab.

Other object selected (no “axis attributes”)

These objects include generators, bouncers, deformations, surfaces, texture maps, shadows, geometries, and 3D text. When one of these objects is selected in the schematic, an ascending (bottom-to-top) scanning of branches is performed. The hierarchical scanning stops for any given branch when an axis or object with axis attributes is encountered.

The menu of the selected object appears on the right side of the Object menu, and a limited number of the parent objects' menus appear on the left side of the Object menu.

For example, in the following schematic, Image 1 is selected.

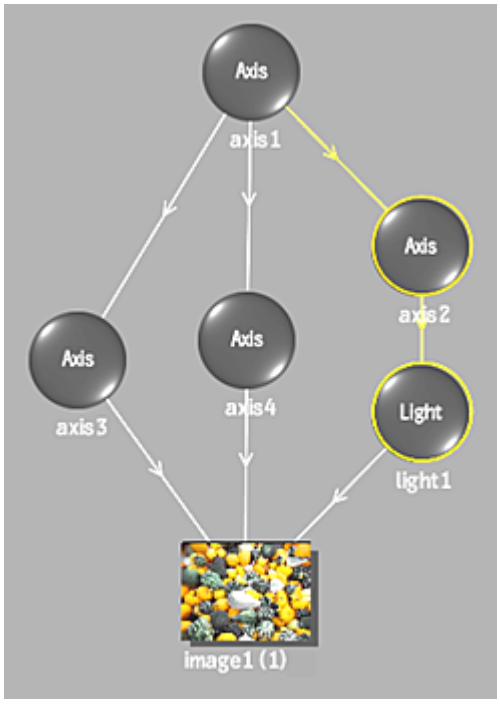


The tabs in the Object menu appear as follows. The image1 tab appears in light blue to signify that multiple objects are connected down to it. Use the tabs to switch between menus within the Object menu. The Camera tab appears in orange as the first tab on the right side of the Object menu, and is exempt from the tab population rules.

Object with “axis attributes” selected

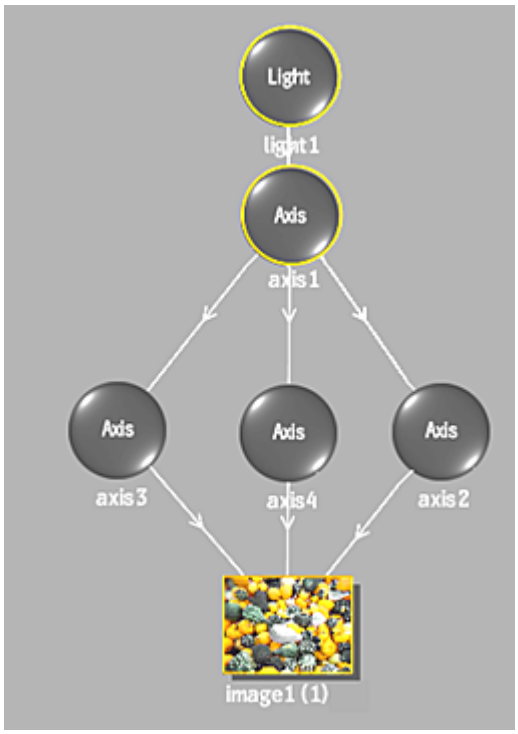
These types of objects include lights, animators, projectors, and cameras. These objects trigger a different scanning behaviour depending on their position within the schematic hierarchy. If the selected object is the first of its branch (top of the hierarchy), it inherits the tab population rules of an axis, that is, descending branch scanning. If the selected object is not the top object of its branch, it inherits the tab population rules of other objects, that is, ascending branch scanning.

For example, in the following schematic, light 1 is selected.



Since Light 1 is not the top object in its branch, the tabs in the Object menu appear as follows.

In the following schematic, light 1 is moved to the top of the branch.



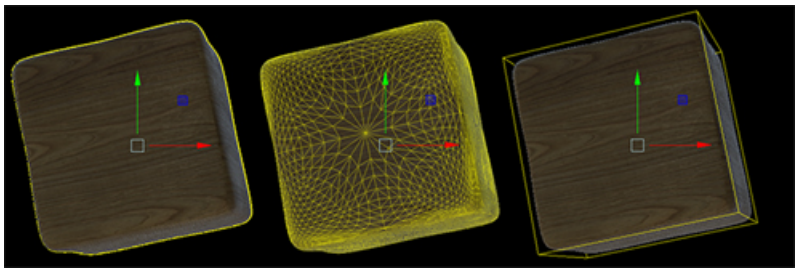
The tabs in the Object menu appear as follows. The image1 tab appears in light blue to signify that multiple objects are connected down to it. Use the tabs to switch between menus within the Object menu.

NOTE The Camera tab appears in orange as the first tab on the right side of the Object menu, and is exempt from the tab population rules. If a camera node is selected in the schematic, the special Camera tab does not appear, and the normal tab population rules apply.

Selecting Objects in Action

Selecting Objects in the Scene

By default, selected objects are highlighted in the Action scene. Use the **Action Preferences ➤ Preferences ➤ Selection Highlight box** (keyboard shortcut `Space+I`) to change this to wireframe or bounding box.



To select objects in Action, you can:

- Click a node in the schematic or an object in the scene.
- Press `Ctrl` drag to multi-select nodes in the schematic.
- Press `Ctrl` and click unselected nodes to add them the multi-selection.
- Press `Ctrl` and click selected nodes to remove them from the multi-selection.
- Press `Ctrl` and click objects in the viewport to add them to the multi-selection.
- Press `Alt` and click a node to select it and its children.
- Press `Shift` and click in the viewport to multi-select along the Z-axis, selecting every object located behind the cursor.

To navigate within a selection:

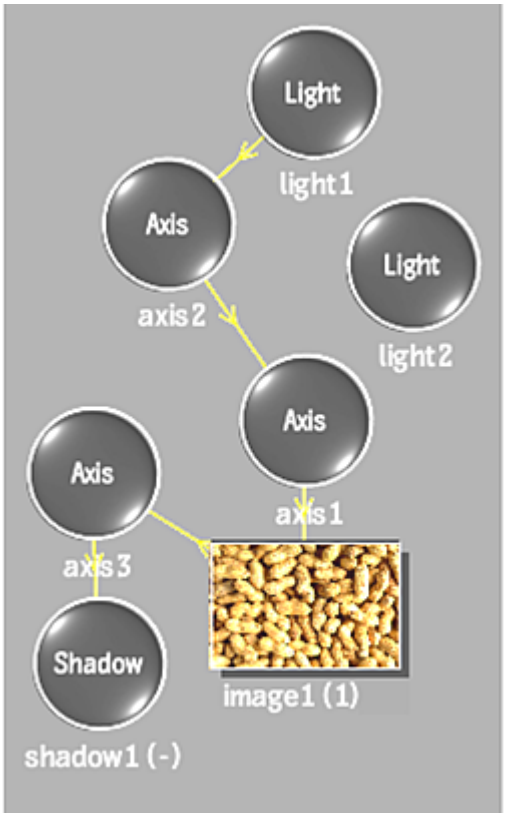
- In the Action schematic or the viewport, cycle through the selected objects by pressing `Q` or `Shift+Q`. You can then use the displayed manipulator to transform the object.

- In the Action schematic, press `Alt+Space+Right` or `Alt+Space+Left` to cycle through the different parent nodes of the selected node.

Using the Object Menu for Multiple Selections

You can select multiple nodes to change some of the settings in the Object menu for all objects of the same type.

Objects that support multiple selection and changing of field settings are axes, images, shadows, and lights. For example, in the following schematic, you can select all of the nodes.



The tabs in the Object menu reflect the multiple selections.

In the Object menu, you can relatively change any of the available fields. For example, if the X Position for Axis1 and Axis2 was originally 100, and for Axis3 was 200, and you drag the X Position slider under the Mult-axes tab to 50, Axis1 and Axis2 are now set to 150 and Axis3 is set to 250.

Because the field values for individual objects can be different, the values in the multiple selected fields display as default values in the Object menu. When you select an individual object and view its Object menu, you can see the true field values.

TIP Switch between multiple selection mode and displaying the individual node settings with `Space+M`.

Connecting Action Nodes

In Action, you can create complex animations where movements applied to one node are passed down to all connecting nodes.

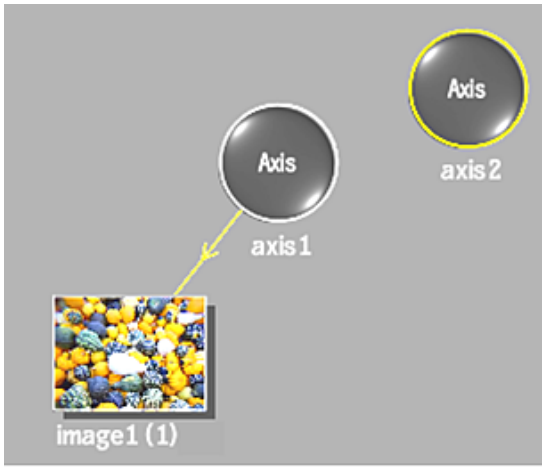
You can add an axis to the scene by itself, then make it the parent of another node. Use this method of parenting additional axes to create complex animations.

For example, create a cube of surfaces by parenting three additional axes to the same surface. Each axis that is parented to a surface places an additional surface in the scene. By changing the position and rotation of each axis, you can create a cube. If you parent the axes with another axis, you can control the position, rotation, scale, and shear of the cube.

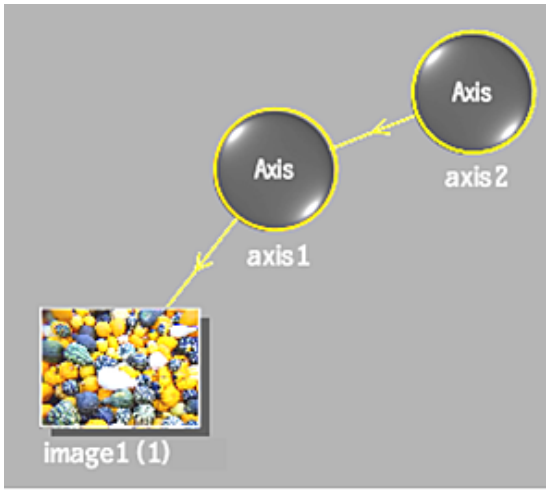
To create a branch:

1. Add an axis (axis2) to the scene.
2. From the View box, select Schematic. The Schematic view should be similar to the *Before* figure.
3. Do one of the following:
   - In the Tools box, select Connect and drag the cursor from the edge of the parent node to the node that will be its child.
   - Press `Shift` and drag a parent node over another node that will be its child. With this method, the Tools box does not have to be set to Connect mode, and can remain in Select mode, for example.
   - In the Action Setup menu, enable Auto Parent in the Schematic section, and then in the schematic, drag the cursor from the edge of the parent node to the node that will be its child. With this method, the Tools box does not have to be set to Connect mode, and can remain in Select mode, for example.

   Axis2 becomes the parent of axis1, as shown in the *After* figure.



Before: The schematic shows axis1 as the parent of image1



After: Axis2 is made the parent of Axis1 using Connect mode

Any transformations applied to axis2 are applied to axis1 and its surface (image1). If axis1 has any transformations, they are added to the transformations from axis2. For example, if axis2 is set to 500, 100, 0 and axis1 is set to -50, 20, -30, the positions are accumulated and applied to the surface. In this case, Image1 is positioned at 450, 120, -30.

TIP You can override the transformations passed from a parent to a child by enabling the Free button in the Axis menu.

Inserting a Node Between Connected Nodes

1. Do one of the following:
   - If Auto Insert is enabled in the Setup menu Schematic settings, drag a node and navigate to the link between two connected nodes. The link is highlighted in orange.
   - If Auto Insert is disabled in the Setup menu Schematic settings, press `Shift` then drag a node and navigate to the link between two connected nodes. The link is highlighted in orange.
2. Release the node.

   The node is inserted, and linked to the two previously connected nodes.

Mimicking, Copying, and Duplicating Objects

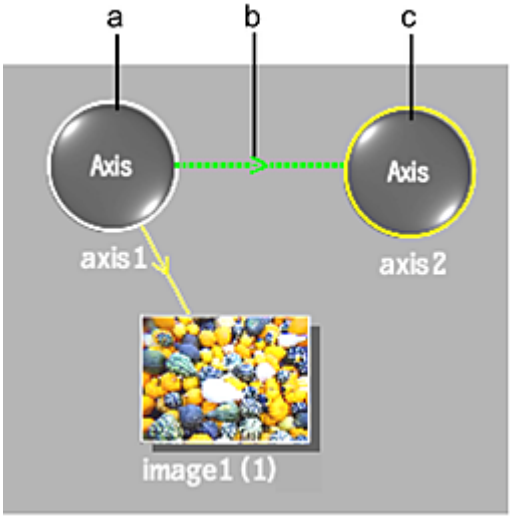
Similar to working in the Batch or Batch FX schematic, you can mimic, copy, or duplicate objects of the same type in the Action schematic.

Using Mimic Link

To create a Mimic link between Action objects:

1. From the Tools box, select Mimic Link.
2. In the schematic, drag between two similar object types, for example Axis to Axis. You can also link different map types, such as from a diffuse map to a reflection map.

   A green arrowed dotted line indicates that the objects are linked as duplicates. The direction of the arrow indicates which object is the master.



(a) Originating object (b) Mimic link (c) Linked object

3. Change the settings of any node.

All of the settings applied to the original object are automatically applied to the linked object. Once objects are linked, any settings applied to either object are applied to both. When linking different map types, only the settings found in the Texture tab are mimicked.

The originating object can link to multiple objects, but only one object can be the originating link.

NOTE To remove the link between mimicked objects, drag the cursor across the green line that joins the two objects. Each object keeps the settings that were applied while they were linked.

Duplicating Objects

To duplicate objects:

1. In the schematic, select a node, or group of nodes.
2. Click Duplicate, or right-click the selection, and choose Duplicate.
   A duplicate of the node or nodes (with all node settings) appears in the schematic.

Copying Objects

To copy objects:

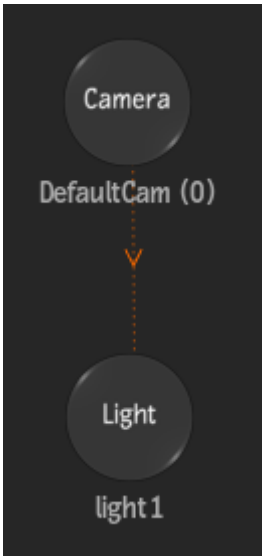
1. In the schematic, select a node, or group of nodes.
2. Right-click the selection, and choose Copy.
   A copy of the node or nodes (with all node settings) is added to the clipboard.
3. Navigate to the location in the schematic (or another Action schematic) where you want the copied node or nodes to reside, right-click and choose Paste.
   A copy of the node or nodes (with all node settings) appears in the schematic.

Applying Look-At Connections

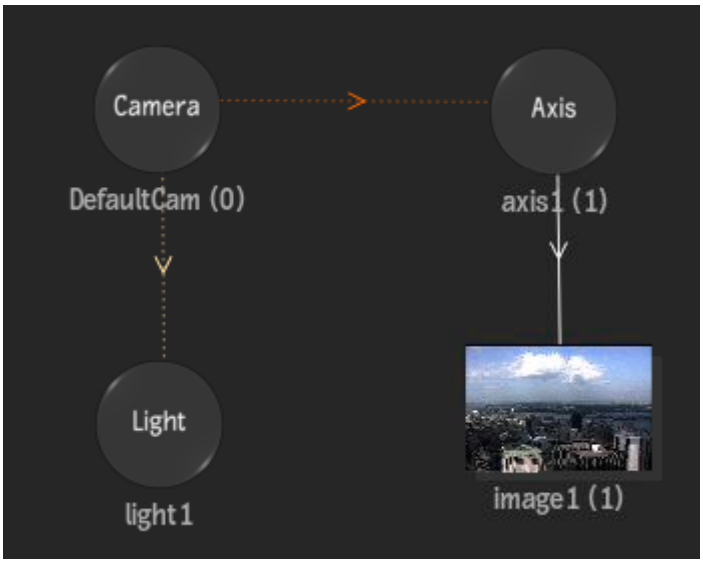
You can attach a look-at connection between Action objects with axis characteristics (Axis, Camera, Light, and Projector). The connection allows the parent object to rotate to face the child look-at object, no matter where it is positioned. Look-At links can be animated; therefore you can link different objects from the parent at different frames. In the channel editor, you can see a *lookAt* channel for every look-at parent.

To apply a look-at connection:

1. Make sure that Auto Key is enabled, and move to the frame where you want to apply the look-at connection.
2. Select Look At in the Tools box.
3. In the schematic, drag from an object with axis characteristics to another object with axis characteristics.
   The selected objects are connected by an orange dotted line with an arrow.

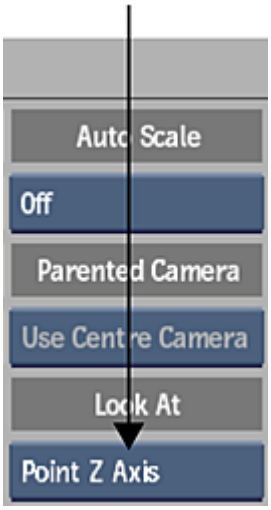


As you move away from that frame, notice that the connection line is dimmed, signifying that you can then add another connection from the same parent to a different node at another frame.



4 To delete a look-at link, make sure that Look At is selected in the Tools box, and return to the frame where the link is active (orange dotted line), then swipe through the line.

5 If an Axis node is a look-at parent, you can set which axis is pointed to the look-at child object, in the Axis menu.



Locking, Soloing, or Hiding Objects

Locking Selection

You can lock the selection to the currently selected nodes. Locking a node selection ensures you can only edit and manipulate that selection and nothing else.

`Alt-Space-L` toggles between selection lock and unlock.

Soloing Objects

Use the Solo button to hide all other objects except the selected object. Using Solo is useful for identifying an object in a scene with many objects, without having to hide all of the other objects.

To solo an object:

1. In the schematic, select the object that you want to solo.
2. From the Selection Mode box, choose Selected.
3. Enable Solo (on the right side of the Action menu).

   All other objects in the scene are hidden, regardless of whether they are set to Hide or not. When Solo is disabled, the Hide settings are restored.

   TIP You can leave Solo enabled and select different objects in the schematic to view each object separately.

To solo a branch:

1. In the schematic, select the parent of the branch that you want to solo.
2. From the Selection Mode box, choose Branch.
3. Enable Solo (on the right side of the Action menu).

   All other objects in the scene are hidden, regardless of whether they are set to Hide or not. When Solo is disabled, the Hide settings are restored.

Hiding Objects

You can temporarily hide an object or objects in the schematic to see different results, for example.

To hide an object or objects:

1. Select and object or multiple objects in the schematic view.
2. Click Hide, or right-click the object, and choose Hide.

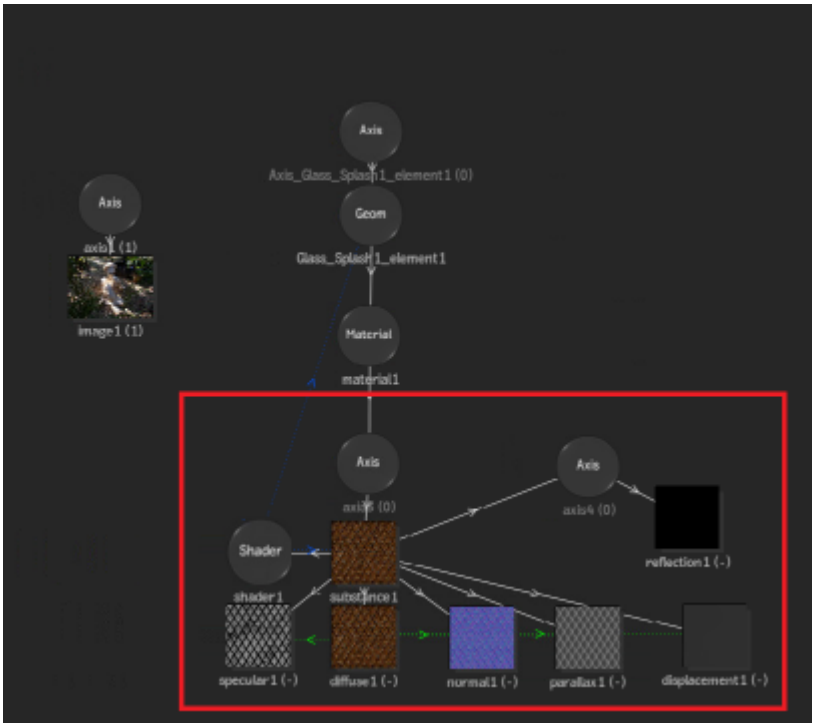
   (hidden) appears after the object name in the schematic.
3. To unhide the object, click Hide again, or right-click the object and select Show.

Grouping Objects

You can collapse any collection of nodes in the schematic into a group to reduce clutter. The result is a cleaner schematic, but you can also quickly edit the contents of the group, or even add nodes inside the group.

To create a group:

1. Select objects in the Action schematic. You can press `Ctrl` and drag to multi-select. Press `Ctrl+Shift` and click to add or remove items from a multi-selection.



NOTE Objects selected for a group do not need to be in the same schematic branch.

2 Click Group, or right-click the selection, and choose Group to collapse the selected objects into a group.

A blue group node appears in the schematic to represent the entire group, and connections are maintained outside of the group.

3 To uncollapse the group, click Ungroup in the Group node menu, or right-click the Group node and select Ungroup.

NOTE You can not create a group within a group, and groups created in the Schematic view are not related to groups created in the Priority Editor.

TIP Set the Icons box to Group Off to hide the icons of objects contained within Group nodes. This can help unclutter the Action schematic. The Icons box is displayed below the Action viewports.

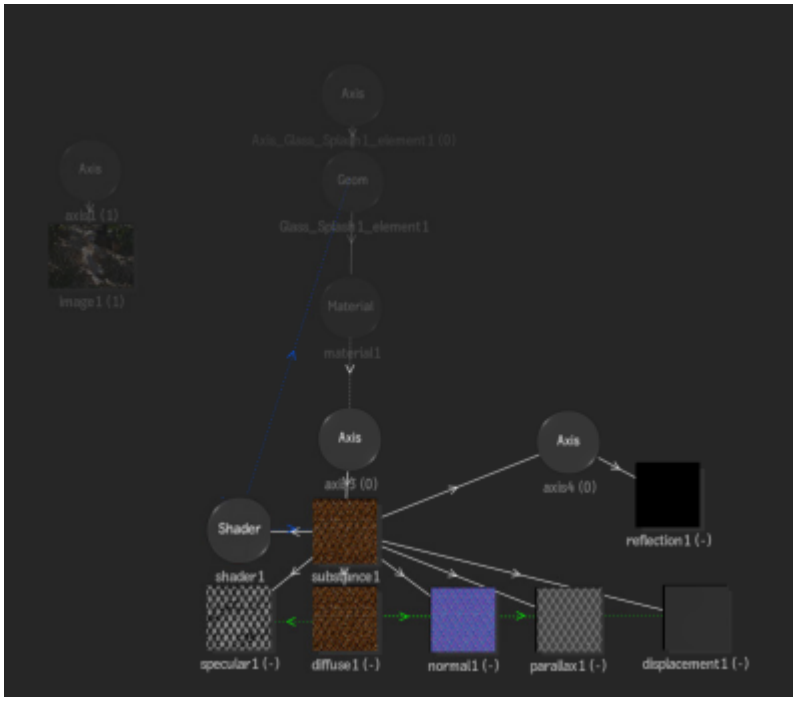
Editing Action Groups

You can easily edit an Action group using the Group Schematic view.

You can access the Group Schematic view using one of these three methods:

- Right-click on a Group node and select Edit Group from the contextual menu.
- Double-click a Group node.
- With the Group node selected, press `F8` or select **Action Object ➤ Group Schematic** from the View box.

The Group Schematic view displays the contents of the group framed in the image window, with all other nodes dimmed.



Once you in Group Schematic view, you can edit or move any node (even the dimmed nodes). You can add nodes to the Group Schematic, and they then become part of the group. You can also add nodes into the regular Action schematic, and see them "live" as dimmed nodes in the Group Schematic.

TIP Use a 2-up view with the regular Action schematic in one view and the Group Schematic in the other. In this case, you can see how adding nodes inside or outside of the group affects your Action scene.

You can perform certain connections to a Group node which are automatically done on all compatible nodes found inside of it. Supported connection are:

- Parent Axis: Connects to all top Axis.
- Lighting link: Connects to all Geometry, Surface, 3D Text, and 3D Shape objects.
- GMask link: Connects to all Geometry, Surface, 3D Text, and 3D Shape objects.
- Child Material: Connects to all Geometry, Surface, 3D Text, and 3D Shape objects.
- Child Texture map: Connects to all Geometry, Surface, 3D Text, and 3D Shape objects.
- Child Substance: On initial creation, does a lighting link and a Material connection to all Geometry, Surface, 3D Text, and 3D Shape objects.

NOTE These connections are made based on the logic of the hierarchy of the group structure. In some cases, you may need to manually create connections or break connections.

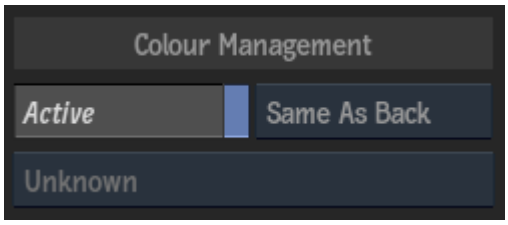
Colour Management in Action

There are a few different settings in Action for colour managing Action-imported media (that is, media being read from an external location). Note that these settings do not affect input media entering Action through Batch or Batch FX, and the timeline, for example.

NOTE For more information on the overall workflow of Colour Management in Flame, see Colour Management (page 1755).

You can set the default Action colour space in Colour Management Preferences (page 1940).

In the Action setup menu Rendering tab, use the Colour Management settings to set the default behaviour of Action:

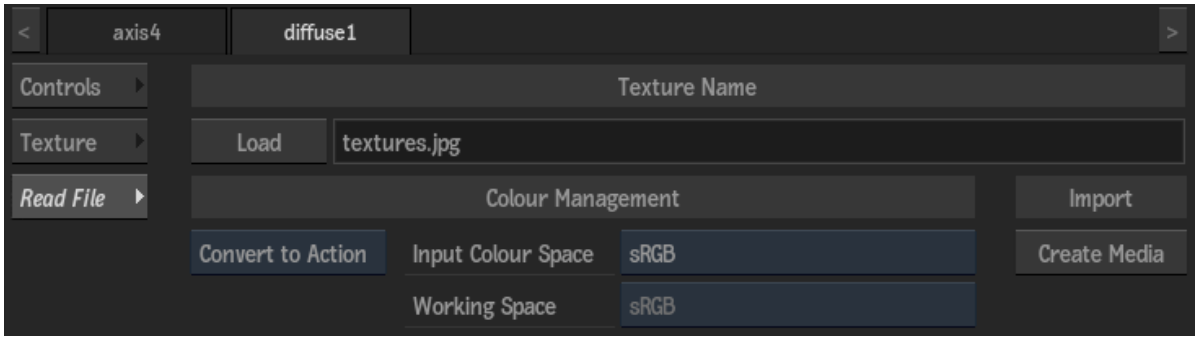


Active button Enable Colour Management in Action (enabled by default). When active, imported media located in the Read File menu of a texture map , as well as for Substance and Lightbox effects using the Scene Linear option are affected. When connecting Back media that is using a non-working Action colour space, the Active button is disabled. Also, if you try to enable the Active button when an unsupported colour space is displayed in the Colour Space field, a warning message appears. The status of this option is also available in the Lightbox API.

Working Colour Space box Select whether to use the colour space defined by the backplate, or select User Defined to set a colour space from the drop-down list below.

User Defined Working Colour Space box Select a working colour space for imported media in Action. Available when User Defined is selected in the Working Colour Space box.

When importing through the Import node (3D models in FBX or 3DStudio formats, and Photoshop PSD files), or editing the Read File menu of a texture map, the following options are available if Colour Management is active:



Colour Management Mode box Select the colour management mode to apply.

Select:	To:
Auto Convert	Convert content using the media file's metadata (such as OpenEXR), using a rule, or by manually selecting the Input Colour Space and the Working Space. Auto Convert allows you to set the media to any Working Space of your liking and does not change if you later switch to Action Working.
Tag Only	Tag content with one of the various colour spaces available in the list, using media file metadata, or using a user-defined colour space. Tagged media is not converted to the destination colour space. Flame displays it in the destination colour space for viewing purposes. The original media remains in its original format and colour space.
Convert to Action	Defer the choice of the working colour space to the Action Setup menu Colour Management settings. This allows you to manage the working colour space of every media imported with Convert to Action from the Action preferences.

Input Colour Space box Select the input colour space of your media.

Working Space box Select the destination colour space of your media.

Setup and Processing Options

When accessing Action from Batch or Batch FX, use the Action Node Prefs menu to customize your Action display and to access tools and guides.

When accessing Action through the timeline or the Tools tab, these options are available from the Setup button in Action.

Use the tabs in the menu to switch between the different options.

Rendering Tab

Certain settings in this menu, such as the Resolution and Over Scan settings, are unavailable if you accessed Action from the timeline.

Resolution Settings

You can set the rendering resolution of clips that are output in Action. For example, if you are working in an NTSC 8-bit project, the default output is NTSC 8 bits. However, if you want to change the output resolution to HD (1920x1080), use the Resolution menu to change the values so the final outcome will be 1920x1080.

You can choose between progressive or interlaced when rendering at video resolution. Action media automatically adjust the rendering mode of each clip. This inherent awareness also makes it possible for Action to properly mix field-based HD / PAL with NTSC and not have dominance problems.

Resolution Mode box Select whether to use the Action background resolution, or a user defined resolution. If you select User Defined, use the settings below to set the resolution. User defined rendering bit depth is limited to 8-bit or 16-bit FP, whereas Same As Background retains the bit depth of the background clip.

Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

Once you make a selection from the Apply and Scale box, the resolution is set for processing clips, and is remembered between Action sessions.

Width field Displays the custom width resolution of the clip. Editable.

Height field Displays the custom width resolution of the clip. Editable.

Aspect Ratio Presets box Select a standard frame aspect ratio. Select the Set to w:h option to set the clip to use square pixels. Select Custom to define a custom frame aspect ratio in the Aspect Ratio field.

Aspect Ratio field Displays the custom render/output aspect ratio. Editable

Bit Depth box Select the render/output bit depth of clips.

Scan Mode box Select the scan mode of clips.

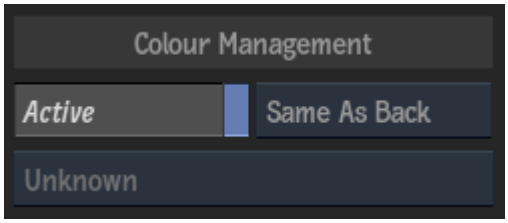
Apply and Scale box Select whether to apply or apply and scale the defined resolution settings. All subsequent processes will use the new render settings.

Select:	To:
Apply	Specify the output resolution.
Apply + Scale	Specify the output resolution and scale a scene to the defined resolution. Use this option if you are working with a low resolution of an Action setup from a previous version of the software. The scaling is applied to geometries, axes, lights, and cameras, as well as their coordinates. Their positions are scaled accordingly. Flame automatically scales textures and images.

NOTE If you do not change the Resolution settings, rendered results and camera settings default to the values set for the current project (set when you created a new project).

Colour Management Settings

Use these settings to colour manage Action-imported media. See Colour Management in Action (page 584).



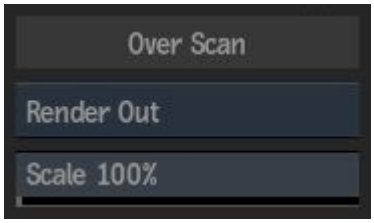
Active button Enable Colour Management in Action (enabled by default). When active, imported media located in the Read File menu of a texture map , as well as for Substance and Lightbox effects using the Scene Linear option are affected. When connecting Back media that is using a non-working Action colour space, the Active button is disabled. Also, if you try to enable the Active button when an unsupported colour space is displayed in the Colour Space field, a warning message appears. The status of this option is also available in the Lightbox API.

Working Colour Space box Select whether to use the colour space defined by the backplate, or select User Defined to set a colour space from the drop-down list below.

User Defined Working Colour Space box Select a working colour space for imported media in Action. Available when User Defined is selected in the Working Colour Space box.

Over Scan Settings

Use the over scan settings to render Action outputs in a larger canvas, without changing the active resolution (allowing you to easily fix a number of edge rendering issues, including AO, GMask Softness, Blooming, etc.).

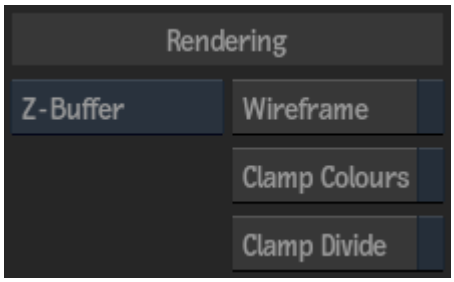


Over Scan Mode box Select whether the over scan settings are applied on an Action render, or only internally when working in Action.

Over Scan Scale field Displays the size of the canvas to be included in the Action rendering output. The over scan border is displayed in the image window as a dotted line, if the Border button is enabled in the Grid settings. Editable.

Rendering Settings

Use the Rendering settings to improve the final output quality of your image.



Z-Buffer box Select an option to determine whether the distance from the camera eye is considered to determine the order in which objects are rendered.

Wireframe button Enable to display each surface and 3D model as a wireframe.

This is useful if interaction with Action becomes slow because of many surfaces and 3D models. Convert objects to wireframe when you want to speed up processing or rendering times. Because lighting and textures are not computed in wireframe mode, interaction time and rendering times are reduced.

Clamp Colours button Enable to clamp colour and luminance in the 16-bit floating point processing pipeline.

Clamp Divide button Enable to allow the Media list divide operation to handle floating-point values correctly (at the cost of losing some colour information).

Stereo Settings

Reset To Stereo Mode button Enable to clear any previous stereo settings in Action, and reset all stereo settings to their defaults.

In the Stereo mode, these are the default settings when you enter Action with the Front/Back/Matte input option using stereo clips:

- A 3D camera (stereo camera) is created and the default camera is hidden.
- In the 3D Camera menu, the Result Camera is set to the stereo camera.
- In the Output menu, the Mode is set to Stereo and the Camera is set to Result Cam.
- A stereo object is created with the clips you selected.

NOTE If you enter Action using mono clips, the default camera is automatically created. If you need a stereo camera, you must add it manually.

Shading Settings

Use the Shading settings to set default Action lighting behaviours.

Active button Enable to light up the scene using added light sources. When disabled, no lighting effects appear in the scene; surfaces and 3D models appear flat. This same setting can be found in the Action Setup menu and the Light menu.

Scene Ambient button Enable to have global ambient lighting in your Action scene. This same setting can be found in the Action Setup menu and the Light menu.

Scene Ambient Intensity field Displays the intensity level of the default ambient lighting in the Action scene. Available when Scene Ambient is enabled. Editable.

Scene Ambient Shading field Displays the shading level of the default ambient lighting in the Action scene. Available when Scene Ambient is enabled. Editable.

Software Anti-Aliasing Settings

You can also access Action hardware anti-aliasing settings in the Common tab when you access Action as a Timeline FX or from the Tools tab, or from Batch or Batch FX Prefs.

Software Anti-Aliasing Sample box Select a software anti-aliasing sampling level. To set an arbitrary amount of animatable samples, select AA N Samples, then choose the number of samples in the Samples field that appears.

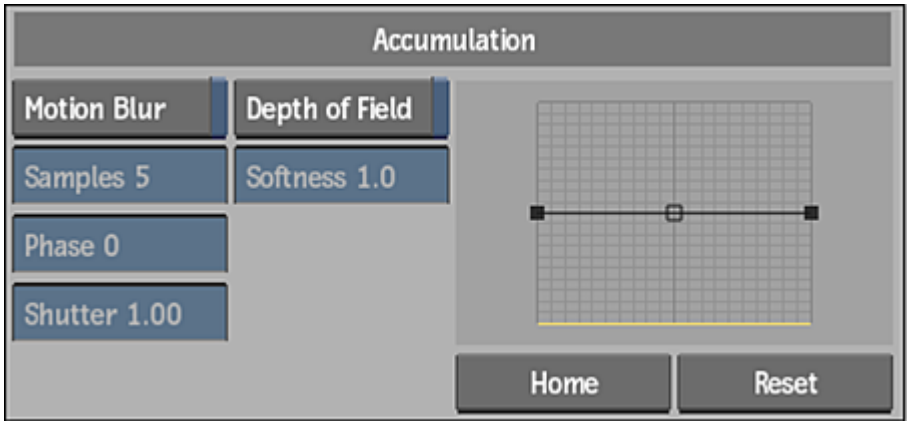
The jagged lines that often occur along the edges of diagonal or curved lines when processing high-frequency images such as text are the result of aliasing. You can increase or decrease the anti-aliasing sampling level. Higher values yield smoother results at the expense of processing time.

Anti-Aliasing Softness field Displays the softness value of the software anti-aliasing sample. Editable.

Samples field Available when AA N Samples is selected in the Software Anti-Aliasing Sample box. Displays the number of software anti-aliasing samples to consider. Editable and animatable.

Accumulation Settings

Use the Accumulation settings to define motion blur properties.



Global Motion Blur button Enable to use motion blur. Once enabled, specific Action object Motion Blur buttons can be enabled or disabled. See About Motion Blur (page 851) for specific examples of creating motion blurs.

Samples field Displays the quality of motion blur and the depth of field produced by the number of samples taken at each frame. Editable.

Increasing this value causes the processing time to increase linearly and affects the quality of the depth of field. The number of motion blur samples is multiplied by the number of anti-aliasing samples. To reduce the total number of passes made for each frame, reduce the level of anti-aliasing when motion blur is enabled.

Phase field Displays the frame that motion blur is based on (before or after the current frame). Editable.

A value of -100 places the motion blur before while a value of 100 places the motion blur after. A value of 0 is centred, which evenly distributes the motion blur. The default value is 0.

Shutter field Displays the duration of motion blur at each frame (essentially the number of frames that the shutter is open). Increasing this value does not increase the processing time. Editable

---

TIP You can animate the Global Motion blur button, as well as the Phase, Shutter, and Samples fields. They can be found in the Channel Editor under the *motion_blur* folder.

Depth of Field button Enable to use the camera's depth of field.

Depth of Field Softness field Defines the softness of the depth of field. A low value yields a sharp falloff between focused and unfocused regions. Editable.

Motion Blur curve Controls the sample weight over the scope of the motion blur.

Home button Resets the position of the motion blur curve after panning.

Reset button Resets the motion blur curve.

Preferences Tab

Surfaces Settings

Default Resolution field Displays the default geometry resolution (number of polygons) of surfaces. You can also change the geometry resolution of specific surfaces in the Surface menu. Editable.

The lower the value, the better the resolution and the greater the processing time required to interact with the image. A value of 1 creates one polygon per pixel on a surface, affording accurate displacement and lighting.

Use this field when Exploding Objects and Surfaces (page 961) with a particle generator and when using hardware texture mapping.

Apply To All button Click to apply the geometry resolution in the Resolution field to all surfaces. Since you can change the resolution of specific surfaces in the Surface menu, you can use this button to re-apply the default resolution to all surfaces.

Filtering Settings

Default Filter box Select the type of filtering to be set as the default when creating surfaces and maps.

Apply To All button Click to apply the default filter to all existing surfaces and maps.

Blending Settings

Default Premultiplication box Select the default blend operation premultiplication setting.

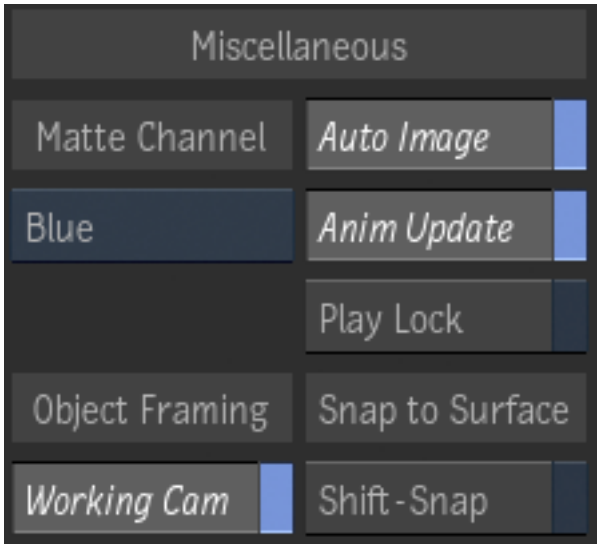
Apply To All button Click to apply the default blend premultiplication setting to all existing surfaces and geometries.

Lightbox Settings

Lightbox Shading button Sets the default behaviour of Lightbox nodes when the attached Light is not active.

Apply To All button Click to apply the default Lightbox Shading setting to all existing lights.

Miscellaneous Settings



Matte Channel box Select which channel is displayed as transparent by default. This can be useful for a multichannel clip to display only the Red channel matte, for example.

Auto Image button Enable to add an image node and axis automatically in the schematic when new media is added.

Animation Update button Enable to update properties such as position, rotation, and colour in the scene. When disabled, animated objects do not move, but keep the position of their current value. Disable to copy keyframe values from one frame to another.

Play Lock button Enable to update the animation settings according to the frame or timebar position as you move through the clip while keeping it locked at the current frame.

When Play Lock is disabled and you use > or < to play the resulting clip, each frame is loaded and displayed in sequence in the image window.

Working Camera button Enable to automatically switch to the Working Camera when framing the contents of the viewport with Frame All (`Space+A`) or Frame Selection (`Space+F`). Disable to frame using the active camera.

Shift-Snap button Enable to use the Shift key as a snap-to-surface modifier key. Hold the `Shift` key when selecting an axis to snap the selected axis to the surface of the underlying geometry. Not available when in Top, Front, or Side views.

Display Settings

Icons box Sets object icon (axes, borders, control points) and wireframe display options. Selected displays icons only for the object currently selected in the scene. Group Off displays every icons except for objects nested in Group nodes. Use option the Cycle Icons Draw Modes keyboard shortcut to cycle through the options.

Icon Transparency field Displays the transparency level for icons and wireframes in the scene. Editable.

Selection Highlight box Select the type of highlight used to identify the objects that are selected in the viewport.

Axis Scale field Displays the size of axis icons in the scene. Editable.

Z-depth Occlusion button Enable to have Z-depth occlusion affect Action widgets such as axis, light, and cameras. It creates a true 3D experience in terms of viewing, but hides occluded widgets.

Space Selection button Select the space for transforms performed using a 3D Manipulator. In Object space, the manipulator sticks to the object's axis orientation, ideal for translations. In World space, the manipulator follows the World's frame of reference, ideal for rotations. In Camera space, the manipulator stays in the camera's XY plane.

Ruler button Enable to display the ruler in the scene area. Use the arrows at each extremity of the ruler to place the beginning and end of the ruler anywhere in your scene view.

TIP Use the `Shift` key to snap the ruler into place either horizontally or vertically.

Ruler Define button Enable to define real unit measurements, such as feet, metres, or inches, instead of pixels. All camera distance or axes measurements thereafter use the defined scale.

Ruler Length field Displays the length and measuring unit (feet, metres) to use in Action. When you enter a length, click Define again to apply the new scale to the ruler and all pertinent fields, such as position, rotation, and scale. Editable.

TIP Hold the `Ctrl` key and click Define to reset the ruler's scale.

Grid box Select the type of grid to display in the scene. Use to position objects in the scene more accurately.

Select:	To:
Grid Off	Disable the grid.
Grid XY	Use a grid constructed on the X and Y planes.
Grid XZ	Use a grid constructed on the X and Z planes. The XZ grid is visible only when the camera is moved from its default position.
Grid YZ	Use a grid constructed on the Y and Z planes. The YZ grid is visible only when the camera is moved from its default position.

NOTE The Action grid is independent from the global grid in the Grids and Guides menu.

Grid Colour pot Displays the custom colour for the grid. Editable.

Ortho Near field Displays the value of the near view in the image window when using Camera or an orthographic view. Editable.

Ortho Far field Displays the value of the far view in the image window when using Camera or an orthographic view. Editable.

Schematic Settings

Auto Parent button Enable to automatically parent nodes in the schematic. Press Alt and drag a node in the schematic to disable Auto Parent temporarily.

Auto Insert button Enable to automatically insert a node when dragged between two connected nodes. When disabled, press Shift to auto insert.

Snap To Grid button Enable to position objects with precision in the scene. When you move an object in the scene, the object is automatically aligned to the snap grid.

NOTE When enabled, this snap only applies to the grid defined in Action. The grid defined from the Grid & Guides menu remains visible, but its snap is overridden as long as the Action Snap is enabled.

Schematic Transparency field Displays the level of transparency of unselected nodes in the schematic. Editable.

Display Information box Select what clip information is displayed in the schematic.

Proxy Update button Enable to automatically update proxies in the schematic. Interaction is slower when enabled. When disabled, Action updates proxies when you switch views. You can also update proxies by clicking Update.

Output Naming Settings

Output Name button Enable to allow newly created outputs in the Output List to inherit the name of the node. If disabled, it will use a default output [number] for additional outputs. You also can rename any output manually by clicking the Rename button below the Output List. Once a setup is saved, the output will use the setup name when processed.

Append Type button Enable to append the type of output to the name of a processed file.

Adaptive Degradation Tab

Use these settings to temporarily deactivate taxing operations during interactive manipulations. These settings do not affect the final renders.

Active button Enable to activate adaptive degradation display settings. Use to prevent slowdowns of the image window display when changing Action settings. Press Ctrl+D to activate or deactivate degradation settings.

You'll see an icon

displayed at the bottom right corner of the image window when degradation is active in Action.

Synchronize All button Enable to synchronize all Batch FX Action nodes with the Adaptive Degradation settings of the current Action node.

Filtering Degrade box Select whether to degrade filtering selections in the image window when changing any Action settings.

Filtering Update box Select whether to always degrade filtering selections, or on manipulation only.

EWA Filtering box Select whether to apply EWA or EWA+Linear filtering on maps (if selected in the Filter box) only when processing (On Process Only), or all the time when working in Action (Degrade).

Particles Degrade box Select a particle quality display setting.

Particles Update box Select whether to always degrade particles selections, or on manipulation only.

Shading Degrade box Select whether to degrade shading selections in the image window when changing any Action settings.

Shading Update box Select whether to always degrade shading selections, or on manipulation only.

Shadows Degrade box Select a shadow quality display setting.

Shadows Update box Select whether to always degrade shadows selections, or on manipulation only.

Surfaces Degrade box Select a surface quality display setting.

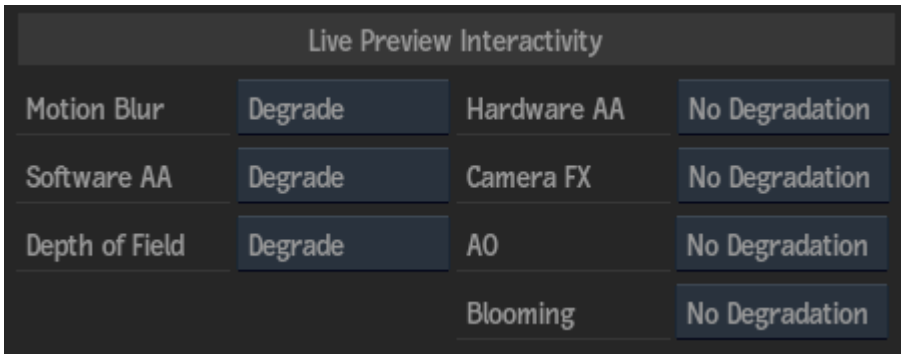
Surfaces Update box Select whether to always degrade surfaces selections, or on manipulation only.

NOTE When Adaptive Degradation is active for a Batch FX Action node, you'll see an icon

over the node in the schematic, and in the bottom right corner of the image window in result view.

Live Preview Interactivity

Use these adaptive degradation settings to alleviate possible performance issues.



Motion Blur Degrade box Select whether or not to degrade motion blur effects while the viewport is in Live Preview mode.

Software AA Degrade box Select whether or not to degrade software anti-aliasing effects while the viewport is in Live Preview mode.

Depth of Field Degrade box Select whether or not to degrade depth of field effects while the viewport is in Live Preview mode.

Hardware AA Degrade box Select whether or not to degrade hardware anti-aliasing effects while the viewport is in Live Preview mode.

Camera FX Degrade box Select whether or not to degrade Camera FX post-processing effects (Matchbox, Lens Flares, Rays) while the viewport is in Live Preview mode.

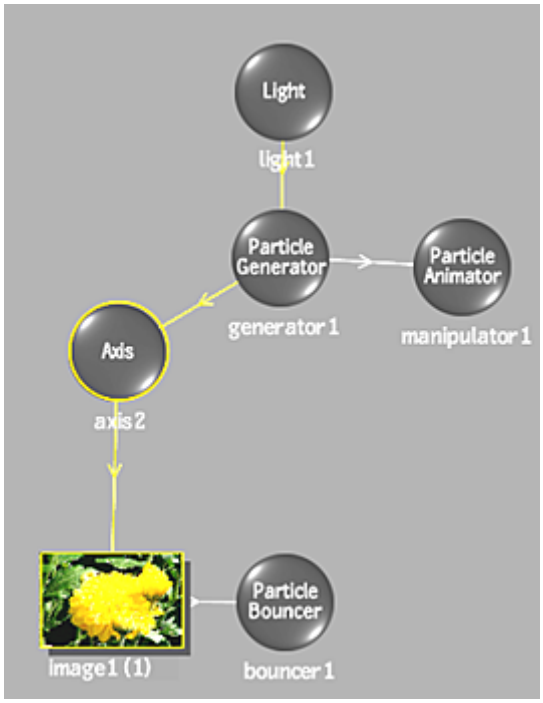
AO Degrade box Select an ambient occlusion quality display setting while the viewport is in Live Preview mode.

Blooming Degrade box Select whether or not to degrade Blooming while the viewport is in Live Preview mode.

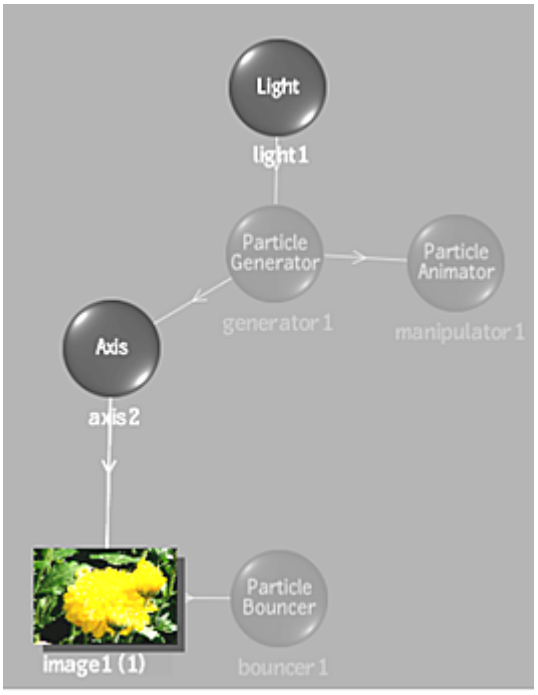
Saving and Loading Action Setups

You can save, load, and delete Action setups.

When importing a setup that contains non-supported objects (such as particles and deformations in Flame Assist), these objects are displayed as greyed out nodes in the schematic, and greyed out menus in the Object menu. These greyed out objects are read-only objects that can be viewed (in the image window and the Object menu), but not modified or linked to or from. In Channels view, you can view the parent channel for the non-supported object, and in Tracks view, you can slide or stretch the parent channel. If you modify the setup and re-save it, you can then open it in the application that supports the objects, and modify all objects.



Schematic of Flame setup



Schematic of same setup opened in Flame Assist

In some cases, due to hardware differences, multiple instances of objects may not be supported when importing a saved setup. In this case, a warning message appears detailing which objects are not supported. The names of the non-supported objects also appear in red in the Action schematic.

To save an Action setup:

1. In Batch or Batch FX, select the Action node. Click Save Node (found next to the node name field). If you entered Action through the timeline or Tools tab, you can click Save directly in the Action menu.

   The file browser and Save menu appear.

2. From the Save Action box, select the format for saving the setup.

Select:	To save:
All	References to all clips in the Media list.
Selected Objects	The selected objects, their media and media settings. Any expressions on channels of selected nodes are baked to the channels before saving, and any Mimic links to or from non-selected nodes are removed before saving. If a node is selected that is mimicked from a non-selected node, though, the animation channels are copied from the node before saving.
Raw	Selected animation channels as a user-readable ASCII file. In the file, each line corresponds to one frame, and each column consists of one frame number and the value of the animation curve at that frame. All saved information starts at frame one. A file with the extension *.raw* is saved in the *.../action* directory by default. At least one channel must be selected in the Channel Editor.
Text	The current text settings, including font, character size, kerning, italics, depth, beveling curve, and text string properties, all of which can be loaded in another Action session. A file with the extension *.atext* is saved in the *.../action* directory by default. A 3D text node must be selected in the schematic.
Preferences	The current Action settings as user preferences. A file with the extension *.pref* is saved in the */opt/Autodesk/user/<user_name>/action/pref* directory.
Defaults	The current Action preferences as Action's new default settings. To restore Action's factory default settings, select Factory Defaults in the Load menu.

3. Type a name for the setup file and click Enter.

   The name appears in the Name field.

   By default, the setup is saved in the */opt/Autodesk/project/<project_name>/action* directory. Using the file browser, you can save setups to the directory of your choice.

4. Once you have saved an Action setup, you can quickly re-save it by clicking Save again.

   TIP If you accessed Action from the Tools tab, you can click Revert to revert to the last saved setup. All changes made since the previous Save operation are undone.

To load a setup in Action:

1. In Batch or Batch FX, select an Action node. Click Load Node (found next to the node name field). If you entered Action through the timeline or Tools tab, you can click Load directly in the Action menu.

The file browser and Load menu appear.

NOTE When you access the file browser through Load, you also have the option of deleting existing setups.

2 In the Load box, select the format for loading the setup.

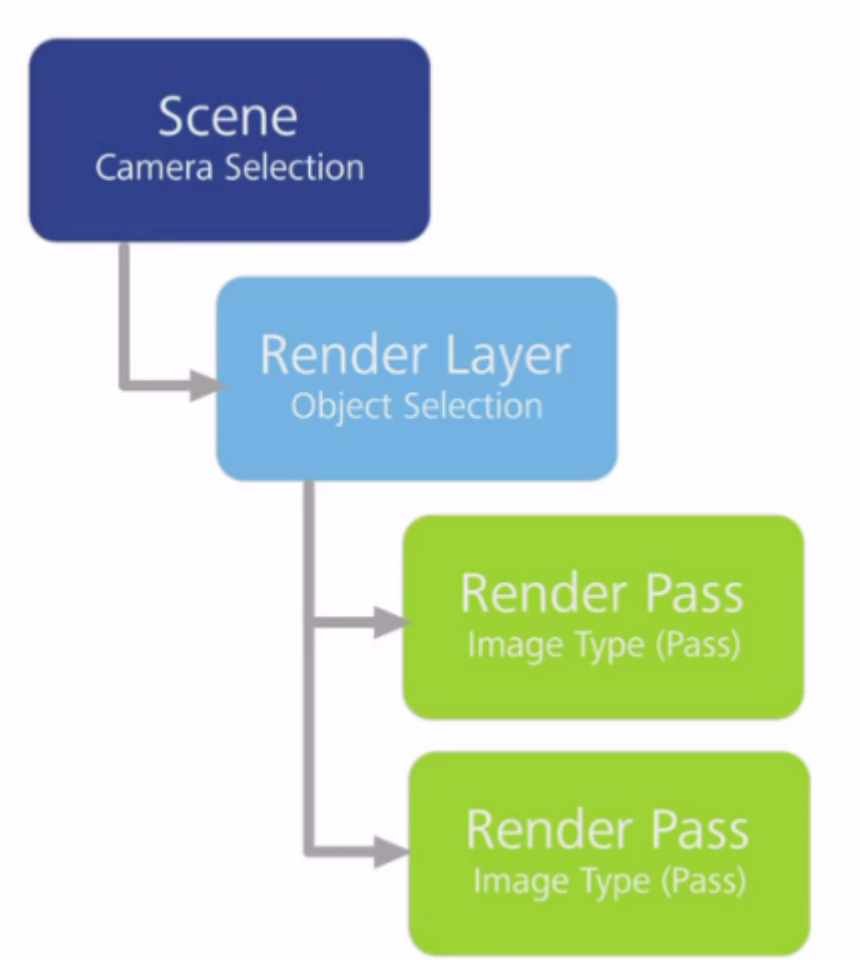
Select:	To:
All	Load the clips in the selected setup into their corresponding media. The media in the Media list is replaced with the loaded clips. If a clip cannot be found, Action searches for it and loads it automatically to Action. If the clip still cannot be found, Action displays the missing clip name in red in the Media list. A surface that uses a missing clip appears as an outline in the image area and is shown in red in Schematic view.
No Clips	Load a setup without its clips. The current media in the Media list remains the same.
Add Nodes+Media	Add nodes and media from the setup file. This option appends the schematic from the setup file to the current schematic, and wherever possible, media from the setup file fills empty slots in the Media list. Remaining media from the setup file is appended to the end of the Media list. Enable Load Cameras if you want to include the cameras saved with the setup.
Add Nodes	Add only the nodes from the setup file. This option appends the schematic from the setup file to the current schematic. Enable Load Cameras if you want to include the cameras saved with the setup.
Raw	Load raw animation data to a selected channel in the Channel Editor.
Text	Load the text setup files. The text settings are loaded into Action's Text menu.
Preferences	Load a file containing Action preferences.
Factory Defaults	Load original Action default settings. Selecting this option prompts you to confirm that you want to restore factory defaults and returns you to the Action menu.

3 Select the setup you want to load.

NOTE Sample Action setups are provided in the *~/examples/action* directory.

About Rendering Outputs from Action

Use the multi-pass rendering capabilities of Action's Output menu to manage your outputs. Options are available to help you set up and prepare many different types of outputs (based on a specific camera), much as you would with render passes in a 3D application. You can process multiple outputs at once, from your complete Action scene, to specific Shadow or Z-Depth outputs, for example.



NOTE If you accessed Action from the timeline, the output consists of a single Render Layer with Comp and Matte enabled.

Action Output Workflow

When using the Action Output menu, you usually follow the options from left to right.

Step:	Action:
1	Use the Render Layers list (page 600) to add, copy, delete, and rename render layers.
2	Set Outputs Options (page 601): select the Primary output type, the camera to output, and any render passes per render layer.

Step:	Action:
3	Set Rendering and Ambient Occlusion Options (page 603), as necessary.
4	Select objects (page 605) to output.
5	Render your outputs from Action (using the options in the Render button and dropdown list).

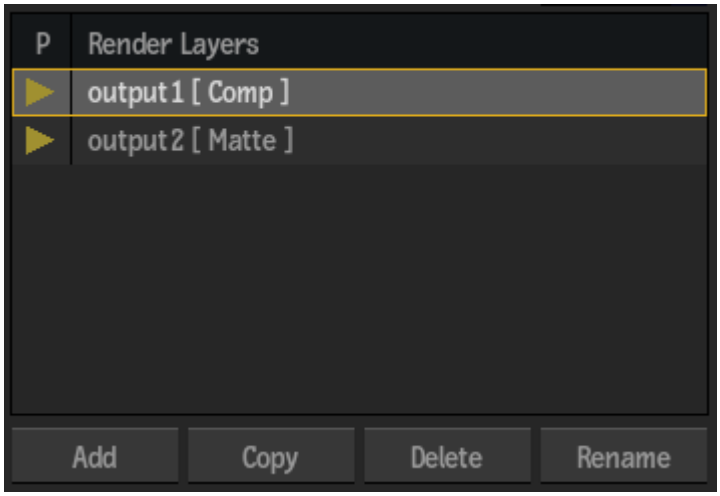
To access the Output menu, click the Output button.

Render Layers List Settings

Use the Outputs List to organize the outputs you want to render from Action. You must have at least one output in the Outputs List. By default, a Comp output exists, which is set to render the entire comp and result camera.

NOTE If you accessed Action from the timeline, you are limited to a single Render Layer with Comp and Matte enabled, and certain settings in the Outputs List are unavailable.

When you save an Action setup, your outputs are also saved.



Render Layers List Lists all of the existing render layers. Use the buttons below to populate the list with the render layers you want. To deselect an render layer from rendering (but keep it in the list), click the yellow arrow beside the output.

Add button Adds a new render layer to the list. Use the Rename button to change the name to something more meaningful.

TIP Use the Output Naming settings in the Action Setup Preferences (page 590) menu to set default naming conventions for your outputs.

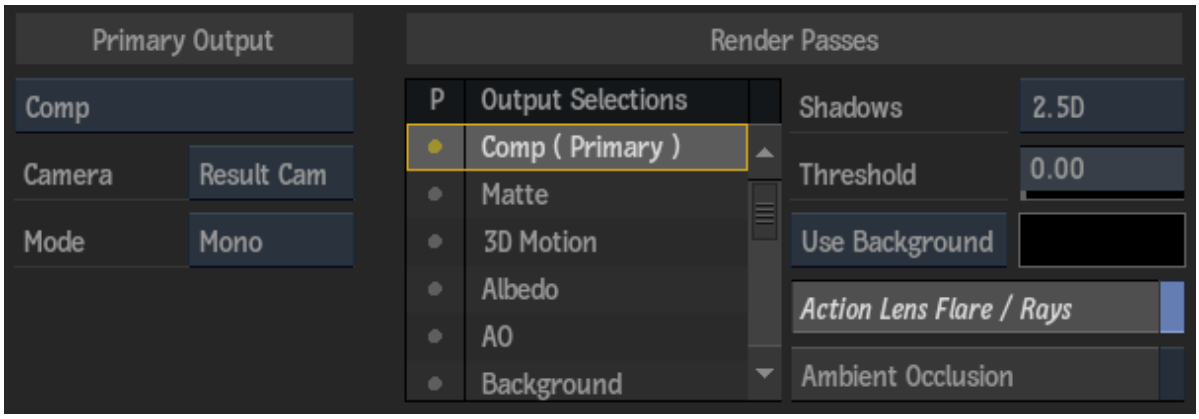
Copy button Creates a copy of a selected render layer.

Delete button Deletes the selected output from the list.

Rename button Opens the onscreen keyboard to rename the selected render layer in the list. Renaming a Render Layer automatically renames all output sockets enabled from that Render Layer.

Outputs Options

For each item in the Render Layers List, you can refine the parameters using the Outputs tab.



Primary Output Section

Primary Output box Select a primary output type. Render Passes options change based on the primary output type.

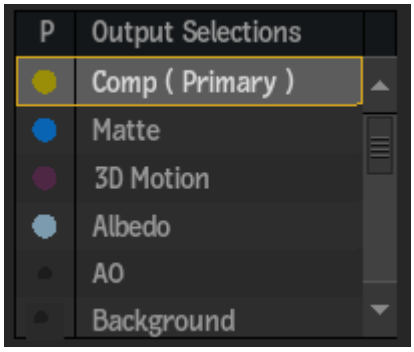
NOTE If you accessed Action from the timeline (or when using the GMask Tracer), a single Render Layer is available with Comp and Matte enabled.

Camera box Select which camera or stereo rig to assign to the render layer.

Mode box Select a render mode. Selecting the Stereo mode and a stereo camera results in left and right clips when processed. Selecting Stereo mode and any monoscopic camera results in two identical clips. Selecting Left, Right, or Mono results in a single output, regardless of the camera selected.

Render Passes Section

In the Output Selections List, the primary output is indicated, such as Matte (Primary). You can add as many other render passes to an output as you want. Simply click the dot to the left of the pass name (a coloured dot indicates that a render pass is enabled for the selected render layer).



TIP To see the result of a render pass without enabling it for the selected render layer, click the name of the render pass in the Output Selections List.

The following render passes are available in Action. Some settings may be different depending on the selected render pass:

Comp Renders selected objects in the scene with their parameters.

Matte Renders the matte of the selected objects in the scene.

3D Motion Renders 3D absolute motion vectors in world space.

Albedo Renders the diffuse colour of selected objects in the scene.

AO (Ambient Occlusion) Renders a grayscale output that is dark in areas that light cannot reach and bright in areas where it can. Specific AO settings can be found in the Ambient Occlusion tab.

Background Renders the Action background from the Media list.

Emissive Renders the colours of selected objects in the scene.

GMask Renders the GMask transparency. Equivalent to selecting GMask Only in the Use GMask box of a Matte render pass.

Lens Flare (also includes Rays) Renders just the lens flares and rays in the Action scene. There is also an option to include lens flares and rays in a Comp render pass.

Media Matte Available as a primary output type, but not as a render pass. Renders the matte of the media specified in the Media field.

Motion Vectors Renders 2D motion vector information.

Normals Renders the normals of selected objects in the scene.

Occluder Renders a GMask occluder when attached to a post-processing effect, such as a lens flare.

Position Renders world absolute pixel coordinates of a scene or a selection of objects.

Reflection Renders the reflection map settings of selected objects in the scene.

Roughness Renders the roughness in a physically-based rendering (PBR) shader.

Shadow Renders a white image with greyscale regions that represent the shadow cast coverage.

Specular Renders the specular map settings of selected objects in the scene.

UV Renders the UV map settings of selected objects in the scene.

Z-Depth Renders the Z-depth of selected objects in the scene.

Z-Depth HQ to be used with Matchbox shaders. Renders a 32-bit Z-depth output encoded into a 16-bit buffer.

Outputs Settings

Depending on the output selection, some of the following settings may appear:

Shadow Output Type box Select the type of shadow cast to output for a Comp or Shadows render pass.

NOTE When set, this becomes the default Shadow Type for any new Shadow Cast nodes.

Threshold field Displays the value at which the alpha is included in the depth of the output.

Comp box Select whether to render the scene over a background or over a colour you choose using the colour picker.

Comp colour pot Displays the colour that the comp is rendered over. Editable.

Action Lens Flares / Rays button Enable to include lens flares and rays in a Comp render pass.

Ambient Occlusion button Enable to process a Comp output with the ambient occlusion (AO) effect. Other AO settings appear in the Ambient Occlusion tab.

Matte Background box Select whether the object's matte is rendered on top of a black (default) or white background. When using the GMask Tracer node, you also have Input as an option. Select Input to set the Media Front and Matte as the Background for both the Comp and Matte outputs respectively.

Use Gmask box Select whether to use the garbage mask transparency or render the GMask colour. Available for Matte render passes.

Blending box Select whether to use a master blend option (Override) or to use the blend set in the Blending Mode box (Keep Blend). Available for Matte render passes.

Blend Mode box Select an option to determine the blending mode of the mask when it overlaps with another mask. Available for Matte render passes when Override is selected in the Blending box.

Bit Depth field This locked field displays that this output type is locked to a 16-bit floating point output. Non-editable.

Motion box Select an option to determine the direction of the motion. Select Average to produce a mix between previous and next frame. Available for 3D Motion and Motion Vector render passes.

Backward button Enable to set the direction of the vectors to backwards. Available for 3D Motion and Motion Vector render passes.

Motion Vector Range box Select whether the vector data is absolute or normalized. Available for Motion Vector render passes.

Camera GMask Only button Enable to display GMasks connected to the current camera. In this case, GMasks do not interact with the scene, and are only used in the post-processing pipeline. Available for GMask render passes.

Light box Select which light to include in a shadow render pass.

Filtering field Displays the amount of smoothing in a shadow render pass. Editable.

Maximum Z Difference field Displays the maximum amount of Z depth information to take into account for smoothing a shadow render pass. Editable.

Media field Displays the number that corresponds to the media in the Media list for a Media Matte output. Editable.

Rendering and Ambient Occlusion Options

Rendering Tab

Use the settings in the Rendering tab to further refine your output. These settings are applicable to all render passes of the same render layer.

Ambient button Enable to process an output with the ambient effect.

Emissive button Enable to process an output with the effect of an emissive map in the scene. An emissive map uses colours to simulate a glowing effect within the texture.

Specularity button Enable to process an output with the specularity effect.

Reflection button Enable to process an output with the reflection effect.

Transparency button Enable to include the alpha transparency of the objects in the output.

GMask button Enable to process the output with garbage mask effects.

Accumulation box Select an accumulation processing option for the output (accumulation comprises AA, Motion Blur, DOF, and HWAA settings). You can select to apply accumulation to RGBA outputs only (Comp, Matte, Albedo, Emissive, Reflection, Shadows, and Specular), or to all outputs that support it. This setting also affects Camera FX rendering speeds.

Anti-Aliasing button Select whether to apply anti-aliasing to the output (when the Accumulation box is set to On or RGBA).

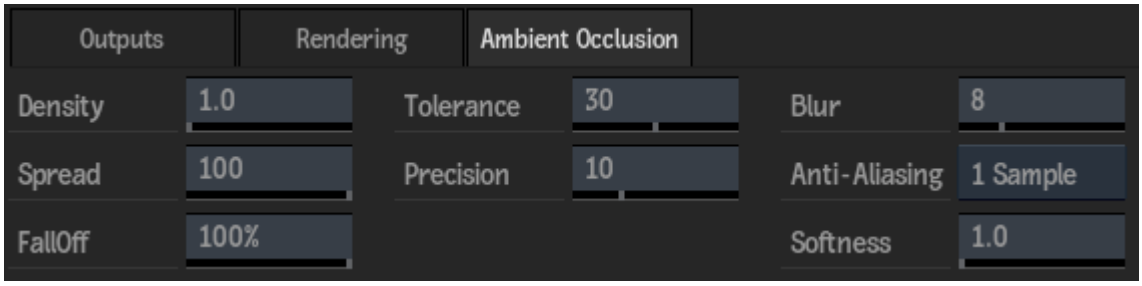
Motion Blur button Enable to include motion blur processing in the output.

Depth of Field button Enable to process the selected output with the Depth of Field settings defined in the Action Setup menu.

Shading button Enable to include shading effects for each output.

Ambient Occlusion Tab

Ambient occlusion refers to the blocking of indirect or diffuse light on an object. Ambient occlusion is caused by indirect light's inability to bounce around and illuminate areas that are blocked by a nearby object that absorbs the light rays. These subtle variations in lighting are visual clues for our eyes to detect surface details and distinctions that would otherwise be washed out and unnoticeable. Ambient occlusion adds realism to your scene by adding shadows in crevices, nooks and crannies, and so on. For each surface point, it calculates how much light is blocked by other geometry.



These settings are available for the AO type, and for the Comp type, if Ambient Occlusion is enabled in the Render Passes options.

Density field Displays the amount of darker areas to include in the ambient occlusion output. Editable.

Spread field Displays the width of the ambient occlusion effect. Editable.

Falloff field Displays the amount of falloff around the edge of the ambient occlusion effect. Editable.

Tolerance field Displays the starting point at which darker areas are included in the ambient occlusion output. Editable.

Precision field Displays the number of steps to take into account on rays cast from the surface of the object. Editable.

Blur field Displays the amount of blur applied to the ambient occlusion. Editable.

Anti-Aliasing Samples box Select an ambient occlusion sampling level.

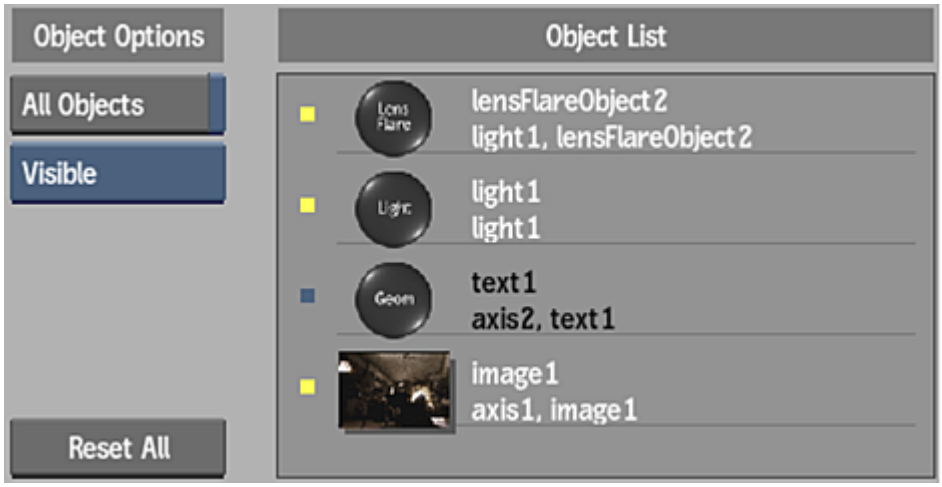
Softness field Displays the softness value of the ambient occlusion sample. Editable.

Regen button Enable to dynamically refresh the image as changes are made to the ambient occlusion settings.

Selecting Objects to Output

You can select objects to include in your output by using the Object Options and Object List, or the Edit Output mode. Objects that are part of an output are displayed in the Object List. For each selected object, you can decide if you want it to be visible in the final render or not. Non-renderable objects, such as cameras, are always included in outputs.

TIP When you create a new output in the Outputs list, it is empty by default. Enable All Objects to display all objects in the scene.



To select an object:

1. In the Object List, click an object.

   Selected objects are highlighted in the Object List, schematic, and image window.

2. To select additional objects, `Ctrl`-click another object.

   NOTE When you select multiple objects with different visibility settings (Visible or Occluded), Mixed is indicated in the Visibility box.

To output all objects:

1. In the Object Options, enable All Objects.

   All objects in the scene, as well as any object you subsequently create, are included in the selected output.

To set the visibility of an object:

1. Select an object from the Object List.
2. From the Visibility box, select one of the following:

Occlude Only Renders the object as black, excluding it from the shading effect. The result is a perceptual hole where the object was originally located.

Visible Renders the object in the final render.

NOTE When the Output type is set to Normal or Z-Depth, the objects in Occlude Only mode are rendered normally; it will be part of the Normal or Z-Depth output. The Visibility mode is ignored.

To add or remove an object from the Object List:

1 From the Tools box, select Edit Output.

NOTE In Edit Output mode, All Objects in the Object Options is disabled when you remove objects from the selected output.

2 Add or remove objects from the output by doing one of the following in the schematic:
- Click an object. The complete tree of the object in the schematic must either be selected or grayed out for it to be included or excluded from the output.
- Press `Ctrl` and drag in the schematic to add or remove multiple objects (or trees) from the output. This acts as a toggle between selected or greyed out (included or excluded from the output).

To set all objects to Visible mode:

1 Click Reset All.

All objects in the selected output are set to the Visible mode.

Object Options

All Objects button Enable to output all objects in the Object List.

Visibility box Select a visibility option for the selected object.

Reset All button Click to set all objects to Visible mode in the Object List.

Action Media

You should have a working understanding of Action media and the relationship between media and surfaces/maps to be successful in using Action. Media has the following characteristics:
- Each front and matte clip combination that you load into Action is called *media*.
- The clips you load into Action are listed in the Media list.
- When you first open Action, you load the front clip and matte clip for the first media and a common back clip for all media.
- Media must have a front and matte of the same resolution, but each media can have a different resolution from the other.
- You can work with front-only or matte-only media. In this case, the empty front or matte is replaced with an internal white frame of the same resolution.
- The same media can be applied to multiple surfaces. Any cropping, blurring, or colour correcting that you apply to one instance of media is applied to all the surfaces for that media. For example, if you blur media, all of the surfaces using that media are blurred.

NOTE If you access Action from the timeline, you are limited to one front/matte media. Therefore, some Media menu settings are not available.

Working With Media in the Media Menu

When you open Action, you load the front and matte clips for the first media. You can then load any additional media.

NOTE In Batch or Batch FX, you can add direct or indirect media to an Action node. See Adding Media to an Action Node (page 518)

The clips you load for each media appear in the Media list. The back clip you selected to access Action appears in the first row of the Media list and is assigned the letter B in the # column. The second row in the Media list contains the first front and matte clips you selected when accessing Action. The length of each clip appears in parentheses beside the clip name. A new line is added to the Media list each time you add media. For stereoscopic projects, you can apply media to the left and right eye.

You can sort the Media list by list number, front clip name, or matte clip name. As you view clips, you can look at the original front, back, or matte clip individually, as well as the results of colour correcting and keying a matte.

To access the Media menu, click Media in the Action menu. To help identify media, `Alt+click` a surface in the schematic, result, or camera view to automatically select the media in the Media list. If the Media menu is not selected, press `Alt` and double-click the surface to switch to the Media menu, and automatically select the media in the Media list. Also, when you select multiple surfaces, you can press `Alt` to select all associated media in the Media list. In this case, the first item selected is framed in yellow in the Media list as the current selection.

Media Menu Tips

Use these tips when working with clips and media in the Media menu.

- Different colours and shades are used as visual cues within the Media menu. The current media is outlined in yellow, and the selected media is highlighted in light grey. Selected fields of the media are highlighted in grey.
- `Ctrl`-click fields or media to add to a selection, or `Shift`-click to add a range to a selection. If you `Ctrl`-click a field that is already selected, all selected fields of this type in other selected media become deselected. However, if you continue to press `Ctrl`, you can then click any field, and all fields of the same type in other selected media become selected.
- Click a numeric field to display the calculator, or click and drag to increase or decrease the value in a field.
- `Ctrl+Alt`-click a field to reset the field (and all other selected fields).
- Double-click the Divide or Crop field to toggle the effect on or off.
- Double-click the MK, CC, CW, or Blur field to enter its menu.
- Click a front or matte media to select all the displayed fields.
- Use the `F` keyboard shortcut to toggle the matte media display in the Media list on or off. You can also set this behaviour with the Show Front Only button in the Action Setup menu.
- Resize the column widths in the Media menu by dragging the column dividers.
- `Ctrl`-click to multiselect media for applying to the left or right eye of a stereo object. The media that you select first (odd numbers) are applied to the left eye. Even numbered media apply to the right eye.

Adding Effects to Media

The Media list allows you to easily add and edit the following effects and tools to your clips. The same media can be applied to multiple surfaces, so any effects that you apply to one instance of media is applied to all

the surfaces for that media. For example, if you blur media, all of the surfaces using that same media are blurred.

#	Name	Lock	Slip	MK	Blur	X Blur	Y Blur	CC	CW	Divide	Crop	Top	Bottom	Left	Right
B	[rusty_silos] (31)	Off	0		✓	15.00	0.00					0	0	0	0
1F	GWindow (96) #1	7	11		✓	9.00	0.00	✓			C	0	0	0	0
1M	GWindow (96)		11			0.00	0.00				S	0	0	0	0
2F	[running_dog] (1)	Off	0			0.00	0.00				C	0	0	0	0
2M	[running_dog] (1)		0			0.00	0.00				S	0	0	0	0
3F		0	0			0.00	0.00				C	0	0	0	0
3M			0			0.00	0.00				S	0	0	0	0

NOTE If you are looking for Shadow Softness settings in the Media list, they can now be found directly in the Shadow (page 645) menu, allowing you to set shadow softness by object, instead of by media.

The effects and tools in the Media list appear from left to right in the order that Action processes them:

Lock and Slip

Use Slip to create a timing offset for your clip and Lock to lock media at a certain frame.

1. In the Media list, drag the slip field for your chosen media.

   A positive value starts the clip at the specified frame. A negative value creates a freeze frame effect where the first frame of the clip is held by the specified number of frames.
2. To lock the media at a certain frame, drag the Lock field to the desired frame number, or click the Lock field to display the calculator to enter a frame number. `Ctrl+Alt` click the field to return it to Off.

NOTE If you want to animate the Slip values, use the Channel Editor. The channels are found in their respective media and are named *slip* and *matte_slip*. The Background Slip channel is found in **Scene ➤ Media ➤ Background ➤ Slip**.

Modular Keyer

When you load media into the Modular Keyer from Action, the front and matte clips for the selected media are loaded with the back clip and appear at the beginning of the processing pipeline. The front clip is used as the Front clip and the matte clip is used as the Key In clip for the processing pipeline. The back clip is used as the Back clip in the processing pipeline and is loaded as a reference only. Any modifications you make to the Back clip in the Modular Keyer are not used when you return to Action.

NOTE If the Action back clip is a different resolution than the front clip, a Resize node is added to the Back pipeline when entering the Modular Keyer.

1. In the Media list, double-click the corresponding MK field for your chosen media to enter the Modular Keyer.
2. As you work in the Modular Keyer, select Action Context from the View box to preview the result.
3. Click Return to return to Action. You do not have to click Render; the modifications are automatically applied to the media in Action.
4. To mute a Modular Keyer effect, press `Alt` and click the MK media. The MK in the field turns black to signify the effect is muted. To unmute the MK effect, press `Alt` and click the field again.

   The MK field is displayed in red if there is an error within the Modular Keyer, such as a broken input connection. Open the Message History viewer to see the error, and return to the Modular Keyer to fix the error.

NOTE Since source nodes are applied before any Media list effects, if you do not want to apply the Modular Keyer effect on a source node, you'll need to create a new media layer for the source node output.

Blur

You can quickly add Gaussian blur along the X and Y axes in the Media list, or enter the full Blur editor to have access to all blur settings, including other blur types, such as Radial and Defocus.

1. In the Media list, drag the X or Y blur values to apply a Gaussian blur, or double-click the corresponding Blur field for your chosen media to enter the full Blur editor.

   TIP `Alt`-drag over a front or matte X or Y Blur field to change both values proportionally; or press `Alt+Shift` and drag over any front or matte blur field to change all four fields proportionally.

2. As you work in the Blur editor, select Action Context from the View box to preview the result.
3. Click Exit to return to Action. You do not have to click Render; the modifications are automatically applied to the media in Action.

   If you applied a blur type other than Gaussian, notice that the X and Y blur fields are not editable, and display as black. If you switch to a Gaussian blur type in the Blur editor, and return to Action, you are able to use the X and Y blur fields in the Media list (they display as white).
4. To mute a Blur effect, press `Alt` and click the white check mark in the media field. The check mark turns black to signify the effect is muted. To unmute the Blur effect, press `Alt` and click the check mark again.

Colour Corrector and Colour Warper

You can apply separate Colour Corrector and Colour Warper effects on Action media.

1. In the Media list, double-click the corresponding CC or CW field for your chosen media to enter the Colour Corrector or Colour Warper.
2. As you work in the Colour Corrector or Colour Warper, select Action Context from the View box to preview the result.
3. Click Exit to return to Action. You do not have to click Render; the modifications are automatically applied to the media in Action.
4. To mute a CC or CW effect, press `Alt` and click the white check mark in the media field. The check mark turns black to signify the effect is muted. To unmute the CC or CW effect, press `Alt` and click the check mark again.

Divide

When compositing with images generated by rendering 3D objects over a black background, unwanted black often appears on translucent or anti-aliased areas of the media. Use the Divide tool to divide the front media by its associated matte media, which eliminates the black edges in the composite.

1. In the Media list, select the media containing the unwanted black.
2. Double-click the Divide field of the media containing the unwanted black.

   The selected media's front is divided by its matte, resulting in correct colour in areas of transparency.

NOTE A blending premultiplication setting is also present in individual surface and geometry menus.

Crop

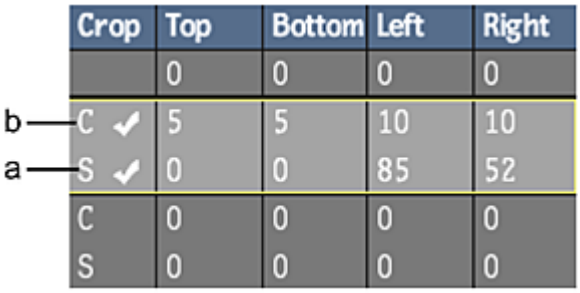
When you crop media, the front and matte clips are cropped together. You cannot crop the back clip. If you applied the media to more than one surface, all the surfaces are cropped.

You can animate a crop by changing the size and shape of the Crop box at different keyframes. You can also animate the softness of the Crop box.

NOTE Animating a crop in the Channel Editor does not enable the Crop field in the Media list.

To crop a clip with the Media list:

1. In Action, click Media.
2. Set values for cropping and softness in the Top, Bottom, Left, and Right fields. Once you set a value in one of the Crop or Softness fields, white checkmarks indicate that Crop and Softness are enabled.



Crop	Top	Bottom	Left	Right
	0	0	0	0
C ✓	5	5	10	10
S ✓	0	0	85	52
C	0	0	0	0
S	0	0	0	0

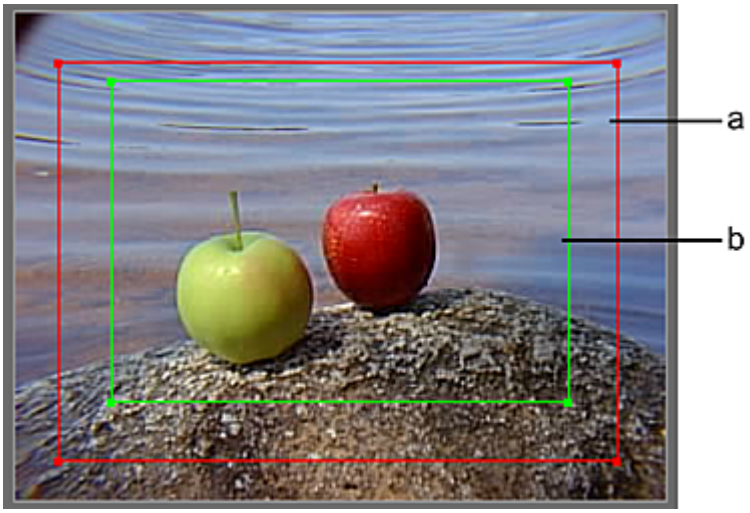
(a) Softness area of Crop field (b) Crop area of Crop field

TIP `Alt`-drag over one of the fields to change all four values proportionally.

To crop gesturally in the image window:

1. Double-click the Crop field to enable Crop and Softness.
2. In the View box, select Media Front or Media Matte.

   The clip appears in the image window with a red outline, which indicates the Crop box, and a green outline, which indicates the Softness box.



(a) Crop box (b) Softness box

TIP You can use two viewports to view the crop interactively on both the front and the matte clips.

3 Set the corners of the Crop box and Softness box by dragging either the corners or edges of the red and green outlines.

The crop and softness values in the Media list are updated as you drag.

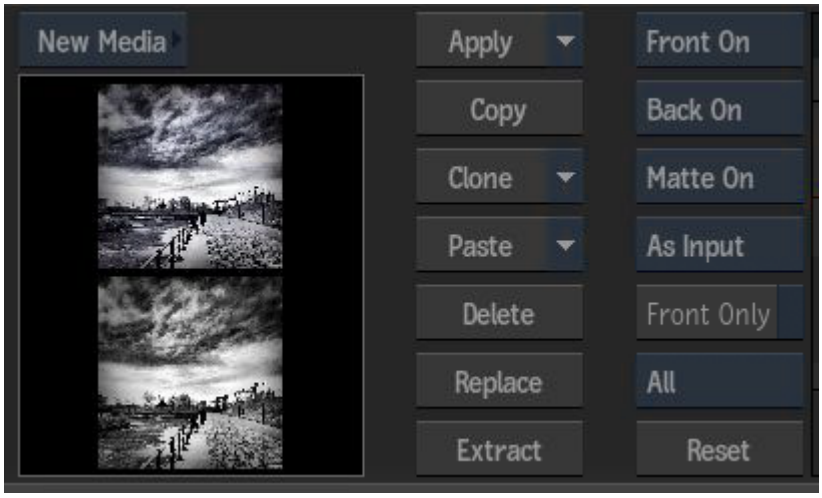
4 Click Result.

The cropped media appears in the image window, and the front and matte clip share the same crop.

5 To mute the Crop and Softness values, double-click the Crop field. To mute only the Softness, double-click the Softness area of the Crop field. When muted, a black checkmark appears in the Crop field.

Media Menu Settings

NOTE If you access Action from the timeline, you are limited to one front/matte media. Therefore, some Media menu settings are not available.



New Media button Click to open the workspace to add new media. Available when Action is accessed from the Tools tab.

Media List box To add media directly to the Media list, select New Media. To add an indirect layer to the Action node, select New Input. Available when Action is accessed from Batch or Batch FX.

Image Proxy Displays a proxy of the currently selected media.

Apply button Applies the selected media in the Media list to the selected surface in the scene.

Apply dropdown list For stereoscopic clips, select how the selected media in the Media list is applied to the selected surface in the scene.

Copy button Copies media and effects (for example, Blur and Crop settings) of the selected media in the Media list. Use the Paste options to decide how you want to use the copy/paste operation.

Clone button Clones the selected media, or media and effects onto a new media entry in the Media list.

Clone dropdown list Select whether the clone operation clones just selected media, or media and effects.

Paste button Pastes the copied parameters onto the selected media or media entry in the Media list.

Paste dropdown list Select how the paste operation is applied in the Media list.

Select:	To:
Paste Media Only	Paste the copied media onto a selected empty media entry, without any effects (such as Blur or Crop settings).
Paste FX/Media+FX	If pasting onto an existing media entry, pastes only the copied effects. If pasting on an empty media entry, pastes both the copied media and effects.

Delete button Deletes the selected media from the Media list.

Replace button Click to replace the selected media in the Media list from the Clip Select area.

Extract button Click to convert the selected media in the Media list from direct to indirect media.

Front Clip box Select an option to edit the front clip's visibility.

Select:	To:
Front On	Display the front clip for the selected media.
Front Off	Hide the front clip for the selected media.

Back Clip box Select an option to edit the back clip's visibility.

Select:	To:
Back On	Display the back clip.
Back Off	Hide the back clip.

Matte Clip box Select an option to edit the matte clip's visibility.

Select:	To:
Matte On	Display the matte clip for the selected media.
Matte Off	Hide the matte clip for the selected media.
Matte Invert	Invert a matte. Black areas will be made white, and white areas will become black. **TIP** To invert multiple matte clips, `Ctrl`-click the matte media and select Invert in the Matte Clip box.

TIP Use options in the View box to display selected media in the image window: Media Front (or press `F1`), Media Back (`F2`), and Media Matte (`F3`).

Media Rendering box Select a rendering option per media.

Front Only button Enable to display only the front media in the Media menu. Disable to display both the front and matte media.

Reset Choice box Select the Media list properties to reset.

Reset button Resets the properties selected in the Reset Choice box.

Action: Moving, Rotating, and Scaling Objects

Transformations change an object's position, size, and orientation without changing its shape. "Transform" is basically a fancy way of saying "move, scale, and rotate".

Transformations are relative to an object's (or component's) pivot point, and take place along/around either the world axes, object axes, or local axes.

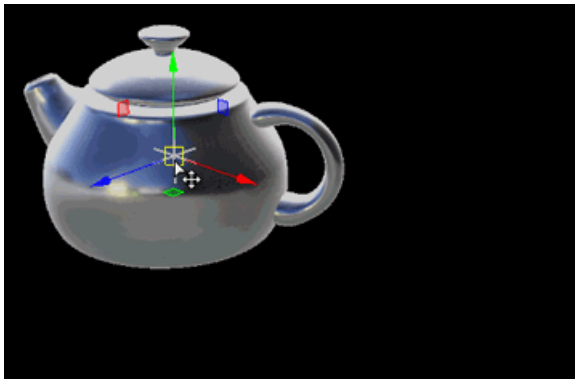
Moving an Axis, a Light, or a Camera

There are two methods to move an axis, a light, or a camera:

- With a manipulator in the viewport
- With the numeric fields in the object's properties

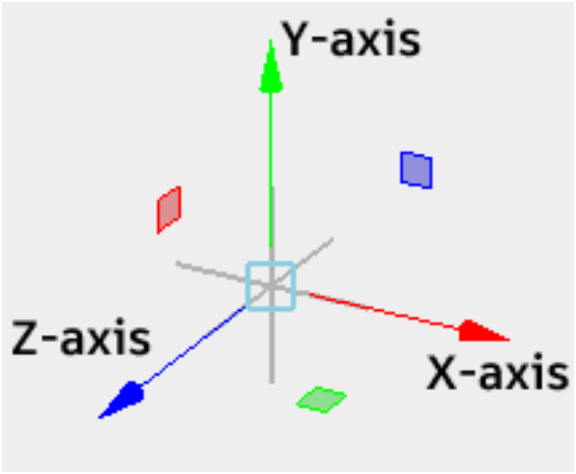
To freely move an object:

1. In the Tools box, choose Translate. Or press **T**.
2. Click and hold the axis, camera, or light to move.
3. Drag the object where you need it.



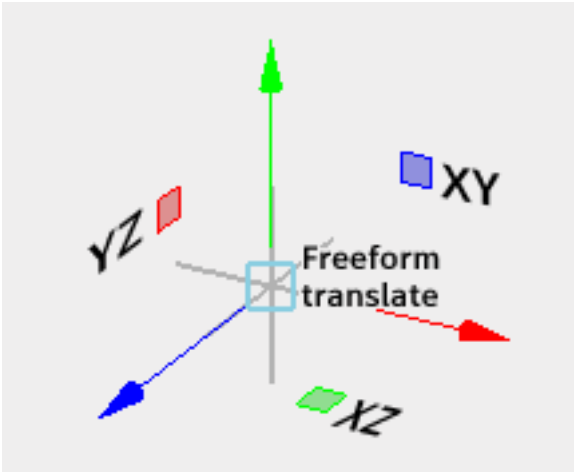
To constrain the movement of an object along one axis:

1. In the Tools box, choose Translate. Or press **T**.
2. Click the axis, camera, or light to move to display its manipulator.
3. Move the cursor over one of the manipulator's axes to make it active (yellow).
4. Click and drag the axis to move the object where you need it.



To constrain the movement of an object to a plane:

1. In the Tools box, choose Translate. Or press **T**.
2. Click the axis, camera, or light to move to display its manipulator.
3. Move the cursor over one of the manipulator's planes to make it active (yellow).
4. Click and drag the plane to move the object where you need it.



To move an object precisely along one axis:

1. In Schematic view, click the object to move.
2. Do one of the following to edit the X, Y, and Z position fields:
   - Drag a position field to move the object.
   - Click a position field to display the calculator. The object moves as soon as you click Enter.

Rotating an Axis, a Light, or a Camera

There are two methods to rotate an axis, a light, or a camera:

- With a manipulator in the viewport
- With the numeric fields in the object's properties

To rotate an object:

1. In the Tools box, choose Rotate. Or press **R**.
2. Click the axis, camera, or light to display its manipulator.
3. Move the cursor over one of the manipulator's ring to make it active (yellow).
4. Click and drag the ring to rotate the object.

To precisely rotate an object:

1. In Schematic view, click the object to rotate. This can be an axis, a light, or a camera.
2. Do one of the following to edit the X, Y, and Z rotation fields:
   - Drag a rotation field to rotate the object around that axis.
   - Click a rotation field to display the calculator, and then enter a value. The object rotates as soon as you click Enter.

Scaling an Axis

There are two methods to scale an axis:

- With a manipulator in the viewport
- With the numeric fields in the object's properties

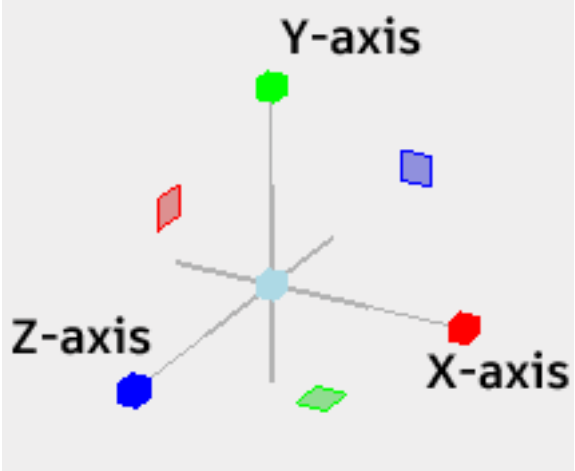
To scale an object uniformly in all directions:

1. In the Tools box, choose Scale. Or press **E**.
2. Click and hold the axis of the object to scale.
3. Drag cursor to scalle the object to scale uniformly in all directions.

To constrain the scaling of an object along one axis:

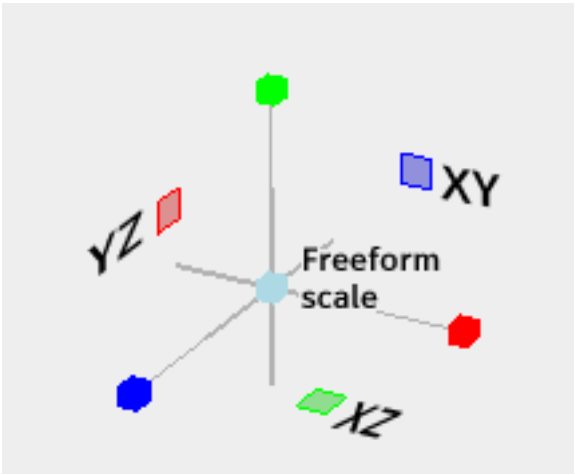
1. In the Tools box, choose Scale. Or press **E**.

2 Click the axis to display its manipulator.
3 Move the cursor over one of the axis' handles to make it active (yellow).
4 Click and drag the handle to scale the object along that axis.



To constrain the scaling of an object to a plane:

1 In the Tools box, choose Scale. Or press **E**.
2 Click the axis to display its manipulator.
3 Move the cursor over one of the manipulator's planes to make it active (yellow).
4 Click and drag the plane to scale the object in that plane.



To scale an object precisely along one axis:

1 In Schematic view, click the object to scale.
2 Do one of the following to edit the X, Y, and Z position fields:
- Drag a position field to move the object.
- Click a position field to display the calculator. The object moves as soon as you click Enter.

NOTE Enable **Prop** to scale uniformly in all directions.

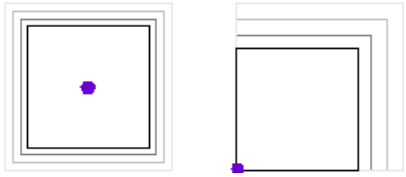
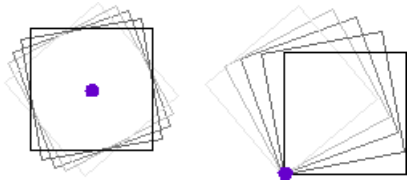
Moving the Centre

The centre point defines the position around which objects are rotated or scaled. By default, the centre point of an object is located at its centre. If you want to rotate an object around a specific point, like a forearm rotates from an elbow, you need to adjust the centre's position.

An axis' manipulator position represents the centre's position in 3D space.

To move the center of an axis:

1. In the Tools box, choose Select. Or press **M**.
2. Select the axis to re-centre. This displays its manipulator.
3. In the Tools box, choose Centre. Or press **Y**.
4. Drag the manipulator where you need the centre: the manipulator is now offset from the actual axis. The manipulator is what you use to transform an axis, any transformation to the axis is performed from the manipulator's point of view.

Transformation:	**Effect of the Centre on the Transformation:**
Move	Moves the object parallel to the centre.
Scale	Scales the object from or to the centre. 
Rotation	Rotates the object around the centre point. 

Camera Space, Object Space, and World Space

3D positions and transformations exist within coordinate systems called spaces. Action has three spaces:

- *World* space is the coordinate system for the entire scene. Its origin is at the center of the scene. The grid you can display from the Grid menu is in the world space.
- *Object* space is the coordinate system from an object's point of view. The origin of object space is at the object's manipulator, and its axes rotate with the object.
- *Camera* space is the coordinate system from the camera's point of view. The origin of object space is at the object's manipulator. The manipulator's XY plane is always the plane of the camera, and the Z-axis always points to the camera. Camera space makes it easy to change X, Y positions, and rotate in Z, no matter where you are looking at.

To change space, do one of the following:

- Use the **Space Selection box** button in the Viewport.
- Cycle through all three spaces using the keyboard shortcut `Space-W`.

The numeric fields displayed in an Object's menu always use the World space. Transformations using an object's manipulator in Object or Camera space are converted into World space coordinates in the object's menu.

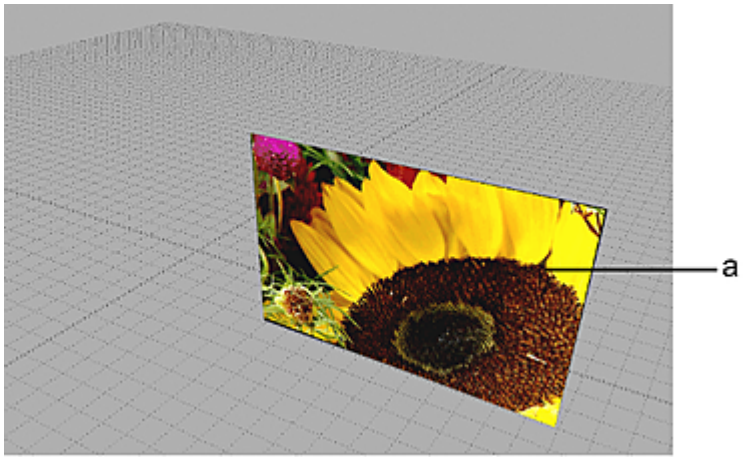
Action Axes

An axis is the element of an object or media that can be manipulated to determine the object's 2D or 3D space, position, and movement.

When you add certain objects to the scene, such as models or images, they are added with their own axes. Each axis is used to place its respective object in the scene. An axis is represented by the following icon.

TIP Simplify the Action scene: hide every axis parenting a texture by pressing `Space+H`.

All transformations that you apply to the selected axis are applied to the objects connected to the axis. For example, if the position of a surface's axis is set to 500, 100, 0, then its surface is placed at 500 on the X-axis, 100 on the Y-axis, and 0 on the Z-axis.



(a) Surface is placed in the scene at 500, 100, 0

You can also add an axis manually to the scene from the node bin.

Selecting an Axis

You can select an axis in any of the following ways:

- Click directly on the axis in the scene.

TIP If you have numerous overlapping axes in the scene, press Q to cycle through each axis until the axis you need is selected.

- Go to Schematic view and click the node for the axis.
- Display the Channel Editor and select the Axis folder or one of its channels.
- Use the Prev and Next buttons in the Axis menu to select the previous or next axis.

TIP If the size of an axis icon prevents you from selecting the right axis, resize the icons using **Action Preferences ➤ Preferences ➤ Axis Scale field**.

Action Views

Front, top, and side view are orthographic views used to position an object's axis in the scene. These views are useful for viewing the scene from another angle than the camera. From the View box (on the viewing/layout toolbar under the viewports), select an option to set the view in the image window.

Creating a Look-At Connection from an Axis

You can attach a look-at connection (page 579) between Action objects with axis characteristics (Axis, Camera, Light, and Projector). The connection allows the parent object to rotate to face the child look-at object, no matter where it is positioned. Look-At links can be animated; therefore you can link different objects from the parent at different frames. In the channel editor, you can see a lookAt channel for every look-at parent.

Applying Tracking Data to an Axis

To remove jitter, or track the movement of a feature in any Action media, you can apply tracking data to an axis. You can also perform more complex planar tracking of the media using the planar tracker.

Stabilize a Clip

To stabilize a clip from Action Axis:

1. From the Action Axis menu's Tracking tab, select the media to be stabilized from the Use Media field (to stabilize the Back media, select 0).
2. In the Axis menu for the selected media, make sure that the motion path is disabled (Path button).

   NOTE Do not move the axis from its default position at the centre of the image window before entering the Stabilizer, or else the tracking data will be overwritten. Use the offset axis to add an offset to the tracker movement instead.

3. Enable Rotation, Scale or both, as needed.
4. In the Tracking Mode box, select Stabilize and click the Enter Tracker button.

   The media is automatically loaded into the Stabilizer and the Stabilizer menu appears.
5. In the Stabilizer menu, enable Context to use the Action scene as the reference.
6. In the Stabilizer, position Tracker1 over the reference point that you want to track.
7. If you enable Rotation or Scale (or both), position Tracker 2 over a point that represents the rotation or change in size of the pattern.
8. Click Analyze to generate the translation data.

   NOTE You can fine-tune the tracking data if necessary. See Tracking Difficult Shots and Correcting Errors (page 1091).

9 Once the analysis is complete, click Return.

When you exit the Stabilizer, the X and Y Shift values for the reference position are automatically copied into the X and Y translation channels for the selected axis in Action.

TIP After stabilizing, you can set a reference frame that has no transformations by enabling the Adjust Offset button (in the Axis tab) at the selected frame.

Track a Clip

To perform one-point or two-point tracking in Action Axis:

1 From the Action Axis menu's Tracking tab, select the media to be stabilized from the Use Media field (to stabilize the Back media, select 0).

2 In the Axis menu for the selected media, make sure that the motion path is disabled (Path button).

3 In the Tracking tab, enable Rotation, Scale or both, as needed. The choice depends on the movement of the object to track and on the camera movement in the clip.

4 In the Tracking Mode box, select Track and click the Enter Tracker button.

The media is automatically loaded into the Stabilizer and the Stabilizer menu appears.

5 In the Stabilizer menu, enable Context to use the Action scene as the reference.

6 Position the tracker(s) over the pattern(s) that you want to track, and click Analyze to generate the tracking data. For one-point tracking, you use only one tracker to generate position information. For two-point tracking, you use a second tracker to generate rotation and/or scaling information. The Stabilizer obtains this information by comparing the position of Tracker2 to that of Tracker1.

- Tracker1 follows the horizontal and vertical translation of the reference point. You should position Tracker1 over a point on the pattern that you want to track.

  NOTE The position of the axis is synced with the position of Tracker1. Therefore, any existing animation on the Axis is overwritten by the tracking data.

- Tracker2 tracks the rotation and/or the change in size of the pattern. In the first frame, the rotation is always 0 and the scaling factor is always 100%. In subsequent frames, a rotation and/or scaling factor is added if the relative position of the two trackers changes. You should position Tracker2 over a point that represents the rotation or change in size of the pattern.

NOTE You can fine-tune the tracking data if necessary. See Tracking Difficult Shots and Correcting Errors (page 1091).

7 Once you are satisfied with the tracking, click Return.

The Axis menu in Action reappears. The tracking data is applied to the Axis media.

Planar Track a Clip

Planar tracking in Action uses tracking algorithms along with combinations of settings to perform the track. Results can vary depending on the footage, as well as settings applied. See Getting Better Results with the Planar Tracker (page 656).

To apply planar tracking to axis media:

1 Select the Axis you want to track.

2 In the Axis menu's Tracking tab, set the Use Media field to assign the back or front/matte media (select 0 if you want to track the Back media).

3 Adjust any Tracking Settings (page 910), as needed (such as Rotation and Scale).

4 Set Analysis Constraints. Since the Axis itself does not contain media, the planar tracker needs GMask splines to include or exclude to perform the tracking analysis.

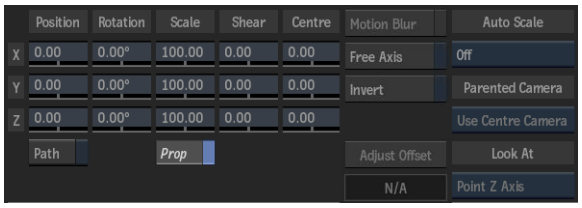
5 Click the Analyze button to perform the planar tracking analysis.

6 To perform additional analysis in a different direction from the first frame of the analysis, toggle the Direction box and click Analyze again.

Axis Menu Settings

When accessing Action as a Timeline FX, you have access to a quick menu with some of these Axis settings. To see the full Axis menu, click the Editor button to enter Action.

Axis Tab



X Position field Displays the position of the X axis. Editable.

Y Position field Displays the position of the Y axis. Editable.

Z Position field Displays the position of the Z axis. Editable.

Motion Path button Enable to animate the position of the axis using a spline drawn in the scene. Disable to animate the position of the axis using explicit animation.

X Rotation field Displays the rotation of the X axis. Editable.

Y Rotation field Displays the rotation of the Y axis. Editable.

Z Rotation field Displays the rotation of the Z axis. Editable.

X Scale field Displays the scale of the X axis. Editable.

Y Scale field Displays the scale of the Y axis. Editable.

Z Scale field Displays the scale of the Z axis. Editable.

Proportional button Enable to change the fields proportionally.

X Shear field Displays the shear of the X axis. Editable.

Y Shear field Displays the shear of the Y axis. Editable.

Z Shear field Displays the shear of the Z axis. Editable.

X Centre field Displays the centre of the X axis. Editable.

Y Centre field Displays the centre of the Y axis. Editable.

Z Centre field Displays the centre of the Z axis. Editable.

Motion Blur button Enable to use a motion blur effect for the selected axis (can only be used once the global Motion Blur is enabled in the Setup menu).

Free Axis button Enable to ignore transformations from parent axes.

Invert button Enable to invert numerical Axis menu settings. This does not affect settings of nodes connected to the Axis.

Adjust Offset button Enable to set the current frame as a reference frame that has no transformation data.

Reference Frame field Displays the frame number of the reference frame that has no transformation data. Non-editable.

Autoscale option box Select whether to autoscale when Position settings are changed relative to the camera (On Position Changes), or when Position, Rotation, Shear, or Centre settings are changed (On All Transforms).

Select:	To:
Off	Not use autoscaling on the image.
On Position Changes	Preserve the projected size of the image by automatically scaling when changing any Position parameters that affect the depth of the image relative to the camera.
On All Transforms	Preserve the projected size of the image by automatically changing Position, Scale, and Centre parameters (when any Position, Rotation, Shear, or Centre parameters are changed).

Parented Camera box Select which camera is used for the parenting offset when the immediate parent is a stereo camera. Choose left, right, or center camera. Active only when there is exactly one parent that is a stereo camera, or if a look-at connection is attached between the axis and the stereo camera.

Point Axis box Select which axis is pointed to the attached look-at object. Available only when objects are attached with a Look At connection. See Applying Look-At Connections (page 579).

Tracking Tab

Common Settings

Tracking Mode box Select which tracking mode to apply.

Enter Tracker button Click to open the Tracker menu to apply tracking data for the selected tracking mode.

Rotation button Enable to apply rotation data to the track.

Skew button Enable to apply skew data to the track.

Scale button Enable to apply scale data to the track.

Perspective button Enable to apply perspective data to the track.

Camera box Specify which camera to take into account when tracking or viewing in Action Object view (F8).

Use Media field Displays the number of the media associated with the analysis. To track the Back media, select 0.

Tracking View box When the media associated to the analysis is a Source node, select to track the media in the Source Front, Source Matte, or Source Result view.

Media FX button Enable to take into account blur and colour correct settings from the Action Media list in the analysis.

NOTE Specific Planar Tracking settings are the same whether you access them from the Axis, Perspective Grid, or GMask node. See Planar Tracking Settings (page 659).

Display Tab

Shape Reference button Enable to display an overlay of the reference frame when tracking in Object view (F8).

Shape Current button Enable to display an overlay of the current frame when tracking in Object view (F8).

Action Surfaces

You use surfaces to display media in the scene and then composite front and matte clips with a common back clip.

You should have a working understanding of Action media and the relationship between media and surfaces to be successful in using Action.

A surface is used to place a media in the scene. To use media that you load into Action, you must add a surface to the scene for that media. A surface has the following characteristics:

- A surface type can be flat, bilinear, perspective, or extended bicubic.
- The same media can be applied to multiple surfaces. Any cropping, blurring, or re-colouring that you apply to one media is applied to all the surfaces for that media. For example, if you blur a media, all of the surfaces using that media are blurred.

The first time you open Action during a session, an image surface is added to the scene using the first media (if Auto Image is enabled in the Action Setup menu). You must add a surface for each additional media before its clips can be used in the scene. You can then add textures and lighting effects, or change the surface's properties such as its shape, transparency, and specular highlight. Once added, the media appears in the Media list.

By default, a Flat surface is added. You can change the surface type in the Shape box of the Surface menu. See Changing the Shape of a Surface (page 633).

You control the position of the surface using axis, rotation, scale, shear, and other attributes related to its placement. See NO LABEL .

Adding Surfaces

To add a surface:

1. In the Media list, select the media containing the back or front/matte that you want to add to the scene.
2. Do one of the following:
   - Drag an image node from the node bin and place it in the schematic. An axis and an image are created and linked together.
   - Drag an image node from the node bin to the Result view, so you can see its effect on the scene before placing it exactly where you want.
   - Double-click an image node. The node appears next to the last added object. You do not need to be in Schematic view to add a node in this manner.

   The surface is added to the scene with its own axis. The selected media in the Media list is automatically applied to the surface.

Notice that when front/matte media is applied to the image surface, the front and matte clip are combined. You can turn off the matte in the Matte Clip box in the Media menu to show the entire front clip.

3 If you later decide to change the media on the surface, select the surface in the schematic, then select the new media in the Media list and click Apply.

Modifying Surfaces

The Surface menu includes properties common to all surfaces, and specific controls for bilinear, perspective, and extended bicubic surfaces. You can change a surface's shape, position, and transparency, as well as apply lighting effects. You can also apply tracking data to, and access the UV points of bilinear, perspective, and extended bicubic surfaces.

To access the Surface menu:

1 Double-click the selected surface in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 571).

The Object Image menu appears with the Image controls for the selected surface displayed on the right side of the menu. Some of the tabs for the Surface menu are contextual, and are available depending on the circumstance.

NOTE When accessing Action as a Timeline FX, you have access to a quick menu with some of Surface settings. To see the full Surface menu, click the Editor button to enter Action.

Surface Tools

From the Tools drop down menu, you can select additional tools to modify surfaces.

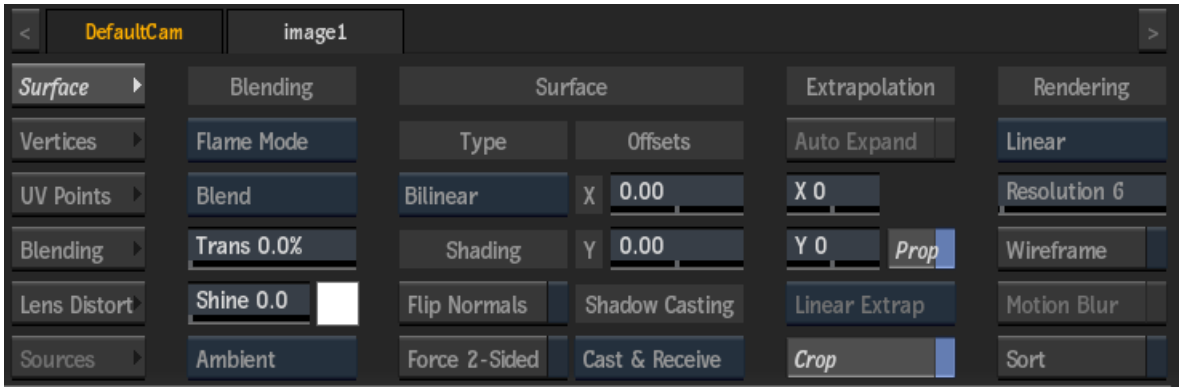
Slide Texture Enables you to slide the texture under the vertices, without moving the vertices, effectively moving the UV points behind the scene.

Move UV/Vertex Enables you to move the UV points and the vertices simultaneously and does not introduce any deformation.

NOTE If the UV points and Vertices grids are already in different positions, the relative position offset is kept when moving the UV points and vertices in this mode.

Surface Settings

The Surface tab is available at all times, and is divided into a number of sections.



Blending Section

Blend Type box Select whether to use Flame or Photoshop blend modes.

Blend Mode box Select how the front and matte clips or the front and back clips are combined.

See Surface Blending Modes (page 634).

Blend Premultiplication box Select whether the blend operation is premultiplied or not. You can set this as a default preference in the Action Setup menu.

Transparency field Displays the transparency of the image. Enter 0 for a completely opaque image, or 100 for a completely transparent image.

Shine field Displays the level of shine for the specular highlights (there are no specular highlights when Shine is set to 0). Editable.

Specular Highlights colour pot Displays the colour of the surface highlights. The specular highlight is visible only if Shading is enabled in the Setup menu and if shine is greater than 0. Editable.

See Specular Highlights (page 667).

Lighting box Select Ambient or Diffuse lighting so that the surface can reflect incidental light.

See Incidental Light Reflection (page 667).

Surface Section

Shape box Select a shape for the selected surface. For Stereo Objects, you are limited to using a Flat surface. For shapes other than flat, you have access to the Vertices and UV Points settings.

Shape	Description
Flat	This is the simplest surface. It is added to the scene by default the first time you enter Action with a Front, Back, and Matte. When working with a Stereo Object, Flat is the only shape available. You cannot change its shape because a flat image does not have vertices. You can, however, scale and shear a flat image using its axis.
Bilinear	A bilinear surface has four vertices: one for each corner. The vertices are joined using linear interpolation (straight lines). You can animate the shape of a bilinear surface by changing the position of the corners.
Perspective	A perspective surface has four vertices similar to a bilinear surface. The vertices form a perspective plane of the surface. You can animate the shape of a perspective surface by changing the position of the corners.

Shape	Description
Extended Bicubic	An extended bicubic surface has vertices in sections that can be subdivided up to eight times to increase the number of vertices. The vertices are joined using bicubic interpolation (curved lines). You can animate the shape of a bicubic surface by changing the position of the corners and moving the tangent handles to adjust the curve between corners.

Flip Normals button Enable to flip the selected surface normals to light the back side of a surface.

Force 2-Sided button Enable to have lights in the scene light both sides of the surface (when shading is turned on).

Surface Offset X field Displays the level of offset for a surface along the X axis. Editable.

Surface Offset Y field Displays the level of offset for a surface along the Y axis. Editable.

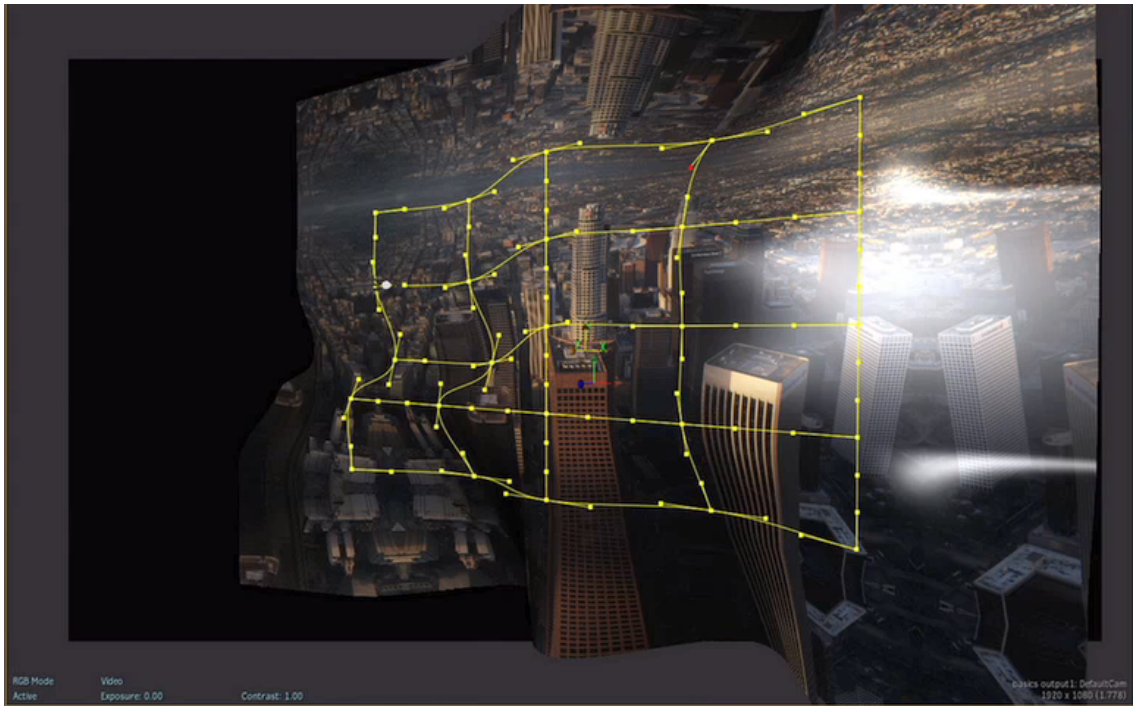
See Offsetting a Surface (page 634).

Shadow Casting box Select how the selected image object will be affected by a Shadow Cast object in the scene.

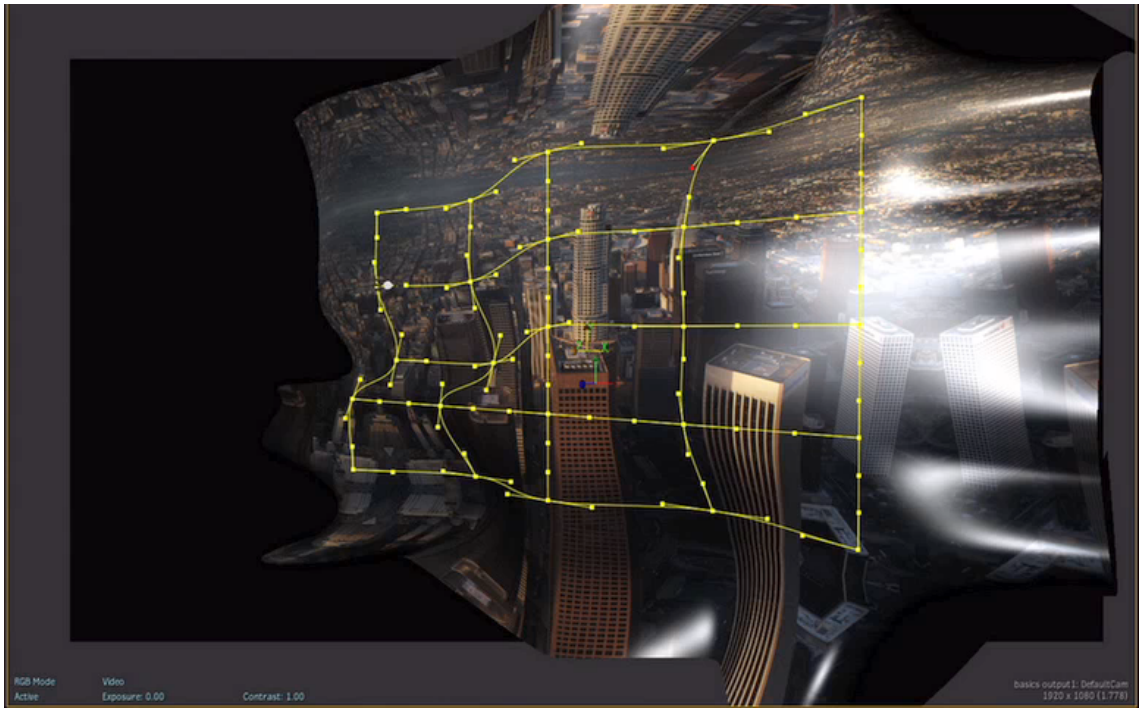
See Surface and Geometry Shadow Casters (page 672).

Extrapolation Section

Using the extrapolation sliders, you can extrapolate the texture beyond the geometry, in X and Y. You can assign negative extrapolation values and effectively crop the texture. The extrapolation is in mirror mode, by default. You can change this by adding a Diffuse node in the Action schematic. When working with an Extended Bicubic surface, you can choose between Linear Extrapolation and Smooth Extrapolation. Smooth extrapolation produces a more organic deformation, extrapolating the deformation created by the vertices and tangents, while Linear extrapolates the surface in a straight line after the last vertices, in the direction of the tangents.



Linear extrapolation



Smooth extrapolation

Auto Expand button Enable to automatically resize the surface when an attached diffuse map's Axis settings are changed. Only available for Flat surfaces, and when a diffuse map is a child of the surface. Always available for stereo objects.

Vertical Size Offset field Displays the vertical size offset (expansion or restriction) of the surface while respecting the surface shape. Editable.

Horizontal Size Offset field Displays the horizontal size offset (expansion or restriction) of the surface while respecting the surface shape. Editable.

Proportional button Enable to change the size offset fields proportionally.

Extrapolation box Available for extended bicubic surfaces only, and when the size offset fields are not at 0. Select Linear to extrapolate the surface in a straight direction. Select Smooth to extrapolate smoothly based on the position of the vertices of the surfaces.

Crop button Enable to restrict a bilinear or perspective surface to the dimensions of the UV points.

You can crop the parts of the image that are outside of the geometry, or let the texture go beyond the surface wireframe, by enabling the Crop button.

Rendering Section

Filter box Select the type of filtering to apply to the surface. For example, to be able to mix various ratios in an Action setup, such as a PAL image in a 1080 HD setup, requires filtering.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.

Select:	To apply:
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter that determines the proportions of an elliptical area around every pixel to perform a weighted sampling of the colours (slower to process than other filters). EWA is always calculated in screen space, so you should always view the result with a 100% zoom factor.
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

EWA and Anisotropic filtering are especially useful to deal with rotations and perspective effects.

The majority of the filters do not alter the image in any way if you have not moved it, or if you move it in actual pixel increments. The only exception is the EWA+Linear filter which always filters the image, even if you have not moved it (inherent to this type of filtering).

Surface Resolution field Displays the geometry resolution of the selected surface. By default, the value is equal to the default Action Resolution setting (in the Rendering section of the Setup menu).

The lower the value, the better the resolution and the greater the processing time required to interact with the image. For example, a value of 1 on an NTSC image creates a mesh with 720 horizontal and 486 vertical subdivisions on the selected surface, affording accurate displacement, normals, and lighting.

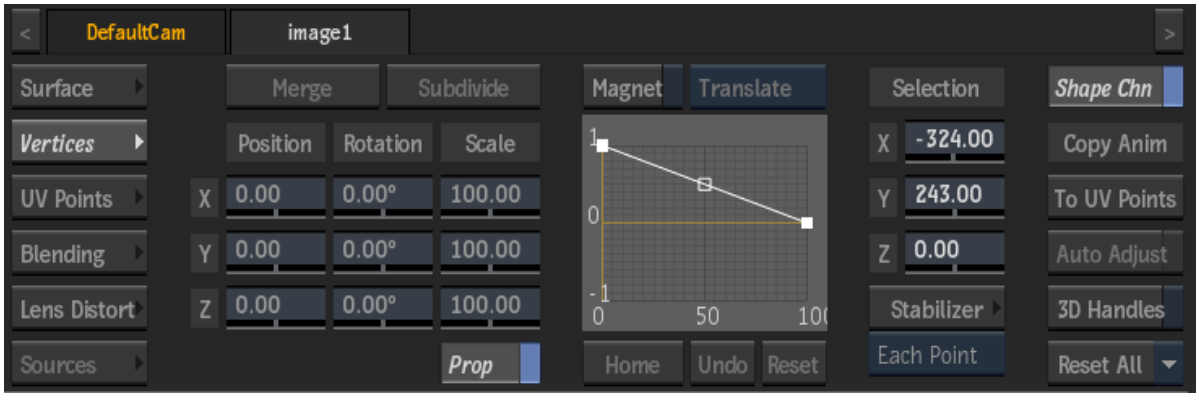
Wireframe button Enable to convert the selected surface to a wireframe representation (lighting and transparency properties are kept).

Motion Blur button Enable to use motion blur for the selected surface (can only be used if the global Motion Blur is enabled in the Action Setup menu).

Sort button Enable to sort non-flat surfaces where overlapping transparent regions are causing artefacts.

Vertices Settings

Vertices settings are available for all surface shapes, except for flat.



Merge button Click to merge the extended bicubic surface sections (can be clicked multiple times).

Subdivide button Click to subdivide the extended bicubic surface into more sections (can be clicked multiple times). See Subdividing an Extended Bicubic (page 640).

X Position field Displays the position of the selected surface points along the X axis. Editable.

Y Position field Displays the position of the selected surface points along the Y axis. Editable.

Z Position field Displays the position of the selected surface points along the Z axis. Editable.

X Rotation field Displays the rotation of the selected surface points along the X axis. Editable.

Y Rotation field Displays the rotation of the selected surface points along the Y axis. Editable.

Z Rotation field Displays the rotation of the selected surface points along the Z axis. Editable.

X Scale field Displays the scale of the selected surface points along the X axis. Editable.

Y Scale field Displays the scale of the selected surface points along the Y axis. Editable.

Z Scale field Displays the scale of the selected surface points along the Z axis. Editable.

Proportional Scale button Enable to scale the surface points proportionally.

Edit Track button Enable to edit tracking data directly from the image window without having to switch to the Animation channel. Make sure the Auto Key is enabled, so as not to lose any keyframe modifications.

Magnet button Enable to transform a range of surface points. Use in conjunction with the Magnet Transformation box. See Transforming Multiple Points (page 641).

Magnet Transformation box Select a transformation type to use when Magnet is enabled.

Magnet Curve Editor Displays the weighted polarity from the centre to the edge of the magnet.

Magnet Curve Home button Resets the position of the magnet curve after panning.

Magnet Curve Undo button Undoes a change to the Magnet Curve Editor.

Magnet Curve Reset button Resets the Magnet Curve Editor.

Vertex Position X field Displays the position of the vertices on the X axis. Editable.

Vertex Position Y field Displays the position of the vertices on the Y axis. Editable.

Vertex Position Z field Displays the position of the vertices on the Z axis. Editable.

Stabilizer button Opens the Stabilizer menu to apply stabilizing data to selected surface vertices.

Tracking Mode box Select wether to track each selected vertex or UV point individually (Each Point) or grouped via two proxy trackers, and from which parameters from the proxy trackers extrapolate the movement (Pos/Scale, Pos/Rot, Pos/Rot/Scale). With Perspective tracking, you can further select whether to enter the Stabilizer with four trackers associated to the 4 Corners of the perspective surface (4 Corners), or use an unlimited number of trackers to track the Perspective transformation of the vertices or UV points (Perspective).

- **Each Point**: Perform the tracking using a tracker for each selected vertex or UV point.
- **Pos/Rot**: Perform the tracking using two proxy trackers that extrapolate the position of the selected vertices or UV points, based on the detected position and rotation motion.
- **Pos/Scale**: Perform the tracking using two proxy trackers that extrapolate the movement of the selected vertices or UV points, based on the detected position and scale motion.
- **Pos/Rot/Scale**: Perform the tracking using two proxy trackers that extrapolate the movement of the selected vertices or UV points, based on the detected position, scale and rotation motion.
- **4 Corners**: Associates a tracker to each corner of your image (equivalent to a four-point track on a bilinear surface).

- **Perspective**: Allows you to add as many trackers as you want and position them in the same plane as your surface, to cover as much of the perspective transformation as possible.

Shape Channel button Enable to use the Shape channel in the Channel Editor. Disable to use individual channels in the Channel Editor. See Reshaping Using the Channel Editor (page 635).

Copy To UV Points button Click to copy vertex data to the UV point animation channels.

Auto Adjust button Enable to scale adjacent tangents automatically. Auto Adjust creates a smooth curve between points in the deformation. If you want to work on a specific area of the image without affecting other tangents, disable Auto Adjust.

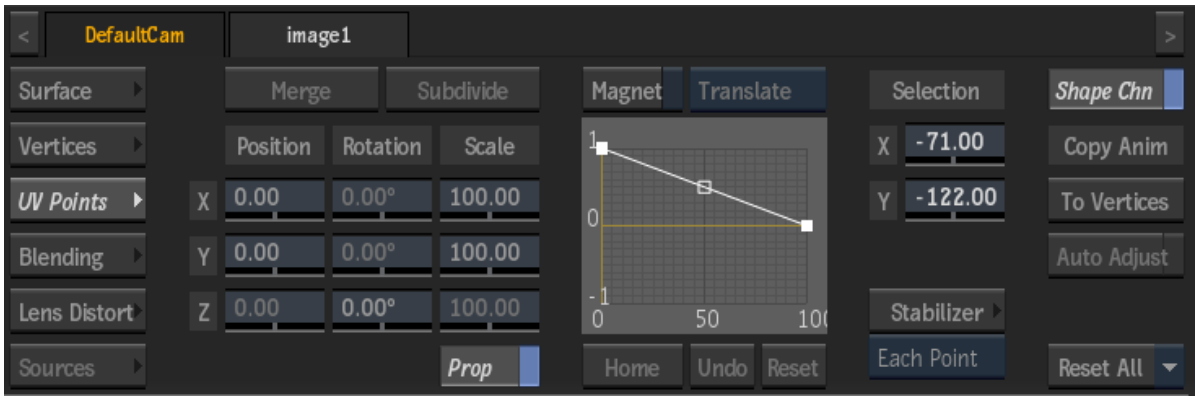
NOTE If you move a tangent explicitly, it is not affected by Auto Adjust. Click Reset Points or Reset Shape so that the tangents will be affected by the Auto Adjust mode.

3D Handles button Enable to allow Z buffering of the vertices. By default, the vertices are always visible, regardless of their position in Z space in relation to other media.

Reset Selection box Select whether to reset selected points, tracking data, or the shape of the surface. Click Reset All to reset all data.

UV Points Settings

UV Points settings are available for all surface shapes, except for flat.



Merge button Click to merge the extended bicubic surface sections (can be clicked multiple times).

Subdivide button Click to subdivide the extended bicubic surface into more sections (can be clicked multiple times). See Subdividing an Extended Bicubic (page 640).

X Position field Displays the position of the selected surface points along the X axis. Editable.

Y Position field Displays the position of the selected surface points along the Y axis. Editable.

Z Rotation field Displays the rotation of the selected surface points along the Z axis. Editable.

X Scale field Displays the scale of the selected surface points along the X axis. Editable.

Y Scale field Displays the scale of the selected surface points along the Y axis. Editable.

Proportional Scale button Enable to scale the surface points proportionally.

Magnet button Enable to transform a range of surface points. Use in conjunction with the Magnet Transformation box. See Transforming Multiple Points (page 641).

Magnet Transformation box Select a transformation type to use when Magnet is enabled.

Magnet Curve Editor Displays the weighted polarity from the centre to the edge of the magnet.

Magnet Curve Home button Resets the position of the magnet curve after panning.

Magnet Curve Undo button Undoes a change to the Magnet Curve Editor.

Magnet Curve Reset button Resets the Magnet Curve Editor.

UV Point Position X field Displays the position of the selected UV points along the X axis. Editable.

UV Point Position Y field Displays the position of the selected UV points along the Y axis. Editable.

Stabilizer button Opens the Stabilizer menu to apply stabilizing data to the selected UV points.

Tracking Mode box Select wether to track each selected vertex or UV point individually (Each Point) or grouped via two proxy trackers, and from which parameters the proxy trackers extrapolate the movement (Pos/Scale, Pos/Rot, Pos/Rot/Scale). With Perspective tracking, you can further select whether to enter the Stabilizer with four trackers associated to the 4 Corners of the perspective surface (4 Corners), or use an unlimited number of trackers to track the Perspective transformation of the vertices or UV points (Perspective).

- **Each Point**: Perform the tracking using a tracker for each selected vertex or UV point.
- **Pos/Rot**: Perform the tracking using two proxy trackers that extrapolate the position of the selected vertices or UV points, based on the detected position and rotation motion.
- **Pos/Scale**: Perform the tracking using two proxy trackers that extrapolate the movement of the selected vertices or UV points, based on the detected position and scale motion.
- **Pos/Rot/Scale**: Perform the tracking using two proxy trackers that extrapolate the movement of the selected vertices or UV points, based on the detected position, scale and rotation motion.
- **4 Corners**: Associates a tracker to each corner of your image (equivalent to a four-point track on a bilinear surface).
- **Perspective**: Allows you to add as many trackers as you want and position them in the same plane as your surface, to cover as much of the perspective transformation as possible.

Shape Channel button Enable to use the Shape channel in the Channel Editor. Disable to use individual channels in the Channel Editor. See Reshaping Using the Channel Editor (page 635).

Copy To Vertices button Click to copy UV point data to vertex animation channels.

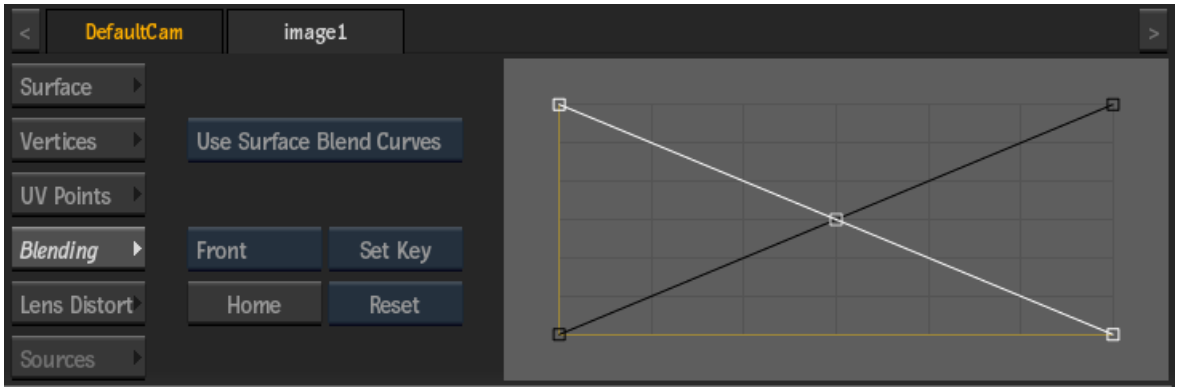
Auto Adjust button Enable to scale adjacent tangents automatically. Auto Adjust creates a smooth curve between points in the deformation. If you want to work on a specific area of the image without affecting other tangents, disable Auto Adjust.

NOTE If you move a tangent explicitly, it is not affected by Auto Adjust. Click Reset Points or Reset Shape so that the tangents will be affected by the Auto Adjust mode.

Reset Selection box Select whether to reset selected points, tracking data, or the shape of the surface. Click Reset All to reset all data.

Blending Settings

Use the settings in the Blending tab to adjust blending curve for each surface.



Blend Curves option box Select whether to work with the surface blend curves or view the existing keyer blend curves.

Matte box Select the matte curve you want to adjust. Selecting a curve in the curve editor automatically sets the correct curve in this box.

Keyframe Option box Select an option to Set, Delete, or Reset keyframes. If Auto Key is enabled, a keyframe is added automatically when you adjust the blending curve.

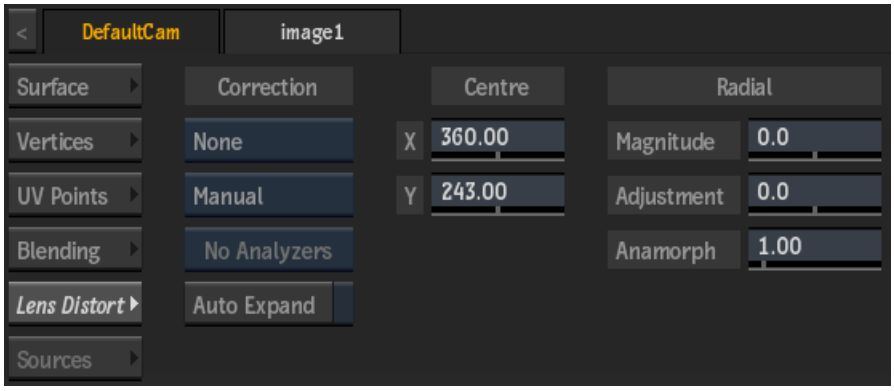
Home button Click to reset the curve view.

Reset Selection box Select whether to reset the selected curve (Reset) or all curves (Reset All) to their default settings.

For more information on using the blending curves, see Applying Blending Curves per Surface (page 637).

Lens Distort Settings

Differences in camera lenses or perspective irregularities cause lens distortion that results in skewed angles. Use the settings in the Lens Distort tab to rectify or simulate these types of distortions in your images.



Distortion box Specify whether you want perform a lens distortion or rectification.

Distort Corrections box Select whether to apply radial corrections manually in this menu, or use the automatic settings derived from the selected analyzer.

Distort Analyzer box Select which analyzer is used to provide automatic radial corrections.

Auto Expand button Enable to automatically resize the surface when an attached diffuse map's Axis settings are changed. This same button appears in the Surface tab.

Centre X field Displays the horizontal position of the centre of the lens. Editable.

Centre Y field Displays the vertical position of the centre of the lens. Editable.

Magnitude field Displays the magnitude of radial distortion or rectification. Editable.

Adjustment field Displays the level of secondary adjustment of radial distortion or rectification. Editable.

Anamorph field Displays the ratio of radial distortion or rectification along the X or Y axis. Editable.

Sources Settings

If a source node is parented above a surface node in the schematic, the Source tab is available. These settings allow you to change source settings on a per surface basis, independent of the source type set in the Source menu for the parent source node or nodes. See Replacing the Front or Matte Clip (page 896).

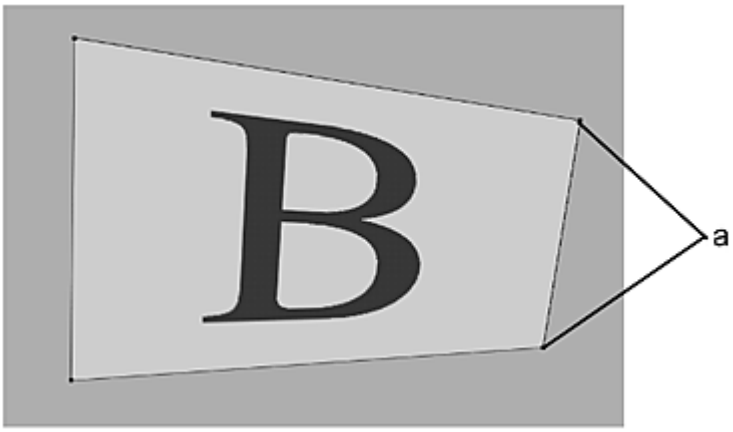
Changing the Shape of a Surface

When you first activate media or access Action, the media's surface is set to Flat by default. In Action, you can also represent a clip using custom bilinear, perspective, or extended bicubic surfaces.

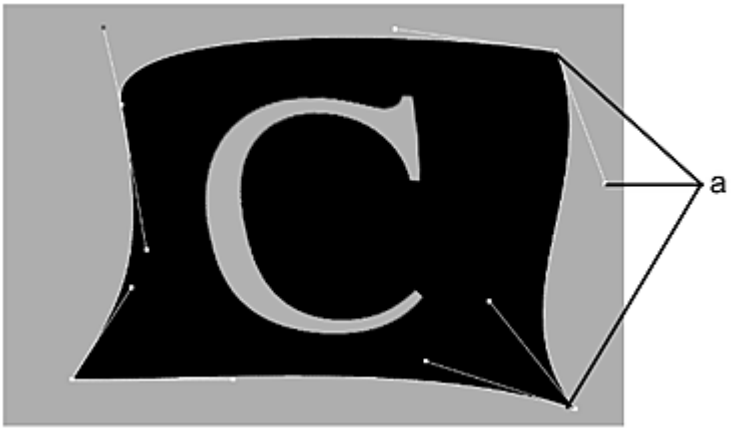
To change the shape of a surface:

1. Double-click the surface in the Schematic view.
2. Select the shape of the surface from the Shape box.
3. Click the vertex you want to edit and drag it to its new position.

   You use the vertices—or handles—on surfaces to change a surface's shape. You can move, rotate, shear, and scale a surface using the Axis (page 621) menu.



(a) Vertices on a bilinear surface



(a) Vertices and tangent handles on an extended bicubic surface

To move a handle on a bilinear or extended bicubic surface, click the handle that you want to edit. The selected handle appears in red. Use the cursor to drag the handle to its new position.

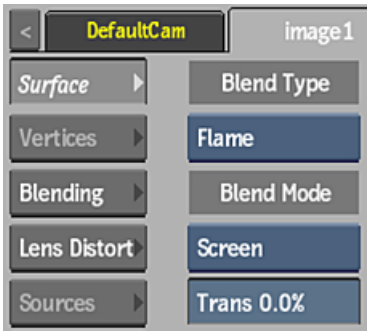
Each corner has two tangent handles. Lengthen or move the tangents to change the shape of the bicubic between corners.

All of the animation modifications you make to a shape are saved. If you decide to change a shape, then change it back to the original shape, your modifications are not lost.

Surface Blending Modes

Use the Blend Mode box to select how the front and matte clips or the front and back clips are combined. For example, use Screen for blending a fire or a lightning bolt shot on black that you want to composite with a different background.

You can select between a number of Flame or Photoshop blend modes by making a selection in the Blend Type box. Some of the blend modes are similar (or identical) between the different types, but are repeated with the mode names you are used to working with.



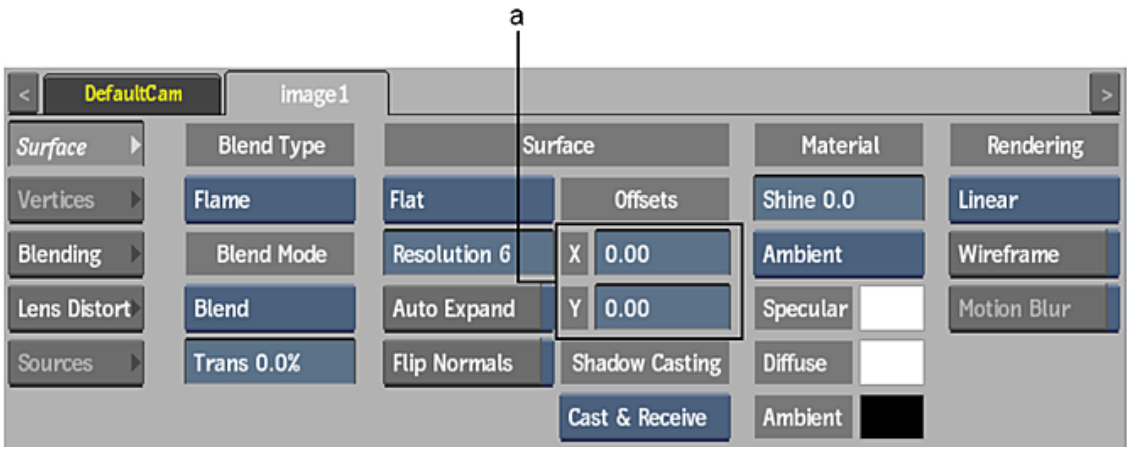
Offsetting a Surface

By default, a surface's axis is at the centre of the surface. Use the Surface Offset X and Y fields to offset a surface along the X-axis or Y-axis. All rotations, scaling, and shearing applied to a surface are applied about its axis. The location of the axis is indicated by the axis icon in the scene.

To offset a surface:

1 Select the surface you want to offset.

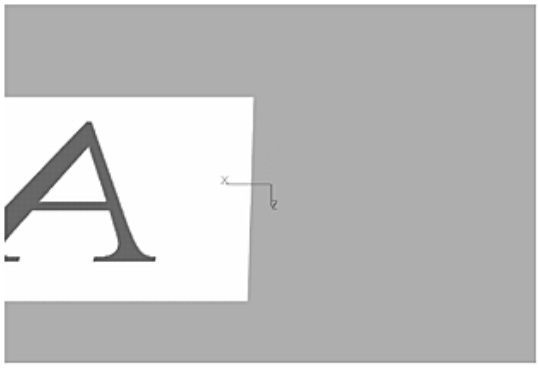
2 In the Surface menu, change the Surface Offset X and Surface Offset Y fields accordingly.



(a) Surface Offset fields

The following figure illustrates the difference between rotating a surface that has not been offset and a surface offset using the Surface Offset X field.

Rotated image with no offset. Notice that the axis is in the centre of the image.



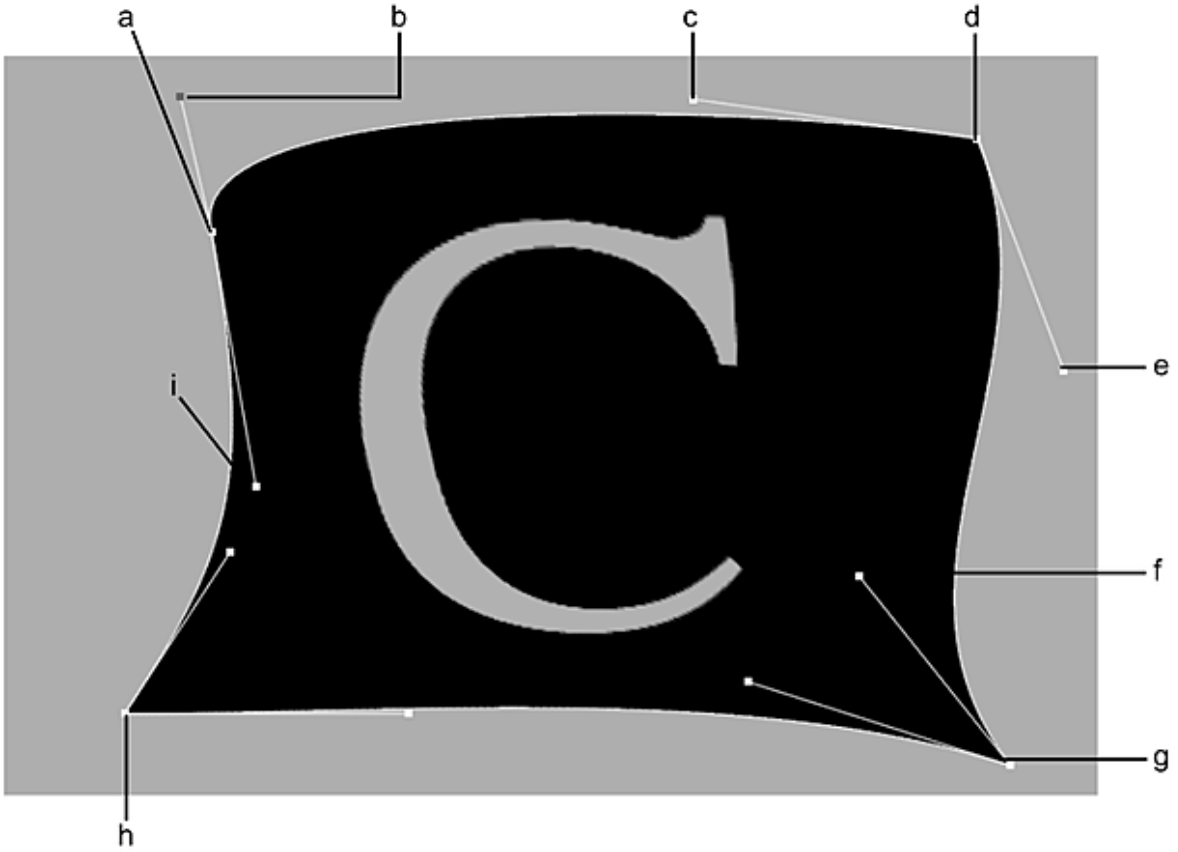
Rotated image with an offset along the X-axis. Notice that the axis is offset from the image centre.

Reshaping Using the Channel Editor

You can select whether you want to create an animation for bilinear, perspective, or extended bicubic surfaces using the Shape channel or individual Vertex channels in the Channel Editor. The Shape channel shows when the shape of the surface changes during the animation. Each time you move a surface handle, a shape key is added at the current frame, provided that Auto Key is enabled.

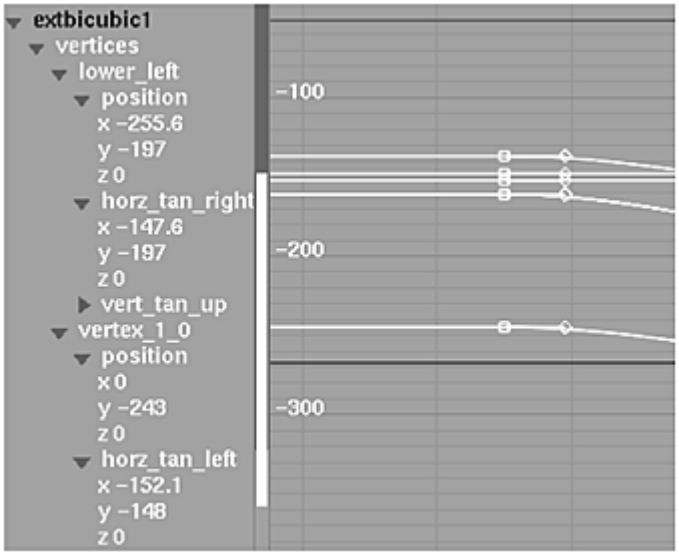
Vertex channels correspond to the vertices—or handles—that appear on the four corners of bilinear, perspective, and extended bicubic surfaces. The channel names for each corner are: upper_left, upper_right, lower_left, and lower_right, and are listed in the channel hierarchy each with a position x,y,z. To use the vertex channels in the Animation editor, make sure that Shape Channel is disabled in the Vertices or UV Points menu.

Each corner vertex has two tangent handles—the tangent handles for upper_left are named horz_tan_right and vert_tan_down.



(a) upper_left (b) horz_tan_right (c) horz_tan_left (d) upper right (e) vert_tan_ down (f) vert_tan_ up (g) lower_right (h) lower_left

Extended bicubics have additional vertex channels that appear in the Channel Editor for subdivided vertex channels. These channels appear only when you create a keyframe for the channel and its value changes. These vertex channels are named vertex_0.1, vertex_1.0, vertex_2.1, and so on, according to their position on the surface. Click the vertex on the surface to highlight its channel in the channel hierarchy. See Warping an Extended Bicubic Surface (page 638).



Blending Curves

You can adjust the blending curves of each surface separately. The blending curve is similar to the Keyer luminance blending curve, but you can adjust it per surface.

When you create a matte for the front clip, a matte for the back clip is automatically created to specify which part of the back clip is used for the composite. By default, the back matte is the inverse of the front matte.

You can adjust the luminance of the front matte and back matte separately in the Action blending curve. For example, increase the luminance of the back matte so that more of the back clip shows through at the edges of the key. This creates a better blend at the edges.

The following calculation is applied to each pixel of the image to create the composite. The calculation is applied in three passes, one each for the R, G, and B values of the front and back images, and the pixel is given the resulting R, G, and B values.

Result = F * FrontLUT + B * BackLUT

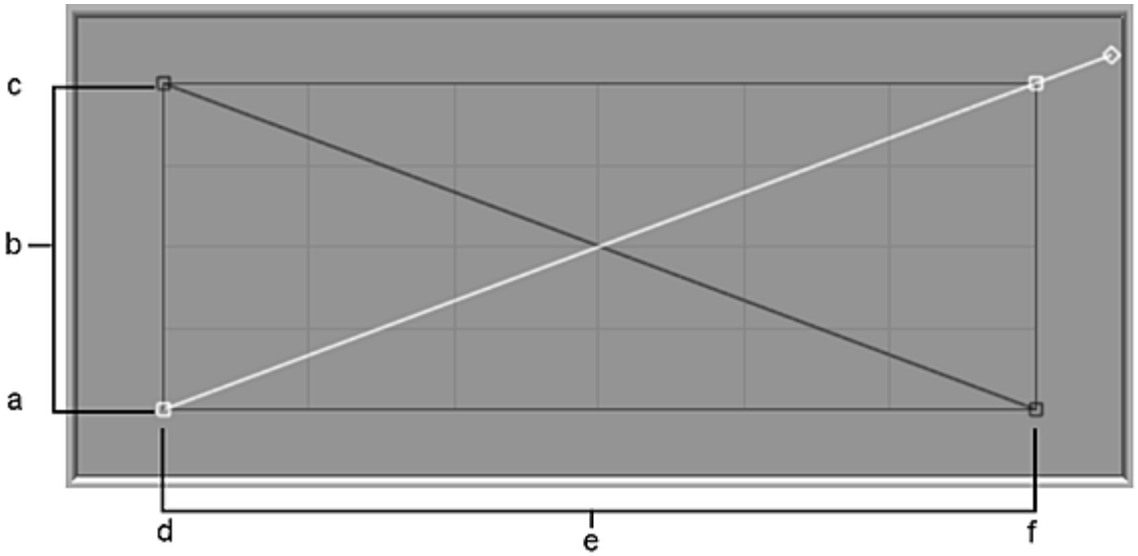
where:

- F = the R, G, and B values of the front image
- B = the R, G, and B values of the back image
- FrontLUT is the front matte pixel value, re-mapped according to any luminance curve change made in the blending curve. The value is expressed as a decimal, where, for example:
  - in 8-bit mode, 0 = 0, 127.5 = 0.5, and 255 = 1
  - in 12-bit mode, 0 = 0, 2047.5 = 0.5, and 4095 = 1
  - 16-bit floating point images, the values are represented on a logarithmic scale between 0 and 1.
- BackLUT is the back matte pixel value, re-mapped according to any luminance curve change made in the blending curve. The value is expressed as a decimal, as is the FrontLUT.

Applying Blending Curves per Surface

To adjust the blending curve:

1. In the Surface menu, click the Blending tab.
2. Select Result view from the View box. This allows you to view a particular image as you adjust the curve.
3. To adjust the luminance curve for the front matte, select Front from the Matte box. To adjust the back matte curve, select Back. Alternatively, click a curve to select it.



(a) 255 (White) (b) Output (remapping of luminance values) (c) 0 (Black) (d) 0 (Black) (e) Input (current luminance values) (f) 255 (White)

In Select edit mode, click a point to display its tangent handle and drag the handle to adjust the curve. Use other modes in the Tools box (Add Points or Break Tangent, for example) to further adjust the curve, adding or deleting points, or breaking tangent handles as needed.

4 Use the options in the Keyframe option box to Set, Delete, or Reset keyframes. If Auto Key is enabled, a keyframe is added automatically when you adjust the blending curve.

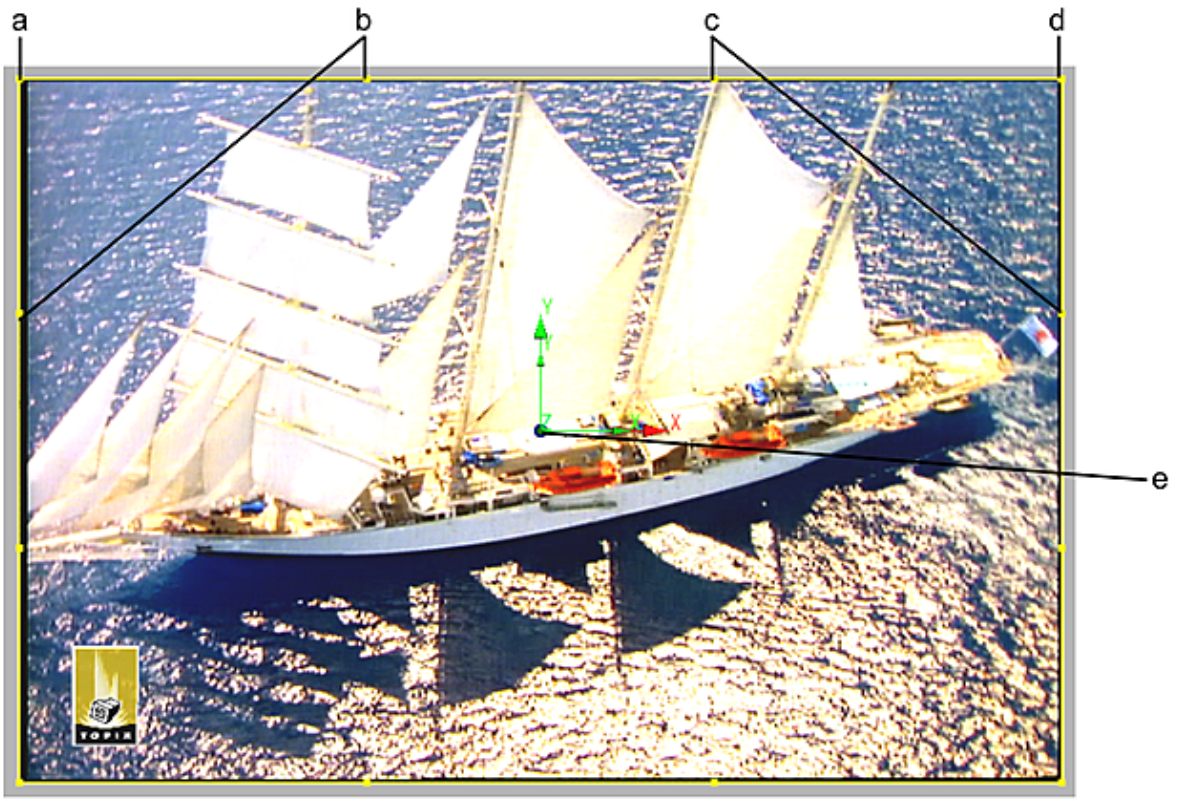
5 Use the Blend Curves option box to switch between the surface blend curves and the keyer blend curves. This gives you a good comparison to luminance curve work you have already done in the Keyer. When Use Keyer Blend Curves is selected, all of the other Action blend curve settings are unavailable.

Warping an Extended Bicubic Surface

Use an extended bicubic surface to warp specific areas of a surface. An extended bicubic surface can be subdivided into many sections, which are controlled by vertices. The vertices allow you to bend and contort the surface. The vertices are joined using bicubic interpolation—curved lines.

You can animate the shape of a bicubic surface by changing the position of the vertices and moving the vertex tangent handles. See Reshaping Using the Channel Editor (page 635).

By default, the extended bicubic surface has four vertices and eight tangents. You can translate, scale, or rotate these points and tangents individually or as a group. Subdividing the bicubic surface increases the number of vertices in the image and allows for more precision.



(a) vertex 1 (b) tangents of vertex 1 (c) tangents of vertex 2 (d) vertex 2 (e) Reference point

Image courtesy of Topix

Extended bicubics have three types of points: vertices, tangents, and the reference point. Vertices and tangents are collectively called *surface points*.

Vertices lie on the surface and its tangents determine the curvature of the deformation at each vertex. The reference point indicates the axis of origin for applying rotation and scaling to surface points.

Use the surface points to control the deformation of the surface. Each vertex has tangent handles. By moving the tangents and their handles, you can warp specific areas of the surface. You can translate, scale, or rotate these points and tangents individually or as a group.

TIP If a tangent point is hidden behind a vertex or reference point, press Q and click the point in the image window to select the tangent.

Tangent handles have three modes.

This mode:	Indicates:
■	The handles are not broken. If you move this handle, the adjacent handle will move in the opposite direction and by the same magnitude (length).
□	The handles are broken. If you move this handle, no other handles are affected.
○	The handles are geometrically continuous. If you move this tangent, the adjacent handles will move in the opposite directions but maintain its magnitude (length).

Subdividing an Extended Bicubic

Subdivide the bicubic surface up to eight times to increase the number of vertices and tangents. Then translate the vertices for a smaller region of the image for more precision. You should warp the surface with a small number of subdivisions to obtain the best results. After applying some deformations to a large portion of the surface, subdivide the surface further and perform deformations on a more localized region of the surface.

To subdivide extended bicubics:

1. From the Vertices menu, click Subdivide.

   You can click the subdivide button up to eight times to further subdivide the surface. Click Merge to undo subdivide operations.
2. Transform the tangents to achieve the effect you want.

To move the reference point:

1. Select the reference point.

   The reference point turns red when selected.
2. Drag the reference point to a new location, or use the X, Y and Z vertex fields to assign a new coordinate for the reference point.

To rotate multiple surface points:

1. Set the reference point values you want to use as the axis of origin using the X, Y, Z Vertex Position fields.
2. Select multiple surface points by pressing `Ctrl` and dragging to select the surface points.
3. From the Tools box, select Rotate.

   A 3D trackball appears on the reference point.
4. Use the 3D trackball to rotate the reference point and selected points.

   All selected points rotate around the reference point.

To scale multiple surface points:

1. Set the reference point values you want to use as the axis of origin using the X, Y, Z Vertex Position fields.
2. Select multiple surface points by pressing `Ctrl` and dragging to select the surface points.
3. From the Tools box, select Scale.
4. Drag in a direction in the image window to scale accordingly.

About Transforming Multiple Points

Use the magnet to transform a range of extended bicubic points. Use the magnet when there are many surface points as a result of more than one subdivision. In the Vertices or UV Points menu, the magnet controls include the Magnet button, the Magnet Transformation box, and the Magnet Curve Editor.

There are two ways to transform points with the magnet. You can use the magnet to select a range of points and transform them gesturally. Or, you can use the invisible magnet in conjunction with the Translation, Rotation, and Scale fields to transform selected points numerically.

Before you can use the magnet, you must set the polarity of the magnet and the magnet area of focus. The magnet's area of focus is determined by where you click the surface and the polarity of the magnet is set by the Magnet Curve Editor. The radius is determined by the distance from the centre to the farthest selected point.

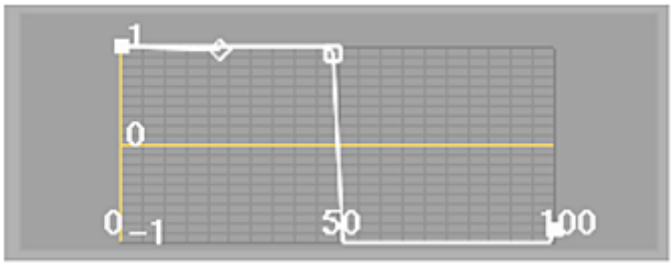
Using the Magnet

Setting the Magnet Polarity

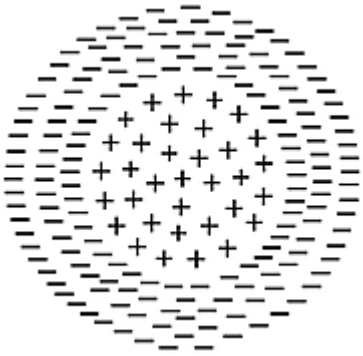
Use the Magnet Curve Editor to define the weighted polarity from the centre to the edge of the magnet. Points are either attracted or repelled depending on their location in the magnet and the shape of the magnet curve.

The area of the magnet is plotted on the X-axis where 0 is the centre of the magnet and 100 is the edge of the magnet. The polarity is plotted on the Y-axis where 1 is maximum positive strength and -1 is maximum negative strength.

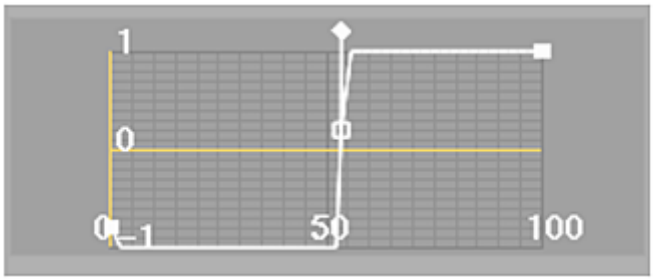
A positive polarity value attracts nearby points. A negative polarity value repels nearby points. The following example illustrates a magnet that has a strong positive polarity near the centre and strong negative polarity near the edges.



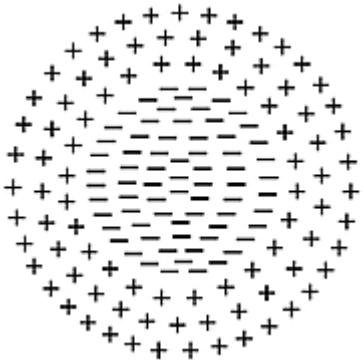
Magnetic Curve Editor



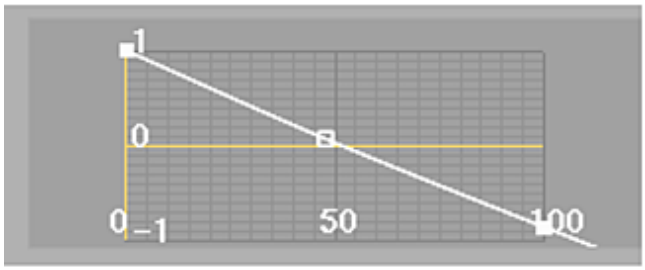
The following example illustrates a magnet that has a strong negative polarity near the centre and a strong positive polarity near the edges.

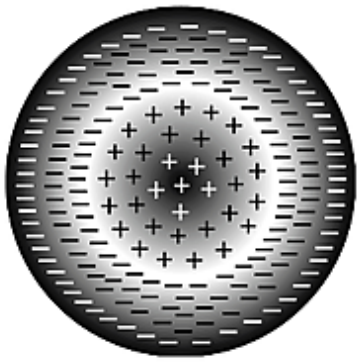


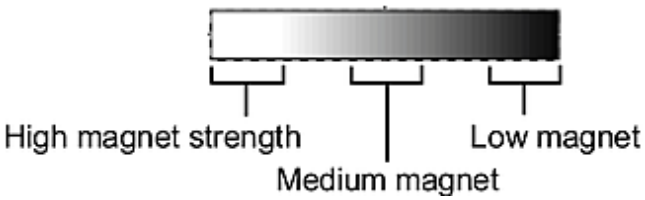
Magnetic Curve Editor



The following example illustrates a magnet whose positive polarity decreases from the centre and whose negative polarity increases toward the edges.







When you transform points, the direction and size of the transformation is determined by the weighted polarity of the magnet at the location of the points.

Using the first example, if you use the magnet to translate the points on the X-axis by +200, the points in the centre of the magnet move in the positive direction on the X-axis—they are attracted to the magnet—while the points near the edge of the magnet move in the negative direction on the X axis—they are repelled by the magnet.

To change the polarity of the magnet:

1. In the Vertices or UV Points menu, click Reset to return the magnet curve to its default.
2. Click the left-most handle on the curve and drag to define the polarity for the centre of the magnet.

Drag the point:	To:
Up	Increase the positive polarity.

Drag the point:	To:
Down	Increase the negative polarity.
To the middle	Assign no polarity.

3 Click the right-most handle on the curve and drag to define the polarity for the edge of the magnet.

Drag the point:	To:
Up	Increase the positive polarity.
Down	Increase the negative polarity.
To the middle	Assign no polarity.

4 Click the middle handle on the curve and drag to define the transition of polarity from the centre to the edge of the magnet.

Transforming Points

Use Magnet mode to transform points gesturally. The magnet's area of focus is determined by where you click the surface and the radius of the magnet is set by the Magnet Curve Editor. The radius is determined by the distance from the centre to the farthest selected point.

To transform points:

1 Use the Magnet Curve Editor to determine the polarity for the magnet.

2 From the Tools box, select Magnet.

The magnet appears as a red circular outline.

3 To resize the magnet, press `Alt+M` and drag left or right.

4 From the Magnet Transformation box, select the type of transformation.

Select:	To:
Translate	Translate the selected points along the X-, Y-, or Z-axis.
Rotate	Rotate the selected points about the centre of the magnet.
Scale	Scale the selected points about the centre of the magnet.

5 Click the points you want to transform in the image window.

6 Drag in the image window to apply the transformation to the selected points.

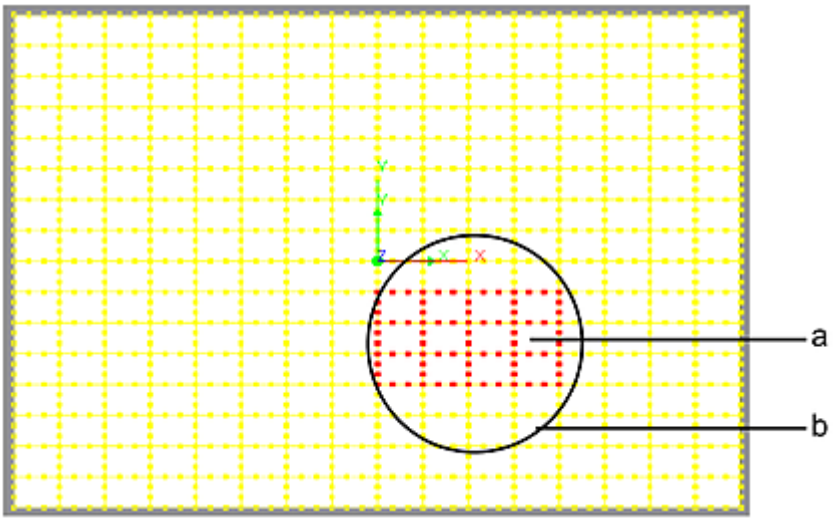
Transforming Unselectable Points

Use the invisible magnet to transform points you cannot select with the magnet. The centre of the magnet is determined by the centre of the selection and the radius is determined by the distance from the centre to the farthest selected point. Any transformations to the selected points are affected by the magnet.

To transform unselectable points:

1. Use the Magnet Curve Editor to determine the polarity for the magnet.
2. From the Tools box, choose Select.
3. Hold `Ctrl` and drag to select a range of points.
4. In the Vertices or UV Points menu, enable Magnet.

   The magnet is not visible, but is illustrated here to show how the points will be affected when you transform them.



(a) Selected Points (b) Magnet (not visible)

5. From the Magnet Transformation box, select the transformation type.

Select:	To:
Translate	Translate the selected points along the X-, Y-, or Z- axis.
Rotate	Rotate the selected points about the centre of the magnet.
Scale	Scale the selected points about the centre of the magnet.

6. Change the values in the Translation, Rotation, or Scale fields.

Adding Drop Shadows

In Action, drop shadows are cutouts based on a surface's matte. A drop shadow can be fully opaque or slightly transparent to simulate a real shadow.

Shadows cast by lit objects are also supported in Action (see Casting Shadows (page 672)).

To add a drop shadow:

1. Select the surface that you want to use.
2. Do one of the following:
   - Drag the shadow node from the node bin and place it in the schematic. An axis and a shadow are created and linked together.
   - Drag the shadow node from the node bin to the Result view, so you can see its effect on the scene before placing it exactly where you want.
   - Double-click the shadow node. An axis and a shadow are created and linked together. You do not need to be in Schematic view to add a node in this manner.

   If a surface node is selected in the schematic, the axis of the shadow node is automatically connected to the surface.

   NOTE A shadow has its own parent axis. It can be moved, rotated, scaled, and sheared independently of its parent surface. Because a shadow is also the child of the surface, moving the surface axis also moves the shadow.

Adjusting Drop Shadows

To change drop shadow colour and transparency:

1. Double-click a shadow node to display the Shadow menu.

2. Make sure that the Shadow Active button is enabled.

   NOTE You can turn the shadow on or off from the Shadow Active button. This button is independent of the Hide option.

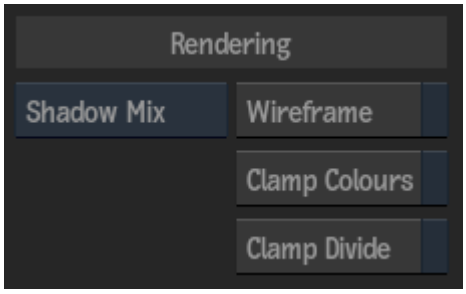
3. To change the colour of the shadow, do one of the following:
   - Enter colour values in the three colour fields in the Shadow menu.
   - Click the colour pot below the fields. Use the colour picker to pick the shadow colour.
4. To adjust the transparency of the drop shadow, use the Transparency field. When this field is set to 100, the shadow is completely transparent. When set to 0, the shadow is completely opaque.

   TIP You can lock the shadow's transparency to the transparency of its parent surface using the Lock To Surface button. Once a shadow transparency is locked to its parent surface, changes to the surface transparency also affect the shadow's transparency. You can then adjust transparency of the shadow as an offset of the surface's transparency.

5. Adjust the softness of a shadow using the softness fields. Use a value of 0.0 to have a distinct shadow. Increase the value to soften the shadow.

Using Shadow Mix

Use Shadow Mix to render shadows and surfaces according to their order in the Priority Editor, independently of their position in Z-space. This option makes it possible to have the shadow of one media on top of the shadow of another media. Shadow Mix is an option in the Z-Buffer box in the Rendering section of the Action Setup menu.



Shadow Menu Settings

Shadow Active button Enable to turn the shadow on.

Shadow Transparency field Displays the transparency level of the drop shadow. Editable.

Lock To Surface button Enable to lock a shadow's transparency to the transparency of its parent surface.

Red Shadow field Displays the red shadow value. Editable.

Green Shadow field Displays the green shadow value. Editable.

Blue Shadow field Displays the blue shadow value. Editable.

Shadow colour pot Displays the colour of the shadow. Editable.

X Softness field Displays the shadow softness level on the X axis. Editable.

Y Softness field Displays the shadow softness level on the Y axis. Editable.

Stereoscopic Workflow in Action

A stereoscopic workflow in Action allows you to create stereo composites using 3D and stereo elements. With the stereo camera rig (the 3D camera), you can access stereo rigs and monoscopic cameras. With support for multiple outputs, you can experiment with any number of passes, including left and right scene output, as well as normals, Z-Depth, matte, media matte, and composition outputs.

When working in a stereoscopic compositing workflow in Action, there are three essential elements: a stereo camera, a stereo object for viewing and adjusting the result, and outputs. The following table outlines the stereoscopic workflow in Action.

Step:	Refer to:
1. Bring stereo clips into Action.	Starting a Stereoscopic Session (page 647).
2. Work with the stereo camera and stereo object to make any adjustments to the scene.	Action 3D Cameras (page 856) and Working with the Stereo Scene (page 648).

Step:	Refer to:
3. Output various passes of your work.	About Rendering Outputs from Action (page 598).

NOTE Since Action as a timeline FX uses only one front/matte media, stereo clips are not supported. If you need to work with stereo clips, access Action from Batch or Batch FX, or from the Tools menu.

Starting a Stereoscopic Session

You can start a stereoscopic session in Action by loading stereo clips and using the Stereo Startup mode. This creates a stereo camera. The output is set to the stereo camera, a stereo object is created for visualizing the scene, and the clips are placed on separate lines in the Media list.

NOTE A stereo clip cannot be loaded in Action if it is selected as the Back input, or if it is selected to replace a clip in the Media list. Also, since Action as a timeline FX uses only one front/matte media, stereo clips are not supported.

When using a stereo clip, its left and right eye tracks are automatically split and placed on individual lines in the Media list. Also, new clips appear in the workspace. They retain the name of the original clips, and are appended with a "_Left" or "_Right" suffix. It is important to save these clips, as they are used for loading an Action setup or loading the previous Action session.

NOTE It is not possible to select a mono clip as the Matte input if the Front input is stereo, and vice versa.

To start a stereo session from the Tools tab:

1. Click Action.
2. From the Input Mode box, select one of the following:
   - Clear All to delete all media and objects from the previous setup. Select stereo clips for the front and matte input. This is the Stereo Startup mode.
   - Front/Back/Matte to enter Action using the previous setup. Select stereo clips for the Front and Matte input, and a mono clip for the Back input.

   NOTE If you enter Action using mono clips, the default camera is automatically created. If you need a stereo camera, you must add it manually.
3. Select the destination.

   If you selected Clear All and selected stereo clips for the Front and Matte input before entering Action (Stereo Startup mode), the following is created:
   - In the schematic, a 3D camera (stereo camera) is created and the default camera is hidden.
   - In the Output menu, the Mode is set to Stereo and the Camera is set to Result Cam.
   - In the Camera menu, the result camera is set to the 3D stereo camera.
   - A stereo object is created with the clips you selected.
   - In the Media list, the Front and Matte inputs are split into separate lines for the right and left eye.

   NOTE If you entered Action using the Front/Back/Matte option and want to clear all previous settings and use the Stereo Startup mode, click the Reset To Stereo Mode button in the Action Setup menu. You will now have to bring in new clips.

   When you exit Action, the workspace contains left and right clips for the Front and Matte inputs for a total of four new clips.

Working with the Stereo Scene

When working in a stereoscopic workflow, the stereo object lets you visualize the scene. The stereo object lets you composite stereo sources in Action, ensuring the stereo effect of the stereo source is preserved during the compositing process. You can combine stereo objects with 3D geometry, such as 3D text or FBX models within the same scene.

The stereo object is a single image surface which contains two diffuse maps (left and right) for handling stereo sources. When working in a stereoscopic workflow in Action, you must handle the left and right eye as separate media in the Media list. When a stereo object is filmed by a stereo camera, a link is created between the left image of the object and the left camera of the stereo rig. This is the same for the right image and right camera. The link ensures that left-eye material is only visible through the left camera, and likewise, right-eye material is handled with the right camera.

NOTE Entering Action with a stereo clip will automatically create a stereo object with the stereo clip used as the left and right material.

To add a stereo object to a scene (not applicable if you accessed Action as a timeline FX):

1. In the Media list, select the media for the left eye. Press `Ctrl` and select media for the right eye.

   NOTE By default, the first clip you select is the media for the left eye. You can select multiple pairs of left-right media. Odd numbered selections are considered as left media and even numbered selections are the right media when creating stereo objects.

2. Create a stereo object for the media by doing one of the following:
   - Drag the Stereo Object node from the node bin and place it in the schematic.
   - Drag the Stereo Object node from the node bin and place it where you want it in the Result view.
   - Double-click the Stereo Object node. You do not need to be in Schematic view to add a node in this manner.

   The stereo object is added to the scene.
3. To display a selected viewport in any of the stereo modes (Anaglyph Mono, Anaglyph Dubois, Blend or one of the Difference modes), select one from the Stereo mode button in the lower-left corner of the viewport.

   NOTE The viewport must be set to Result.

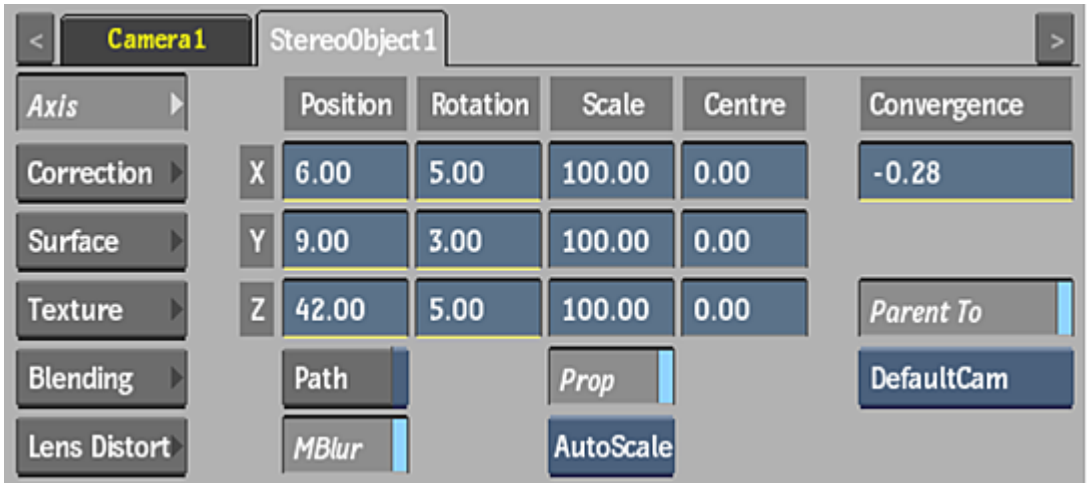
4. To open the Stereo Object menu, double-click the StereoObject node in the schematic.

Stereo Object Settings

Since Action as a timeline FX uses only one front/matte media, stereo clips are not supported. If you need to work with stereo clips, access Action from Batch or Batch FX, or from the Tools menu.

Stereo Object Axis Settings

Use the Axis tab of the Stereo Object menu to position, rotate, scale, and shear an axis, as well as adjust the convergence, parenting, and autoscaling of stereo objects.



The Position, Rotation, Scale, and Centre settings are the same as in the Axis menu. The following Axis settings are specific to the Stereo Object:

Autoscale box Select whether to autoscale when settings are changed relative to the camera.

Select:	To:
Auto Off	Not use autoscaling on the image.
Auto Z	Link the Position Z parameter with Convergence. This allows you to see how a change in the Convergence value makes the object appear to move closer or farther from the camera. This preserves the same visual aspect while scaling the textures up or down accordingly. **NOTE** The result camera must be set to the stereo camera.
AutoScale	Change the left and right texture parameters when the Position Z or a Scale parameter is changed. When you create a stereo object, it is automatically oriented towards the camera and automatically scales. That is, the apparent size of the images scale to compensate when you move the stereo object along the Z axis. If you move a stereo object away from the camera, it grows, and vice versa. **NOTE** The result camera must be set to the stereo camera.

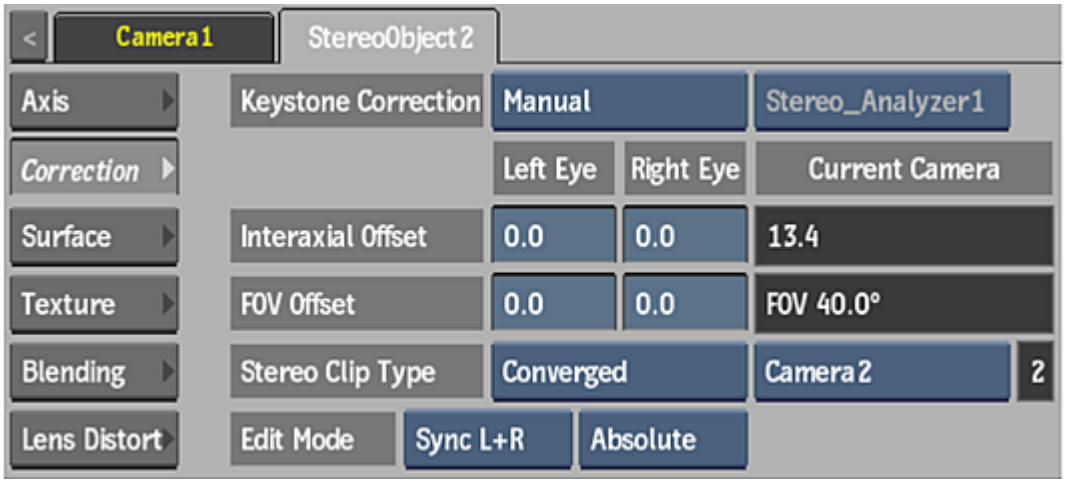
Convergence field Displays the equal amount of convergence in opposite directions horizontally, between left and right images. Positive values make the image appear farther from the camera. Editable.

Parent To button Enable to associate a stereo object with the camera selected in the Parent Camera box. Enabled by default to force the stereo object to face the 3D camera at all times when the camera is moved around.

Parent Camera box Select which camera in the scene is associated with the stereo object. Setting found in the Axis and Correction tabs.

Stereo Object Correction Settings

Use the Correction tab of the Stereo Object menu to indicate how the footage was shot, and make interaxial offset and FOV offset adjustments to the left/right images.



Keystone Correction box Select whether to apply keystone corrections manually in this menu, use the automatic settings derived from the selected analyzer, or inherit the Stereo Mode settings from the stereo camera (Pass Through).

In most cases, a keystone correction is needed if the stereo types of the camera and the stereo object do not match, but there may be cases when you want to sync the two.

Correction Analyzer box Select which analyzer is used to provide automatic keystone corrections.

Interaxial Offset Left Eye field Displays the offset distance between the left eye compared to the 3D camera. Editable.

Interaxial Offset Right Eye field Displays the offset distance between the right eye compared to the 3D camera. Editable.

Interaxial Offset Current Camera field Displays the interaxial offset of the selected camera. Non-editable.

FOV Offset Left field Displays the offset field of view between the left eye compared to the 3D camera. Editable.

FOV Offset Right field Displays the offset field of view between the right eye compared to the 3D camera. Editable.

FOV Offset Current Camera field Displays the offset field of view of the selected camera. Non-editable.

Stereo Clip Type box Specify how the footage was shot: Parallel, Off Axis, or Converged.

Parent Camera box Select which camera in the scene is associated with the stereo object. Setting found in the Axis and Correction tabs.

Camera Number field Displays the number of the selected camera. Non-editable.

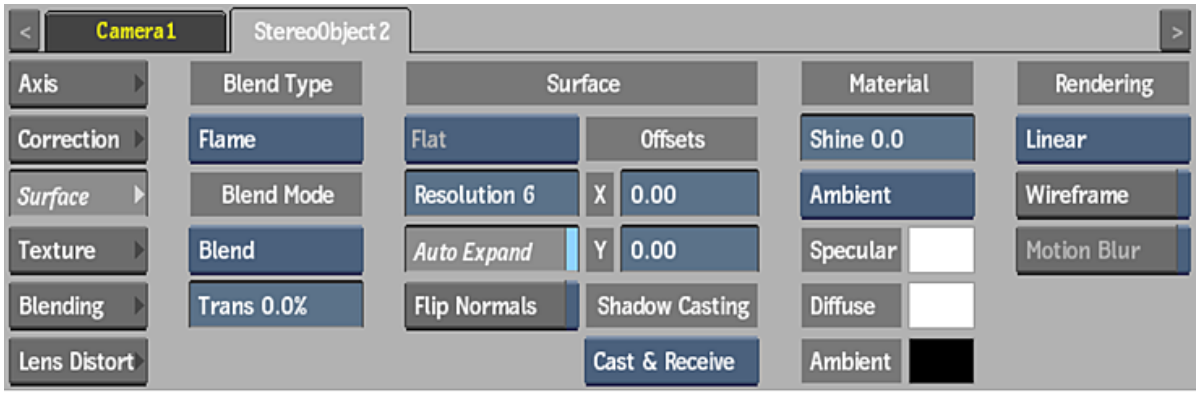
Tools box Select whether to sync the left and right eye correction values.

When synced, a change to one value affects the other value. The left eye position is a combination of the camera's left eye position and the left interaxial offset; the same applies for the right eye. The effective FOV for the correction is the sum of the camera's FOV and the FOV offset.

Sync Type box Select whether the relationship between the left and right eye correction values is absolute or relative. Available when Sync L+R is selected in the Tools box.

Stereo Object Surface Settings

Use the Surface tab of the Stereo Object menu to adjust surface and shading properties of the stereo object.



The Surface tab settings are the same as in the Object Image menu for non-stereo objects. See Surface Settings (page 624).

Stereo Object Texture Settings

Use the Texture tab of the Stereo Object menu to set the built-in diffuse maps (as well as any added texture maps) of a stereo object.

X Position field Displays the position of the X axis. Editable.

Y Position field Displays the position of the Y axis. Editable.

Z Position field Displays the position of the Z axis. Editable.

X Rotation field Displays the rotation of the X axis. Editable.

Y Rotation field Displays the rotation of the Y axis. Editable.

Z Rotation field Displays the rotation of the Z axis. Editable.

X Scale field Displays the scale of the X axis. Editable.

Y Scale field Displays the scale of the Y axis. Editable.

Z Scale field Displays the scale of the Z axis. Editable.

Proportional button Enable to change the fields proportionally.

X Shear field Displays the shear of the X axis. Editable.

Y Shear field Displays the shear of the Y axis. Editable.

Z Shear field Displays the shear of the Z axis. Editable.

X Centre field Displays the centre of the X axis. Editable.

Y Centre field Displays the centre of the Y axis. Editable.

Z Centre field Displays the centre of the Z axis. Editable.

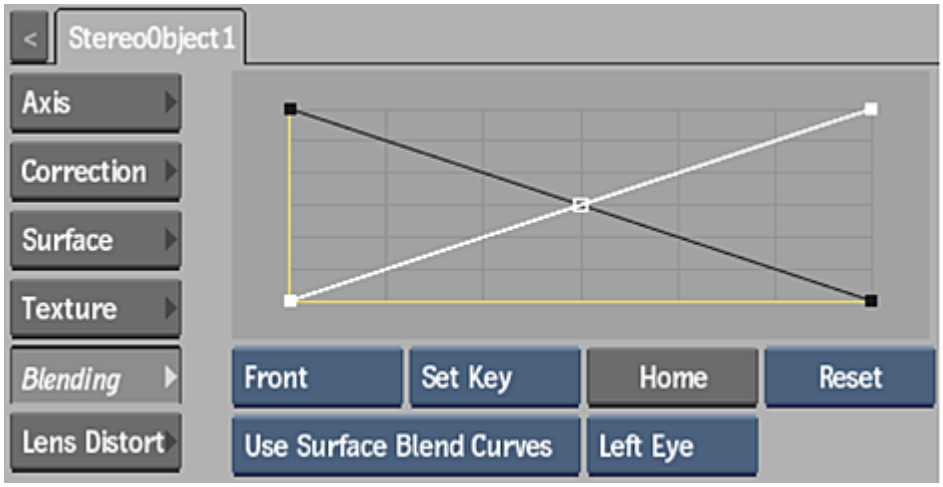
Tools box Select whether to sync the left and right eye texture values.

Sync Type box Select whether the relationship between the left and right eye texture values is absolute or relative. Available when Sync L+R is selected in the Tools box.

Texture Eye box Select whether to apply changes to the left or right eye and display the values of the left or right texture.

Stereo Object Blending Settings

Use the Blending tab of the Stereo Object menu to adjust the blending curves of each stereo surface separately. The blending curve is similar to the Keyer luminance blending curve, but you can adjust it per eye.



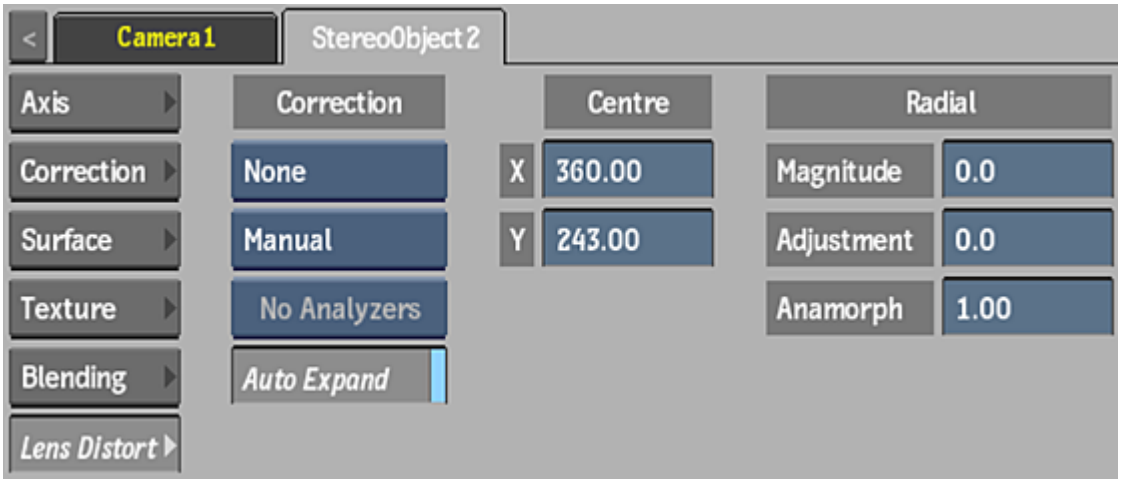
You can switch between the stereo surface blend curves and the keyer blend curves. This gives you a good comparison to luminance curve work you have already done in the Keyer.

The Blending tab settings are the same as in the Object Image menu for non-stereo objects. See Applying Blending Curves per Surface (page 637). One extra setting is available for stereo objects, allowing you to select which eye to apply blending curves.

Eye Selection box Select which eye to apply the blending curves. Select Both Eyes to apply the same blending settings to both eyes. In this case, the settings of the last selected eye apply to both eyes.

Stereo Object Lens Distort Settings

Differences in camera lenses or perspective irregularities cause lens distortion that results in skewed angles. Use the settings in the Lens Distort tab to rectify or simulate these types of distortions in your stereo images.



The Lens Distort tab settings are the same as in the Object Image menu for non-stereo objects. See Lens Distort Settings (page 632).

Action Perspective Grid

Use the Perspective Grid node to help you with perspective alignments in your Action scene. The perspective grid creates a rectangle on a plane within the perspective of the clip. The accuracy of the plane determines whether any adjustments are properly scaled and oriented in your clip. Objects can be placed anywhere on this plane. Aligning objects to the perspective grid has multiple benefits:

- You can perform a 4-corner pin with a surface or geometry that respects the perspective.
- Aligning the grid establishes the orientation of the plane in 3D space and the Field of View (FOV) of the camera that shot the scene.
- You can straighten the perspective of attached objects in 2D mode.
- Because the Perspective Grid node has an axis-like behaviour, objects attached to it inherit the grid's 3D transformations.
- Use the Tracking tab of the Perspective Grid menu to perform 4-corner or planar tracking.

Aligning an Object with a Perspective Grid

To align an object with a perspective grid:

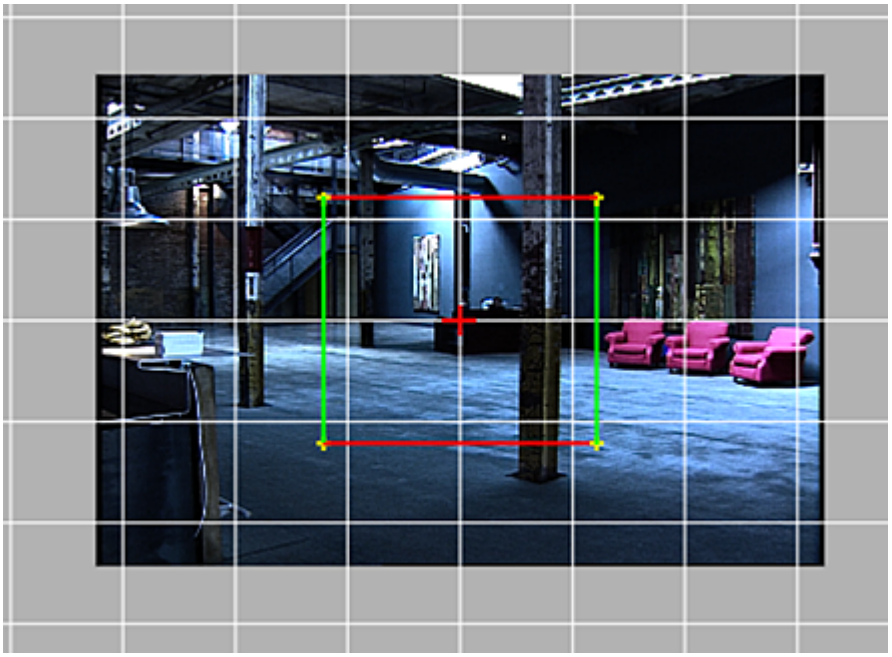
1. Click Media.
2. In the Media menu, select the media you want to use for the perspective grid.

   This media becomes the reference for the perspective grid alignment.
3. Do one of the following:
   - Drag the Perspective Grid node from the node bin and place it in the schematic.
   - Drag the Perspective Grid node from the node bin and place it where you want it in Result view.
   - Double-click the Perspective Grid node. You do not need to be in Schematic view to add a node in this manner.

   The Perspective Grid object is added to the schematic. In the Schematic view, the number in brackets next to the name of the node indicates the media used for the perspective grid reference (0 refers to the back clip).

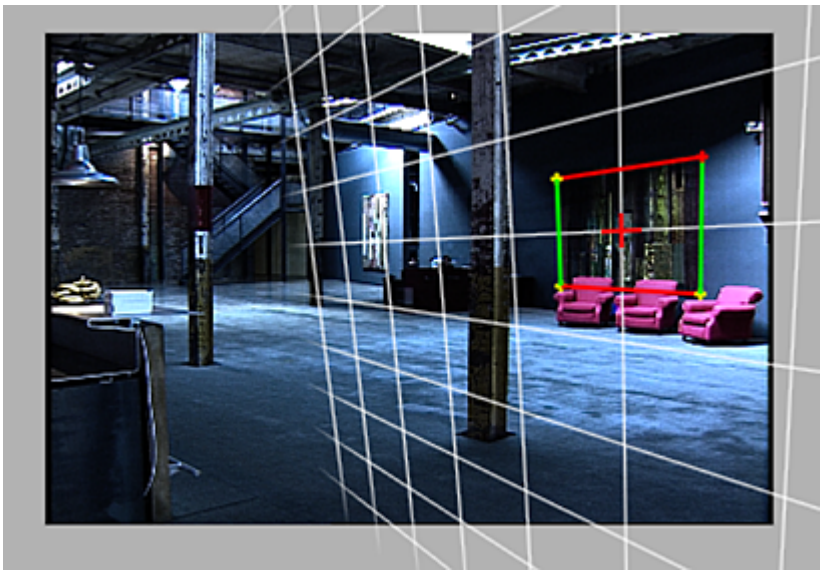
To specify different perspective grid reference media, select the media in the Media menu, then click Apply.

4 With the Perspective Grid object selected in the schematic, select Action Object Solo view from the View box (or press F8) to see the grid and the selected media.



5 From the Perspective Grid menu's Grid tab, select whether you want a 3D or 2D grid mode.

6 While in Action Object Solo view, drag each of the four corners of the rectangle to the desired location. The grid automatically aligns to the new perspective.



TIP If the Magnifier button is active in the Display tab of the Perspective Grid menu, each corner is magnified as you drag it, helping you to find the exact location to place it.

7 Set other Grid Settings (page 657), as needed.

8 Once you are satisfied with the alignment of the perspective grid rectangle, parent an object to the Perspective Grid node. The object inherits the perspective transformation of the Perspective Grid node. Switch to Result view (F4) to see the complete scene.

NOTE You can also manipulate the perspective grid in the Result view, provided that Lock is not selected in the FOV Adjustments box of the Perspective Grid menu, or the perspective grid camera is the same as the Result camera.

Working with FOV

The FOV Adjustments box in the Perspective Grid menu allows you to animate the perspective grid by positioning corner points in other frames. To create an animated transform, place the grid in the first frame, then select Lock in other key frames. You can also select Lock if the camera FOV does not change over the length clip.

If you select Dynamic in the FOV Adjustments box, Action will solve for the FOV in each frame on which you edit the corners. The target camera FOV and position adjusts to fit the perspective set by the perspective grid. If Lock is selected, the perspective grid does not adjust the target camera. To use the existing value in the camera and not solve for it, regardless if you animate the perspective grid, you should select Lock before positioning in the first frame.

NOTE When the FOV of the camera is computed (not locked), the camera is repositioned in order to frame the media.

Tracking with the Perspective Grid

You can use the Perspective Grid and its settings to track the grid position, and apply this tracking data to your Action scene. The Perspective Grid acts like an Axis, so you can apply this tracking data to other objects in the scene.

You can perform a simple four corner track of the grid corners in the Tracker, and perform more complex planar tracking of the media using the Perspective Grid's own planar tracker.

To perform 4 corner Perspective Grid tracking:

1 From the Action Perspective Grid menu's Tracking tab, select the media to be tracked from the Use Media field (to stabilize the Back media, select 0).

2 Set your perspective grid corners.

3 Select 4 Corners from the Tracking Mode box.

4 Click the Enter Tracker button.

The media is automatically loaded into the Stabilizer and the Stabilizer menu appears.

5 In the Stabilizer menu, enable Context to use the Action scene as the reference.

6 Optional: Refine the position the trackers over the corners that you want to track, and click Analyze to generate the tracking data.

NOTE You can fine-tune the tracking data if necessary. See Tracking Difficult Shots and Correcting Errors (page 1091).

7 Once you are satisfied with the tracking, click Return.

The Perspective Grid menu in Action reappears. The tracking data is applied to the Perspective Grid media.

Planar tracking in Action uses tracking algorithms along with combinations of settings to perform the track. Results can vary depending on the footage, as well as settings applied. See Getting Better Results with the Planar Tracker (page 656).

To apply planar tracking to a perspective grid:

1. From the Action Perspective Grid menu's Tracking tab, select the media to be tracked from the Use Media field (to stabilize the Back media, select 0).
2. Set your perspective grid corners.
3. Adjust any Tracking Settings (page 659), as needed.
4. Set any Analysis Constraints, if needed.
5. Click the Analyze button to perform the planar tracking analysis.

Getting Better Results with the Planar Tracker

You can use planar tracking in your Action compositing workflow on many different types of footage to produce different results. The tracking menu exposes many settings to help you prepare and refine the planar track. Read the following sections for some tips and best practices to get the best possible results out of the planar tracker.

What Result Are You Looking For?

Before you perform planar tracking, it is useful to think of what type of result you are trying to achieve in Action. Since you can apply planar tracking from the Axis, Perspective Grid, or GMask node, you should have an idea of how each node handles the resulting tracking data:

- **Planar Tracking in the Axis node:** Use when you want another object or surface to follow to transform happening in an images sequence. Since the Axis itself does not contain media, the planar tracker needs GMask splines to include or exclude to perform the tracking analysis.
- **Planar Tracking in the Perspective Grid node:** Use when you have images with planar info that you want the perspective to be sorted out for you. Since the Perspective Grid acts as an Axis, it performs a similar tracking function, but has the advantage of tracking the grid information directly (with or without GMask constraints).
- **Planar Tracking in the GMask node:** Use primarily for rotoscoping work (to remove parts of the image or the Action scene for another tool or compositing), or if you want to use a GMask as an animated occlude.

Pre-Tracking Settings

See the full list of settings here (page 659) or in the tooltips. Here are a few things to keep in mind before you track:

- Look out for rotation, scale, skew, and perspective in your media to be tracked; and enable the appropriate buttons to help the tracking analysis. Of these four buttons, pay particular attention to Perspective; enable it only if the content is planar, and there is perspective information. Enabling Perspective without these may result in instability in the tracking analysis.
- Sampling field (Region Warping algorithm): As a general rule, a larger sampling reference increases the quality of the results, but takes longer. There may be instances, though, when a lower sampling value works better, such as when there is a lot of grain or noise in the media, or if there is fast camera motion. In these instances, down-sampling may actually eliminate these potential sources of confusion for the algorithm.
- Lighting button (Region Warping algorithm): Enable to help with colour variations or high specularity. Enabling Lighting can also help with diffuse reflection in an object, such as clouds in a window.

- Occlusions button (Region Warping algorithm): Make sure that the sequence has occlusions that you don't want to be tracked. Enabling on a sequence where no occlusions appear, may lead to tracking errors.
- Analyze RGB Channels button (Region Warping algorithm): Should only be enabled when colour information can make a difference in a track. When enabled, it could generate a better result than when only Luminance is used, but at the cost of performance.
- In some cases, bad tracking data can be alleviated by selecting Lock in the FOV Adjustments box in the Perspective Grid menu, before tracking.

Analysis Constraints

Use inclusive or exclusive GMask constraints to limit the tracking to certain areas of the media. When planar tracking from the Axis node, you must set GMask constraints, as the Axis itself can not be tracked.

- Inclusive constraints: You can set GMask constraints (which include GMask gradients, if created), or just GMask gradients (inner and outer gradients can be tracked). An example of using only GMask gradients is when you are tracking a highly reflective surface, such as a smart phone screen. In this case, you can create a gradient on the GMask to expand the trackable area.

  NOTE When tracking from the GMask node, Gradients is the only available inclusive constraint.

- Exclusive constraints: You can set areas of the media to not be considered in the tracking. Media GMasks float in the Action scene (that is, they are not attached to the node being tracked), but they must be assigned to the same media as the node being tracked.

Analysis Settings

To help you view the tracking analysis as it occurs, switch to Object Solo (**F8**) view, and make sure that the Shape Reference and Shape Current buttons are enabled in the Display tab.

If you need to fine-tune the tracking analysis, many of the analysis settings are similar to those in the Stabilizer; Step, Snap, and Auto Update Reference should be familiar to you, as well as using the backspace key to delete bad keyframes (it is preferable to delete any bad keyframes before re-tracking). See the full list of settings here (page 659) or in the tooltips.

- Use **Ctrl** for added functionality with the Go To Reference button to go to the last analyzed frame, and with the Set button to add a keyframe on the track without doing an analysis.
- Updating the reference can help the track, since it constantly compares the current frame content to the reference, to extract the tracking data. You have two options to update the reference:
  - Update it manually when the track is failing, using Snap.
  - Enable Auto Update Reference. Using Auto Update Reference generally allows you to perform an easier track, at the cost of stability, since it compares the current frame to the previous (or next) frame, and therefore is likely to introduce offset over time, or instability in the track. Large occlusions or motion blur may also impede the Auto Update Reference.

Perspective Grid Menu Settings

Grid Tab

Transform Mode box Select either 2D (homographic planar projection of the perspective) or 3D (full 3D perspective transformation) mode. Some settings are not available depending on the mode chosen.

NOTE Perspective Grids contained in Action setups from a previous release appear in 3D Legacy mode, maintaining the previous grid position.

Invert button Enable to remove the animated perspective of the object parented to the grid. Available only in 2D mode.

Scale field Displays the size of the object under the perspective grid. Editable.

Reset button Click to reset the perspective grid settings (does not reset the active perspective camera settings).

Top Left X field Displays the top left X position of the grid. You can also edit this position directly in the image window. Editable.

Top Left Y field Displays the top left Y position of the grid. You can also edit this position directly in the image window. Editable.

Top Right X field Displays the top right X position of the grid. You can also edit this position directly in the image window. Editable.

Top Right Y field Displays the top right Y position of the grid. You can also edit this position directly in the image window. Editable.

Bottom Left X field Displays the bottom left X position of the grid. You can also edit this position directly in the image window. Editable.

Bottom Left Y field Displays the bottom left Y position of the grid. You can also edit this position directly in the image window. Editable.

Bottom Right X field Displays the bottom right X position of the grid. You can also edit this position directly in the image window. Editable.

Bottom Right Y field Displays the bottom right Y position of the grid. You can also edit this position directly in the image window. Editable.

Grid Width field Displays the width resolution (in Action coordinates) of a 2D perspective grid with Invert enabled. Editable.

Grid Height field Displays the height resolution (in Action coordinates) of a 2D perspective grid with Invert enabled. Editable.

Constraint box Select whether to lock the height or width of the grid when manipulating the grid points (select Lock Both to lock the width and height). Lock Both is selected automatically when planar tracking.

Z Offset field Displays the distance between the perspective grid and the camera along the Z axis to make a 3D grid larger or smaller while maintaining the visual result. Editable.

Perspective Camera field Displays the active perspective camera number. Non-editable.

Stereo Camera Perspective box Select whether to use the left or right camera from a stereo camera rig when working with the perspective grid.

FOV field Displays the field of view of the active perspective camera. When adjusting the perspective grid, the FOV of the target camera updates automatically (unless Lock is selected in the FOV Adjustments box). Non-editable.

FOV Adjustments box Select an FOV camera adjustment mode.

Select:	To:
Static	Use all corner keyframes to compute an overall FOV and set the camera to non-animated during tracking of the perspective grid. By using multiple keyframes of corners, this is a better estimate of FOV (for sequences where FOV does not change).
Dynamic	Allow adjustments of the target camera FOV and position during tracking of the perspective grid. Use when the FOV changes over the length of the sequence.

Select:	To:
Lock	Lock the camera FOV and position during tracking of the perspective grid.

Tracking Tab

Common Settings

Tracking Mode box Select which tracking mode to apply.

Enter Tracker button Click to open the Tracker menu to apply tracking data for the selected tracking mode.

Rotation button Enable to apply rotation data to the track.

Skew button Enable to apply skew data to the track.

Scale button Enable to apply scale data to the track.

Perspective button Enable to apply perspective data to the track.

Camera box Specify which camera to take into account when tracking or viewing in Action Object view (F8).

Use Media field Displays the number of the media associated with the analysis. To track the Back media, select 0.

Tracking View box When the media associated to the analysis is a Source node, select to track the media in the Source Front, Source Matte, or Source Result view.

Media FX button Enable to take into account blur and colour correct settings from the Action Media list in the analysis.

Planar Tracking Settings

Analyze button Click to start the planar tracking analysis.

Direction box Select whether to track forward or backward.

Go to Reference button Click to move to the frame of the last Snap operation, or press Ctrl and click to go to the last analyzed frame.

Step button Click to advance to the next frame and analyze.

Set button Click to analyze the current frame, based on the reference frame, to realign the shape used for tracking. Press Ctrl and click Set to add a keyframe on the track without doing any analysis.

Skip button Click to advance to the next frame and delete the selected point on the tracking path (performs a combination of the Step and Delete buttons).

Delete button Click to delete the selected point on the tracking path. You can also use the Backspace key to delete the current keyframe, and move to the previous keyframe.

Auto Update Reference button Enable to update the tracking reference at the current frame after each analysis. Disable to track the movement of the reference point specified in the reference frame.

Snap button Click to redefine the tracking reference at the selected frame.

GMask Inclusive box Select whether to use the union of parented GMasks or GMask gradients as the tracking reference. Select None to use the whole image (Axis), or the four corner region (Perspective Grid).

GMask Exclusive box Select whether to exclude GMasks in the tracking analysis.

Tracking Algorithm box Select which planar tracking algorithm to use. Select Region Warping to use all the pixels defined within the reference, or Multi Feature Detection to use some points defined within the reference. Different settings appear below based on the algorithm chosen.

Sampling field Displays the size of the reference for Region Warping. A smaller reference increases performance at the expense of the quality of its results. Editable.

Lighting button Enable to estimate local lightning variations of the tracked surface.

Occlusions button Enable to automatically exclude parts of the reference to be analyzed based on their likelihood to be occluded.

Analyze RGB Channels button Enable to include all colour channels in the tracking analysis. Disable to only track luminance.

Average field Displays the blending factor to apply to the tracking analysis, based on the neighbouring frames. Use to remove jitter, for example. Note that this operation is destructive, and once applied, the value returns to 0 (you can use Undo to revert, if needed). Editable.

Size field Displays the area in pixels of each tracker. Editable.

Min Number field Displays the minimum number of trackers to use in the analysis. Editable.

Tolerance field Displays the level of acceptable changes in the pixel environment. For example, a low tolerance value ensures that the motion of an external object passing in front of the tracked object does not get tracked through the analysis. Editable.

Display Tab

Shape Reference button Enable to display an overlay of the reference frame when tracking in Object view (F8).

Shape Current button Enable to display an overlay of the current frame when tracking in Object view (F8).

Magnifier button Enable to display the magnifier while dragging a corner of the perspective grid rectangle.

Zoom field Displays the zoom factor of the magnifier. Editable up to 5x.

Display Grid button Enable to display the grid lines in Result or Object view.

Grid Colour pot Displays the colour of the grid lines. Click to change the colour.

Action Lights and Lighting Effects

When you enter Action, you can activate a global shading option (from the Rendering section of the Action Setup menu) to light your complete scene. Once you add a light to the scene, the shading becomes active, and you can also enable Scene Ambient lighting from the Rendering section of the Action Setup menu, or in individual Light menus. Action objects are lit up in the scene according to the number, position, direction, and colour of added light sources, as well as the rotation and spread of each light source.

Action lights also serve as the parent node for a number of lighting effects, such as lens flares, rays, Lightbox shaders, and shadow casts.

Keep in mind the following when working with Action lights:

- You can add up to 64 active light sources to a scene. You can also control each light individually.
- You can set individual lights to different types, such as spotlights or area lights. Ambient is also available as an individual light type, and is different than the global Scene Ambient lighting.
- Lights can respect priority order in Action. Use the Light Source box in the Light menu Basics tab to select whether a light is added to the result of previous lights in the Priority Editor (page 565) (Additive

Light) or to the source diffuse (Solo Light). The Light Priority Editor is also useful for setting the order of Lightbox shaders attached to a light.

- A GMask (or multiple GMasks) connected to a Light with a selective GMask Link (page 901) can help you create interesting effects, where the light is cast through the GMask shape. You can track the GMask, and combine GMask Links with Look At Links and Replica nodes to easily build a composite, such as simulating sunlight shining through windows.
- Use the Action Object Solo (F8) view on a light to see the scene through the point of view of the light, as if it was a camera.
- Similar to cameras in Action, you can set a Free or Target light mode. Select Free to light the scene in the direction that you aim the light, or Target to aim the light at a target object in the scene based on a point of interest. Different settings are available in the Light menu based on the choice of light mode.
- You can attach a look-at connection (page 579) between Action objects with axis characteristics (Axis, Camera, Light, and Projector). The connection allows the parent object to rotate to face the child look-at object, no matter where it is positioned. Look-At links can be animated; therefore you can link different objects from the parent at different frames. In the channel editor, you can see a lookAt channel for every look-at parent.
- By default, the light you add to the scene is applied to all surfaces. However, you can also apply a light source to specific surfaces with inclusive and exclusive light links.
- You can also use image-based lighting (IBL) to light objects in the scene. IBL maps behave differently than other lighting effects, such as lens flares, since you don't attach it to a light in the scene. The IBL performs lighting based on media as an environment texture. Global ambient lighting can also be enabled or disabled from the IBL menu.
- Create your own Lightbox shaders to add interesting custom lighting effects that are cast from lights.

Adding a Light Source

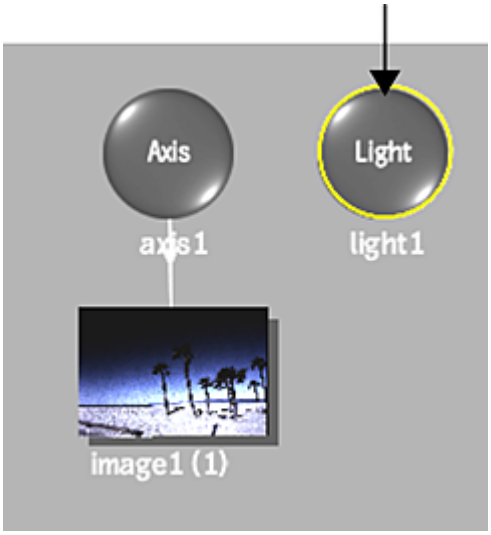
When you enter Action, the scene is lit by global scene ambient lighting. This global scene ambient lighting can be enabled or disabled in the Rendering section of the Action Setup menu or in individual Light menus. You can also add individual light objects in Action. When you add a light to a scene, it is applied to all objects. Before adding another light, you may want to position the light and modify its attributes. In the Lights menu, you can set lighting properties such as position, orientation, spread, falloff, and colour.

NOTE When accessing Action as a Timeline FX, a light object is automatically added to your Action scene.

To add a light to the scene:

1. Do one of the following:
   - Drag the light node from the node bin and place it in the schematic.
   - Drag the light node from the node bin to Result view, so you can see its effect on the scene before placing it exactly where you want.
   - Double-click the light node. The node appears next to the last added object. You do not need to be in Schematic view to add a node in this manner.
   - With the cursor in the viewport, press Space-L to add a new light node that uses the current camera position.

   A light is added to the scene. An icon representing the light source is added to the schematic.



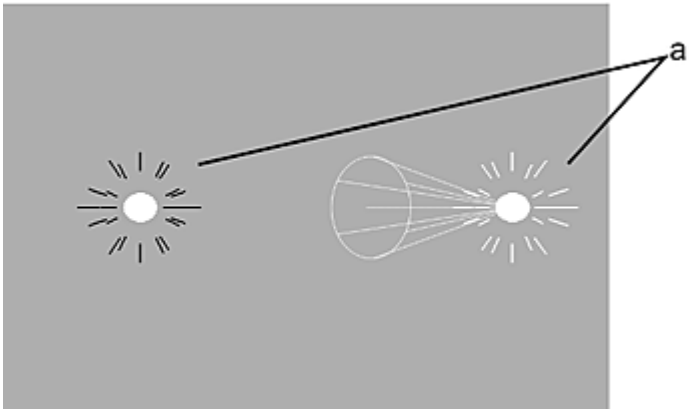
Unlike many objects, a light is added without an axis. To set the position and rotation of a light source, use the Position and Rotation fields in the Light menu.

2 To display the Light menu, double-click the selected light in the schematic.

NOTE The default Z position of the light is set at 50% of the Z position of the default camera in your Action scene.

3 Enable the Active button in the Light menu to activate the light source.

The light icon appears in the scene.



(a) Light sources in the scene.

To work with global scene ambient lighting in the scene:

1 Enable the Scene Ambient button in the Rendering section of the Action Setup menu, or in an individual Light menu.

This is a global setting for the Action scene, and is enabled by default.

2 In the Action Setup menu, you can also adjust the intensity and shading levels of this global scene ambient light.

Selecting a Light Source

You can select a light source in the following ways:

- Click the light source in the scene.
- In Schematic view, click the icon corresponding to the light source. All light sources initially appear at the same X, Y, and Z position in the scene (0, 0, 0). If you add two light sources, for example, you need to move one light source in order to see the other.
- Select a Light folder in the Channel Editor.
- Use the Prev and Next buttons to select the previous or next light source.

Importing Lights

FBX and Alembic files may have lights and their data included, and you can use these same lights in Action. See NO LABEL .

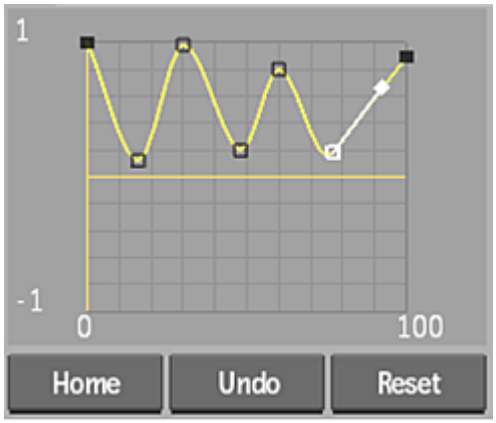
Using the Light Bevel Curve

The bevel curve allows you to create unique lighting effects in conjunction with the Spread field. Since a spread value of 90° or less creates a spotlight, you can then create a bevel curve to act as a multiplier of the spotlight intensity. You have two falloff model options for working on the bevel curve in the Light Profile tab. Different settings appear depending on the model selected.

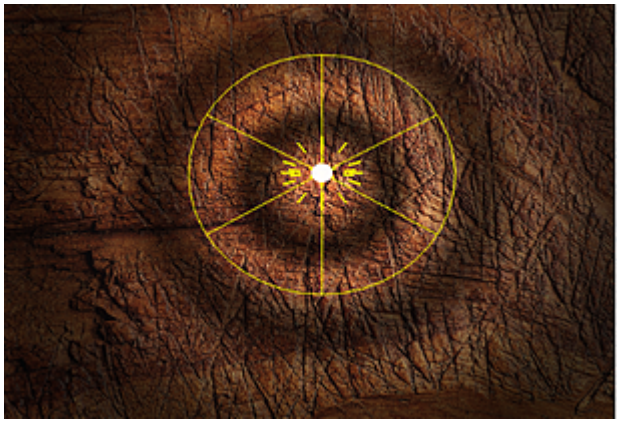
Custom Falloff Model

When selecting Custom as the falloff model, you have full control of the bevel curve. Use the options in the Tools box to add, select, delete, or move keyframes on the bevel curve. The bevel curve behaves in much the same way as an animation curve in the Channel Editor. Make sure that the spread value is less than 90°, and use the settings and the curve itself to create lighting effects.

For example, a curve such as this.



Results in a lighting ripple-like effect.



TIP To create black holes in your lighting effect, use values below zero on the Bevel curve.

Parametric Falloff Model

When using a Parametric model, you have access to falloff in and out settings, as well as an attenuation setting. In this case, if the spread is below 90°, you can see the falloff curve, but are not able to directly manipulate the curve as you can with the custom model.

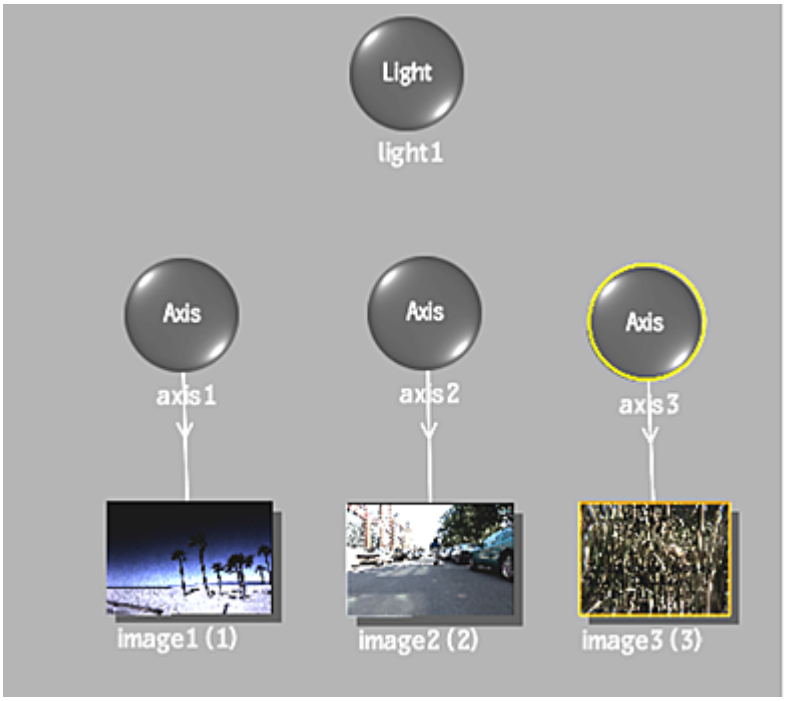
Applying Selective Lighting

When you add a light source to a scene, the light is applied to all surfaces. You may want a light source to only illuminate an individual or specific group of surfaces, or prevent a light source from illuminating an individual or specific group of surfaces.

TIP You can also use selective light links with the Projector nodes (page 819).

To apply selective lighting:

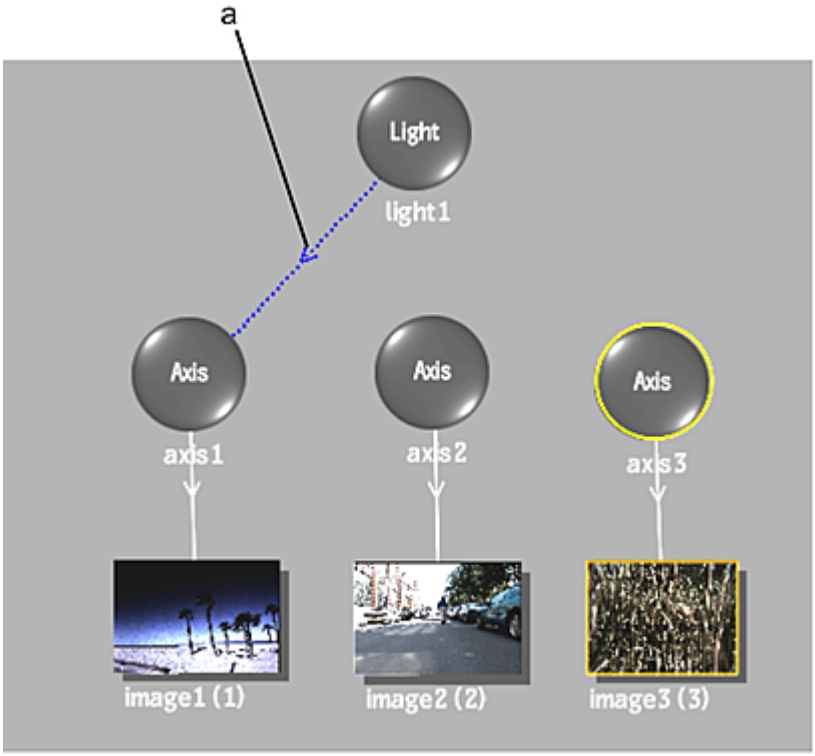
1. Add multiple surfaces to the scene.
2. Add a light to the scene.
   All surfaces are illuminated.



3 Select Light Link from the Tools box.

4 To illuminate only a selected surface, click the light node, and drag it to an axis or image you want illuminated.

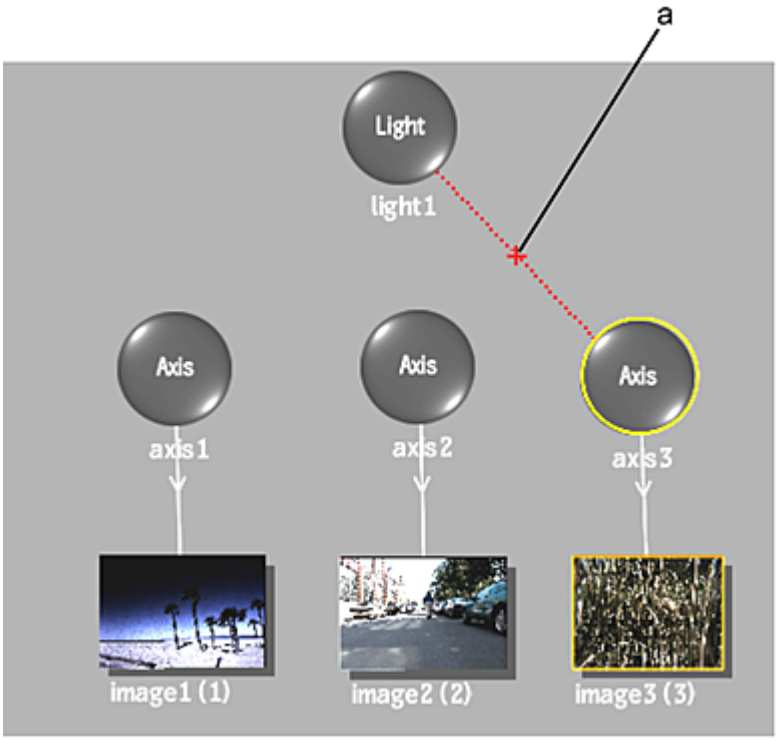
The selected object is connected to the light source by a blue dotted line with an arrow, and only the selected surfaces are illuminated.



(a) Light inclusion link

5 To exclude a surface, hold the `Alt` key while clicking and dragging from the light source to the surface you do not want illuminated.

Excluded surfaces are connected to the light source by a red dotted line with an "X", and they are not illuminated. In the following example, all surfaces are illuminated, except for image3, which is excluded.

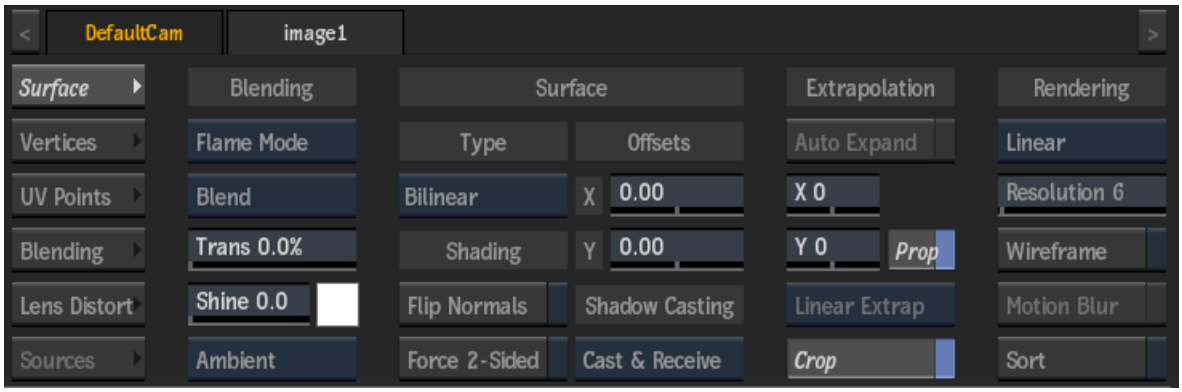


(a) Light exclusion link

TIP To remove the inclusion or exclusion link, while still in Light Link mode, click and drag over the line that connects the light source to the axis or image.

Modifying Surface Lighting

After you set the parent-to-child relationships between lights and surfaces, you can set specific surface lighting properties. You control a surface's lighting and adjust the surface's specular highlight in the Surface menu. To access the Surface menu, double-click the selected surface in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).

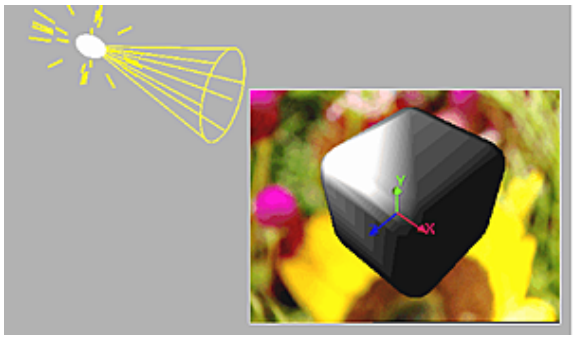


The Surface settings that relate to lights are described as follows.

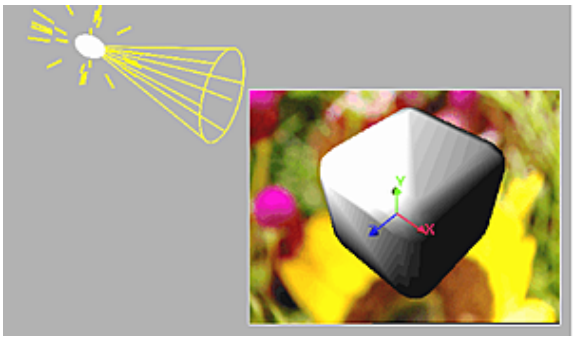
Specular Highlights

A specular highlight is a reflection of a light source. The position of the specular highlight depends on the position and number of light sources surrounding a surface and the angle of the camera.

Use the Shine field to change the intensity of the specular highlight. When the Shine value is set to 0, the specular highlight is disabled. To change the size of the specular highlight, use the Falloff field in the Light menu.



This surface is lit using a Falloff of 30 and a Spread of 27. The surface's Shine is set to 10.



This surface is lit using a Falloff of 20 and a Spread of 27. The surface's Shine is set to 1.

By default, the specular highlight is the same colour as the light source. You can change the colour of the specular highlight by changing the specular colour values.

The specular colour is the colour of light that is reflected by the surface. For example, if the specular colour is red and the light source is white, the specular highlight is red. If the specular colour is yellow and the light source is red, the highlight is orange.

Incidental Light Reflection

Set how a surface reflects incidental light by applying ambient or diffuse lighting. The actual colour of the reflection depends on both the colour value of each pixel and the colour of the incidental light. The intensity of the reflection depends on the orientation of the light source relative to the surface; it is greatest where the incident light strikes the object perpendicular to its surface. The intensity of the reflection is independent of the camera eye position.

When you turn shading on, you do not have to enable a light source to see the lighting effect, as a default infinite light source supplies ambient light at 20% intensity. The infinite light source is located behind the camera eye and cannot be moved. As soon as you add a light source, the infinite light source is replaced by the new light source.

In the Surface menu, make a selection in the Lighting box:

Select:	To reflect incidental light:
Ambient	To all parts of a surface that are not directly illuminated.
Diffuse	Equally in all directions, producing a flat reflection on the object.

TIP You can optionally adjust the colour of the incidental light using the ambient or diffuse RGB channels. To display the RGB channels, click Animation to display the Channel Editor. Expand the surface's folder (it should already be selected), expand its Material folder, then expand the Ambient or Diffuse folder.

Flipping a Surface's Normals

Flips the normals of the surface so that light is applied to the opposite side of the surface. Use to create a two-sided shaded surface. To control both surfaces, parent them by a new axis and use this axis to rotate, scale, shear, and move the two surfaces.

NOTE There may be a priority problem causing one surface to be drawn over the other. To correct this problem, use the Priority Editor (page 565) to animate the drawing priority of surfaces, or change the Z position of one surface by one pixel.

Converting to Wireframe

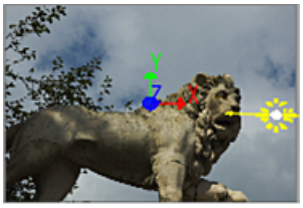
You can remove some lighting effects for a selected surface by converting surfaces to a wireframe depiction of the surface. When Wireframe is enabled, specular values and any applied textures are replaced with a wireframe view of the surface. When used on an image, the surface will adopt a screen-like look.

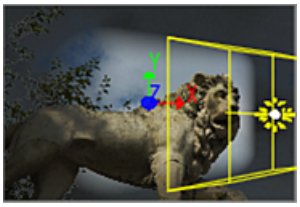
Light Menu Settings

When accessing Action as a Timeline FX, you have access to a quick menu with some of these Light settings. To see the full Light menu, click the Editor button to enter Action.

Basics Tab

Light Type box Select the type of light to apply to the scene.

Light Type:	Description:	Example:
Point / Spot	A point light radiates light uniformly in all directions. A spotlight radiates a cone of light centred along the spotlight direction. Use the Spread field to change the spread angle. A spread of 90° or less creates a spot-light.	
Directional	A directional light shines evenly in one direction only. The light icon in the scene displays an arrow showing the direction of the light. Use a directional light to simulate a very distant point light source (for example, the sun as viewed from the surface of the Earth).	

Light Type:	Description:	Example:
Ambient	An ambient light shines in two ways—some of the light shines evenly in all directions from the location of the light (similar to a point light), and some of the light shines evenly in all directions from all directions (as if emitted from the inner surface of an infinitely large sphere). Use the Shade field to set the percentage on ambient light applied. **NOTE** The ambient light type is not the same as the global Scene Ambient lighting available for the whole Action scene.	
Rectangle Area	A rectangle area light is similar to a point or spot light except that it produces a hotspot based on the shape of the rectangle. Use the area size fields to set the size of the rectangle.	
Ellipse Area	An ellipse area light is similar to a point or spot light light except that it produces a hotspot based on the shape of the ellipse. Use the area size fields to set the size of the ellipse	

Different settings may appear directly below the Light Type box depending on the type selected.

Width field Displays the width of an area or directional light. Editable.

Height field Displays the height of an area or directional light. Editable.

Shade field Displays the percentage of ambient light applied. Editable.

Light Mode box Select Free to light the scene in the direction that you aim the light, or Target to aim the light at a target object in the scene based on a point of interest.

Light Intensity field Displays the intensity of the selected light. Available in the Light menu Basics and Profile tabs. Editable.

Light Spread field Displays the spread angle. A value of 90° or less creates a spotlight. Available in the Light menu Basics and Profile tabs. Editable

X Position field Displays the position of the selected light along the X axis. Editable.

Y Position field Displays the position of the selected light along the Y axis. Editable.

Z Position field Displays the position of the selected light along the Z axis. Editable.

Path button Enable to animate the position of the light using a spline drawn in the scene. Disable to animate the position of a light using explicit animation.

Motion Blur button Enable to use a motion blur effect for the selected light (can only be used if the global Motion Blur is enabled in the Setup menu).

X Rotation field Displays the rotation of the selected light along the X axis. Available when Free is selected in the Light Mode box. Editable.

Y Rotation field Displays the rotation the selected light along the Y axis. Available when Free is selected in the Light Mode box. Editable.

Z Rotation field Displays the rotation the selected light along the Z axis. Available when Free is selected in the Light Mode box. Editable.

Distance field Displays the position of the light's focus. Available when Free is selected in the Light Mode box.

X Point of Interest field Displays the position of the point of interest along the X axis. Available when Target is selected in the Light Mode box. Editable.

Y Point of Interest field Displays the position of the point of interest along the Y axis. Available when Target is selected in the Light Mode box. Editable.

Z Point of Interest field Displays the position of the point of interest along the Z axis. Available when Target is selected in the Light Mode box. Editable.

Roll field Displays the amount of light roll. Available when Target is selected in the Light Mode box.

Red Light field Displays the red value of the selected light. Editable.

Green Light field Displays the green value of the selected light. Editable.

Blue Light field Displays the blue value of the selected light. Editable.

Light colour pot Displays the colour of the light source. Editable.

Active button Enable to turn the selected light source on.

Lightbox Order box Select whether an attached Lightbox effect occurs before (Pre) or after (Post; default) the light.

Light Source box Select whether the light or Lightbox effect is added to the result of previous lights in the Priority Editor (Additive Light) or to the source diffuse (Solo Light).

Active button Enable to light up the scene using added light sources. When disabled, no lighting effects appear in the scene; surfaces and 3D models appear flat. This same setting can be found in the Action Setup menu and the Light menu.

Scene Ambient button Enable to have global ambient lighting in your Action scene. This same setting can be found in the Action Setup menu and the Light menu.

Profile Tab

Settings for controlling the light bevel curve are located in the Light Profile tab.

Light Intensity field Displays the intensity of the selected light. Available in the Light menu Basics and Profile tabs. Editable.

Light Spread field Displays the spread angle. A value of 90° or less creates a spotlight. Available in the Light menu Basics and Profile tabs. Editable

Light Attenuation field Displays the level of amplitude of the light. Available when Parametric is selected in the Falloff Model box. Editable.

Light Falloff field Displays the amount of falloff around the edge of the light source (also changes the size of the specular highlight). Available when Custom is selected in the Falloff Model box. Editable.

Decay Type box Select the type of decay to apply to the light source.

Decay field Displays the rate at which light decreases for the chosen decay type. Editable.

Falloff Model Settings

Falloff Model box Select whether to use a parametric or custom falloff model. A custom falloff allows you to interactively use the profile curve, while a parametric falloff displays the curve for visual reference only.

Falloff In field Displays the level of incoming light falloff. Available when Parametric is selected in the Falloff Model box. Editable.

Falloff Out field Displays the level of outgoing light falloff. Available when Parametric is selected in the Falloff Model box. Editable.

Light Bevel curve Displays the specific profile of the selected light. Use the options in the Tools box to add, select, delete, or move keyframes on the bevel curve. When using a parametric falloff model, the curve is read-only. See Using the Light Bevel Curve (page 663).

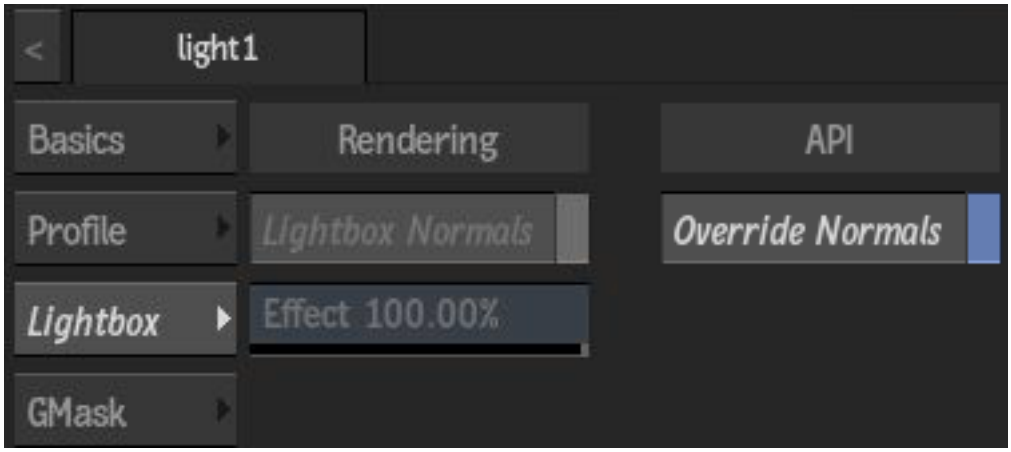
Home button Resets the Bevel curve viewer to show the whole curve.

Undo button Undoes the last set of Bevel curve operations.

Reset button Resets the Bevel curve.

Lightbox Tab

Use these settings when a Lightbox node is attached to a Light.



Lightbox Normals button Enable to take normals into consideration to compute the transparency of the Lightbox effect on the scene.

Effect field Displays how much the orientation of the normals affects the result of the shader, relative to the light position in the scene. Available when Lightbox Normals is enabled. Editable.

Override Normals button Enable to force all surfaces to be affected by the Lightbox shader, when Light Normal attenuation isn't appropriate for the Lightbox effect. Disable to only affect front-facing surfaces, relative to the light. The default behaviour of this button can be set as an XML tag in the shader.

GMask Tab

Use this setting when a GMask is connected to a Light with a GMask link.

GMask Link Texture Size box Select the texture size of a GMask connected to a Light through a GMask Link.

Relighting: Casting Shadows

You can create realistic 2D and 3D shadows in your Action scene by using lights to cast and receive shadows, as well as self-shadow (an object can cast a shadow on itself).

After adding and setting up shadow casts in Action, you can output the shadow by itself, or as part of your overall composition. A shadow pass is a white image with greyscale regions that represent the shadow coverage. See Output Options (page 601) for information on the specific shadow output settings.

Action also supports drop shadows (see Adding Drop Shadows (page 644)).

Adding a Shadow Cast to a Light

Lights in the scene are able to cast shadows. You can parent a Shadow Cast object to multiple lights at once. This allows you to control the overall attributes of the shadow (for example, colour, softness, and transparency).

To add a shadow cast to the scene:

1. Add and position a light to your scene.
2. Do one of the following:
   - Drag the shadow cast node from the node bin and place it in the schematic.
   - Double-click the shadow cast node.

   If there is only one light in the scene, the shadow cast node is automatically connected to it. If there are several lights in the scene, select the light in the schematic that you want parented to the shadow cast before added the shadow cast node. Otherwise, you can parent the shadow cast node to the light or lights manually in the schematic.
3. To display the Shadow Cast menu, double-click the shadow cast object in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 571).

Surface and Geometry Shadow Casters

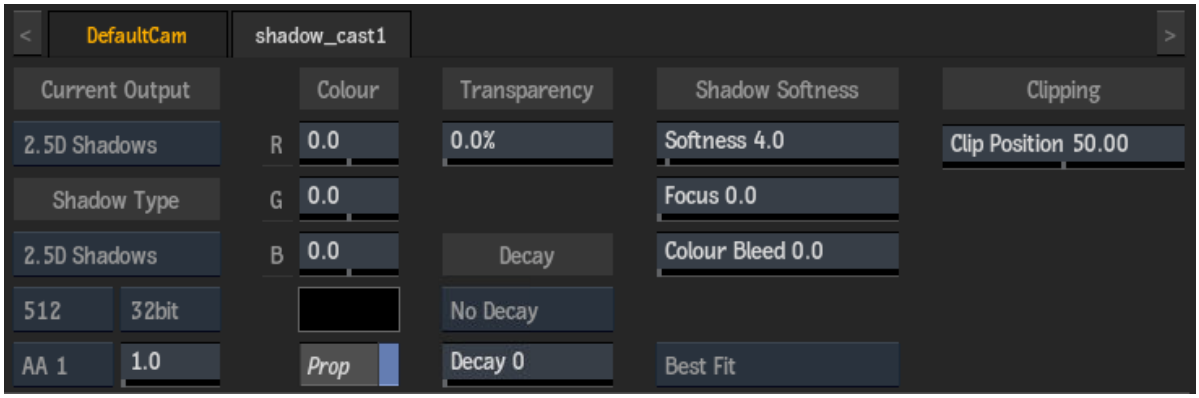
By default, all objects in the scene cast and receive shadows from a light attached to a Shadow Cast object. You can control this on a per object basis in the Object Surface or Geometry menu.

The Shadow Casting box in the Surface or Geometry menu allows you to select how the each image or geometry object is affected by a Shadow Cast object in the scene.

Select:	For the Surface or Geometry to:
Invisible Receiver	Be invisible in the scene while still receiving shadows. All object properties are maintained, so you can still use blend modes, transparency, or even texture the surface or geometry for more flexibility. **TIP** You can use a "dummy" geometry or surface as the receiver, such as a white frame, angled to match the background. In this case, set the Blend Mode to Multiply, and use other object properties, such as transparency and diffuse colour.
Invisible Caster (was previously called Shadow Only)	Not be displayed, but the shadow is displayed.
Cast & Receive	Cast and receive a shadow (this is the default).
Receiver	Receive, but not cast a shadow.
Caster	Cast, but not receive a shadow.
Off	Not cast or receive a shadow.

Shadow Cast Menu Settings

The Shadow Cast menu settings are described as follows. Shadow types are dependant on your graphics card, so you may not see all of these settings.



Shadow Output Type box Select the type of shadow to output. This setting is repeated in the Render Passes section of the Output menu for a Shadow Output Selection.

NOTE When set in the Output menu, this becomes the default Shadow Type for any new Shadow Cast nodes.

Source Shadow Type box Select the type of shadow cast for the source node. This setting is available in the Shadow Cast menu only if the shadow cast node is a child of a source node, and is repeated in the Source menu.

Shadow Type box Select a mapping type for the shadow.

Shadow Type:	Description:
3D Hard Shadows	Use 3D Hard Shadows for higher precision shadows when lights are close to the objects in the scene. These are best for hard edge shadows with penumbra effects, but can be slower.
3D Soft Shadows	Use 3D Soft Shadows if you want faster soft shadows with lights further away from your objects. These shadows offer explicit control over softness.
2.5D Shadows	2.5D Shadows work best on transparent or semi-transparent objects, for example a shadow projected on a wall by smoke.

NOTE Some of the settings in the Shadow Cast menu differ based on the shadow type you choose.

Resolution box Select a resolution to determine the quality of the selected casting type. You can select up to 8K resolution, but a message may appear if too much memory is used with multiple Shadow Cast nodes. In this case, you can Confirm to move to the next lower resolution, or Cancel to disable the shadow cast.

Texture Bit-Depth box Select a 16 or 32-bit quality for the shadow cast. 32-bit offers more precision, but at the cost of memory usage.

Anti-Aliasing Sample box Select an anti-aliasing sampling level for the shadow cast.

NOTE Depending on your graphics card and the size of your Action scene, shadows may not appear or render properly with higher anti-aliasing sampling levels.

Anti-Aliasing Softness field Displays the softness value of the anti-aliasing sample for the shadow cast.

NOTE Jitter on 3D shadows is inherent to the shadow map technique. However, in most situations it can be go completely unnoticed. Some lighting situations which cause stretched shadows, such as lights close to the horizon, are very likely to create visible jittering. To alleviate jittering problems, you can try to use higher resolution, anti-aliasing, and anti-aliasing softness levels. In these cases, interactive manipulations in the image window may become taxing; therefore, you should activate Adaptive Degradation (page 594) for Shadows.

Red Colour field Displays the amount of red in the shadow (based on the colour of the attached light). Editable.

Green Colour field Displays the amount of green in the shadow (based on the colour of the attached light). Editable.

Blue Colour field Displays the amount of blue in the shadow (based on the colour of the attached light). Editable.

Shadow Colour pot Displays the colour of the shadow (based on the colour of the attached light). Editable.

Proportional button Enable to change the Red, Green, and Blue colour fields proportionally.

Transparency field Displays the transparency level of the shadow. Editable.

Decay Type box Select the type of decay to apply to the shadow.

Decay field Displays the rate at which the shadow decreases for the chosen decay type. Editable.

Softness field Displays the softness of a shadow. Editable.

Density field Displays the level of height/depth discontinuity in a 3D soft shadow. Use to fix light bleeding issues. Editable.

Dark Threshold field Displays the distance of the object from a 3D soft shadow. Use to create a spatial offset, and to remedy step-ladder or levitating problems. Editable.

Focus field Displays the softness of the 2.5D shadow based on the distance from the light. Objects closer to the focus distance are less blurred. Editable.

Colour Bleed field Displays the amount of colour bleed in the 2.5D shadow from semi-transparent objects in the scene. Editable.

Flattening Mode box Select a flattening mode for the 2.5D shadow. Most of the time, Best Fit gives the best quality, but if you see clipping artefacts in the shadow, try one of the other modes.

Penumbra field Displays the softness of a 3D hard shadow. Editable.

Sampling Mode box Select a softness sampling mode for the 3D hard shadow.

Filter Samples field Displays the amount of filter samples to take into account when creating softness (X xY) for a 3D hard shadow. Available when Regular sampling is chosen in the Sampling Mode box. Editable.

Caster Details field Displays the amount of shadow caster samples to take into account when creating softness (X x Y) for a 3D hard shadow. Editable.

Auto Near/Far button Enable to automatically set the near and far parameters for the 3D shadow based on the objects in the scene.

Near field Displays the near distance of the start of the 3D shadow. Editable.

Far field Displays the far distance of the end of the 3D shadow. Editable.

Matte Threshold field Displays the value at which the alpha casts a 3D shadow. Editable.

Overlap field Displays the amount of overlap from the light source. Increase to remove imperfections in the 3D shadow. Editable.

Clip Position field Displays the position of the clipping plane for 2.5D shadows, relative to the depth of the shadow. Use to define at what depth the shadow begins to be shown. Editable.

Relighting: Lens Flares

Use lights in your scene to generate procedural lens flares with built-in 3D occlusions.

With lens flares in Action, you can control:

- How lights change as they move behind 2D or 3D layers.
- How flares behave when a light exits or enters the camera field of view.

Lens Flares in Action are comprised of a Lens Flare object, attached to one or more Border FX objects, and any number of texture components, such as irises, streaks, and glows. To help you get accustomed to working with lens flares, a preset with typical settings is loaded when you first add a lens flare object to your scene.

Multiple lens flares can be attached to a light, and multiple lights can attach to a lens flare.

TIP Use a GMask link to connect a GMask directly to the Lens Flare node in the schematic. If you then select Use As Occluder in the Post Processing box in the GMask menu, you can create interesting effects by having the GMask occlude only the Lens Flare effect.

Rendering Lens Flares

Lens Flares can be added as a render pass output selection. The lens flare (and rays) result can also applied by default in the Comp output (using the Action Lens Flare / Rays button).

You can deactivate the Action Lens Flare / Rays button, and put a lens flare anywhere in the post processing pipeline, using its own output pass via a Lens Flare Matchbox shader (by default Lens Flares are at the very end of the Camera FX pipeline).

See Outputs Options (page 601) for more information.

Adding a Lens Flare to a Light

In Action, a lens flare is a child of a light. Multiple lights can be parented to the same Lens Flare object.

To add a lens flare to the scene.

1. Add and position a light to your scene.
2. Do one of the following:
   - Drag the lens flare node from the node bin and place it in the schematic.

- Double-click the lens flare node.

  A default lens flare (including a Lens Flare object, Border FX object, and multiple component texture objects) is added to the schematic. In Result view Live Preview mode, you can see the lens flare (you may need to move the light in the scene to see the full lens flare effect).

  If there is only one light in the scene, the lens flare node is automatically connected to it. If there are several lights in the scene, select the light in the schematic that you want parented to the lens flare before added the lens flare node. Otherwise, you can parent the lens flare node to the light or lights manually in the schematic.

3. To add a texture component to a lens flare, select the Border FX object in the schematic, then double-click a texture node in the Relighting tab of the node bin.
4. To display the Lens Flare menu, double-click the lens flare object in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 571).

You can also load and save lens flare presets. See Using Lens Flare Presets (page 677).

Using Lens Flare Presets

You can save and load lens flare presets, offering you the flexibility to create a collection of your own lens flares with the objects and settings you want.

NOTE You can save and reload Lens Flare presets containing animation channel expressions. Note that only animation channel expressions found inside the setup will reload properly (Lens Flare nodes and its children). Also keep in mind that node names need to be unique for expressions to resolve properly (if you load the same Lens Flare setup twice, you'll have an expression resolving clash).

To load a lens flare preset:

1. From an existing lens flare in the schematic, double-click the Lens Flare node to access the Lens Flare menu.
2. From the Basics tab, click the Load button in the Preset section to open the Preset browser.

   By default the browser points to *opt/Autodesk/shared/presets/lensflare*.
3. Select a preset from the default path (in the Creative or RealLenses subfolders), or navigate to a path of your choosing to select a preset, then click Load.

   The Lens Flare object, as well as any other nodes included in the preset are attached to the light, replacing the current lens flare.

To save a lens flare preset:

1. Set up a lens flare object with any settings and attached texture components that you want.
2. From the Basics tab of the Lens Flare menu, click the Save button in the Preset section.
3. From the Preset browser that appears, name and select a location for your saved preset. Be careful not to overwrite an existing preset, unless you want to.

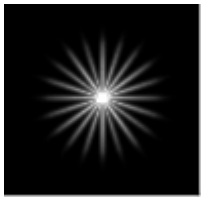
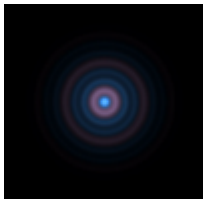
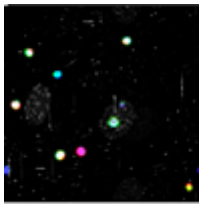
   The Lens Flare preset is saved with all nodes that are connected under the Lens Flare node, including Border FX nodes, and texture components (even a Diffuse Map or Matchbox node, if used to re-texture a component).

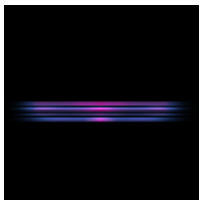
NOTE Legacy Lens Flare presets are also available through the Action Presets node. These presets are not available in the same directory path as the presets accessed from the Lens Flare menu.

Using Texture Components With Lens Flares

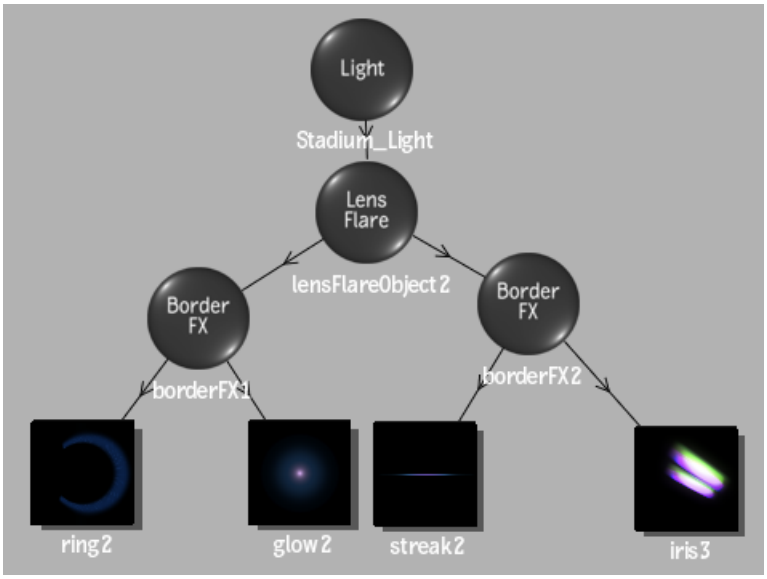
You must add texture components to the Border FX object of a lens flare to see a result. These components are textures that are attached to a Lens Flare Border FX object (or optionally, directly to a Rays or Blooming object). The Lens Flare Object menu has global settings for the complete lens flare, but each component texture has its own menu to control settings particular to it.

Each type of component can be added multiple times, each with its own settings. The following are the types of texture components you can add:

Component:	Description:	Example:
Glint Texture	Star-like texture.	
Glow Texture	Glowing loops of different colours.	
Iris Texture	Multiple shapes, such as polygons, discs, orbs, or caustics.	
Lens Texture	Lens "defects" such as hair, scratches, or fingerprints.	
Ring Texture	Rings with multi-coloured loops.	

Component:	Description:	Example:
Streak Texture	Lines streaking across the image.	

In the Action schematic, the components are attached to a Border FX object that is itself attached to the Lens Flare object. You can use multiple Border FX objects to achieve the lens flare look you want. For example, attach a ring and glow to one Border FX object, while a streak and iris are attached to another Border FX object. In this case, you can control how the lens flare behaves when it reaches the borders of your image differently for the components under each Border FX object.



TIP You can also attach multiple Border FX objects to the same texture component, to compare different Border FX settings, for example. In this case, you can use the Action Hide button to hide each selected Border FX node to see the different results.

Re-Texturing Components

Since the components are textures, you can also re-texture a component using a Diffuse Map with your own texture media applied. To do so, select the component in the schematic, then select the media you want to use from the media list, and double-click the Diffuse Map node in the node bin. In this case, the Pattern settings in the component menu are not applicable, though you can still use the settings in the Basics and Border FX tabs of the component menu, as well as the Diffuse menu settings.

Using Matchbox Textures with Lens Flares

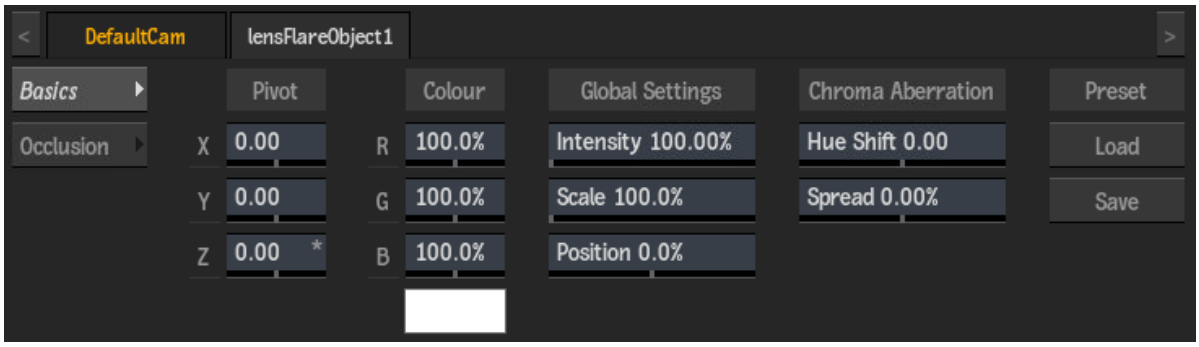
You can also use a Matchbox shader to re-texture a lens flare component, by connected the Matchbox node to the component that you want to replace. Since Matchbox nodes can contain embedded textures in a texture grid, this is a good way to add your own real world lens flare elements, or even hijack the lens flare by texture bombing with your own graphic elements, such as logos. A number of lens flare Matchbox preset shaders are included in Flame.

You can also use lens flare components on their own as Substance Texture presets, as a *LENSFLARE* subfolder is available in the Substance presets browser. See Substance Textures (page 762) for more information.

Lens Flare Menu Settings

The Lens Flare Object menu lets you control settings for the complete lens flare effect. The Border FX object and each attached component texture also have their own specific menus.

Basics Tab



Position X field Displays the screen space position along the X axis of the Lens Flare pivot point. Editable.

Position Y field Displays the screen space position along the Y axis of the Lens Flare pivot point. Editable.

Position Z field Displays the screen space position along the Z axis of the Lens Flare pivot point. Editable.

Red Colour field Displays the amount of red in attached flare components (based on the colour of the attached light). Editable.

Green Colour field Displays the amount of green in attached flare components (based on the colour of the attached light). Editable.

Blue Colour field Displays the amount of blue in attached flare components (based on the colour of the attached light). Editable.

Flare Colour pot Displays the colour of the attached flare components (based on the colour of the attached light). Editable.

Global Intensity field Displays the intensity of the attached flare components (multiplied by the intensity of the attached light). Editable.

Global Scale field Displays the scale of the attached flare components. Editable.

Global Position field Displays the position of attached flare components in relation to the light (0%) and pivot point (100%). Editable.

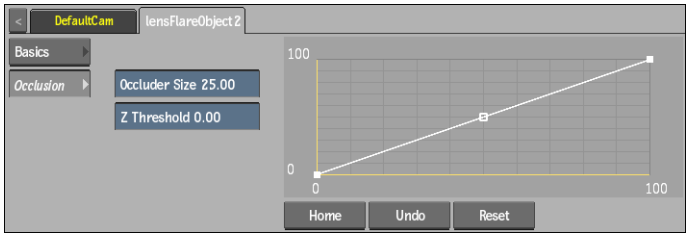
Hue Shift field Displays the RGB offset applied to the edges of attached flare components. Editable.

Spread field Displays the amount of refractive distortion applied to the edges of attached flare components. Editable.

Load button Click to open a preset browser to select a lens flare preset.

Save button Click to open the preset browser in order to save the current lens flare (including all components attached under the lens flare node).

Occlusion Tab



Occlusion curve Displays the occlusion profile of the lens flare. Use to set the behaviour of the lens flare components when behind other objects in the scene.

Occluder Size field Displays the size of fade in/out of occluded lens flare components. Editable.

Z Threshold field Displays the transparency value at which objects start occluding the flare. Editable.

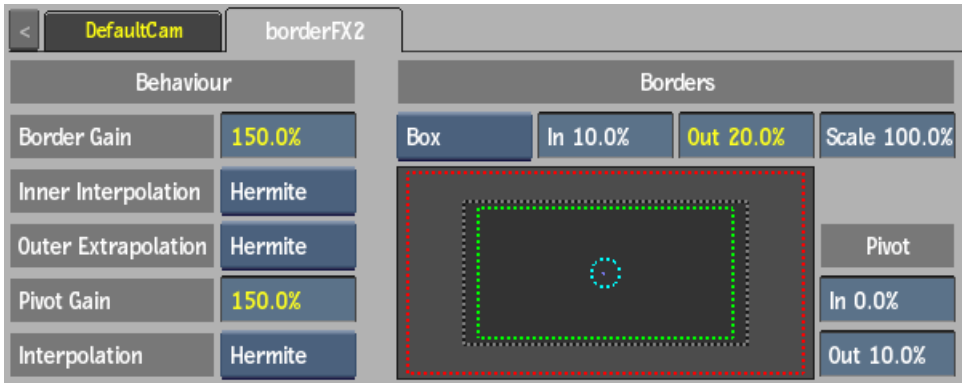
Home button Resets the bevel curve viewer to show the whole curve.

Undo button Undoes bevel curve operations.

Reset button Resets the bevel curve.

Border FX Menu Settings

Use the Border FX settings to control how the lens flare behaves when it reaches the borders of your image. A visual representation of the inner and outer borders, as well as the pivot area is displayed in the menu.



Border Gain field Displays the amount of gain to apply to the attached lens flare components when the light reaches the defined border. Editable.

Inner Interpolation box Select an interpolation type to define the transition between the border gain and the inner border.

Outer Extrapolation box Select an extrapolation type to define the transition between the border gain and the outer border.

Pivot Gain field Displays the amount of gain to apply to the attached lens flare components when the light reaches the defined pivot area. Editable.

Pivot Interpolation box Select an interpolation type to define the transition between the pivot gain and the pivot point.

Border Mode box Select whether Border FX settings are applied vertically, horizontally, or in both directions.

Inner Border Margin field Displays the position of the inner border. Editable.

Outer Border Margin field Displays the position of the outer border. Editable.

Border Scale field Displays the scale of the border that controls where the lens flare effect occurs. Editable.

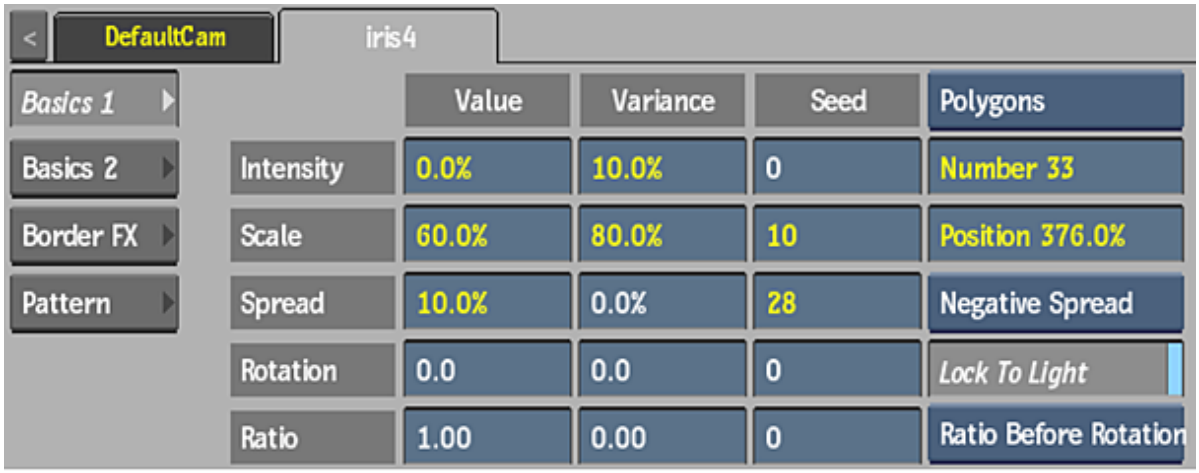
Pivot Radius In field Displays the inner radius of the pivot point. Editable.

Pivot Radius Out field Displays the outer radius of the pivot point. Editable.

Component Menu Settings

Each flare component has its own menu to control settings particular to the component. Some of these settings are common among the different component types, while some are specific to the component type.

Basics 1 Tab



Intensity Value field Displays the brightness of the component. Editable.

Intensity Variance field Displays how much the intensity varies. Available when the Number field value is greater than 1. Editable.

Intensity Seed field Displays the random intensity seed value. Available when the Number field value is greater than 1. Editable.

Scale Value field Displays the size of the component. Editable.

Scale Variance field Displays how much the scale varies. Available when the Number field value is greater than 1. Editable.

Scale Seed field Displays the random scale seed value. Available when the Number field value is greater than 1. Editable.

Spread Value field Displays the position of components in relation to each other. Available when the Number field value is greater than 1. Editable.

Spread Variance field Displays how much the spread varies. Available when the Number field value is greater than 1. Editable.

Spread Seed field Displays the random spread seed value. Available when the Number field value is greater than 1. Editable.

Spread option box Select a behaviour for the spread settings: Centre (equal), Positive (light to pivot direction), or Negative (pivot to light direction).

Rotation Value field Displays the level of rotation of the component. Editable.

Rotation Variance field Displays how much the rotation varies. Available when the Number field value is greater than 1. Editable.

Rotation Seed field Displays the random rotation seed value. Available when the Number field value is greater than 1. Editable.

Ratio Value field Displays the aspect ratio of the component. Editable.

Ratio Variance field Displays how much the ratio varies. Available when the Number field value is greater than 1. Editable.

Ratio Seed field Displays the random ratio seed value. Available when the Number field value is greater than 1. Editable.

Iris Shape box Select the shape of iris component. For other component types, this box is grayed out and displays the type of component.

Number field Displays the amount of components. Editable.

Position field Displays the offset applied to the component in relation to the light (0%) and pivot point (100%). Editable.

Lock To Light button Enable to lock the orientation of the component to the light.

Order box Select whether ratio is applied before rotation or vice-versa.

The Lens Texture component Basics 1 tab only has the following two settings (not shown):

Overall Brightness field Displays the brightness level of the Lens Texture component. Editable.

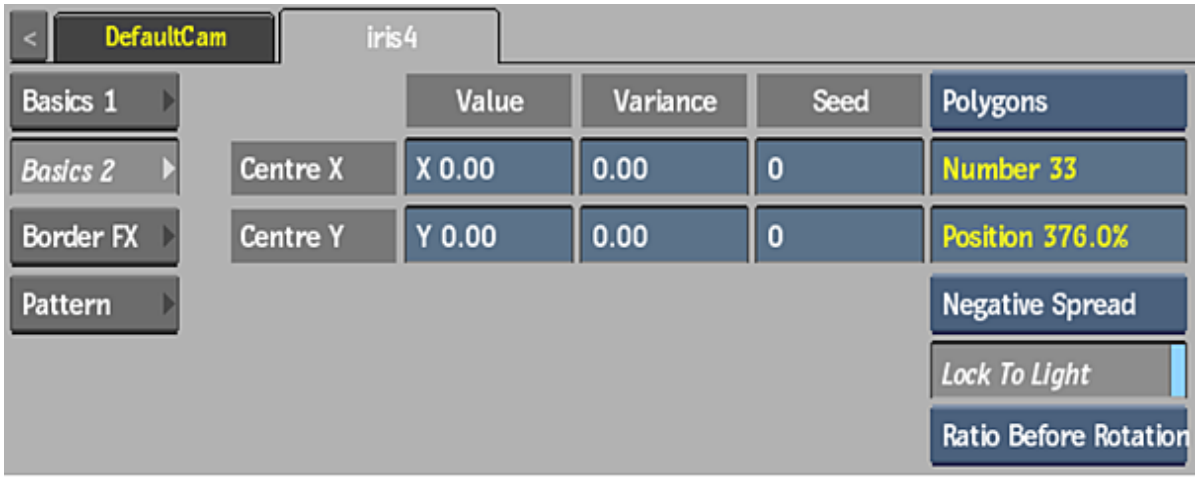
Inner Brightness field Displays how much of the lens is revealed. Editable.

The Glint component Basics tab also has the following two settings (not shown):

Overall Speed field Displays the rate at which the Glint animation plays.

Time Offset field Displays the start point of the Glint animation. With a value of 0, the animation starts at frame 1. With a value of 100, the animation begins as if it has been generating for 99 frames. You cannot animate this field.

Basics 2 Tab



Centre X Position field Displays the position of scaling and rotation of the component along the X axis. Editable.

Centre X Variance field Displays how much the centre position varies along the X axis. Available when the Number field value is greater than 1. Editable.

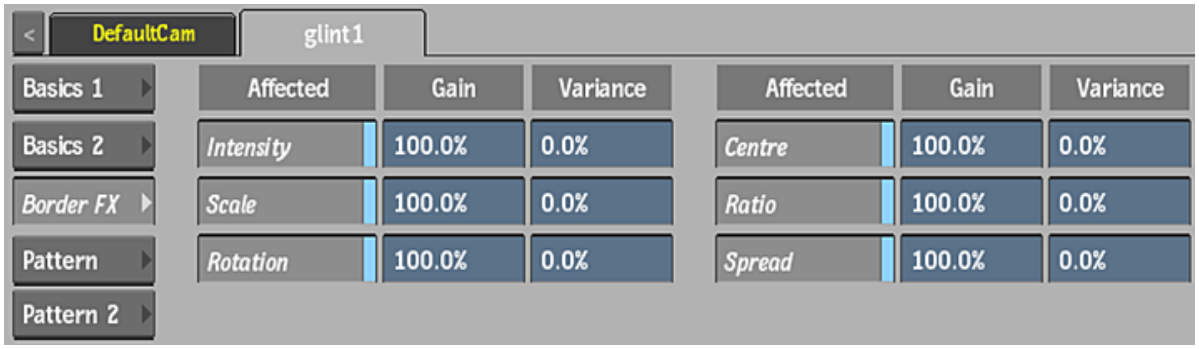
Centre X Seed field Displays the random centre X seed value. Available when the Number field value is greater than 1. Editable.

Centre Y Position field Displays the position of scaling and rotation of the component along the Y axis. Editable.

Centre Y Variance field Displays how much the centre position varies along the Y axis. Available when the Number field value is greater than 1. Editable.

Centre Y Seed field Displays the random centre Y seed value. Available when the Number field value is greater than 1. Editable.

Border FX Tab



The component Border FX settings allow you to choose which how the component interacts with the parent lens flare border settings.

Border Intensity button Enable to take into account intensity values of the parent relighting object.

Intensity Gain field Displays the amount of gain applied to the intensity. Editable.

Intensity Variance field Displays how much the intensity varies. Editable.

Border Scale button Enable to take into account scale values of the parent relighting object.

Scale Gain field Displays the amount of gain applied to the scale. Editable.

Scale Variance field Displays how much the scale varies. Editable.

Border Rotation button Enable to take into account rotation values of the parent relighting object.

Rotation Gain field Displays the amount of gain applied to the rotation. Editable.

Rotation Variance field Displays how much the rotation varies. Editable.

Border Centre button Enable to take into account centre values of the parent relighting object.

Centre Gain field Displays the amount of gain applied to the centre. Editable.

Centre Variance field Displays how much the centre varies. Editable.

Border Ratio button Enable to take into account ratio values of the parent relighting object.

Ratio Gain field Displays the amount of gain applied to the ratio. Editable.

Ratio Variance field Displays how much the ratio varies. Editable.

Border Spread button Enable to take into account spread values of the parent relighting object.

Spread Gain field Displays the amount of gain applied to the spread. Editable.

Spread Variance field Displays how much the spread varies. Editable.

Pattern Tab(s)

Use the pattern settings to control the texture before it is applied to the lens flare or ray. The settings in the Pattern tabs vary depending on the component type. You can get a quick description of each setting by viewing its tooltip.

Relighting: Rays

Use rays to simulate volumetric effects. You can attach a Rays object to a light in Action.

You can position rays behind 2D or 3D objects to generate effects, although rays are still visible even without an object to outline them. When semi-transparent objects are positioned in front of the light, it is possible to generate volumetric rays that use the colour of the object.

TIP Use a GMask link to connect a GMask directly to the Rays node in the schematic. If you then select Use As Occluder in the Post Processing box in the GMask menu, you can create interesting effects by having the GMask occlude only the Rays effect.

Adding a Rays Object to a Light

In Action, a ray is a child of a light. Multiple lights can be parented to the same Rays object.

To add a ray to the scene.

1. Add and position a light to your scene.
2. Do one of the following:
   - Drag the rays node from the node bin and place it in the schematic.
   - Double-click the rays node.

   A Rays object is added to the schematic. In Result view, you can see the ray.

   If there is only one light in the scene, the rays node is automatically connected to it. If there are several lights in the scene, select the light in the schematic that you want parented to the rays node before added the rays node. Otherwise, you can parent the rays node to the light or lights manually in the schematic.

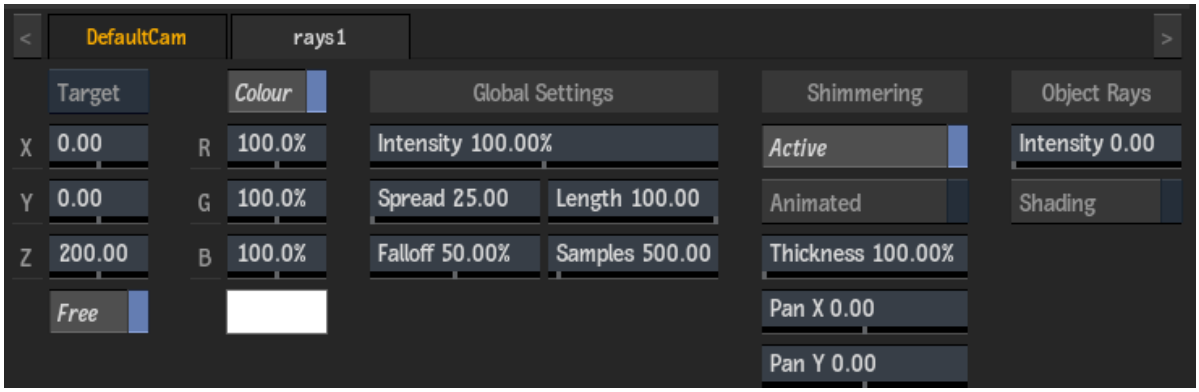
NOTE Since a Rays object is considered a post-processing effect in Action, make sure that your image window is set to Live Preview mode to be able to see the results.

3 To display the Rays menu, double-click the Rays object in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 571).

Similar to the workflow of the Lens Flare, you can add texture components (page 678) to a rays node to enhance the effect.

Rays Menu Settings

The Rays Object menu lets you control settings for the complete rays effect. If you have added component textures to the Rays effect, each component texture also has its own menu that is specific to the component (see Component Menu Settings (page 682)).



Rays Mode box Select whether the Rays pivot point position is relative to the parent Light target position (Target mode), or independent of the parent Light (Pivot mode).

Position X field Displays the screen space position along the X axis of the Rays point-of-interest. Unavailable if Free is disabled. Editable.

Position Y field Displays the screen space position along the Y axis of the Rays point-of-interest. Unavailable if Free is disabled. Editable.

Position Z field Displays the screen space position along the Z axis of the Rays point-of-interest. Editable.

Free button Enable to ignore transformations from the parent light object. When disabled, the ray is affected by the parent light's position, rotation, spread, and falloff settings.

Colour button Enable to add the ray colour to the colour bleed applied to semi-transparent 3D objects placed in front of the attached light.

Red Colour field Displays the amount of red in the effect and in any attached components (based on the colour of the attached light). Editable.

Green Colour field Displays the amount of green in the effect and in any attached components (based on the colour of the attached light). Editable.

Blue Colour field Displays the amount of blue in the effect and in any attached components (based on the colour of the attached light). Editable.

Colour pot Displays the colour of the effect and any attached components (based on the colour of the attached light). Editable.

Intensity field Displays the intensity of the effect and any attached components (multiplied by the intensity of the attached light). Editable.

Spread field Displays the shape of the ray cone. Editable.

Falloff field Displays the amount of smoothness applied to the borders of the ray cone. Editable.

Length field Displays the amount of softness applied to the ray. Editable.

Samples field Displays the quality of the ray based on the radial distance to the attached light. Editable.

Active button Enable to use shimmering settings to modulate the ray with noise particles.

Animated button Enable to automatically apply a noise effect to the shimmer at each frame.

Thickness field Displays the thickness of the shimmer noise particles. Editable.

Pan X field Displays the amount of movement of the shimmer noise particles along the X axis. Editable.

Pan Y field Displays the amount of movement of the shimmer noise particles along the Y axis. Editable.

Object Intensity field Displays the amount of ray colour bleed applied to semi-transparent 3D objects placed in front of the attached light. Editable.

Shading button Enable to allow the ray to inherit the shaded colours of a 3D object as it passes through the object.

Relighting: Blooming

Attach a Blooming node to a light in the scene to help define highlight areas that generate a glowing effect. You can add textures to stamp a blooming node with particular patterns, such as streaks and glints.

Blooming can affect surfaces and geometries in your Action scene, and you can also use lighting links from the attached lights to selectively include or exclude blooming from objects in the scene.

Since Blooming works in Action screen space, the effect is not limited to the surface or geometry in your scene. Note however, that while blooming can occur outside of the Action scene while you are viewing the effect in the image window, in this case you may not see the full blooming effect when you preview or process in Action. You may notice this also if you zoom in or out while viewing in the image window.

TIP Use a GMask link to connect a GMask directly to the Blooming node in the schematic. If you then select Use As Occluder in the Post Processing box in the GMask menu, you can create interesting effects by having the GMask occlude only the Blooming effect.

Adding a Blooming Object to a Light

In Action, a blooming node is a child of a light. Multiple lights can be parented to the same Blooming object.

To add a bloom to the scene.

1. Add and position a light to your scene.
2. Do one of the following:
   - Drag the blooming node from the node bin and place it in the schematic.
   - Double-click the blooming node.

   A Blooming object is added to the schematic. In the Result view, you can see the bloom effect.

   If there is only one light in the scene, the blooming node is automatically connected to it. If there are several lights in the scene, select the light in the schematic that you want parented to the blooming node before adding the blooming node. Otherwise, you can parent the blooming node to the light or lights manually in the schematic.
3. Optional: To add a texture component, select the blooming object in the schematic, then double-click a texture node in the Relighting tab of the node bin.
4. To display the Blooming menu, double-click the Blooming object in the schematic, or follow the tab population rules for the Object menu.

Blooming with Texture Components

Similar to the workflow of the Lens Flare, you can add texture components to a blooming node to enhance the effect. When using blooming in Stamping mode, texture components must be attached to see any result.

The Stamping tab of the Blooming menu has global settings for all attached components, but each component texture has its own menu to control settings particular to it.

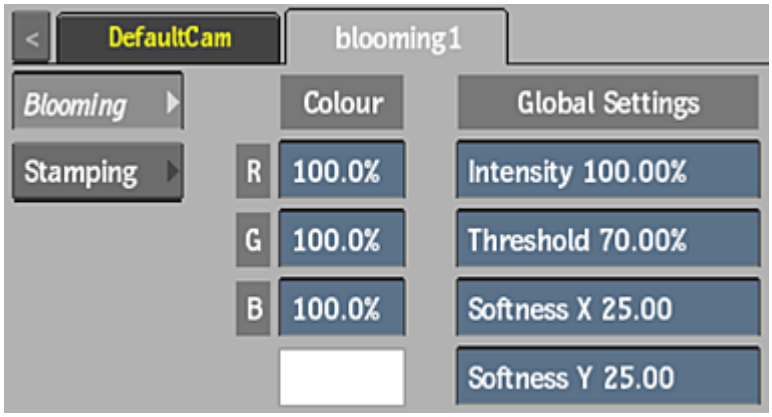
Be aware of the following when working with texture components attached to a blooming node:

- Each type of component can be added multiple times to a blooming object, each with its own settings.
- Some settings in the Basic tab of the component menu do not affect stamping, such as Number, Position, Variance, and Seed. Only one instance of the texture pattern is used for stamping, so these settings have no effect.
- You can use the Border FX settings for each component to decide how the luminance of the objects that the blooming is affecting modulates the Basics settings (Intensity, Scale, Rotation, and Ratio) of the texture pattern. Centre and Spread are not used for blooming.
- You can re-texture a component using a Diffuse Map with your own texture media applied. To do so, select the component in the schematic, then select the media you want to use from the media list, and double-click the Diffuse Map node in the node bin. In this case, the Pattern settings in the component menu are not applicable, though you can still use the settings in the Basics and Border FX tabs of the component menu, as well as the Diffuse menu settings.

Blooming Menu Settings

The Blooming menu is divided into two tabs, each with its own type of blooming effect, which can work independantly, or in combination with each other.

Blooming Tab



Red Colour field Displays the amount of red in the effect and in any attached components (based on the colour of the attached light). Editable.

Green Colour field Displays the amount of green in the effect and in any attached components (based on the colour of the attached light). Editable.

Blue Colour field Displays the amount of blue in the effect and in any attached components (based on the colour of the attached light). Editable.

Rays Colour pot Displays the colour of the effect and any attached components (based on the colour of the attached light). Editable.

NOTE The same colour settings are also found in the Stamping tab.

Intensity field Displays the intensity of the effect and any attached components (multiplied by the intensity of the attached light). Editable.

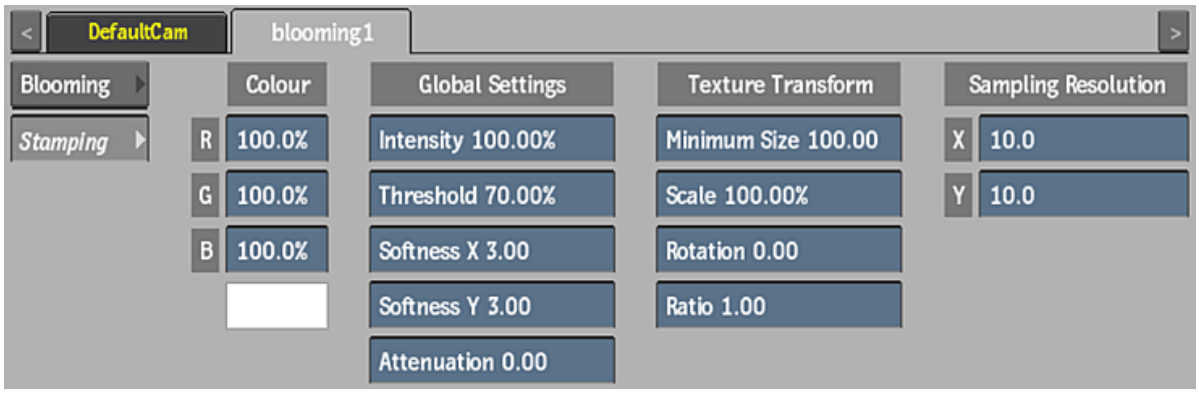
Threshold field Displays the minimum luminance value at which blooming occurs. Editable.

Softness X field Displays the amount of softness along the X axis of the blooming effect. Editable.

Softness Y field Displays the amount of softness along the Y axis of the blooming effect. Editable.

Stamping Tab

For stamping settings to have any effect, you need to attach one or more texture components to the blooming node. In this case, stamping settings are global for all attached components, and each component texture has its own menu that is specific to the texture.



Red Colour field Displays the amount of red in the effect and in any attached components (based on the colour of the attached light). Editable.

Green Colour field Displays the amount of green in the effect and in any attached components (based on the colour of the attached light). Editable.

Blue Colour field Displays the amount of blue in the effect and in any attached components (based on the colour of the attached light). Editable.

Rays Colour pot Displays the colour of the effect and any attached components (based on the colour of the attached light). Editable.

NOTE The same colour settings are also found in the Blooming tab.

Stamping Intensity field Displays the global stamping intensity of all attached texture components. Editable.

Stamping Threshold field Displays the minimum value at which stamping occurs for all attached texture components. Editable.

Stamping Softness X field Displays the softness along the X axis for all attached texture components. Editable.

Stamping Softness Y field Displays the softness along the X axis for all attached texture components. Editable.

Stamping Attenuation field Displays the smoothing level of the blooming effect. Use to fade out regions that have too much blooming. Editable.

Texture Minimum Size field Displays the minimum size of all attached texture components. Editable.

Texture Scale field Displays the size of all attached texture components. Editable.

Texture Rotation field Displays the level of rotation of all attached texture components. Editable.

Texture Ratio field Displays the aspect ratio of all attached texture components. Editable.

Sampling X field Displays the size of the grid along the X axis to affect the number of samples taken to calculate the stamping effect. A higher value yields faster results, but may be less precise. Editable.

Sampling Y field Displays the size of the grid along the Y axis to affect the number of samples taken to calculate the stamping effect. A higher value yields faster results, but may be less precise. Editable.

Relighting: Image-Based Lighting IBL

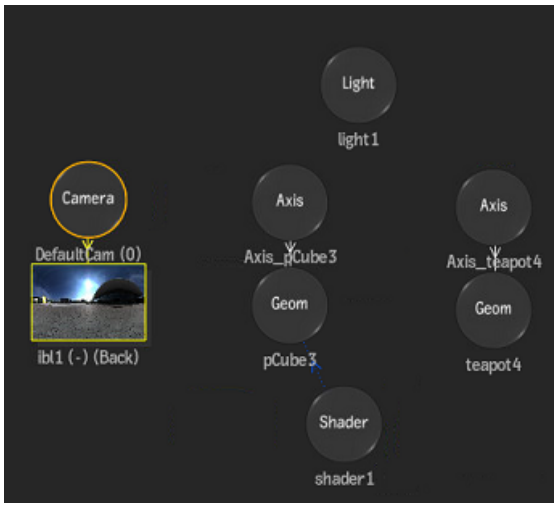
Image-based lighting (IBL) is the process of illuminating scenes and objects (real or synthetic) using images of light from the real world. This is in contrast to using only direct light sources such as point lights or spotlights, which are more localized. A typical use of IBL is taking high-dynamic photos of a chrome ball placed in the original environment of a live-action shoot, then using the photos as an IBL map to simulate the lighting conditions of the shoot. In Action, you can use the IBL node in Reflection mode or Ambient mode (which more closely simulates a global illumination effect).

IBL maps can affect standard shading objects and physically based shading objects at the same time, depending on each object's rendering mode. In physically based mode, the IBL behaviour is controlled by the individual object material parameters, either from the Shader node, or the corresponding PBS Map. When an object is not physically based shaded, then the Standard Shading section of the IBL menu dictates how it affects the object (Ambient or Reflection mode).

Light Probe Images courtesy of Paul Debevec, www.debevec.org

Tips for working with IBL Maps:

- You can colour blend images onto the backplate (the IBL map must be a shot of the environment that the backplate was taken from). This technique is useful for green screen work.
- IBL maps can be useful for lighting 3D geometry or text objects from every direction using the colours in the environment.
- You can use IBL maps to make surfaces perfectly reflective or perfectly diffuse. You can also make an IBL map act as a virtual spherical background plate.
- Action supports angular, cubic, and cylindrical (latlong) IBL mapping types. You can use the Map Convert (page 1595) tool to convert images to one of these types.
- IBL maps work well with PBS shading (page 775) in Action. One IBL map can be used for both PBS and standard shading in the same scene:



The cube has a Physically Based Shader attached, so its settings override the IBL shading settings.

The teapot uses the IBL menu's Standard Shading.

Adding an IBL Map

IBL maps behave differently than other Light FX nodes, such as Lens Flares, since you don't attach it to a light in the scene. The IBL performs lighting based on media as an environment texture.

NOTE On a Mac system, you are limited to one IBL map per Action object.

To add an IBL map to the scene:

1. Select the media to be used as the IBL in the image list. Any type of media can be used, including, but not limited to, *.hdr* images.

   NOTE If no media is selected, you can create an IBL with a default cylindrical image (presets are available at `/opt/Autodesk/presets/<product version>/action/ibl`).

2. Select the node in the schematic that you want to attach the IBL map to.
3. Do one of the following:
   - Drag the IBL node from the node bin and place it in the schematic.
   - Double-click the IBL node.

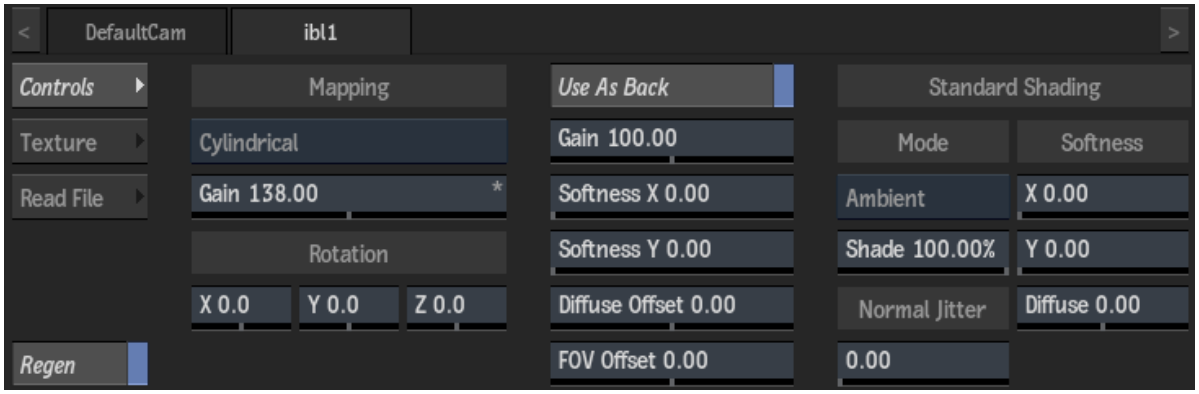
To affect the entire scene, add an IBL map and make it a child of the Result camera (or any camera used for the comp output). To affect a specific object, make the IBL map a child of the object. You can use only one IBL to affect the entire scene, and only one IBL to affect an object.

To specify different media as the IBL source, select the media in the Media menu, then click Apply, or use the Read File tab to manage the IBL media (see Working With Textures in Map Nodes (page 762))

4 To display the IBL menu, double-click the IBL object in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 571).

IBL Menu Settings

The settings in the IBL menu are as follows.



Mapping Settings

Mapping Type box Select the type of texture mapping. Use the Map Convert (page 1595) tool to convert different mapping types (for example from spheric to one of the supported types).

IBL Type:	Description:	Example:
Angular	Usually a high-dynamic picture of a mirrored ball.	

IBL Type:	Description:	Example:
Cubic	A series of images shown as an unfolded cube displaying six sides of the image.	
Cylindrical	An image mapped onto an unfolded cylinder (also known as latitude and longitude, or latlong maps).	

Light Probe Images courtesy of Paul Debevec, *www.debevec.org*

Gain field Displays the overall brightness of the IBL map. Editable.

Regen button Enable to dynamically refresh the image as changes are made to the IBL settings.

Rotation Settings

Rotation X field Displays the level of rotation of the IBL map along the X axis. Editable.

Rotation Y field Displays the level of rotation of the IBL map along the Y axis. Editable.

Rotation Z field Displays the level of rotation of the IBL map along the Z axis. Editable.

Background Settings

Use As Back button Enable to use the selected IBL map as the background in the scene. Background settings appear when enabled.

Background Gain field Displays the overall brightness of the IBL map when used as a background. Editable.

Background Softness X field Displays the amount of X-axis blur applied to the IBL map when used as a background (not available when Cubic is chosen as the mapping type). Editable.

Background Softness Y field Displays the amount of Y-axis blur applied to the IBL map when used as a background (not available when Cubic is chosen as the mapping type). Editable.

Background Diffuse Offset field Displays the offset based on an attached diffuse map or the existing diffuse light in the image when used as a background. Editable.

FOV Offset field Displays the field of view offset applied to an IBL map that is used as a background. Use to simulate a zoom in or out of the background. Editable.

Standard Shading Settings

PBS shading may override these settings, but one IBL map can be used for both PBS and standard shading in the same scene.

IBL Type box Select whether to apply Ambient or Reflection IBL mapping. Extra settings appear when Ambient is selected.

Shade field Displays the amount of lighting used. Editable.

Jitter field Displays the sampling area used to calculate the normal information for ambient lighting effects. Available when Ambient is selected as the IBL type. Editable.

Softness X field Displays the amount of X-axis blur applied to the IBL map (not available when Cubic is chosen as the mapping type). Editable.

Softness Y field Displays the amount of Y-axis blur applied to the IBL map (not available when Cubic is chosen as the mapping type). Editable.

Diffuse Offset field Displays the offset based on an attached diffuse map or the existing diffuse light in the image. Editable.

Texture Tab Crop And Resize Settings

Fit Method box Select a fit method option to be applied to the IBL map.

Keep Aspect button Enable to preserve the aspect ratio of non-square pixels (not available for the Fill fit method).

Use Cropped Size button Enable to replace the IBL map with the cropped size of the IBL media. Disable to use the cropped IBL media as is.

Texture Tab Filtering Settings

Filter box Select the type of filtering to apply to the IBL map.

About Lightbox

Unlike Matchbox GLSL shaders, Lightbox shaders are available exclusively in Action, relying on the Light framework, to which a Lightbox node is parented in the Action scene. Lightbox shaders allow you to create and use a wide range of effects that affect one fragment at a time, allowing a casting of colour effects through the light cone, respecting every aspect of Light behaviour, including shading, decay, and feathering. Lightbox nodes can be useful in the lighting pipeline to develop a look at the scene level of Action, or to help with blue screen or green screen work.

Lightbox effects are based on the OpenGL Shading Language (glsl) programming language and can be created by users to use in Action, or to share with other Flame users (a number of preset and example shaders are also available). For this reason, a Lightbox API is available at *http://www.autodesk.com/shader_builder_api_guide*, that allows you to access Action scene information (such as distance, normals, camera position, and other 3D environment variables).

To add a Lightbox node in Action:

1. Do one of the following:
   - With a Light node selected in the schematic, double-click the Lightbox node in the All Nodes or Relighting bin.

The file browser opens to allow you to select a Lightbox effect. When selecting a shader from the file browser, you can use the File Format box to display shaders in .glsl format or encrypted Lightbox .lx format.

- With a Light selected in the schematic, double-click a Lightbox preset node from the dynamic Lightbox node bin (page 550).

  The Lightbox node is added to the scene as a child of the selected Light. If no light was selected prior to adding the Lightbox node, a new Directional Light is added to the scene as the parent of the Lightbox node.

TIP Lightbox nodes are displayed as white in the schematic to help you identify them. If you added the Lightbox from the dynamic node bin, the name of the effect also appears on the node in the schematic.

2. Double-click the Lightbox node in the schematic to display its menu.

   The Lightbox node populates the menu dynamically, based on the .glsl (and .xml) code, but there are a few UI elements that are constant, located in the Shader tab.

3. If you added a Lightbox effect from the All Nodes or Relighting bin, you can change the effect by clicking Change Shader in the Shader tab of the Lightbox menu.

Lightbox Usage Tips

Here are a few highlights and tips to help you when working with Lightbox in Action:

- Multiple Lightbox nodes can be parented under the same Light, and the order of processing can be controlled in the Light Priority Editor (page 565).

  Since multiple Lightbox nodes parented under the same Light node respect a priority order, you can create a pipeline of Lightbox effects. For example, you can use a Selective 3D Lightbox shader to create a selective, then apply a second Lightbox shader, such as Colour Correct to affect only the selected areas.

- A GMask (or multiple GMasks) connected to the parent Light with a selective GMask Link (page 901) can help you create interesting effects, where the Lightbox is cast through the GMask shape. You can track the GMask, and combine GMask Links with Look At Links and Replica nodes to easily build a composite, such as simulating sunlight shining through windows.

- Lighting links (page 664) can be useful to connected Lightbox nodes in creating inclusive or exclusive constraints.

- Use the Action Object Solo (`F8`) view on the parent light to see the scene through the point of view of the light, as if it was a camera. The `F8` view also allows you to interact directly in the image window with icons, if they are present in the shader.

- If the loaded Lightbox shader has any preset starting points built into it (such as in the Duotone shader), you can find them in the Presets list in the Shader tab of the Lightbox menu.
- A folder is available in the channel editor for each Lightbox effect in your Action scene.
- Apart from the Lightbox menu with its dynamic settings generated from the .glsl code, there are a number of settings in the Light menu (page 668) that can help you set up your Lightbox effect (mostly in the Rendering section of the Basics tab, and in the Lightbox tab of the Light menu). By default, when a new Lightbox node is added to the scene with a new Light, the light is inactive, and global shading and Scene Ambient settings are set to active.

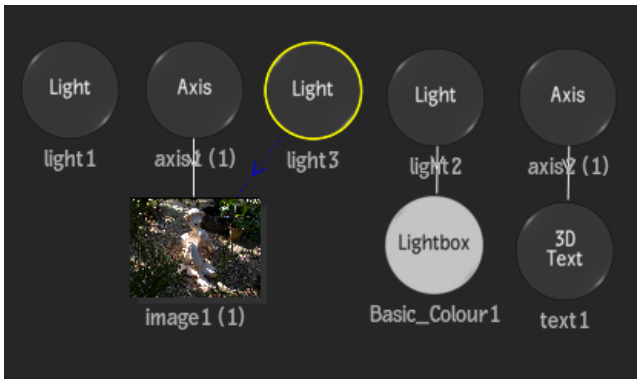
  NOTE When the parent Light is inactive, certain Light settings are unavailable, such as colour and intensity.

Lightbox Examples

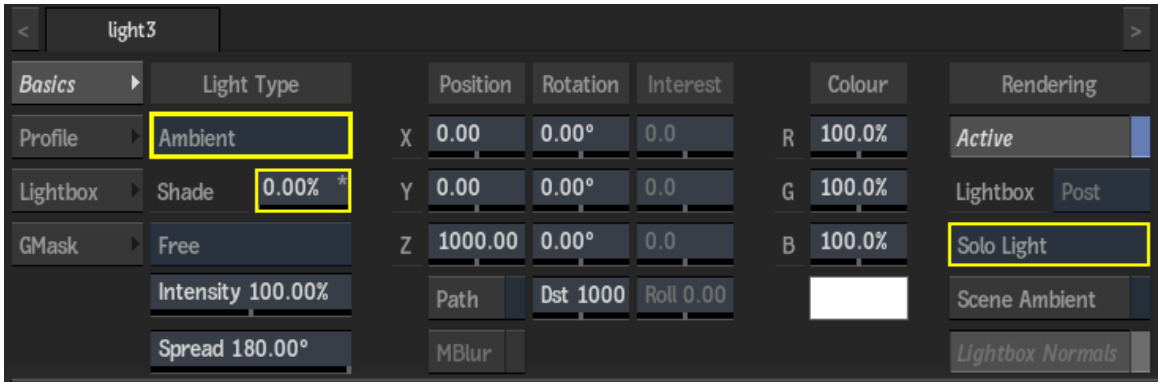
Use the following examples of Lightbox shaders to get a feel for the workflow of using different types of Lightbox effects in Action.

Combining Multiple Lights with Lightbox

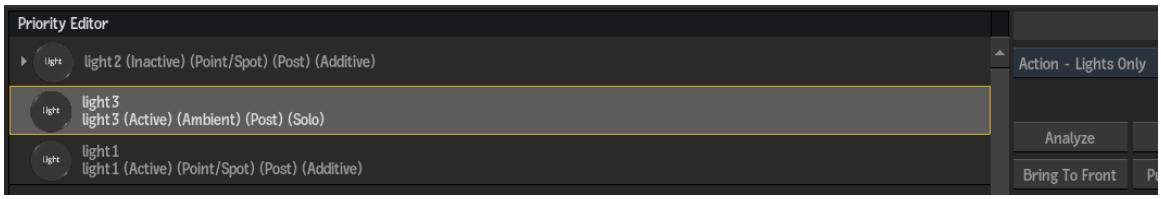
A common scenario that you may encounter when working with Lightbox is how to combine images that you don't want to shade with a geometry that you do want to shade. You can use selective lighting links and the Priority Editor to solve this issue.



- Light 1 is a global light, set to Additive Light in the Light Source box.
- Light 2 is the parent of the Lightbox node, and is not set to Active.
- Light 3 is connected to the image node with a Light Inclusion link; the Light Type is Ambient, with the Shade set to 0%, and the Light source box set to Solo Light.



- These settings, when combined with Light 3 appearing in the Priority Editor after Light 1, produces the desired effect.



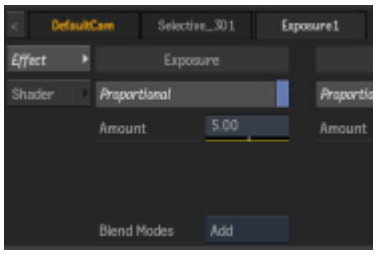
Lightbox Selectives

The goal when using the Selective 3D or Selective Noise 3D Lightbox shader is to create an alpha to use in the next Lightbox node in the pipeline.

Some tips for working with Lightbox Selective nodes:

- Add the Selective node as the first Lightbox to affect other Lightbox nodes that you can add later. It should appear at the bottom of the Priority list for the selected Light.
- Enable Selective Viewing in the Selective menu to display the selected area in magenta. Don't forget to disable Selective Viewing when you are done setting the selection.
- Choose a type of selective to create, such as 3D Sphere or 3D Cube for a Selective 3D node, or a noise type for a Selective Noise 3D node, and adjust other settings, as necessary. Enable Invert, if needed.

- Once the Selective is created, disable Selective Viewing, and add other Lightbox nodes under the same Light. The effect of this new Lightbox node only occurs in the Selective area that you created in the Selective node:

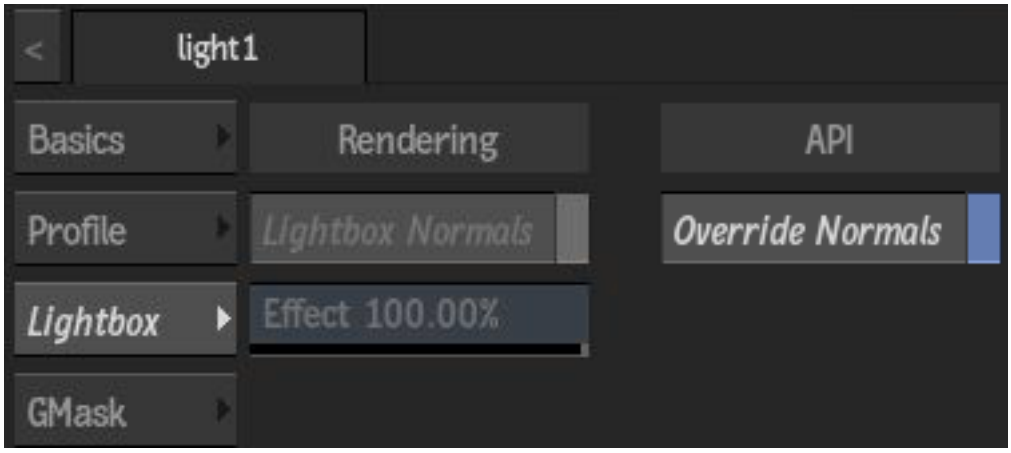


- You can have more than one Selective node parented under a Light. The Selective areas created by all the Selective nodes are combined to create one alpha selection.

Environmental Lightbox

You can use the Clouds and Fog Lightbox presets to simulate these environmental conditions in your Action scene.

For these effects, you do not want to use the normals attenuation from the parent Light, so make sure that the Override Normals button is enabled in the Lightbox tab of the Light menu. The state of this button can be coded directly in the shader (it is enabled by default for the Clouds and Fog presets).



Action 3D Geometry

A powerful feature of Action is its ability to import 3D models created in other applications and combine them with existing clips. Compositing 3D models and characters with other media or a background can be done quickly and with a finite level of control.

3D geometric objects are manipulated like an image or a clip: you can animate their position and shape, apply textures and media, and light objects to produce a variety of effects.

You can import 3D polygon objects such as 3ds Max files, FBX files, Alembic files, Wavefront files, Inventor files, SVG files, garbage masks, and Paint geometry. 3ds Max files contain object data, specifically, texture and materials. The FBX format acts as the intermediary between different file types. Files can be exported from another product to the FBX or ABC format and then imported into Flame.

Scalable Vector Graphics (*.svg*) files are open-standard 2D graphics. Once imported, an *.svg* file appears in the Action schematic as a Group node; once ungrouped, the graphic is converted to a number of geometry elements: 3D Shape (page 752) nodes (with accompanying GMasks and Axes).

NOTE To preserve colour information in *.svg* files from Adobe Illustrator, make sure that Presentation Attributes is enabled when exporting from Illustrator. Other vector graphics programs may have similar settings.

Paint geometry files are created by the Flame Paint tool. If you want to work with polygon geometry in Action, import Paint geometry. Action ignores its animation and attributes, such as its colour, outline, and gradient. For example, in Paint, if you create a blue polygon, animate its scale, and save it as geometry, it is imported in Action as a white polygon with no animation.

NOTE Sample 3ds Max and FBX model files are located in the */opt/Autodesk/presets/<product_home>/models/*(FBX or 3DS) libraries. All models are textured with an identical image. The library consists mostly of geometric primitives such as cubes, cylinders, and spheres.

Importing 3D Models

You can import a 3D model into one or several geometry nodes. You can also import multiple 3D models into a single animated geometry node.

To import a 3D model:

1. Do one of the following:
   - Right-click the schematic and select Import.

- Drag the Import node from the node bin and place it in the schematic.
- Drag the Import node from the node bin and place it where you want it in Result view.
- Double-click the Import node. You do not need to be in Schematic view to add a node in this manner.

The Import menu and file browser appear.

NOTE When selecting Paint, GMask, or Photoshop, the subsequent settings described in this section do not apply.

2 From the Import Type box, select the import format.

If you select FBX or Alembic, you can then choose an Import Type:

- **Action Objects:** to create local copies of the FBX or Alembic files, without any connection to the source file (similar to a hard import). This option can impact auto-save time and space in the file system for large geometries.
- **Read File:** to create an FBX or Alembic Scene node, similar to a soft-imported Group node, with added functionality. This option maintains a live link to the source file, insuring a constant update with file changes, as well as having no impact on auto-save time and file system disk space.

NOTE Make sure that you have write permissions to the folder where the FBX file is located, as Flame decompresses the textures embedded in the FBX to an *.fbm* folder co-located with the FBX file.

3 If the file you want to import has a file extension different from the one specified, type a file extension.

4 Set any Import Settings (page 701), as needed.

5 Select the file to import from the file browser. You can set the default Action geometry import path (or use Python hooks to configure customized Setup paths) in the Preferences (page 1944) menu.

6 Click Load.

The 3D model (Geom node) and axis is added to the scene. SVG files appear as Action 3D Shape (page 752) nodes. For FBX and Alembic geometries, depending on your selection in the Import Type box, you may see a Scene Node, or all of the component nodes of the geometry.

You can change the 3D model's colour, specular highlight, shine, and other material properties. See 3D Geometry Menu Settings (page 711).

NOTE If an imported FBX or 3DS file contains one or more textures, a Batch reel is created to house these textures. It is closed by default.

To import multiple 3D models into an animated sequence:

1 Follow the same steps for adding a single 3D model, but from the Import file browser, select multiple 3D models by holding the `Shift` or `Ctrl` key.

2 Once the models are selected, click Load.

The selected 3D models are loaded to the same line in the Media list.

3 Double-click the Geometry node in the schematic to access the Geometry menu, then click the Timing tab.

Importing Photoshop Files into Action

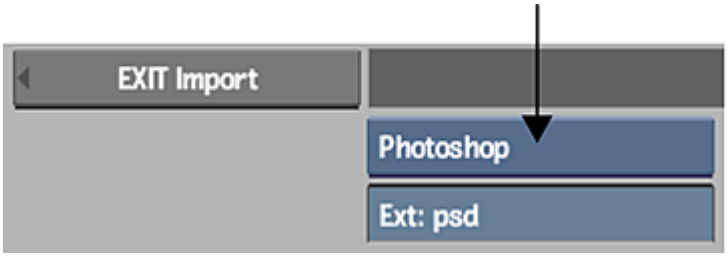
You can also import Adobe Photoshop® format files into Action without having to convert them into a TIFF or other format. Once imported, the PSD file keeps its inherent layer structure, which can be used or modified within Flame.

NOTE This functionality is not available if you accessed Action from the timeline.

You also have the option of automatically importing all the layers at their native resolution. Each layer is parented by an axis that gives it the correct offset in the X and Y axes. Hidden layers are imported, but will remain hidden. Photoshop blend modes are maintained for each layer.

To import a PSD file into Action:

1. From the Action Node bin, double-click the Import node.
   The file browser appears.
2. From the Import Type box, select the Photoshop format.



3. Select a *.psd* format file.
4. Click Load.
   The Photoshop file is loaded into Action. Each layer is parented by an axis, and blend modes are maintained. A Batch reel is created to house the images imported in the layers. It is closed by default.

Import Settings

Geometry Import Settings

Depending on the type of import, some of the options differ. See below for specific FBX and Alembic import settings.

Import Type box Select the 3D model type to import.

File Extension field Displays the default extension for the file type selected in the Import Type box.

Smooth button Enable to build normals for the 3D model. Enable if you are importing polygons that do not have normals.

Auto Fit In Scene button Enable to scale the imported model to fit into the current frame. When disabled, the imported model maintains the same size in which it was created.

SVG DPI field Displays the desired dots per inch for coordinates in the imported SVG file. Used if the SVG file uses real-world units (for example inch, cm, or mm); has no effect if the SVG is in pixel units. Editable.

Separate Nodes button Enable to create individual nodes for all 3D models contained in a file. When disabled, the 3D model is added to the scene with its own axis.

Rotate Axis button Enable to rotate the 3D model by 90 degrees on the X-axis so that it is compatible with the target's coordinate system.

Create Media button Enable to load the textures of the geometry to the Media list. If a texture is used in multiple geometry maps, it is loaded only once in the Media list. Not available if Action is accessed as a Timeline FX.

FBX and Alembic Import Settings

Import Type box Select an import type for the FBX or Alembic model. Select Read File to create an FBX or Alembic Scene node, similar to a soft-imported Group node. Select Action Objects to create local copies of the FBX or Alembic files (similar to a hard import).

Auto Fit In Scene button Enable to scale the imported model to fit into the current frame. When disabled, the imported model maintains the same size in which it was created, or you can use the Scene Units to Pixels field to manually set a scaling factor.

Scene Units to Pixels field Displays the scaling factor used on the imported FBX file to fit the scaling used in the application. One unit in the FBX file (default is cm) is converted to the number of pixels (default is 10) that you specify.

Models button Enable to import the geometry from the file.

Lights button Enable to import lights from the file.

Cameras button Enable to import cameras from the file. Free cameras are imported as target cameras (with a point of interest).

Mesh Animations button Enable to import the scene animations and preserve complex geometry animations , such as nCloth animations.

Normals button Enable to import the model's normal information.

FBX Specific Options

Keep FBX Frame Rate button Enable to use the frame rate of the FBX file as the frame rate in Action. It might be useful to enable Keep FBX Frame Rate when importing an FBX file before starting to build your animation, otherwise the timing of your animation may be affected.

Bake FBX Animation button Enable to add a keyframe at every frame of the imported FBX file. When disabled, Action translates the FBX keyframes to a comparable animation curve for comparable channels.

Import As Action Object button Enable to import Action objects embedded in an FBX file, that had been previously exported from Action. For example, an exported FBX with extended bicubic data remembers this upon re-import, and creates the Extended Bicubic surface, in addition to the Geometry.

Create Media button Enable to load the textures of the geometry to the Media list. If a texture is used in multiple geometry maps, it is loaded only once in the Media list. Not available if Action is accessed as a Timeline FX.

Alembic Specific Options

Frame Rate box Select the frame rate of the imported Alembic file.

Colour Management Settings

When importing 3D models in FBX or 3DStudio formats, and Photoshop PSD files, there are colour management settings that appear. See Colour Management in Action (page 584) for more information.

Timing Settings

If the 3D model has animation, you can find settings in the Timing tab of the Geometry menu.

Animation Mode box Select the animation mode to use for multiple 3D geometries imported into a single animated geometry node.

Select:	To:
Loop	Play in a continuous loop.
Once	Play once. The 3D geometry is no longer displayed.
Last Still	Play once, and hold the last frame.
Timing	Animate according to the timing in the animation channel.

Hold field Displays the number of continuous frames for Loop, Once, or Last Still Animation modes. Editable.

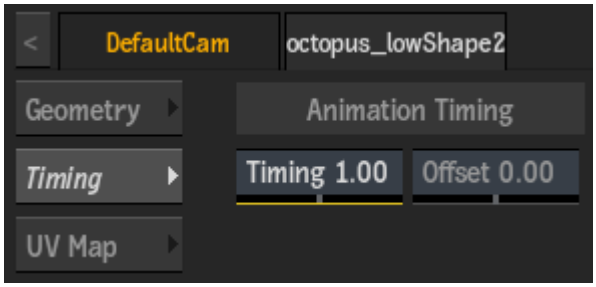
Slip field Displays the offset to the start point for Loop, Once, or Last Still Animation modes. Editable.

Timing Range option box Select an option to determine how Frame Timing values outside the timing range of the animation are handled. Available when Timing is selected in the Animation Mode box.

Select:	To:
Roll	Roll over the Frame Timing value.
Cut	(Geometry is not displayed.)
Round	Display the first or last geometry (with this option, you can select the first and the last geometry of your animation).

Frame Timing field Displays the value for the frame in the timing curve. Available when Timing is selected in the Animation Mode box. Editable.

For Alembic models or FBX models with cache animation, these Timing settings are available:



Timing field Displays the scale of the geometry animation. Editable.

Offset field Displays the offset applied to the animation curve. Editable.

Working with FBX and Alembic Scene Nodes

While you can import FBX and Alembic geometries as Action objects, this may lead to Action interactivity or disk space issues. You can instead use the Read File option to import FBX and Alembic geometries, giving you the benefit of soft-imported files, while still maintaining the option of caching the files, if needed (for archiving, for example).

When importing an FBX or Alembic file as an Action read file node, an FBX or Alembic Scene node is created in the schematic.

Scene nodes are similar to Action Group nodes (page 581), in that the Scene node is a collapsed collection of nodes in the schematic. The result is a cleaner schematic, but you can also quickly expand the contents of the Scene node, or even parent other nodes to or from the Scene node.

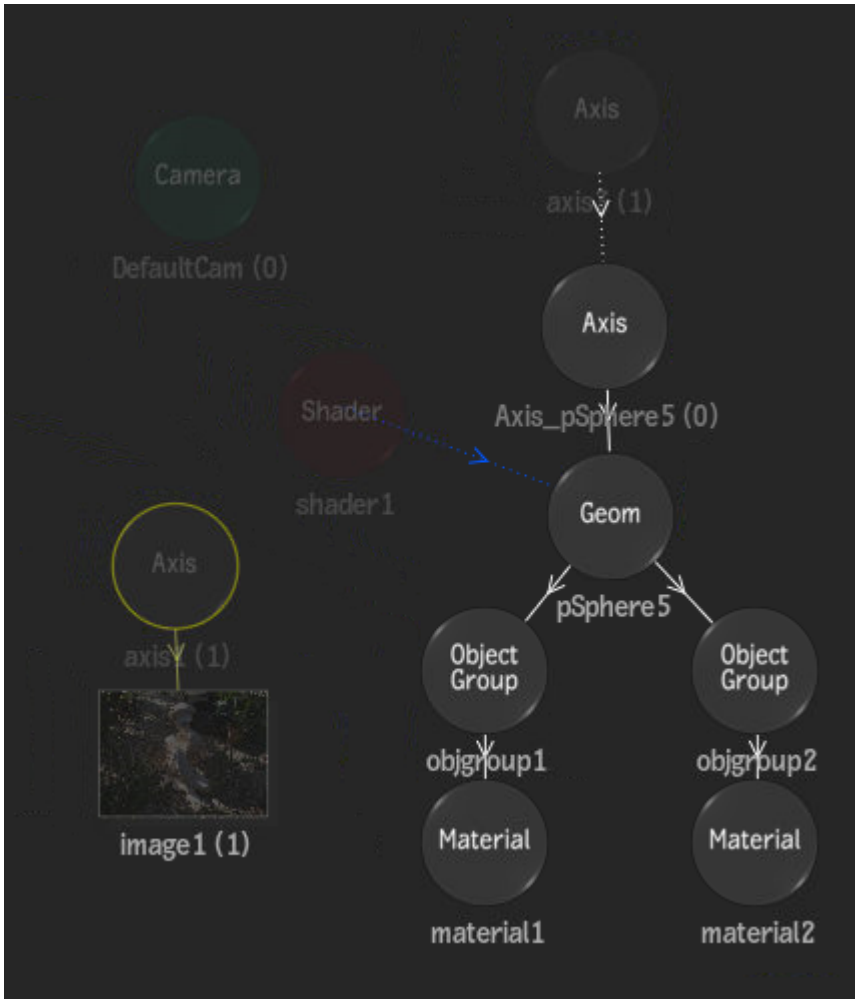
Expanding and Working With Scene Nodes

You can easily see and interact with individual nodes within a Scene node, using the Group Schematic view.

You can access the Group Schematic view using one of these three methods:

- Right-click on a Scene node and select Edit Group from the contextual menu.
- Double-click a Scene node.
- With the Scene node selected, press `F8` or select **Action Object ➤ Group Schematic** from the View box.

The Group Schematic view displays the contents of the Scene node framed in the image window, with all other nodes dimmed.



You cannot edit nodes that are part of the Scene node, since they are soft-imported and live outside of Action (you can see their menu settings, but they are dimmed). You can however, hide nodes that are part of a Scene node Group Schematic.

Once in Group Schematic view, you can move any node (even the dimmed nodes). You can add and connect nodes to the Group Schematic, but they remain dimmed in the Group Schematic view. You can also add nodes into the regular Action schematic, and see them "live" as dimmed nodes in the Group Schematic.

TIP Use a 2-up view with the regular Action schematic in one view and the Group Schematic in the other.

You can perform certain connections to a Group node which are automatically done on all compatible nodes found inside of it. Supported connection are:

- Parent Axis (including Look At and Mimic links): Connects to all top Axis.
- Lighting link: Connects to all Geometry objects.

- GMask link: Connects to all Geometry objects.
- Child Material: Connects to all Geometry objects.
- Child Texture map: Connects to all Geometry objects.
- Child Substance: On initial creation, does a lighting link and a Material connection to all Geometry objects.

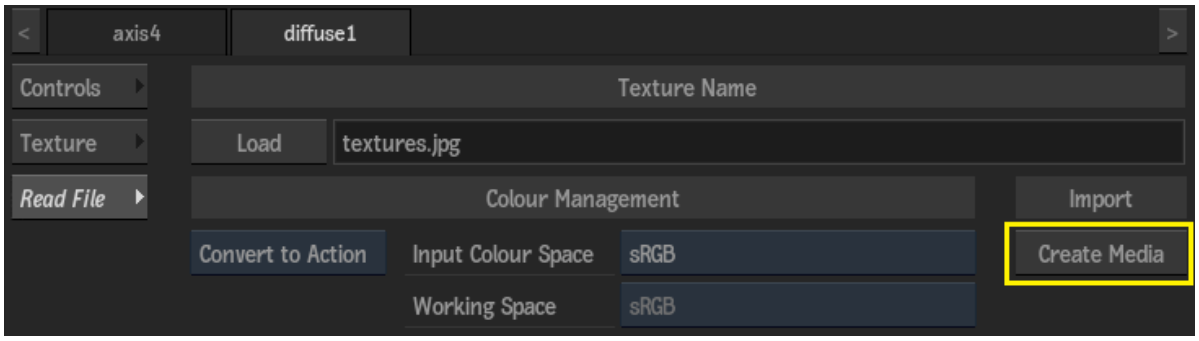
NOTE These connections are made based on the logic of the hierarchy of the group structure. In some cases, you may need to manually create connections or break connections.

If the linked FBX or Alembic file is updated, click Refresh in the Scene menu to update the Scene node in Action.

NOTE If the linked file is moved or changes its file name, the File Path field turns red to indicate that the original file cannot be found, and the geometry in your scene is replaced with a "Missing" text. In this case, click the File Path field to open the browser and locate the moved or renamed file.

Converting a Scene Node to Action Objects

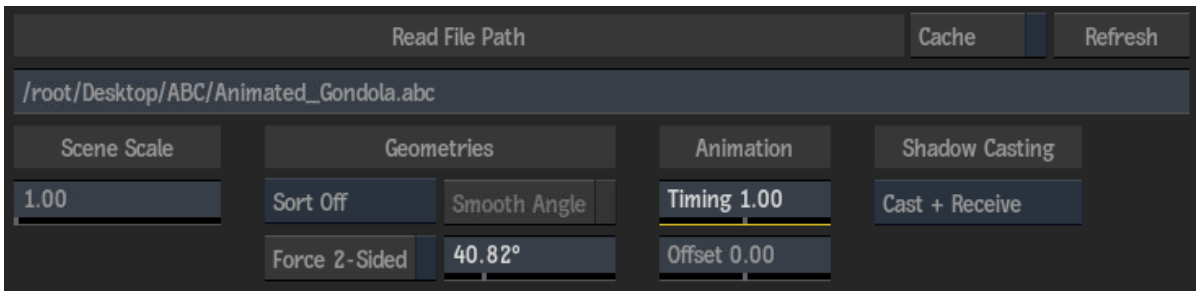
You can copy nodes (or the complete structure) from within the Group Schematic and paste them outside of the Group. In this case, the nodes become hard-imported (that is, local Action managed objects). If the copied nodes include a texture, you can also create the media for the texture in Action. From the Texture menu's Read File tab, click Create Media:



The Texture Name is removed from this menu, and the texture appears in the Action Media list instead, assigned as a proper layer to the texture in the schematic.

NOTE You can also enable Cache in the Scene menu to perform the conversion to hard-import on the complete model. Do this at the end of the project if you want to include the FBX or Alembic scene as part of an archive, for example.

FBX and Alembic Scene Settings



File Path field Displays the path to the file. This field is displayed in red if the linked file can no longer be found. Click to open the browser to locate a moved file, or to replace the currently displayed scene with another one.

Cache button Click to copy all of the scene files locally to Action, in order to perform an archive, for example.

Refresh button Click to update the Scene node with the content of the linked file. Use this if the file has been updated since its import.

Scene Scale field Displays the factor by which to scale the scene. Does not scale the camera if it has already been extracted using the Extract Camera button. Editable.

Sort box Select how the 3D models contained in the scene are drawn.

Select:	To:
Sort Off	Not apply any sorting.
Sort All	Sort all objects according to the Z-order and the order of the object group (Material ID).
Sort Transparency	Give transparent objects lower priority than opaque parts, to solve transparency issues if the scene contains semi-transparent 3D models.

Force 2-Sided button Enable to have lights in the scene light both the inside and outside of the geometry (when shading is turned on).

Smooth Angle button Enable to override existing normals in the geometry, then use the Smooth Angle field to change the value.

Smooth Angle field Displays the angle at which the edges of normals become hard. Changes to this field only affect the shading of the displacement, and not the shape. Editable.

Timing field Displays the scale of the geometry animation. Editable.

Offset field Displays the offset applied to the animation curve. Editable.

Shadow Casting box Select how the selected geometry object is affected by a Shadow Cast object in the scene. See Surface and Geometry Shadow Casters (page 672).

About 3D Data in the FBX Format

The FBX standard format provides a means for exchanging 3D data—3D polygonal models, cameras, lights, keyframe animation—for scene compositions between tools and packages developed by different manufacturers.

FBX support makes it possible to import Autodesk 3ds Max, Autodesk Maya, Autodesk MotionBuilder, Autodesk Mudbox, and Autodesk Softimage files. You can import models, scene compositions, lights, and camera data.

NOTE Flame supports the import of most models, including polygonal, nurbs, and smooth bind skinning, while subdiv primitives are not supported. See the FBX *compatibility maps*.

About Mesh Animation

Mesh animation (geometry caching) records the position of every vertex in an animated 3D scene over time. This allows character animations, animated geometry deformations, or physical simulations to be "baked"

and transported in the FBX file without an understanding of the animation technique used to create the effect. By exporting FBX scenes with geometry caching, and then importing into Action with Mesh Animations enabled, you can preserve complex geometry animations, such as nCloth animations.

All imported cached geometry can still be manipulated in Action like a normal geometry. Deform meshes, texture maps, lighting, and axis manipulations are all supported on the animated geometry. Animations can also be re-timed within Action.

FBX Information Preserved at Import

Flame supports the latest FBX SDK, in sync with Autodesk 3D applications. To ensure compatibility, all applications must use the same version of the FBX SDK.

The following FBX features are preserved in Action:

- Point lights, spotlights, area lights, ambient lights, and directional lights
- Shadow casting
- Object ID and material assignments
- Sub-materials, exposed explicitly in the Action schematic
- Bezier animation curves

Object Group and Material Nodes

FBX models are created with one or more faces, which leads to different representations in Action.

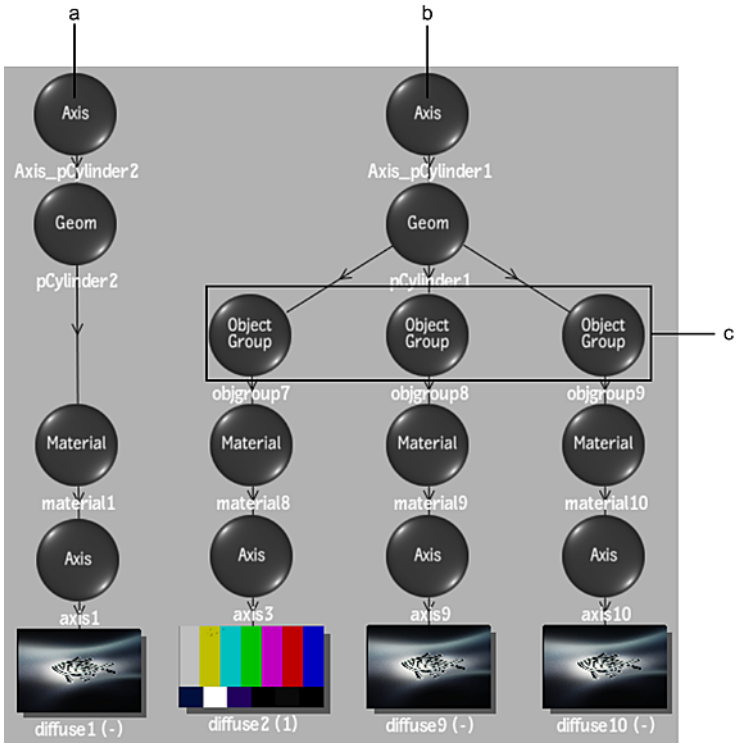
NOTE In this context, *face* actually means either a single face (such as one of six sides of a cube) or faces (such as a group consisting of 2 sides of a cube) that are grouped together to receive a single texture. How faces are grouped (if grouped at all) is decided when the model is created and cannot be modified in Flame.

A model with a single face appears as a simple model, with a single Material node attached to the Geom node.

A model with multiple faces appears as a complex schematic where one Object Group node is created for each face. To each Object Group node is then attached the Material node. Having multiple Object Nodes allows you to modify, even replace, the textures applied to each shader.

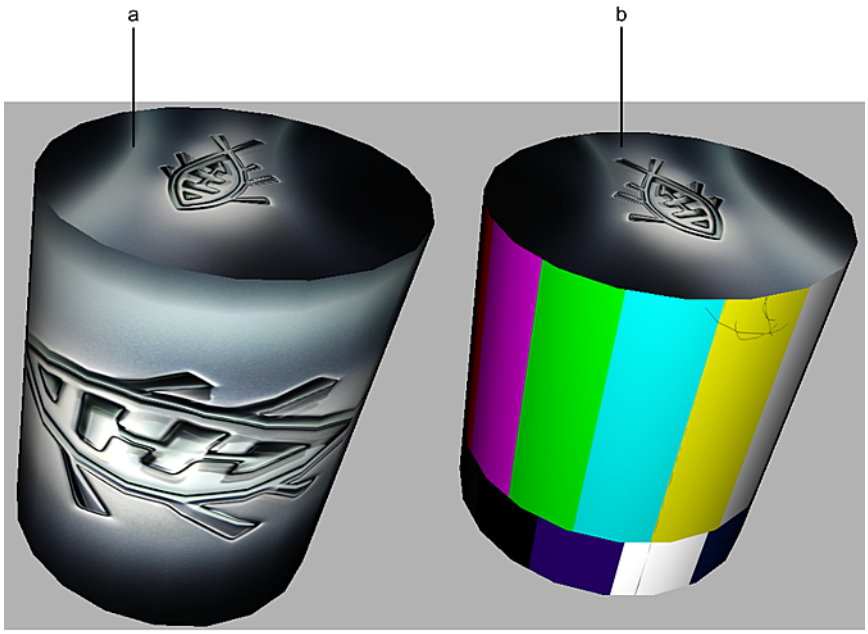
You cannot create new Object Group nodes, and their link to the Geom node is unbreakable; they have no editable attributes. Object Group nodes details how faces were applied to the FBX model, and allow you to texture differently the components of the FBX model.

In the following examples, one imported FBX model has a single face, while the other has three faces, one for each sides of the cylinder.



(a) FBX model with a single face (b) FBX model with multiple faces (c) Added Object Group nodes, one for each face

The schematic above creates the image below. Note how having different Object Group nodes allows you to link different textures.



(a) FBX model with a single face: one texture map for the whole model (b) FBX model with multiple faces: a different texture for each face

Exporting FBX Geometries

If you are prototyping a 3D scene layout in Flame, or you want to send a particular model to another 3D program, such as Maya, you can export the FBX, make changes, then re-import it into Flame.

Supported objects available for FBX export are:

- Surfaces
- Geometries
- Lights
- Cameras
- Point Cache

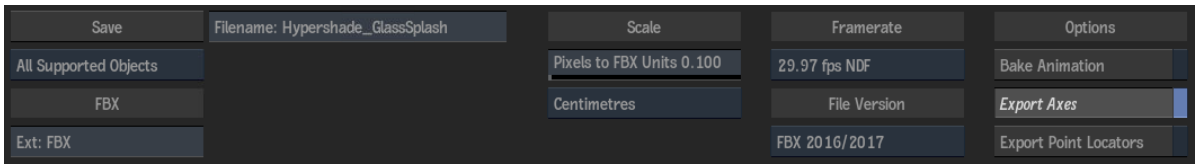
NOTE Textures and Extended Bicubic surfaces are only supported at the current frame (that is, animations are not supported).

To export an FBX geometry:

1. In the schematic, select the FBX geometry that you want to export (or no selection if you are planning on exporting the whole scene).
2. From the right-click contextual menu in the schematic, select Export.
3. In the Export Geom menu, select whether to export Selected Objects or All Supported Objects.
4. Set other export options, as needed (see below).
5. Set the filename and directory path for the export.
6. Click Save.

TIP When re-importing the FBX geometry into Action, enable Import As Action Object in FBX Options to import Action objects embedded in the FBX file, that had been previously exported from Action. For example, an exported FBX with extended bicubic data remembers this upon re-import, and creates the Extended Bicubic surface, in addition to the Geometry.

FBX Export Settings



FBX Mode box Select whether to export all supported FBX objects, or only the selected FBX objects.

File Extension field Displays the default extension for the file type selected (in this case, FBX).

Filename field Displays the file name for the exported FBX model. Editable.

Pixels to FBX Units field Displays the scaling factor to apply to the exported FBX file to be used in the 3D application. Use the Units box to select the unit of measurement. Editable.

Units box Select a unit of measurement to apply to the exported FBX file.

Framerate box Select a framerate for the exported FBX file.

FBX File Version box Select which FBX version to export the file to.

Bake Animation button Enable to add a keyframe at every frame of the exported FBX camera file.

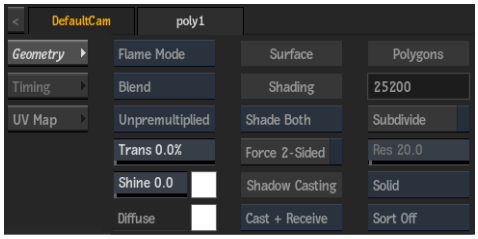
Export Axes button Enable to export the animated axes present in the Action scene.

Export Point Locators button Enable to export the 3D point locators created by the Analyzer.

NOTE You can also export Cameras and 3D Cameras to the FBX format, from their respective menus.

3D Geometry Menu Settings

You can change and animate parameters such as the colour, specular highlight, shine, and transparency of 3D models. You set these parameters using the Geometry menu. To access this menu, double-click a Geometry node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).



Blending Settings

Blend Type box Select whether to use Flame or Photoshop blend modes.

Blend Mode box Select how the 3D model and the scene are combined. The available list of modes depends on the selection in the Blend Type box. See Surface Blending Modes (page 634).

Transparency field Displays the transparency level of the 3D model. Editable.

Shine field Displays the intensity value of the specular highlight. When this value is zero, the specular highlight is disabled. Shine affects both size and intensity. Editable.

See Creating a Specular Highlight on a Model (page 717).

Specular colour pot Displays the colour of light reflected by the 3D model's surface. To enable the specular highlight, the Shine value must be larger than 0. Editable.

Diffuse colour pot Displays the diffuse colour. Editable.

Surface Settings

Shade Order box Select the drawing priority of the 3D model normals.

Select:	To draw the polygons:
Shade Both	That are both facing and opposite the camera.
Shade Front	Facing the camera last.
Shade Back	Opposite the camera last. This option is especially useful for semi-transparent models

Force 2-Sided button Enable to have lights in the scene light both the inside and outside of the geometry (when shading is turned on).

Shadow Casting box Select how the selected geometry object will be affected by a Shadow Cast object in the scene. See Surface and Geometry Shadow Casters (page 672).

Polygons Settings

Polygons field Displays the number of polygons in the 3D model. Non-editable.

Subdivide button Enable to create high-quality shading for polygon models.

Resolution field Displays the geometry resolution of the 3D model. Active when Subdivide is enabled. Editable.

Wireframe box Select a wireframe option for the 3D model. When you render the 3D model with Wireframe or Original Wire selected, it retains its light, shading, and texture attributes.

Select:	To:
Solid	Disable wireframe for the 3D model (filled polygons are drawn).
Wireframe	Display the model as a wireframe outline (triangular polygons only).
Original Wire	Display the model as a wireframe outline (original mesh; any polygon type). May be useful for imported geometries.

Sort box Select how the 3D models contained in the scene are drawn.

Select:	To:
Sort Off	Not apply any sorting.
Sort All	Sort all objects according to the Z-order and the order of the object group (Material ID).
Sort Transparency	Give transparent objects lower priority than opaque parts, to solve transparency issues if the scene contains semi-transparent 3D models.

Timing Settings

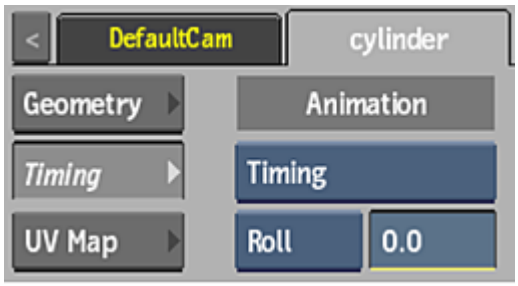
If the 3D model has animation, you can find settings in the Timing tab of the Geometry menu.

Animation Mode box Select the animation mode to use for multiple 3D geometries imported into a single animated geometry node.

Select:	To:
Loop	Play in a continuous loop.
Once	Play once. The 3D geometry is no longer displayed.
Last Still	Play once, and hold the last frame.
Timing	Animate according to the timing in the animation channel.

Hold field Displays the number of continuous frames for Loop, Once, or Last Still Animation modes. Editable.

Slip field Displays the offset to the start point for Loop, Once, or Last Still Animation modes. Editable.

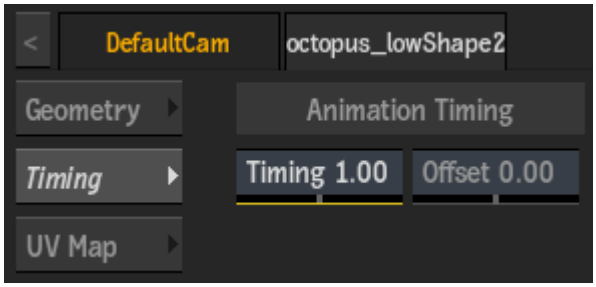


Timing Range option box Select an option to determine how Frame Timing values outside the timing range of the animation are handled. Available when Timing is selected in the Animation Mode box.

Select:	To:
Roll	Roll over the Frame Timing value.
Cut	(Geometry is not displayed.)
Round	Display the first or last geometry (with this option, you can select the first and the last geometry of your animation).

Frame Timing field Displays the value for the frame in the timing curve. Available when Timing is selected in the Animation Mode box. Editable.

For Alembic models or FBX models with cache animation, these Timing settings are available:

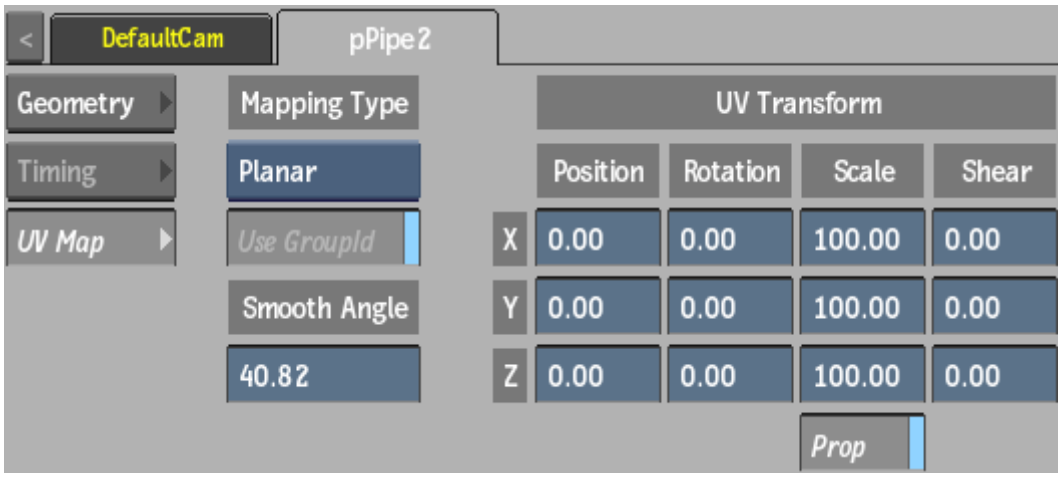


Timing field Displays the scale of the geometry animation. Editable.

Offset field Displays the offset applied to the animation curve. Editable.

UV Map Settings

Use the UV Mapping settings to select how the UV coordinates of an attached displace, normal, or diffuse node are mapped to the 3D model. You can also apply axis transformations to the UV map. These transformations are different from the settings of the parent axis in that they transform the axes of the actual UV map coordinates.



UV Mapping Type box Select the type of UV mapping to apply to the attached node.

When a Displace or Normal node is attached to a geometry, you may need a UV mapping type other than Default for the displace or normal pattern to have any effect on the geometry.

NOTE When a Diffuse node is attached to a geometry, you must select Wrap from the Mapping box in the Diffuse menu to be able to use the UV mapping settings. See Diffuse Mapping (page 800).

Use GroupId button Enable to respect GroupId information in an FBX file created in 3ds Max.

Smooth Angle button Enable to override existing normals in the geometry, then use the Smooth Angle field to change the value.

Smooth Angle field Displays the angle at which the edges of normals become hard. Changes to this field only affect the shading of the displacement, and not the shape. Editable.

Position X field Displays the position of the X axis. Editable.

Position Y field Displays the position of the Y axis. Editable.

Position Z field Displays the position of the Z axis. Editable.

Rotation X field Displays the rotation of the X axis. Editable.

Rotation Y field Displays the rotation of the Y axis. Editable.

Rotation Z field Displays the rotation of the Z axis. Editable.

Scale X field Displays the scale of the X axis. Editable.

Scale Y field Displays the scale of the Y axis. Editable.

Scale Z field Displays the scale of the Z axis. Editable.

Prop Scale button Scales the X, Y, and Z UV axes proportionally.

Shear X field Displays the shear of the X axis. Editable.

Shear Y field Displays the shear of the Y axis. Editable.

Shear Z field Displays the shear of the Z axis. Editable.

NOTE The UV Transform fields are only available if a Mapping Type other than Default is selected.

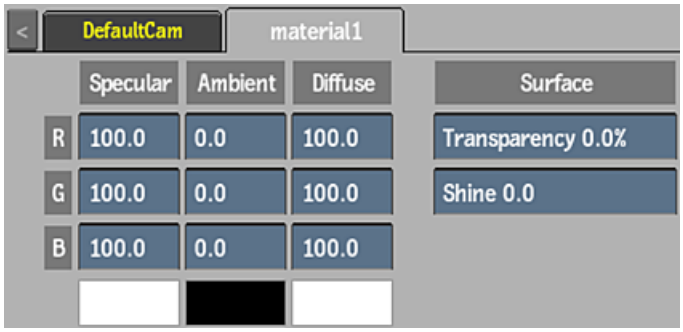
Material Node Settings

The Material node creates a central hub to control the specular, ambient, diffuse, transparency, and shininess of a geometry's children maps and substances.

Flame automatically creates a Material node when you insert a Substance Texture or when you import 3ds Max or FBX models with textures. You can also manually add a material node to any geometry or surface in the Action schematic.

NOTE In a PBS workflow, some Physically Based shader settings or PBS map settings override the Material node settings. In this case, specular and shine settings in the Material menu do not have any effect.

Material Node Menu Settings



Lighting Settings

Red Specular field Displays the red specular highlight value. Editable.

Green Specular field Displays the green specular highlight value. Editable.

Blue Specular field Displays the blue specular highlight value. Editable

Specular colour pot Displays the colour of light reflected by the 3D model's surface. To enable the specular highlight, the Shine value must be larger than 0. Editable.

NOTE Specular lighting sets the colour of light reflected by the 3D model's surface. To enable the specular highlight, the Shine value must be larger than zero.

Red Ambient field Displays the red ambient colour value. Editable.

Green Ambient field Displays the green ambient colour value. Editable.

Blue Ambient field Displays the blue ambient colour value. Editable.

Ambient colour pot Displays the colour of the area of the 3D model that is not illuminated by a direct light source. Editable.

NOTE Ambient lighting sets colour to the area of the 3D model that is not illuminated by a direct light source. The edge of the ambient area mixes with the specular highlight colour and the diffuse colour.

Red Diffuse field Displays the red diffuse colour value. Editable.

Green Diffuse field Displays the green diffuse colour value. Editable.

Blue Diffuse field Displays the blue diffuse colour value. Editable.

Diffuse colour pot Displays the diffuse colour. Editable.

NOTE Diffuse lighting modifies the colour and illumination of the entire 3D model. Diffuse light mixes with the colour of the light sources used to illuminate the 3D model. The diffuse colour may also mix with the ambient colour and the colour of the specular highlight.

Surface Settings

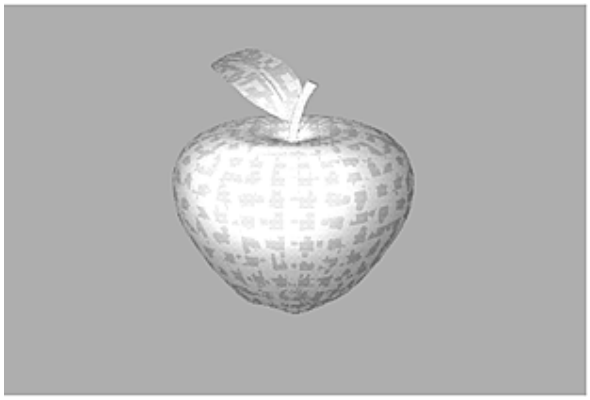
Transparency field Displays the transparency level of the 3D model. Editable.

Shine field Displays the intensity value of the specular highlight. When this value is zero, the specular highlight is disabled. Shine affects both size and intensity. Editable.

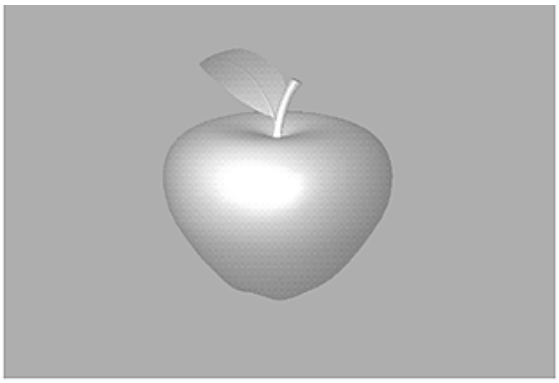
See Creating a Specular Highlight on a Model (page 717).

Adjusting Normals

When using transparency with 3D models, you may sometimes see the back polygons, giving the model a shattered or broken look. This happens when the drawing priority of the normals is not sorted properly. Enable Sort and select an option from the Sort Order box to sort the drawing priority back to front or front to back.



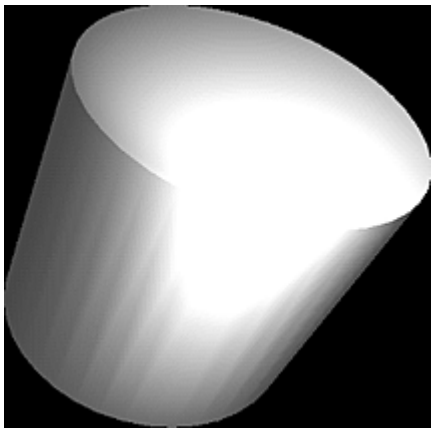
Drawing order of the 3D model's polygons is incorrect. Back polygons are drawn through when the model is transparent.



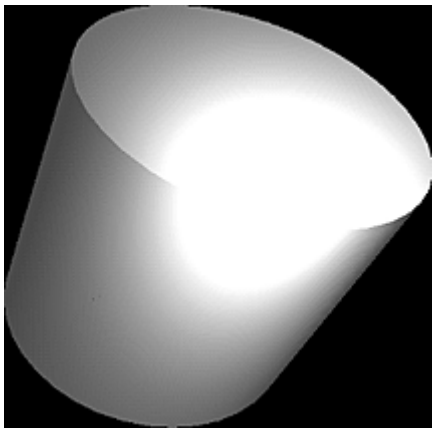
Select Front to sort the polygons front to back. The transparency is drawn correctly.

Subdividing a Model

The Subdivide feature is useful for creating precise highlights and spotlights. The polygons in the 3D model are subdivided at the time of render according to the value you specify, resulting in smoother rendered surfaces.



Spotlight on a polygon model rendered with Subdivide off.



Spotlight on a polygon model rendered with Subdivide set to 2.

To subdivide a model:

1. Select the model or geometry you want to subdivide.
2. In the Geometry menu, click Subdivide to enable the Resolution field.
3. Edit the Resolution value.
4. Click Render or Preview to see the subdivision effect.

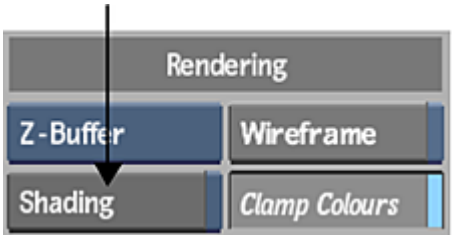
You can animate the Subdivide channel in the Channel Editor; however, expect a longer rendering time when the Subdivide value changes over several keyframes.

Creating a Specular Highlight on a Model

Change the specular colour by entering values in the Specular red, green, and blue channel fields or using the colour picker. For example, if the specular colour is red and the light source is white, the specular highlight is also red. If the specular colour is yellow and the light source is red, the highlight is orange.

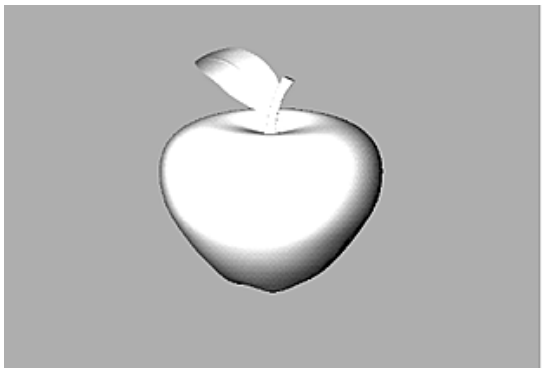
To use a specular highlight with a 3D model:

1. Add and position a light source in the scene.
2. In the scene, select the 3D model to which you want to add the highlight.
3. In the Action Setup menu, enable Shading.

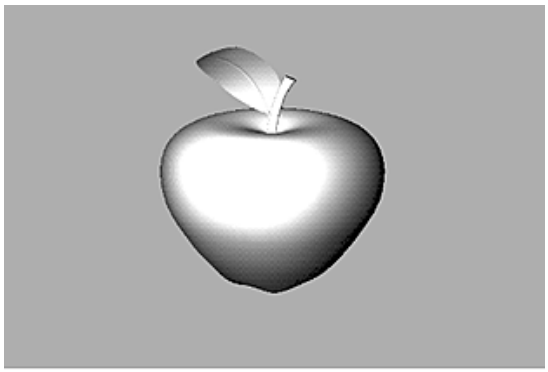


4 In the Geometry menu, set a value in the Shine field.

A high Shine value produces a dimmer highlight while a low Shine value produces an intense highlight. When the Shine field is set to zero, the 3D model does not have a highlight. The following example shows the same 3D model using two different Shine values.



Shine is set to 1. The specular highlight is intense.



Shine is set to 20. The specular highlight is dimmer.

Deforming Models and Surfaces

Use a deformation mesh to deform a 3D model or surface. To use 3D deformations, attach a Deform node to the 3D model or surface, and then use the Deform menu to modify the mesh. You can animate individual points on the deformation mesh, or move the entire mesh over the 3D model or surface to apply the deformation as the mesh passes over the object.

Adding a Deformation Mesh

When you add a Deform Mesh node, the deformation mesh appears over the 3D model or surface. You can view the source mesh, the destination mesh, or both, to assist you as you deform the object. You can also turn both meshes off to view only the deformed object.

The deformation mesh consists of lattices and cells. You can divide the mesh into 1 to 100 lattices (to form a 3D grid of vertices), and each lattice can be divided by 1 to 3 cells. Increase the number of cells and lattices to deform specific areas of the object.

To add a deformation mesh:

1 In the schematic, select the axis for the image, 3D model, or 3D text.

2 Do one of the following:

- Drag the Deform Mesh node from the node bar and place it in the schematic.
- Drag the Deform Mesh node from the node bar and place it where you want it in Result view.
- Double-click the Deform Mesh node in the node bar. You do not need to be in Schematic view to add a node in this manner.

The deformation mesh is added to the selected object.

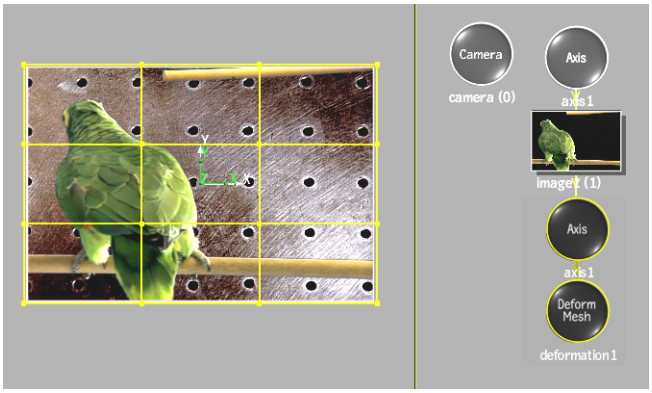


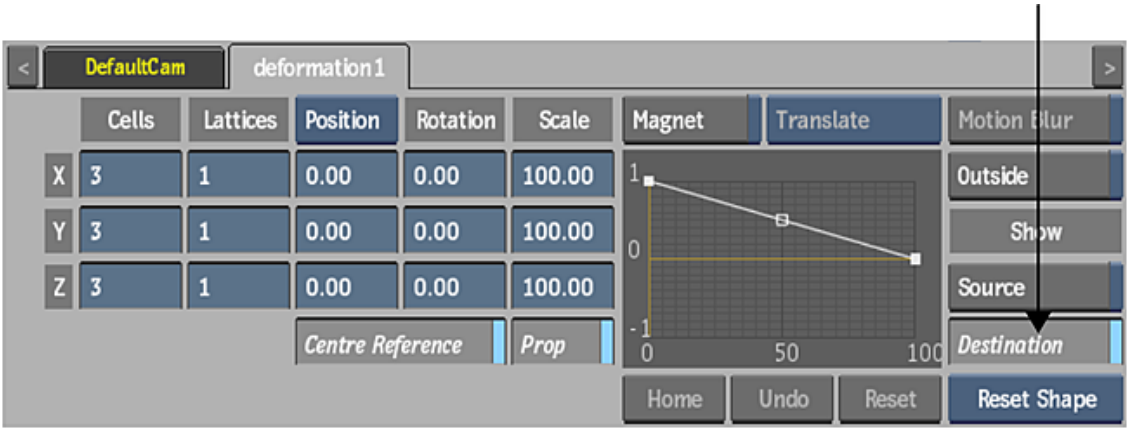
Image courtesy of Quietman

If you do not see the deformation mesh, follow the next steps.

3 Double-click the Deform Mesh node in the schematic.

The Deform Mesh menu appears.

4 In the Deform Mesh menu, enable the Show Destination button to view the deformation mesh in the image window.



Modifying a Deformation Mesh

You define a 3D grid of vertices using lattices and cells to create a deformation mesh. You can divide the mesh into 1 to 100 lattices (to form the 3D grid of vertices), and each lattice can be divided by 1 to 3 cells. Increase the number of cells and lattices to deform specific areas of the object.

NOTE You can only change the number of cells or lattices before you modify the parameters of the mesh. If you modify a parameter, for example, translate a tangent, you cannot change the number of cells or lattices.

To change the number of cells or lattices:

1 In the image window, select the mesh.

2 In the Deform menu, change the number of cells in the Cells X, Y, and Z fields. By default they are set to 3, 3, 3.

3 In the Lattices X, Y, and Z field, change the number of vertices.

The cells or lattices are added to the deformation mesh.

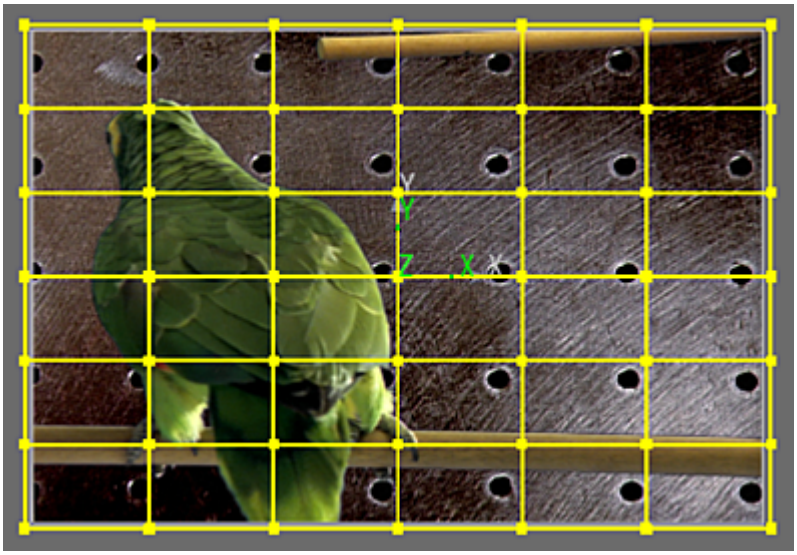


Image courtesy of Quietman

4 To view the original mesh, enable Show Source.

5 Use the Outside button to apply the deformation outside of the mesh (enabled) or constrain the deformation within the mesh (disabled).

To select points:

1 From the Tools box, choose Select.

2 Select the point(s):

- To select a single point, click the point.
- To select a range of points, hold the `Ctrl` key and draw a box around the points you want to select.
- To add a point or range of points to a selection, hold `Shift+Ctrl` and draw a box around the point(s) you want to add to the selection.

When you move a point, all selected points also move.

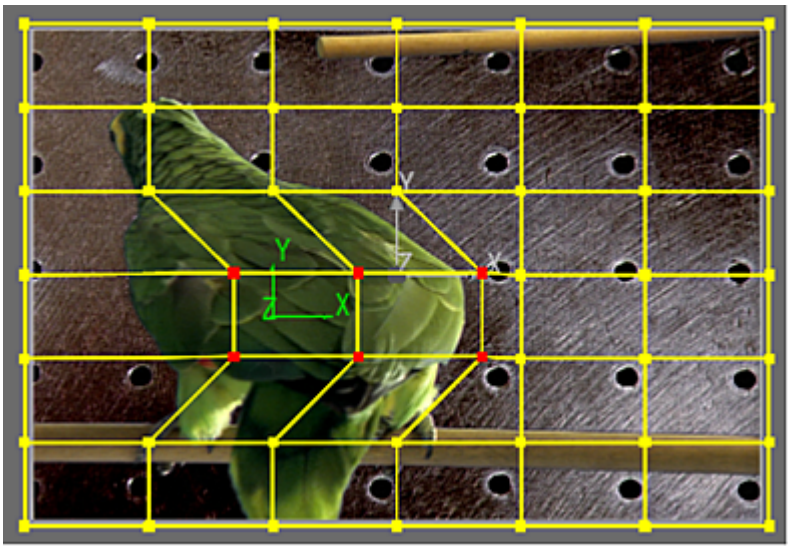


Image courtesy of Quietman

To transform a point on the mesh, drag it in the image window, or change the value in one of the X,Y or Z Translation fields.

Passing a Deformation Mesh Over an Object

Animate the deformation mesh so that it passes over the object. If the Outside button is disabled, the deformation is only applied to the parts of the object within the mesh. To achieve this effect, you must move the mesh over the object. You cannot move the object through the mesh because the object axis parents the mesh axis.

You can also use the channel editor to animate individual or multiple points for the deformation mesh.

To pass a deformation mesh over an object:

1. Add the deformation mesh to the object.
2. In the Deform menu, disable Outside.
3. Modify the points on the mesh to achieve the deformation you want.
4. In Schematic view, select the deformation axis.
5. Click Axis to view the Axis menu.
6. Move the deformation mesh so that it is on one side of the object.
7. Set a new keyframe with the deformation mesh on the other side of the object.

Transforming a Range of Points

Use the magnet to transform a range of deform mesh surface points. In the Deform Mesh menu, the magnet controls include the Magnet button, the Magnet Transformation box, and the Magnet Curve Editor. The magnet controls work the same way as the magnet controls in the Extended Bicubic (page 641) menu.

Deform Mesh Menu Settings

X Cells field Displays the number of cells in each lattice along the X axis. Editable.

Y Cells field Displays the number of cells in each lattice along the Y axis. Editable.

Z Cells field Displays the number of cells in each lattice along the Z axis. Editable.

X Lattices field Displays the number of lattices (to form a grid of vertices) along the X axis. Editable.

Y Lattices field Displays the number of lattices (to form a grid of vertices) along the Y axis. Editable.

Z Lattices field Displays the number of lattices (to form a grid of vertices) along the Z axis. Editable.

Position/Current mode box Select to modify the Position or Current axis controls.

X Position field Displays the position of the selected points along the X axis. Editable.

Y Position field Displays the position of the selected points along the Y axis. Editable.

Z Position field Displays the position of the selected points along the Z axis. Editable.

X Current field Displays the position of the selected vertex along the X axis. Editable.

Y Current field Displays the position of the selected vertex along the Y axis. Editable.

Z Current field Displays the position of the selected vertex along the Z axis. Editable.

X Rotation field Displays the rotation of the selected points along the X axis. Editable.

Y Rotation field Displays the rotation of the selected points along the Y axis. Editable.

Z Rotation field Displays the rotation of the selected points along the Z axis. Editable.

X Scale field Displays the scale of the selected points along the X axis. Editable.

Y Scale field Displays the scale of the selected points along the Y axis. Editable.

Z Scale field Displays the scale of the selected points along the Z axis. Editable.

Proportional Scale button Enable to scale the points proportionally.

Centre Reference button Enable to create a centre reference axis for selected points.

Magnet button Enable to transform a range of deform mesh surface points.

Magnet Transformation box Select a transformation type to use when Magnet is enabled.

Magnet Curve Editor Defines the weighted polarity from the centre to the edge of the magnet.

Magnet Curve Home button Resets the position of the magnet curve after panning.

Magnet Curve Undo button Undoes a change to the Magnet Curve Editor.

Magnet Curve Reset button Resets the Magnet Curve Editor.

Motion Blur button Enable to use a motion blur effect for the selected deformation (can only be used if the global Motion Blur is enabled in the Action Setup menu).

Outside button Enable to apply the deformation outside of the mesh. Disable to constrain the deformation within the mesh.

Show Source button Enable to view the original mesh.

Show Destination button Enable to view the destination mesh.

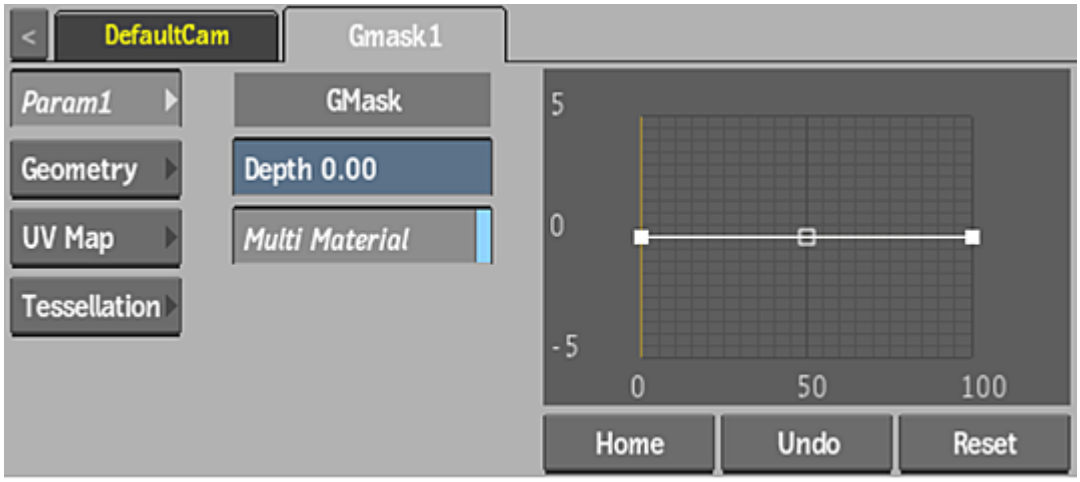
Reset box Select whether to resets the shape of the deformation mesh, or only the selected points.

Using Garbage Masks as 3D Geometry

Garbage mask files are created through the Keyer or by the GMask node in Batch. You can import a saved garbage mask setup into Action, and specify 3D properties, such as depth and geometry settings. If your garbage mask is animated, all shapes are imported into Action. Softness and offset settings in the imported garbage mask are not brought into Action. Any holes in the imported garbage mask are also holes in the 3D geometry. For help creating garbage masks, see Drawing a Mask (page 1106).

TIP A library of preset garbage mask setups is available. To load a preset garbage mask, navigate to the */opt/Autodesk/project/<project_name>/gmask* directory when importing your 3D model.

Once you import a garbage mask as a 3D geometry into Action, use the GMask menu to extrude the garbage mask.



GMask Menu Settings

Depth field Displays the level of depth (thus extruding the selection, making it three dimensional). Editable.

Multi Material button Enable to create an Object Group node for each of the front, back, and extrude of the 3D object. You can then attach a different texture map to apply to the different surfaces.

Profile Curve Displays the selected profile curve (bevel, scale, or rotation). Move and add points to the curve, as well as adjust the tangent handles to produce different effects with the extruded geometry. If you select a Parametric profile type, a separate read-only curve is displayed.

Home button Resets the selected curve viewer to show the whole curve.

Undo button Undoes the last operation for the selected curve viewer.

Reset button Resets the selected curve viewer to the default curve.

Click the Geometry (page 711) and UV Map tabs to apply any of the other geometry settings to your garbage mask.

Click the Tessellation (page 749) tab to control the tessellation of the garbage mask.

Action 3D Text

You can create and manipulate 3D text strings in your Action scenes. With 3D text, you specify typical text properties such as font, font size, kerning, and italics. Since 3D text strings created in Action are also 3D geometries, you can extrude text, offset your text from a path, and apply other geometry settings.

Using 3D Text Presets

A number of 3d text presets are included in Action, such as rotating or fading text to add to your scene. These presets can help you add complex text effects with just a few clicks.

To add a 3d text preset:

1 Do one of the following:
- Drag the Presets node from the node bin and place it in the schematic.

- Drag the Presets node from the node bin and place it where you want it in Result view.
- Double-click the Presets node. You do not need to be in Schematic view to add a node in this manner.

  The file browser opens.

2 From the Preset Type box, select 3D Text.

The 3D Text Preset file browser appears, pointing to the default location of the presets: *opt/Autodesk/<product home>/3d_text_presets*.

3 Optional: Enable Scale to Action Resolution to load the preset in the current Action resolution.

4 Optional: Select which rendering settings to enable or disable in the preset (Z-Buffer, Shading, Polygon Resolution, and Colour Clamping).

NOTE These settings are enabled by default, and by disabling any of them, you may not see the intended results in the preset.

5 Select the 3d text preset you want to load. Hold `Ctrl` and click to select multiple presets.

TIP Switch to Proxies view to see a visual representation of the presets.

6 Click Load.

The 3d text preset is then appended to your Action scene. In the 3D Text menu, you can change the default text string of the preset.

3D Text Presets Usage Tips

A number of 3D text presets are included in Action, such as fading or spinning 3D text effects to add to your scene. The presets are easily added to your scene from the Action node bar. For more information on the 3D Text node and its menu, see About 3D Text (page 723).

The 3D Text presets are designed for ease-of-use, so you should be able to easily identify the various schematic nodes and their corresponding menus. For example, the nodes in the schematic for each preset are renamed to give you an indication of the functionality of the node. Many of the presets include a *transition_ctrl* Axis node, that you can use to modify the timing of the text effect, as follows:

Rotation X=duration of the In effect; Y=start weight; Z=end weight. The Y and Z Rotation fields represent the shape of the "S" curve of the In effect, if applicable. The combined Y and Z values should equal 100.

Scale X=duration before the start of the In effect; Y=duration between the end of In effect and the start of the Out effect.

Shear X=duration of the Out effect; Y=start weight; Z=end weight. The Y and Z Shear fields represent the shape of the "S" curve of the Out effect, if applicable. The combined Y and Z values should equal 100.

Centre These fields include expressions to produce animation curves, and shouldn't be altered.

When loading a 3D Text preset, with a few exceptions, you can replace the **Type Your Own Text** string in the Text field of the 3D Text menu. Some of the presets have multiple text entries. In these cases, replace the text string of each Text Geom node with your preferred text string.

Presets Browser Settings

Preset Type box Select the category of presets to display in the browser.

Scale to Action Resolution button Enable to load the preset in the current Action resolution.

Z-Buffer button Enable to load the Z-buffer rendering settings of the preset.

Shading button Enable to load the shading rendering settings of the preset.

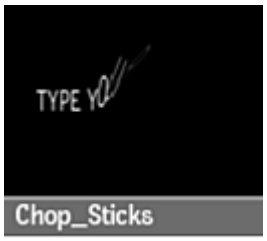
Polygon Resolution button Enable to load the rendering resolution settings of the preset.

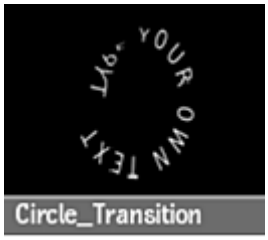
Colour Clamping button Enable to load the resolution colour clamping settings of the preset.

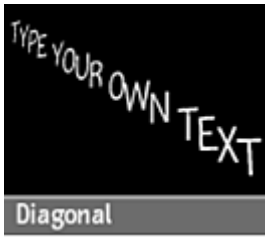
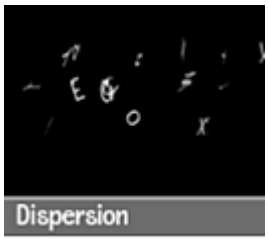
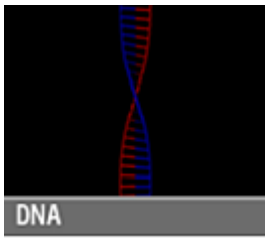
3D Text Preset List

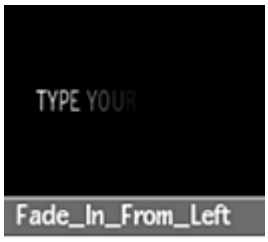
Use the following table to get a quick overview of the 3D text presets available in this release, along with particular comments, if applicable.

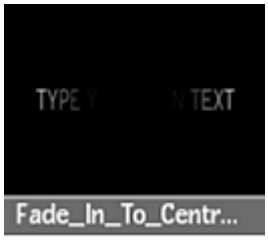
Proxy	Name/Description/Comments
	**3DRotate** Text on a rotating circular path. Edit timing using the Animation Track Editor.
	**3DRotate_Shadow** Text rotating on a circular path with a fake shadow. Edit timing using the Animation Track Editor.
	**3DRotate_Transition** Transition between two text strings rotating on a circular path. Edit timing using the Animation Track Editor.

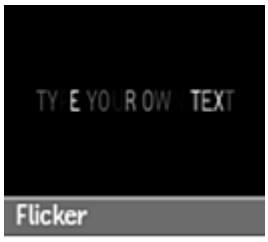
Proxy	Name/Description/Comments
	**Behind_Camera_3D** Characters coming from behind the camera. Edit timing using *transition_ctrl* settings.
	**Bounce** Characters falling and bouncing. Edit timing using *transition_ctrl* settings. Set duration and amplitude of the bouncing.
	**Break_and_Fall** Characters breaking in two parts and falling down. Edit timing using the Animation Track Editor.
	**Bubble_Pop** Characters forming a bubble which pops. Edit timing using *transition_ctrl* settings.
	**Chop_Sticks** Characters arriving one after the other with a scale and rotation effect. Edit timing using *transition_ctrl* settings.

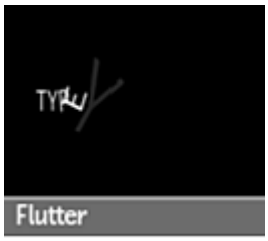
Proxy	Name/Description/Comments
	**Circle** Characters moving along a circular path. Edit timing using the Animation Track Editor.
	**Circle_Transition** Transition between two text strings, in which characters are moving along a circular path with a banking effect. Edit timing using *transition_ctrl* settings.
	**Climber** Characters climbing from the bottom to the top of the frame. Edit timing using *transition_ctrl* settings. Use offset.rotation.X to set the displacement duration and offset.rotation.Y to set the rotation duration.
	**Clock_Transition** Transition between four text strings rotating like clock hands. Edit timing using *transition_ctrl* settings.
	**Colour_Flasher** Moving coloured flashing text. Edit timing using *transition_ctrl* settings. Colour_offset sets the time offset between red, green, and blue channels.

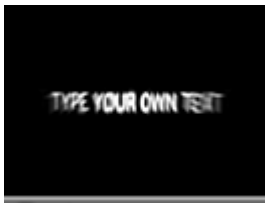
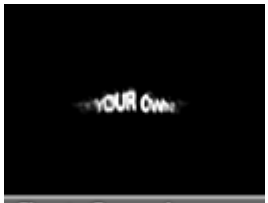
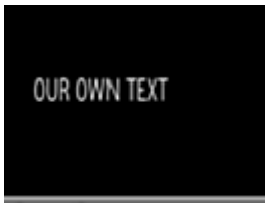
Proxy	Name/Description/Comments
	**Counter** Incremental counter with numbers 000 to 999.
	**Diagonal** Characters dispersing diagonally in the frame. Edit timing using *transition_ctrl* settings.
	**Dispersion** Characters dispersing randomly. Use time_stretch.scaling.X to slow down or speed up the effect.
	**Dispersion_Transition** Transition between two text strings using a dispersion movement. Edit timing using *transition_ctrl* settings.
	**DNA** Animated double helix structure using two text strings of the letter "T".

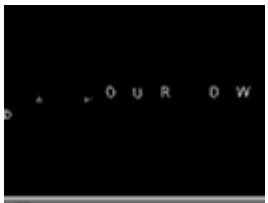
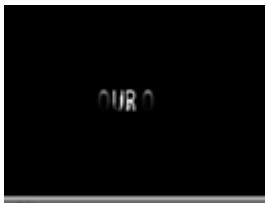
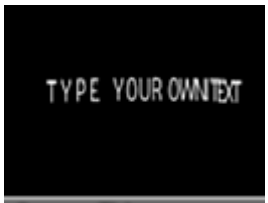
Proxy	Name/Description/Comments
	**Dominos** Characters falling one after the other. Edit timing using *transition_ctrl* settings.
	**Fade_In_Flicker** Fade in of randomly flickering characters. Edit timing using *transition_ctrl* settings.
	**Fade_In_From_Centre** Fade in from the centre to the extremities of the text. Edit timing using *transition_ctrl* settings. Set characters.rotation.X to the number of characters in the Text field.
	**Fade_In_From_Left** Fade in from the left to the right of the text. Edit timing using *transition_ctrl* settings.
	**Fade_In_Out_By_Character** Fade in and out one character after the other. Edit timing using *transition_ctrl* settings. Set characters.rotation.X to the number of characters in the Text field.

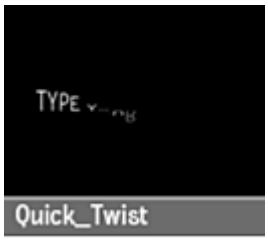
Proxy	Name/Description/Comments
	**Fade_In_Random** Use time_stretch.scaling.X to slow down or speed up the effect.
	**Fade_In_To_Centre** Fade in from the extremities to the centre of the text. Edit timing using *transition_ctrl* settings.
	**Fade_Out_From_Centre** Fade out from the centre to the extremities of the text. Edit timing using *transition_ctrl* settings. Set characters.rotation.X to the number of characters in the Text field.
	**Fade_Out_From_Right** Fade out from the right to the left of the text. Edit timing using *transition_ctrl* settings.
	**Fade_Out_Random** Use time_stretch.scaling.X to slow down or speed up the effect.

Proxy	Name/Description/Comments
	**Fade_Out_To_Centre** Fade out from the extremities to the centre of the text. Edit timing using *transition_ctrl* settings.
	**Fast_Beam** Characters arriving from the right side of the frame, with scaling and flickering. Edit timing using *transition_ctrl* settings. Customize the flicker effect using *flicker* settings.
	**Flasher** Customize the flashing using *period_offset* settings.
	**Flicker** Animate with period.rotation.X.
	**Flip_Transition** Transition between two text strings, with characters rotating on the Y axis. Edit timing using *transition_ctrl* settings.

Proxy	Name/Description/Comments
	**Flutter** Characters arriving one after the other with a scale and rotation effect. Edit timing using *transition_ctrl* settings.
	**Flying_Transition** Transition between two text strings in which characters are flying out from the centre and coming back to it. Edit timing using *transition_ctrl* settings.
	**From_Back** Characters arriving from behind the camera. Use time_stretch.scaling.X to slow down or speed up the effect.
	**From_Bottom** Characters arriving from the bottom of the frame, with rotation. Edit timing using *transition_ctrl* settings.
	**From_Camera_3D** Characters arriving from behind the camera and turning towards the camera. Edit timing using *transition_ctrl* settings.

Proxy	Name/Description/Comments
	**Ghost** Text arriving toward the camera, reflecting, and fading. Edit timing using the Animation Track Editor.
	**Ghost_Cascade** Similar to Ghost, with a cascade applied. Edit timing using the Animation Track Editor.
	**Ghost_Rotation** Similar to Ghost with an animated ParticleDraw axis. Edit timing using the Animation Track Editor.
	**Helix_Transition** Transition between two text strings in which characters are rotating on their X axis. Edit timing using *transition_ctrl* settings.
	**Insertion** Text is revealed frame after frame in insert mode. Edit timing using *transition_ctrl* settings. Use offset.rotation.X to set the starting value of the path.

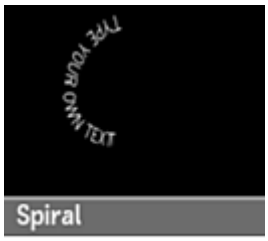
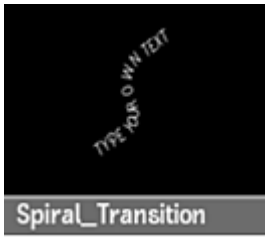
Proxy	Name/Description/Comments
 Kern	**Kern** Characters arriving and leaving the frame with kerning and rotation. Edit timing using *transition_ctrl* settings.
 Mirror	**Mirror** Text on path with a mirror effect. Edit timing using the Animation Track Editor.
 Negate	**Negate** Text revealed using a negate blending effect. Edit timing using *transition_ctrl* settings.
 Ocean_Tide	**Ocean_Tide** Text moving slowly as if characters were floating on the sea. Use period and amplitude axes to customize the wavy effect.
 Ocean_Tide_Transi...	**Ocean_Tide_Transition** Transition between two text strings moving slowly as if characters were floating on the sea. Edit timing using *transition_ctrl* settings. Use period and amplitude axes to customize the wavy effect.

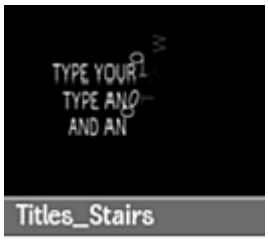
Proxy	Name/Description/Comments
	**Path_Warp** Text on animated path producing a warping effect. Edit timing using the Animation Track Editor.
	**Pendulum** Characters moving as if attached to a pendulum Edit timing using *transition_ctrl* settings. Use period and amplitude axes to customize the oscillating effect.
	**Pulse_Wave** Characters moving vertically with a pulsating effect. Customize using period and amplitude settings.
	**Quick_Rotate** Characters rotating on their X axis. Edit timing using *transition_ctrl* settings.
	**Quick_Twist** Characters scaled on their Y axis. Edit timing using *transition_ctrl* settings.

Proxy	Name/Description/Comments
	**Random_Display** Random letters with a specific starting and ending text string. Edit timing using *transition_ctrl* settings. Type the start text in text_1 and the stop text in text_2.
	**Random_Hide** Characters disappearing randomly. Clipping Plane modified. Use time_stretch.scaling.X to slow down or speed up the effect.
	**Random_Letters** Letters displayed randomly.
	**Random_Reveal** Characters appearing randomly. Clipping Plane modified. Use time_stretch.scaling.X to slow down or speed up the effect.
	**Replace_Transition** Transition between two text strings with rotation. Edit timing using *transition_ctrl* settings.

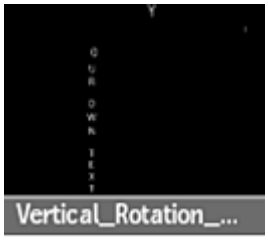
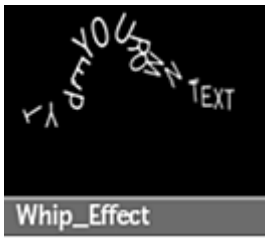
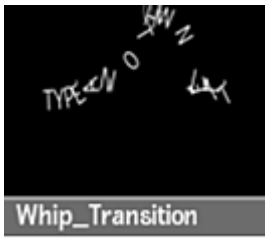
Proxy	Name/Description/Comments
	**Reverse** Transition between two text strings in which characters reverse direction. Edit timing using the Animation Track Editor.
	**Rubber** Characters are stretched and released like rubber. Edit timing using *transition_ctrl* settings. Set period and amplitude of the bouncing.
	**Rubber_Transition** Transition between two text strings using a rubber effect. Edit timing using *transition_ctrl* settings. Set period and amplitude of the bouncing.
	**Scaling_Transition** Transition between two text string using a scaling effect. Edit timing using *transition_ctrl* settings. Set characters.rotation.X to the number of characters in the Text field.
	**Shiver_Transition** Transition between two text strings with a shiver-like effect. Edit timing using *transition_ctrl* settings. Increase or decrease the turbulence using noise_amplitude.

Proxy	Name/Description/Comments
	**Shuffle_In_Out** Characters arriving from the left, stopping in the centre, and leaving to the right. Edit timing using *transition_ctrl* settings.
	**Spin_Compress** Characters revealed with rotation, and leaving with a compression effect. Edit timing using the Animation Track Editor.
	**Spin_In_Out** Characters arriving and leaving with rotation. Edit timing using *transition_ctrl* settings.
	**Spin_Transition** Transition between two spinning text strings. Edit timing using *transition_ctrl* settings.
	**Spinning_Letters** Characters thrown with a spinning effect. Edit timing using *transition_ctrl* settings. Experiment with transition_ctrl.scaling.Y and transition_ctrl.scaling.Z.

Proxy	Name/Description/Comments
	**Spiral** Text on a spiral path. Edit timing using the Animation Track Editor.
	**Spiral_Transition** Transition between two text strings with a spiral effect. Edit timing using *transition_ctrl* settings.
	**Spring_Transition** Transition between two text strings with letters springing left and right. Edit timing using *transition_ctrl* settings. Customize the pulsing effect using period and amplitude settings
	**Squash_Transition** Transition from a text string with letters being squashed to reveal the other text string. Edit timing using *transition_ctrl* settings.
	**Star** Text moving along a star path. Edit timing using the Animation Track Editor.

Proxy	Name/Description/Comments
	**Stretch** Text arriving from the left of the frame with a stretching effect. Edit timing using *transition_ctrl* settings.
	**Subtract** Text revealed using a subtract blending effect. Edit timing using *transition_ctrl* settings.
	**Text_Falling** Characters moving along a line, then falling down. Edit timing using the Animation Track Editor.
	**Titles** Three text strings on a path arriving from the upper left. Edit timing using the Animation Track Editor.
	**Titles_Stairs** Three text strings revealed with a spinning effect. Edit timing using *transition_ctrl* settings.

Proxy	Name/Description/Comments
	**Track_In** Characters appear and disappear with scaling. Edit timing using *transition_ctrl* settings.
	**Tremor** Characters appear as if shaking. Increase or decrease the turbulence effect using noise_amplitude.
	**Twist** Characters arriving and leaving with a twisting effect. Edit timing using *transition_ctrl* settings.
	**Typewriter** Text revealed frame after frame with a typewriter effect. Edit timing using *transition_ctrl* settings. Use transition_ctrl.rotation.X andtransition_ctrl.shearing.X to set the fade in duration of characters.
	**Typewriter_Cursor** Text is revealed frame after frame with a cursor preceding characters. Edit timing using *transition_ctrl* settings. Use transition_ctrl.rotation.X and transition_ctrl.shearing.X to set the fade in duration of characters. Set as many white squares as characters in the text node. Adapt cursor kerning if needed.

Proxy	Name/Description/Comments
 Vertical_In_Out	**Vertical_In_Out** Characters arriving from the left side of the frame and moving vertically before going out to the right of the frame. Edit timing using *transition_ctrl* settings.
 Vertical_Rotation_...	**Vertical_Rotation_Transition** Transition between two text strings moving vertically. Edit timing using *transition_ctrl* settings.
 Waveform	**Waveform** Waveform using Cursor font. Link text.character_anim.scaling.Y to an audio file.
 Whip_Effect	**Whip_Effect** Characters whipped by a rotating effect. Edit timing using *transition_ctrl* settings.
 Whip_Transition	**Whip_Transition** Transition between two text string with characters whipped by a rotating effect. Edit timing using *transition_ctrl* settings.

Proxy	Name/Description/Comments
	**Whirlwind_Transition** Transition between two text string with characters circulating by a rotating effect. Edit timing using *transition_ctrl* settings.
	**White_Background** Characters revealed on a white background. Edit timing using *transition_ctrl* settings. Change the background using white_bkgd_ax settings and white_bkgd kerning value.
	**Windy** Characters moving as if by wind. Customize with period and amplitude axes.
	**Zig_Zag** Characters arriving and leaving following a zig-zag path. Edit timing using *transition_ctrl* settings. Change the path, if needed.

Adding a 3D Text Node

When you add a 3D Text node to your Action schematic, a special geometry node with an axis is added.

To add a 3D Text node to the scene:

1. Do one of the following:
   - Drag the 3D Text node from the node bin and place it in the schematic.
   - Drag the 3D Text node from the node bin and place it where you want it in Result view.
   - Double-click the 3D Text node. You do not need to be in Schematic view to add a node in this manner.

A Geometry object, called Text1 by default, and parent axis appear in the schematic. In Result view, the default Text string appears.

2 To open the 3D Text menu, double-click the 3D Text node in the schematic, or follow the tab population rules for the Object menu.

See Populating Menu Tabs of Selected Objects (page 571).

You can also add a 3D Text node using the Presets node. See Using 3D Text Presets (page 723).

Changing 3D Text Properties

When you add a 3D Text node to your scene, the default text string “Text” appears. You can easily change this text string.

To change a text string:

1 Click the Text field.

The on-screen keyboard appears, representing the character set for the selected font. Enable Up ASCII to access the rest of the character set. If the selected font has special symbol characters, enable Symbols to see them.

2 Type your text string or use the on-screen keyboard.

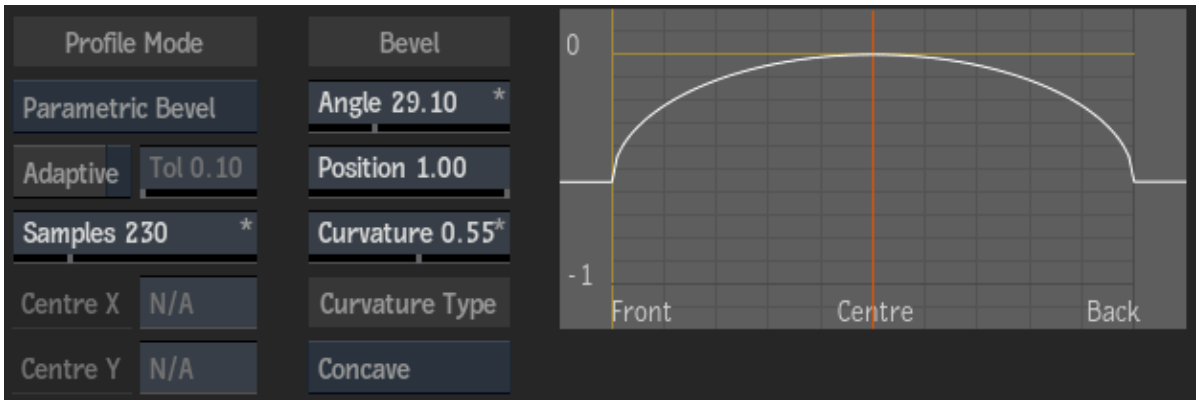
3 Click Exit Keyboard. The text string is displayed in the Text field and automatically updated in the scene.

4 Use the settings in the 3D Text tab to change the font, size, depth, and other text properties.

Using the Profile Curve With 3D Text

Use the profile curve in the Profile tab to scale, rotate, or create a bevelled edge on your extruded 3D text. You also have access to a parametric bevel curve to simplify the creation of common shapes, such as rounded profiles. Select which type of curve to work with, then use the menu settings and the options below the curve or in the Tools box to add, select, delete, or move keyframes on the curves. The curves behave in much the same way as an animation curve in the Channel Editor. Experiment with different curves to create different effects.

The profile curves are additive, that is, you can switch to each type, and change the specific curve without affecting the other curves. The parametric bevel profile type appears with added bevel settings, and a read-only curve to help you visualize the profile.



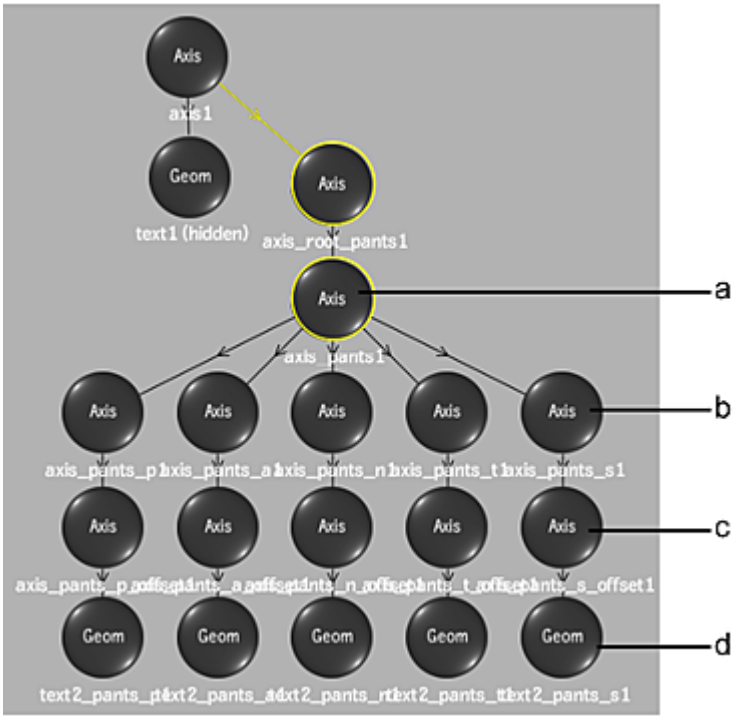
Separating Text

Rather than create a separate pivot point for each letter, you can separate words or sentences so that each letter can be individually manipulated by its own axis in the schematic.

To separate text:

1. Select the text to separate.
2. From the Text tab, click Separate.

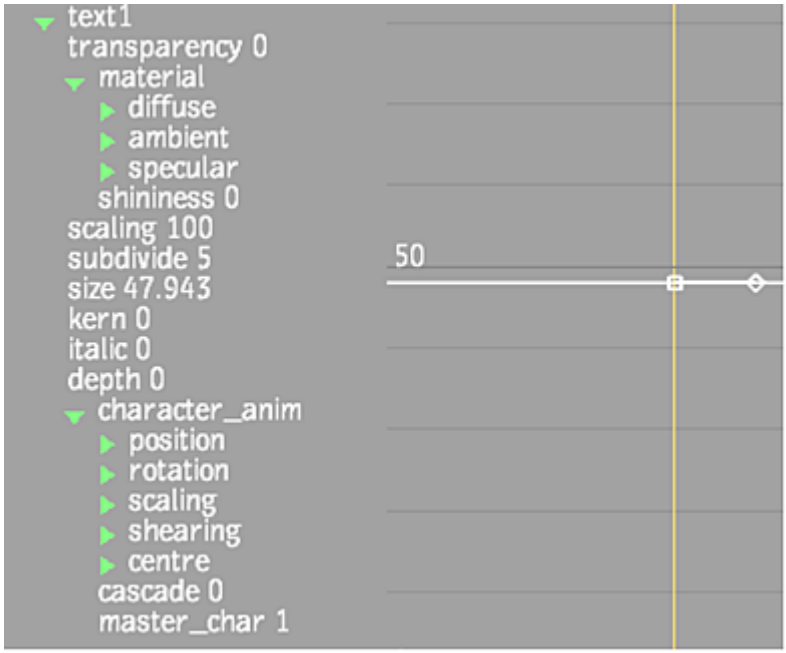
   Each letter of the text geometry is now an independent geometric object, and has its own axis and offset (to separate the letters). Each word is also given its own axis. The original 3D Text node is hidden in the schematic.



(a) Root axis for word (b) Root axis for letter (c) Offset per letter (d) Letter geometry

Animating 3D Text

You can animate the 3D text property and geometry channels in the Channel Editor. However, you cannot animate the text string or its profile curve. The 3D text channels are contained in the text folder.



3D Text Menu Settings

3D Text Tab

Text field Displays the current text string. Click to display the keyboard to enter your text.

Font field Displays the current font. Click to open the font library to select a different font for the text.

You specify the default font in the Preferences menu. Also, you can install additional fonts for use with Flame.

Depth field Displays the level of depth (thus extruding the selection, making it three dimensional). Editable.

Depth Mode box Select where the extruded geometry is positioned along the Z axis. Select Custom to set the distance from front to back manually.

Depth Centre field Displays the position of the extruded geometry along the Z axis, as a percentage. Editable if Custom is selected in the Depth Mode box.

Size field Displays the font size for the text string. Editable.

Kerning field Displays the kerning value for the text string. Editable.

Italic field Displays the level of italics for the characters in the text string. Editable.

Motion Blur button Enable to use a motion blur effect for the selected text (can only be used if the global Motion Blur is enabled in the Action Setup menu).

Rigid Text button Enable to gang the text string characters as a single geometry. Enabling this button is particularly noticeable when attaching the 3D Text node to a 3D path.

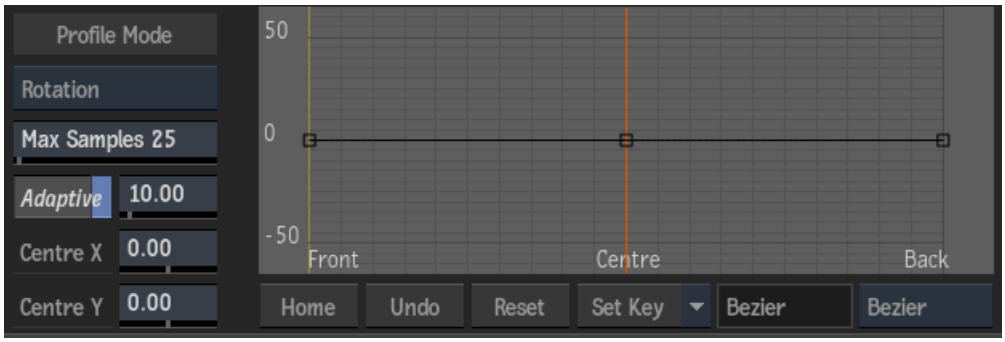
Show Pivots button Enable to display the pivot points for each individual text character in the 3D Text string (displayed in the image window in red). When disabled, only the master character pivot point is displayed (in green).

This setting can also be found in the Character Axis tab.

Separate button Click to separate selected text so that each letter has its own axis. See Separating Text (page 745).

Multi Material button Enable to create an Object Group node for each of the front, back, and extrude of the 3D object. You can then attach a different texture map to apply to the different surfaces.

Profile Tab



Profile Type box Select which profile curve to display. Different settings may appear depending on the type selected.

Adaptive button Enable to extrude based on an adaptive subdivision, adding smaller segments where there is a higher curvature. When disabled, the depth is divided into Max Samples slices of exactly the same size.

Tolerance field Displays the point at which the extrusion is subdivided. Editable.

Samples field Displays the maximum number of extrusion samples applied to the non-flat curves. Editable.

Centre X field Displays the centre along the X axis that the scale and rotation curves use. You can also adjust the centre settings in the image window by dragging the crosshair. Editable.

Centre Y field Displays the centre along the Y axis that the scale and rotation curves use. You can also adjust the centre settings in the image window by dragging the crosshair. Editable.

Parametric Angle field Displays the slope of the beveling. Editable.

Parametric Position field Displays the relative position of the start of the beveling. Editable.

Parametric Curvature field Displays the roundness or curvature of the beveling. Editable.

Curvature Type box Select the type of curvature to apply to the parametric curve.

Profile Curve Displays the selected profile curve (bevel, scale, or rotation). Move and add points to the curve, as well as adjust the tangent handles to produce different effects with the extruded geometry. If you select a Parametric profile type, a separate read-only curve is displayed.

Parametric Bevel Curve Read-only display of the parametric profile curve, based on the profile settings.

Home button Resets the selected curve viewer to show the whole curve.

Undo button Undoes the last operation for the selected curve viewer.

Reset button Resets the selected curve viewer to the default curve.

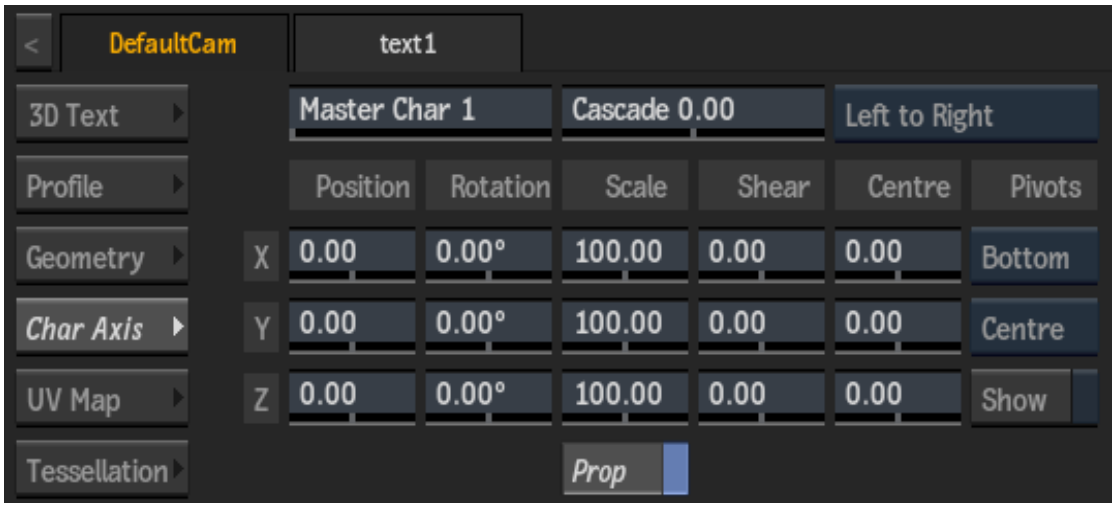
Keyframe Option box Select an option for working with keyframes in the profile curve.

Interpolation Status field Displays the interpolation type for the selected area of the curve. Non-editable.

Interpolation Type box Select an interpolation type to define the shape of the profile curve between keyframes.

Character Axis Tab

You can change the axis properties of your 3D text string characters. This can be useful in offsetting your text from a 3D path.



Master Character field Displays the number of the character in the text string that is considered to be the master. All other text characters follow this character in any character axis settings. Editable.

Cascade field Displays the amount of frames to offset the animation of other characters from the master character. The animation that is offset includes all numeric fields in the Character Axis tab, as well as the Specular, Ambient, Diffuse, Transparency, and Shine fields in the Geometry tab. Editable.

For example, if Cascade is set to 0, all characters have the same animation as the master character. If Cascade is set to a positive number, all characters other than the master character have their animation offset forward in time.

Cascade Alignment box Select the flow of the cascade offset, with respect to the master character.

X Position field Displays the position of the offset along the X axis. Editable.

Y Position field Displays the position of the offset along the Y axis. Editable.

Z Position field Displays the position of the offset along the Z axis. Editable.

X Rotation field Displays the rotation of the offset along the X axis. Editable.

Y Rotation field Displays the rotation of the offset along the Y axis. Editable.

Z Rotation field Displays the rotation of the offset along the Z axis. Editable.

X Scale field Displays the scale of the offset along the X axis. Editable.

Y Scale field Displays the scale of the offset along the Y axis. Editable.

Z Scale field Displays the scale of the offset along the Z axis. Editable.

Proportional Scale button Enable to scale the X, Y, and Z axes proportionally.

X Shear field Displays the shear of the offset along the X axis. Editable.

Y Shear field Displays the shear of the offset along the Y axis. Editable.

Z Shear field Displays the shear of the offset along the Z axis. Editable.

X Centre field Displays the centre of the offset along the X axis. Editable.

Y Centre field Displays the centre of the offset along the Y axis. Editable.

Z Centre field Displays the centre of the offset along the Z axis. Editable.

Vertical Pivot box Select the vertical position of the pivot point for the selected text characters.

Horizontal Pivot box Select the horizontal position of the pivot point for the selected text characters.

Show Pivots button Enable to display the pivot point for each individual text character in the 3D text string, displayed in the image window in red. When disabled, only the master character pivot point is displayed (in green). This setting can also be found in the 3D Text tab.

Geometry and UV Map Tabs

The settings in the Geometry (page 711) and UV Map tabs are the same as for the Geom node.

Changing Tessellation Properties

Tessellation is the process of tiling the curves' shapes with polygons. Flame offers different tessellation methods for use with 3D Text and 3D Shape nodes:

Tessellation Type box Select the tessellation type you want to apply to the geometry. More settings appear if you select Delaunay or Medial Axis. Medial Axis is not available when using the 3D Shape node.

- **Standard (GLU)** is the fastest tessellation option, although it is also the least efficient.
- **Delaunay** generates a mesh composed entirely of triangular polygons. This method gives consistent and predictable results, and in particular, it will not give different results if the tessellated objects are rotated.
- **Medial Axis** creates concentric contour lines along the medial axes (averages between the input boundary curves), morphing from one boundary shape to the next.

The Wireframe box is also provided in the Tessellation menu. It is the same setting as in the Geometry tab, but repeated here for ease-of-use.

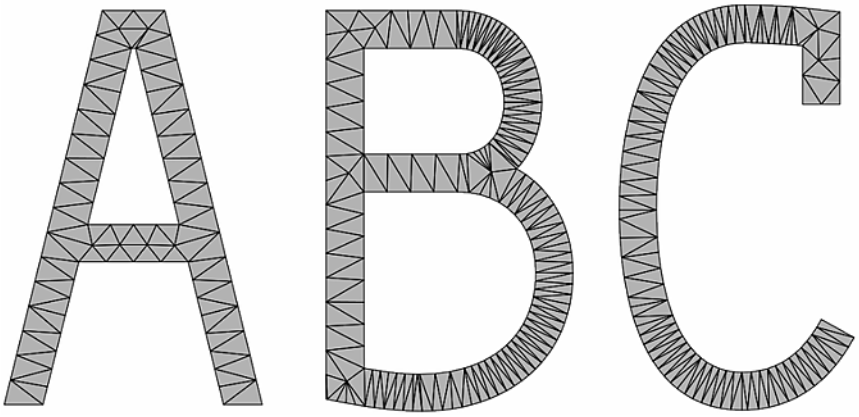
Each method has its own set of options, described in the sections that follow.

Standard (GLU)

The Standard (GLU) tessellation method is the legacy tessellation option; while being very light in its processing requirements, it is also the least efficient and precise. And compared to Delaunay and Medial Axis tessellation methods, it has no options to fine-tune the resulting tessellation.

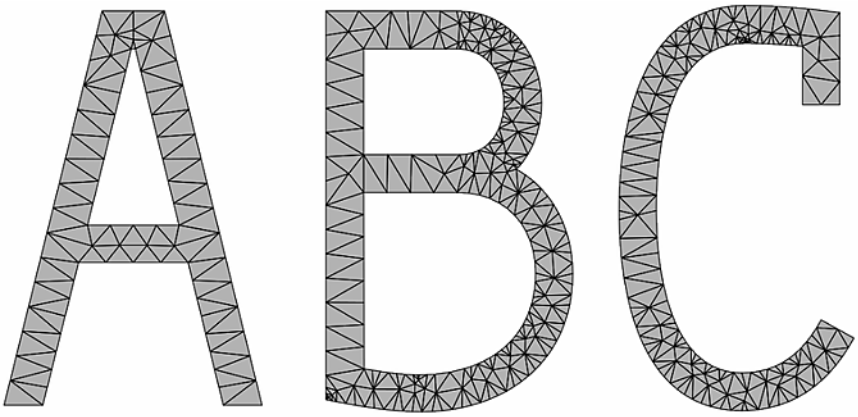
Delaunay

The Delaunay tessellation method (or more precisely, *constrained Delaunay tessellation*) generates a mesh composed entirely of triangular polygons. This method gives consistent and predictable results, and in particular, it does not give different results if the curves are rotated.



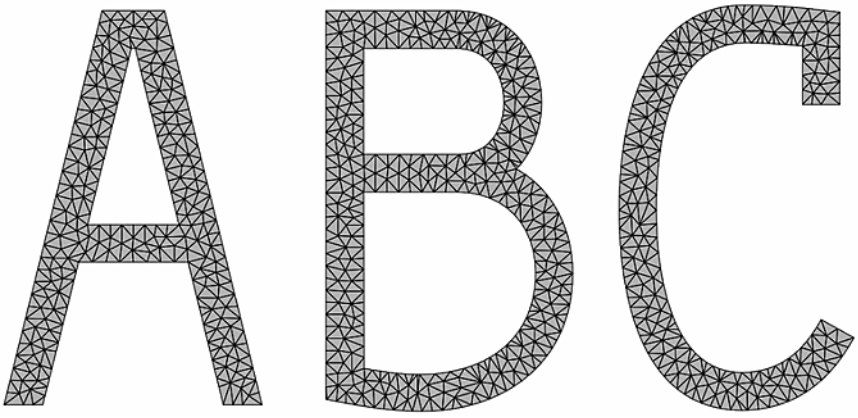
With this method, there are several options for fine-tuning the tessellation further.

Min Angle field Displays the smallest value of angle that the polygons can have. If a triangle contains an angle that is smaller than this value, it gets replaced by better-shaped ones. Eliminating small-angled triangles gives a more uniform shading and is more suited for deformations.



Min Angle = 20

Max Area field Displays the largest value of area that the polygons can have. If a triangle is larger than this value, it gets replaced by smaller ones. This allows the polygon mesh to be deformed more smoothly.



Max Area = 5

Max Vertices field Displays the total number of new vertices that can be added by the Minimum Angle and Maximum Area options. 0 by default.

Use this option as a precaution against accidentally setting the other options to values that would create huge amounts of geometry with long processing times.

It is not recommended that you rely on this option to control the final number of vertices because it can force the tessellation to stop before the process is completed, thereby giving an unpredictable combination of polygon shapes and sizes.

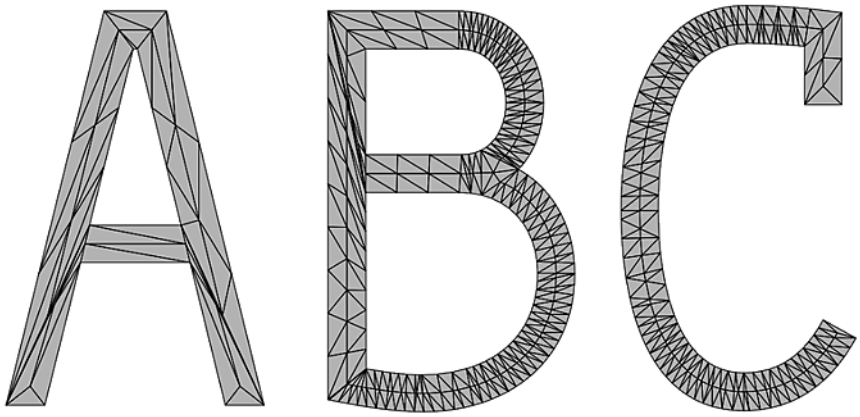
Boundary Split box Select an option to control tessellation along boundaries.

Select:	To:
Free	Allow the boundary edges along the outer contour and inner holes to be split further during tessellation. This is particularly useful for text and other shapes that may contain straight edges that need to be deformed smoothly.
None (Contour Only)	Allow boundary edges along inner holes to be split, but not boundary edges along the outer contour. Note that this may affect the uniformity of the mesh if you enabled Min Angle or Max Area.
None (Contour and Holes)	Prevent any boundary to be split. Note that this may affect the uniformity of the mesh if you enabled Min Angle or Max Area.

Medial Axis

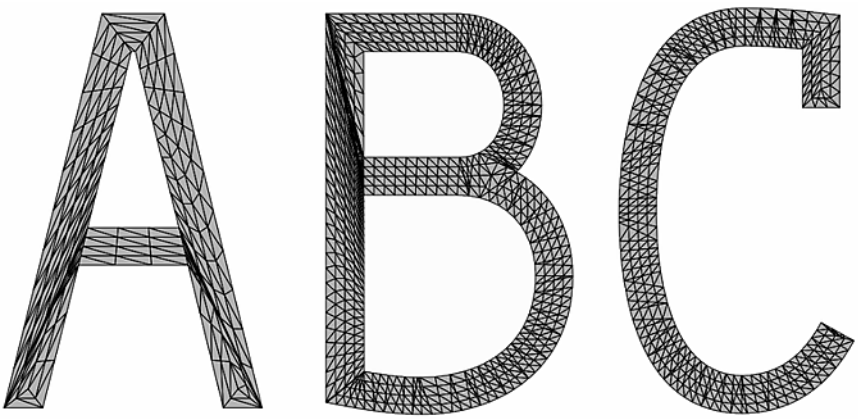
The Medial Axis tessellation method creates concentric contour lines along the medial axes (averages between the input boundary curves), morphing from one boundary shape to the next. This method creates mainly quads with some triangles, so it is well-suited for subdivision surfaces.

NOTE Medial Axis is not available when using the 3D Shape node.



With this method, there are several options for fine-tuning the tessellation further.

Loops field Displays the number of loops used in the tessellation, modified by the Adaptive toggle.



Loops = 2

Adaptive button Enable to set the Loops field as the average number of medial axes drawn and to keep the distance between them fairly constant. Disable to set the Loops field as the exact number of medial axes drawn per boundary (rounded to the nearest integer).

Backtrack Length field Displays the tessellation value at the extremities. Editable.

Set to:	To have:
0	The medial axis intersects boundaries at each point of concavity, which can often create many small triangles especially in sharp extremities.
Positive value	The medial axis does not extend completely to the boundary and the remaining area is tessellated with a fan shape.
Negative value	Sharper embossing effects

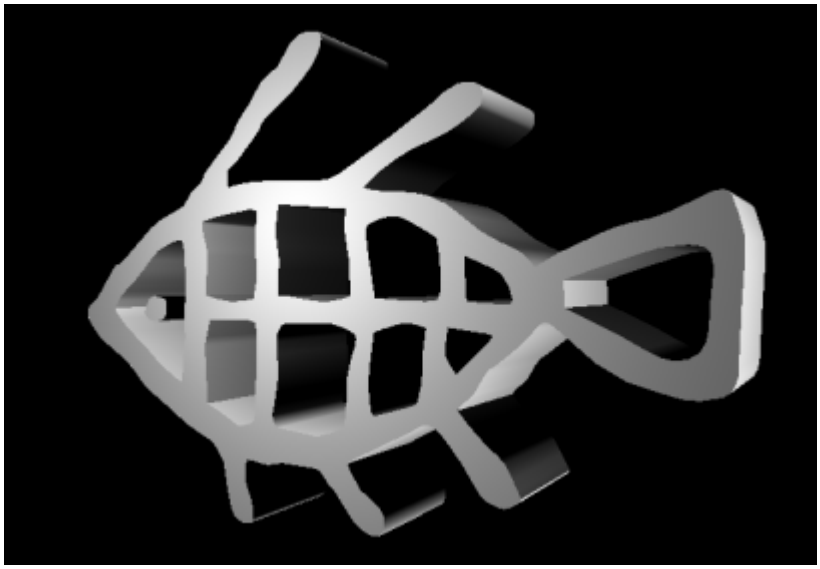
Split Edges to Enhance button Enable to add vertices to allow the contour lines to follow the medial axes more accurately. Turn this option off if there are no holes in the geometry; otherwise, there may be shading artifacts along internal curves.

Edge Tessellation box Select an option to control the shape of the polygons.

Select:	To:
None	Have long edges that are not split. This results in fewer polygons and lighter geometry, but the resulting long, thin polygons may not deform well.
Equal on Both Sides	Have a tessellation made of squarer polygons that deform better, at the cost of a heavier tessellation.

Action 3D Shape

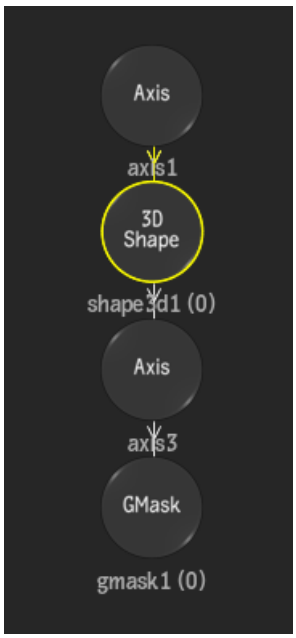
The 3D Shape node combines Action's 3D geometry environment with extrudable GMask splines to create a powerful motion graphics tool that is easy to use. 3D Shape is used to create custom spline geometries (open or closed) on which you can apply all the same functions as on any other Action geometry, as well as bevel, scale, and rotation curves.



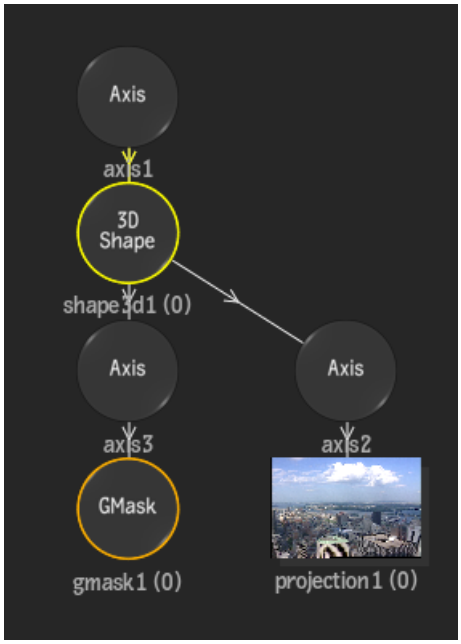
The GMask splines remain "live" in Action, with instant feedback as you modify the spline in the perspective view. The 3D Shape/GMask combination supports shape and vertex animation, shape tracking, and transparency gradients. The splines can be combined to create geometries with holes, and with an understanding of priorities between splines, so you can build animated concentric figures, for example. This hybrid approach to geometry allows you to create interesting effects that can produce an overall look in an image, or a specific effect in a portion of an image.

Here are a few points to help you when working with 3D Shapes:

- When you add a 3D Shape node, in the Action schematic, notice that a hierarchy of a 3D Shape node and GMask node (each with its own Axis) is created. You use the GMask to draw an open or closed spline. The menu structure allows you to see and use the 3D Shape settings on the left and the GMask settings on the right.



- You can add as many GMask nodes to the 3D Shape node as needed. You can decide if an attached GMask is seen as a hole or filled by the parent 3D Shape. A Spline Combination box in the 3D Shape menu allows you to set the combination pattern if you use multiple GMasks.
- You can use the F8 Object view to help you visualize your GMasks attached to a 3D Shape node.
- As with 3D Text geometries, you can attach different textures to the front, back, and extrusion of a 3D Shape with the Multi Material button. With a 3D Shape geometry, you can also apply a media projection with any media in the Action Media list. For example, you can project the same media onto a 3D Shape spline with gradients, then apply selective lighting only to that area.



Importing SVG Files as 3D Shapes

Scalable Vector Graphics (*.svg*) files are open-standard 2D graphics, and can be imported with the Action Import (page 699) node. Once imported, an *.svg* file appears in the Action schematic as a Group node; once ungrouped, you can see that the graphic is converted to a number of geometry elements: 3D Shape nodes (with accompanying GMasks and Axes). In some cases, elements with the same colour are grouped together to allow you to more easily manipulate the elements.

The following *.svg* elements are not supported in Flame:

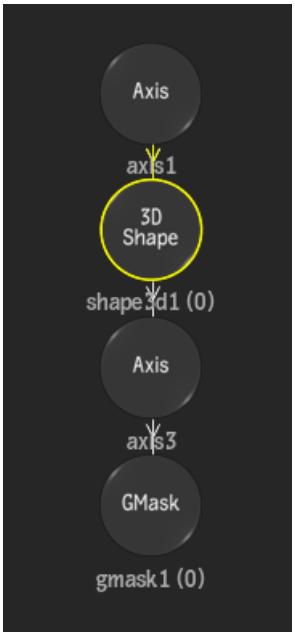
- Animation
- Text and TextPaths
- Images
- Gradients and Patterns
- Clipping, Masking, and Blending
- Paint Styles
- Filters
- Interactive elements and scripts

Adding a 3D Shape Node

To add a 3D Shape node:

1. Do one of the following:
   - Drag the 3D Shape node from the node bin and place it in the schematic.
   - Double-click the 3D Shape node. You do not need to be in Schematic view to add a node in this manner.

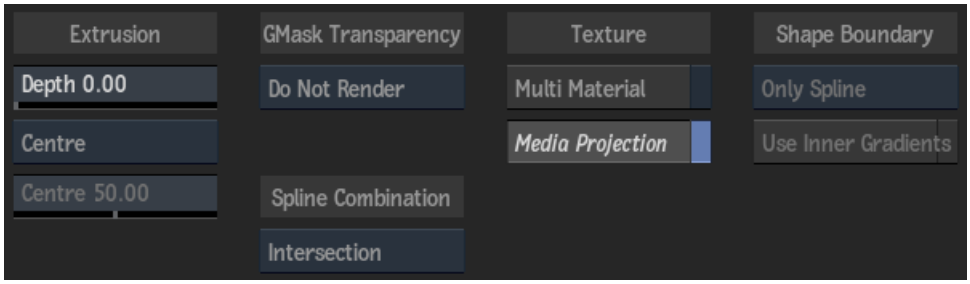
   The 3D Shape node is added to the schematic with its own parent axis. A GMask node (and axis) are also added as children of the 3D Shape node.



2. Draw Shape is automatically selected in the Tools box, and you can create a GMask shape (page 902) (open or closed spline).
3. Double-click the 3D Shape node in the schematic to display the 3D Shape menu, as well as the GMask menu.
4. In the GMask menu's Spline tab, use the Hole button to decide if the spline is seen by the 3D Shape node as a hole or filled.
5. Optional: Select media in the Media list, and click Media Projection in the 3D Shape menu to add a Diffuse (and Axis) node to the 3D Shape node with its mapping type set to Projection.
6. Use the settings in the 3D Shape menu to extrude the spline geometry.

3D Shape Menu Settings

Basics Tab



Depth field Displays the level of depth (thus extruding the selection, making it three dimensional). Editable.

Depth Mode box Select where the extruded geometry is positioned along the Z axis. Select Custom to set the distance from front to back manually.

Depth Centre field Displays the position of the extruded geometry along the Z axis, as a percentage. Editable if Custom is selected in the Depth Mode box.

GMask Transparency box Select how the transparency of an attached GMask is rendered.

Select:	To:
For 3D Shape Only	Render the GMask as if it were linked to the 3D Shape object only. In this case, the GMask blend mode is automatically set to 2 Intersect 1 (Inside) for optimal blending.
Render	Render the GMask globally in the Action scene. In this case, the GMask blend mode is automatically set to Add for optimal blending.
Do Not Render	Not render the GMask in the scene. In this case, certain GMask menu settings are hidden, such as Transparency, Intensity, Smoothing and Blending.

Spline Combination box Select how multiple GMask splines attached to the 3D Shape object are combined.

Option:	Combination Pattern:
Intersection	Splines with the Hole button selected (in the GMask menu) are always rendered as a hole; other splines are rendered according to their relative position. Where two or more splines overlap, these rules apply: An even number of overlaps are rendered as a hole; whereas an odd number are rendered as filled.
Union	Splines with the Hole button selected (in the GMask menu) are always rendered as a hole; other splines are rendered filled, except for their intersection with holes.
Priority	Splines are rendered according to their status (hole or filled, depending on the state of the Hole button), and the GMask Priority list determines the proper combination pattern.

Multi Material button Enable to create an Object Group node for each of the front, back, and extrude of the 3D object. You can then attach a different texture map to apply to the different surfaces.

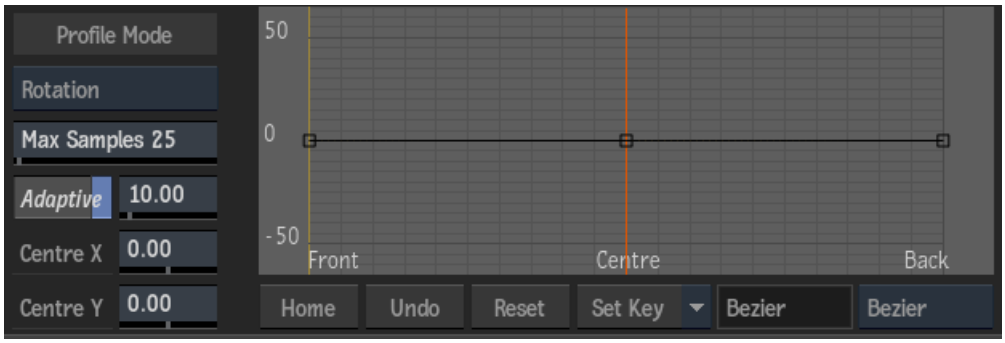
Media Projection button Enable to automatically add a Diffuse (and Axis) node to the 3D Shape node with its mapping type set to Projection.

Shape Boundary box Select how an attached GMask's gradient is extruded. Available when an attached GMask is closed, and has an inner or outer gradient.

Select:	To extrude:
Only Spline	Just the original curve.
Only Gradient	Just the geometry between the gradients.
Spline And Gradient	Everything inside the outer gradient. Set automatically when a gradient is added.

Use Inner Gradients button Enable to include the inner gradient when using the Spline And Gradient option in the Shape Boundary box. Disable if tessellation issues occur when using inner and outer gradients.

Profile Tab



Profile Type box Select which profile curve to display. Different settings may appear depending on the type selected.

Adaptive button Enable to extrude based on an adaptive subdivision, adding smaller segments where there is a higher curvature. When disabled, the depth is divided into Max Samples slices of exactly the same size.

Tolerance field Displays the point at which the extrusion is subdivided. Editable.

Samples field Displays the maximum number of extrusion samples applied to the non-flat curves. Editable.

Centre X field Displays the centre along the X axis that the scale and rotation curves use. You can also adjust the centre settings in the image window by dragging the crosshair. Editable.

Centre Y field Displays the centre along the Y axis that the scale and rotation curves use. You can also adjust the centre settings in the image window by dragging the crosshair. Editable.

Parametric Angle field Displays the slope of the beveling. Editable.

Parametric Position field Displays the relative position of the start of the beveling. Editable.

Parametric Curvature field Displays the roundness or curvature of the beveling. Editable.

Curvature Type box Select the type of curvature to apply to the parametric curve.

Profile Curve Displays the selected profile curve (bevel, scale, or rotation). Move and add points to the curve, as well as adjust the tangent handles to produce different effects with the extruded geometry. If you select a Parametric profile type, a separate read-only curve is displayed.

Parametric Bevel Curve Read-only display of the parametric profile curve, based on the profile settings.

Home button Resets the selected curve viewer to show the whole curve.

Undo button Undoes the last operation for the selected curve viewer.

Reset button Resets the selected curve viewer to the default curve.

Keyframe Option box Select an option for working with keyframes in the profile curve.

Interpolation Status field Displays the interpolation type for the selected area of the curve. Non-editable.

Interpolation Type box Select an interpolation type to define the shape of the profile curve between keyframes.

NOTE The Profile curve for 3D Shape functions the same as that of 3D Text. For more information, see Using the Profile Curve With 3D Text (page 744).

Geometry and UV Map Tabs

The settings in the Geometry (page 711) and UV Map tabs are the same as for the Geom node.

Changing Tessellation Properties

Tessellation is the process of tiling the curves' shapes with polygons. Flame offers different tessellation methods for use with 3D Text and 3D Shape nodes:

Tessellation Type box Select the tessellation type you want to apply to the geometry. More settings appear if you select Delaunay or Medial Axis. Medial Axis is not available when using the 3D Shape node.

- **Standard (GLU)** is the fastest tessellation option, although it is also the least efficient.
- **Delaunay** generates a mesh composed entirely of triangular polygons. This method gives consistent and predictable results, and in particular, it will not give different results if the tessellated objects are rotated.
- **Medial Axis** creates concentric contour lines along the medial axes (averages between the input boundary curves), morphing from one boundary shape to the next.

The Wireframe box is also provided in the Tessellation menu. It is the same setting as in the Geometry tab, but repeated here for ease-of-use.

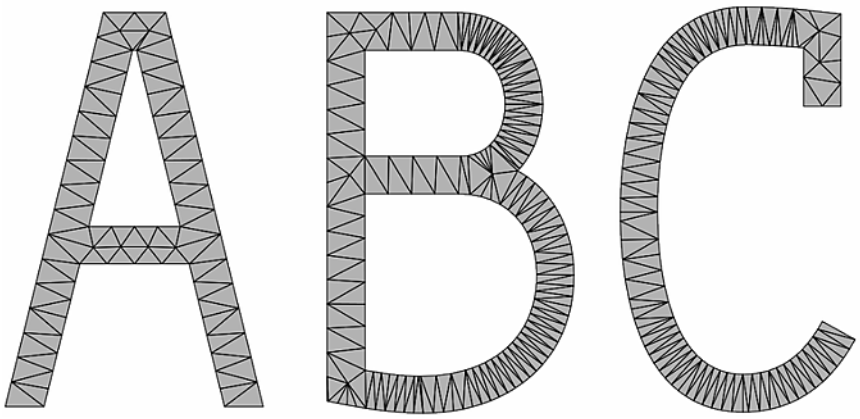
Each method has its own set of options, described in the sections that follow.

Standard (GLU)

The Standard (GLU) tessellation method is the legacy tessellation option; while being very light in its processing requirements, it is also the least efficient and precise. And compared to Delaunay and Medial Axis tessellation methods, it has no options to fine-tune the resulting tessellation.

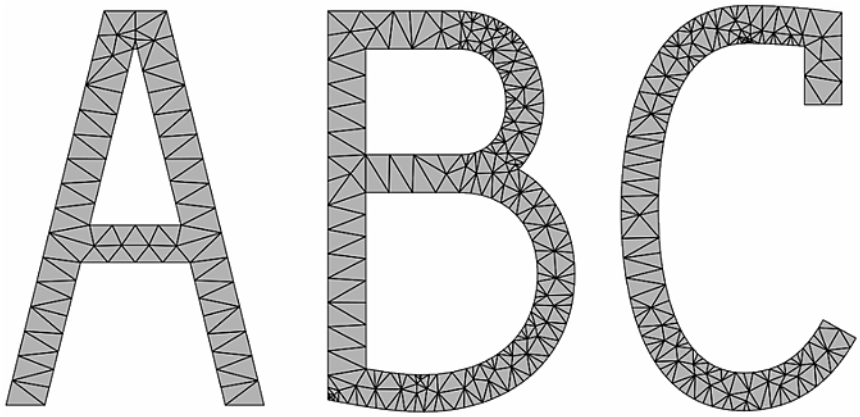
Delaunay

The Delaunay tessellation method (or more precisely, *constrained Delaunay tessellation*) generates a mesh composed entirely of triangular polygons. This method gives consistent and predictable results, and in particular, it does not give different results if the curves are rotated.



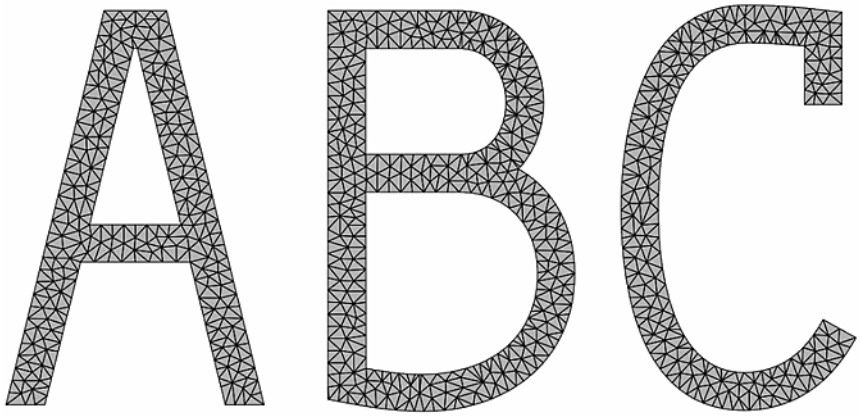
With this method, there are several options for fine-tuning the tessellation further.

Min Angle field Displays the smallest value of angle that the polygons can have. If a triangle contains an angle that is smaller than this value, it gets replaced by better-shaped ones. Eliminating small-angled triangles gives a more uniform shading and is more suited for deformations.



Min Angle = 20

Max Area field Displays the largest value of area that the polygons can have. If a triangle is larger than this value, it gets replaced by smaller ones. This allows the polygon mesh to be deformed more smoothly.



Max Area = 5

Max Vertices field Displays the total number of new vertices that can be added by the Minimum Angle and Maximum Area options. 0 by default.

Use this option as a precaution against accidentally setting the other options to values that would create huge amounts of geometry with long processing times.

It is not recommended that you rely on this option to control the final number of vertices because it can force the tessellation to stop before the process is completed, thereby giving an unpredictable combination of polygon shapes and sizes.

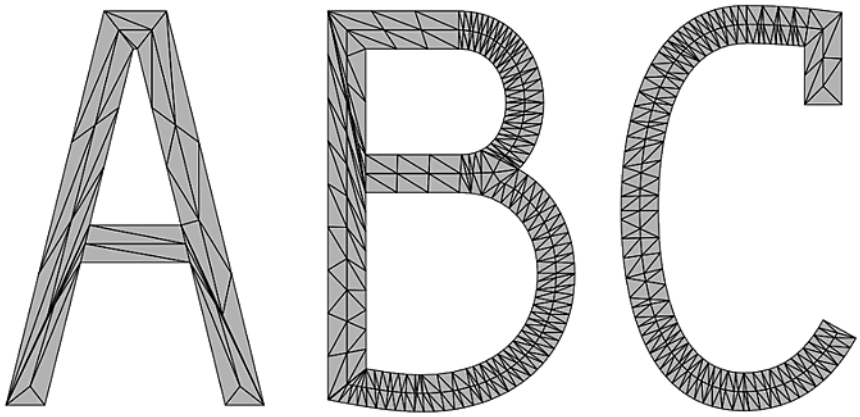
Boundary Split box Select an option to control tessellation along boundaries.

Select:	To:
Free	Allow the boundary edges along the outer contour and inner holes to be split further during tessellation. This is particularly useful for text and other shapes that may contain straight edges that need to be deformed smoothly.
None (Contour Only)	Allow boundary edges along inner holes to be split, but not boundary edges along the outer contour. Note that this may affect the uniformity of the mesh if you enabled Min Angle or Max Area.
None (Contour and Holes)	Prevent any boundary to be split. Note that this may affect the uniformity of the mesh if you enabled Min Angle or Max Area.

Medial Axis

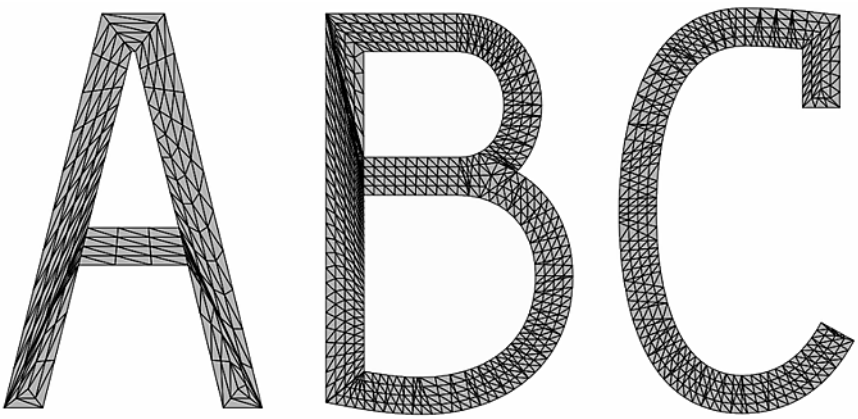
The Medial Axis tessellation method creates concentric contour lines along the medial axes (averages between the input boundary curves), morphing from one boundary shape to the next. This method creates mainly quads with some triangles, so it is well-suited for subdivision surfaces.

NOTE Medial Axis is not available when using the 3D Shape node.



With this method, there are several options for fine-tuning the tessellation further.

Loops field Displays the number of loops used in the tessellation, modified by the Adaptive toggle.



Loops = 2

Adaptive button Enable to set the Loops field as the average number of medial axes drawn and to keep the distance between them fairly constant. Disable to set the Loops field as the exact number of medial axes drawn per boundary (rounded to the nearest integer).

Backtrack Length field Displays the tessellation value at the extremities. Editable.

Set to:	To have:
0	The medial axis intersects boundaries at each point of concavity, which can often create many small triangles especially in sharp extremities.
Positive value	The medial axis does not extend completely to the boundary and the remaining area is tessellated with a fan shape.
Negative value	Sharper embossing effects

Split Edges to Enhance button Enable to add vertices to allow the contour lines to follow the medial axes more accurately. Turn this option off if there are no holes in the geometry; otherwise, there may be shading artifacts along internal curves.

Edge Tessellation box Select an option to control the shape of the polygons.

Select:	To:
None	Have long edges that are not split. This results in fewer polygons and lighter geometry, but the resulting long, thin polygons may not deform well.
Equal on Both Sides	Have a tessellation made of squarer polygons that deform better, at the cost of a heavier tessellation.

Action Shading and Textures

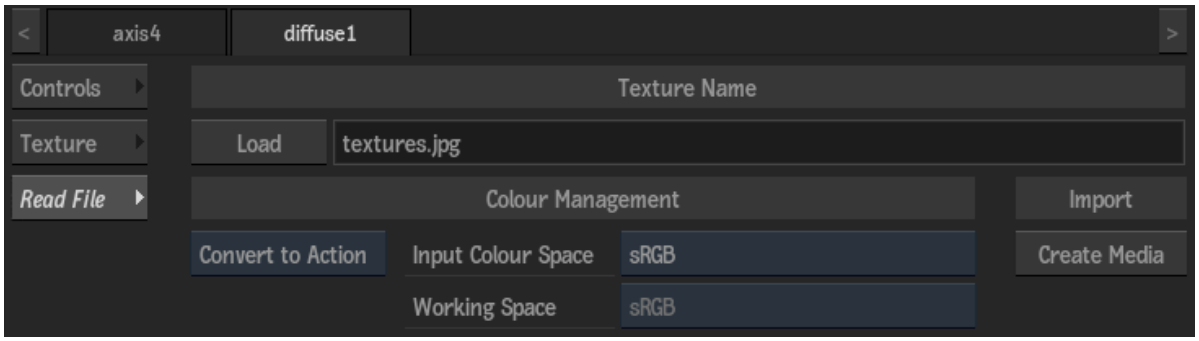
Action uses shaders to compute the colour, lighting, shadows, and other attributes of each pixel or vertex of objects in the scene. Shaders use the processing pipeline of the GPU to accelerate object-specific rendering effects. You can use shaders to control the interaction between surfaces or models and the lights in the scene to contribute to the realism of a material simulated in a texture.

You can use any media to map textures to Action surfaces and geometries, thus adding detail such as depth and reflections to your 3D composites.

NOTE On a Mac system, you are limited to 16 textures per Action object; therefore in large Action scenes, you may notice that some features are not available (shadow casting, for example).

Working With Textures in Map Nodes

You create maps in Action (such as Diffuse and Reflection maps) based on media from the Media list. In some cases, such as with Substance Textures, the media is automatically applied to map nodes. To specify different media as the texture source, select the media in the Media menu, then click Apply. If the media you want to use resides elsewhere on your filesystem, you can use the Read File tab of the map menu to load the texture, and then decide if you want the texture to be managed by Action, or not.



Load button Click to open a browser to select an external texture for the map.

Name field This locked field displays the name of an externally loaded texture.

Create Media button Click to convert the external texture into an Action-managed texture. The Texture Name is removed from this menu, and the texture appears in the Action Media list instead, assigned as a proper layer to the texture in the schematic.

NOTE For information on the Colour Management settings in this menu, see Colour Management in Action (page 584).

Substance Textures and Substance PBR

Using the Substance Texture Node

Use the Substance Texture node to quickly create photorealistic procedural textures using a library of organic, fabric, and material presets. A loaded preset consists of a Material Node, a Substance node, as well as other texture maps and shaders, as needed.

While Flame comes with a selection of Substance Texture presets, you can also use Substance Textures available in Autodesk Maya, Autodesk 3ds Max, or created with the Substance Designer from *Allegorithmic* (any texture file using the *.sbsar* file extension).

TIP You can also create textures with your own input materials, such as photographs, using the Substance Materialize (page 770) node.

To add a Substance Texture to your Action scene:

1. In the schematic, select the surface or 3D geometry to which you want to apply the Substance Texture.
2. Do one of the following:
   - Drag the Substance Texture node from the node bin and place it in the schematic.
   - Double-click the Substance Texture node.

The file browser appears, pointing to the default location of the presets: `opt/Autodesk/presets/<product version>/substance_presets/TEXTURE`. Change the path if you have Substance texture *.sbsar* files stored in a different location.

TIP Switch to Proxies view to see a visual representation of the presets.

3 Navigate and select the Substance Texture you want to load.

The Substance Texture, with applicable maps and shaders, is automatically loaded into Action.

Using the Substance PBR Node

Use the Substance PBR (Physically Based Rendering) node to create physically based textures based on real world presets. A loaded preset consists of a Material Node, a Substance node, as well as other PBS maps, texture maps, and shaders, as needed.

While Flame comes with a selection of Substance PBR presets, you can also add your own Substance PBR assets (obtained from *Substance Database* or *Substance Share*, for example).

NOTE IBL Maps (page 690) work particularly well in a physically based shader workflow. You can connect an IBL node to a camera to have it affect the whole Action scene (and combine PBS and standard shading).

To add a Substance PBR to your Action scene:

1 In the schematic, select the surface or 3D geometry to which you want to apply the Substance PBR.

2 Do one of the following:

- Drag the Substance PBR node from the node bin and place it in the schematic.
- Double-click the Substance PBR node.

  The file browser appears, pointing to the default location of the presets: `opt/Autodesk/presets/<product version>/substance_presets/PBR`. Change the path if you have Substance PBR assets stored in a different location.

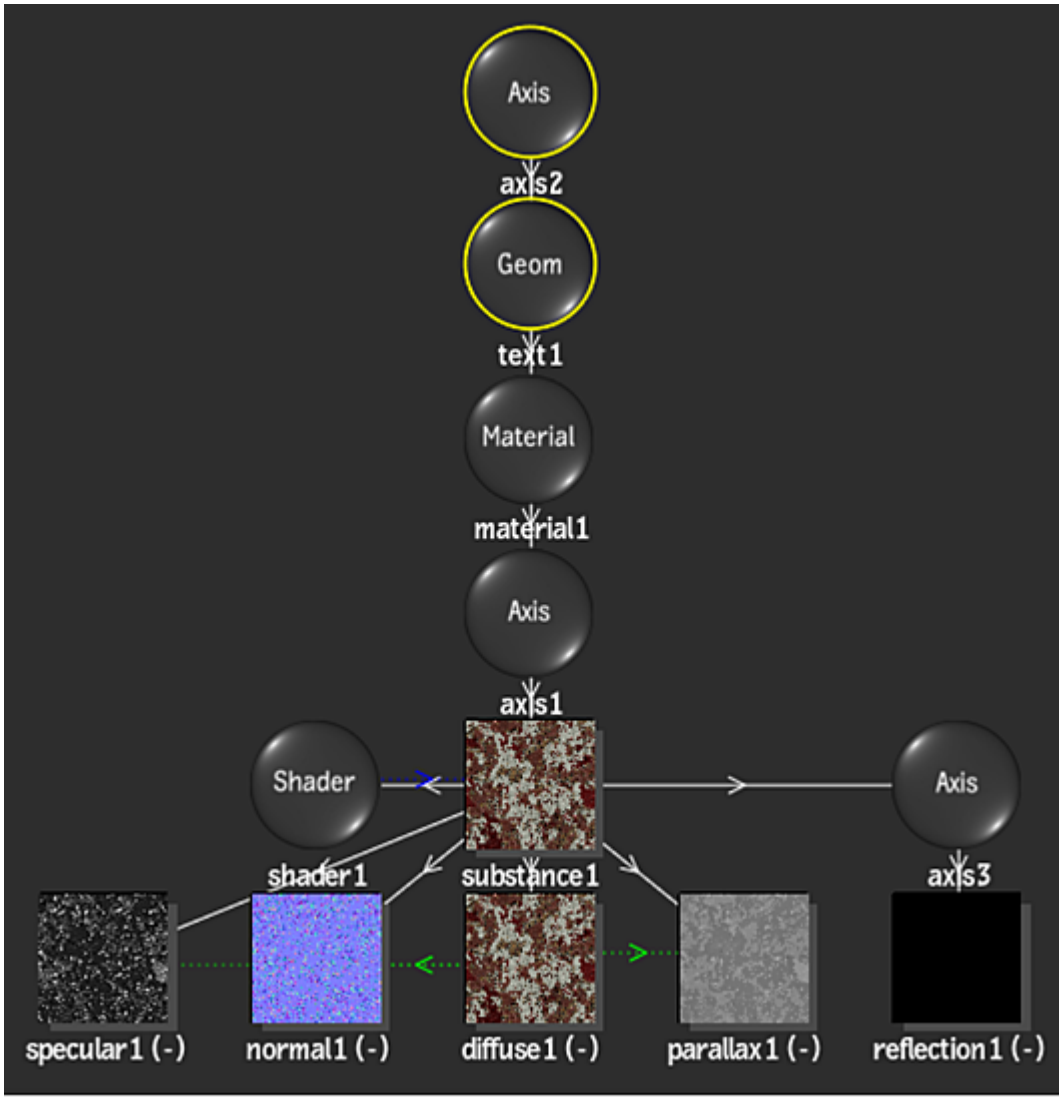
TIP Switch to Proxies view to see a visual representation of the presets.

3 Navigate and select the Substance PBR you want to load.

The Substance PBR, with applicable maps and shaders, is automatically loaded into Action.

Navigating the Schematic with a Substance Texture

When you load a Substance Texture preset into the Action schematic, multiple nodes are added and connected with different types of links, creating a type of Substance group. You can work with the menus of the various nodes (such as the Parallax node) as you would if you had added the object manually.



Keep in mind the following when working with Substance Texture presets in the schematic and menus:

- The Substance node is parented by an Axis node, itself parented by a Material node, and is the parent of some or all of the following map nodes:
  - Specular node — see Specular Mapping (page 792)
  - Normal node — see Normal Mapping (page 788)
  - Diffuse node — see Diffuse Mapping (page 800)
  - Parallax node — see Parallax Mapping (page 784)
  - Displacement node (locked to Hardware Displacement) — see Displacement Mapping (page 779)
  - Emissive node — see Emissive Mapping (page 796)

  Media is automatically applied to these mapping nodes, and cannot be changed.
- The Substance node is also parented to a Reflection node, with its own axis. You can apply media to the Reflection node. See Reflection Mapping (page 804).
- A Shader node is parented from the Substance node. Lighting links (blue dotted lines) are applied from the Shader node to the Substance node and the originally selected surface or geometry. Shader node

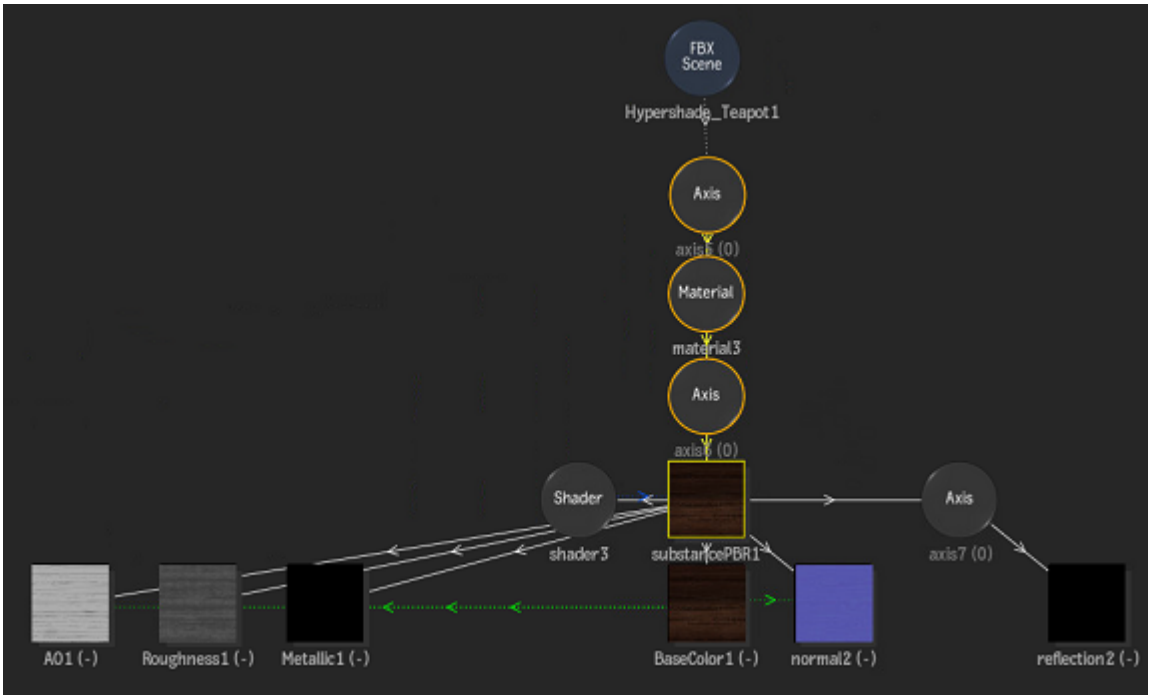
settings override Material node settings (at the exception of Transparency and Ambient settings). See Using the Shader Node (page 771).

- Some presets require you to add input media to feed into the Substance node and attached maps to create a plausible effect. For example, the *Broken_Glass* preset in the *Glass* category requires one input node. See Substance Textures with Inputs (page 766).
- All Map parameters are loaded with default values based on the loaded Preset. The Repeat Mode is automatically set to Tile Repeat on all attached maps. Reflection map parameters sometimes have an expression automatically set on its effect or softness values.
- The Diffuse node acts as an originating duplicate object to the present Specular, Normal, Emissive, and Parallax nodes (green dotted lines). Therefore, any common settings applied to one of these map nodes are applied to all map nodes. The common settings are found in the Texture tab of any of the map menus. See Mimicking, Copying, and Duplicating Objects (page 578).
- Since the Substance preset requires all of the loaded objects, and the parenting, lighting, and duplicate links to function correctly, you are unable to delete any of the objects separately, or break any of the links.
- Some presets load and use only the maps that are needed for that preset. For example, the *Defocused_Light* preset in the *Abstract* category includes only an Emissive map that blends in additive mode with the diffuse colour of the attached surface or geometry. In this case, the preset works best if the diffuse colour in the Geometry or Image menu is set to black.
- You cannot parent an object from any of the Substance objects. You can parent the Material node to another object to substitute for one of the Substance maps. For example, you can add a Diffuse map and connect it to the Material node, and then hide the Substance Diffuse map so that it is no longer applied to the Substance effect.
- If you select a surface or geometry before adding the Substance Texture preset, the connection to the Material node of the Substance node, and the lighting link from the Shader node are automatically applied. If you do not select a surface or geometry first, you have to make these connections manually for the Substance node to function correctly.

TIP Use the Hide Texture Axis keyboard shortcut (`Space+H`) to hide texture-related axes from the Action scene. Hiding these axes reduces the number of axes displayed in the viewport, and makes it easier to pick the correct axis to transform an object.

Substance PBR Schematics

A Substance PBR schematic is similar to a Substance Texture schematic. It consists of a Material node, a Shader node, a Substance node containing all the relevant preset controls, and the corresponding texture maps generated by the Substance engine. Some of these texture maps correspond to PBS Map (page 813) nodes (in this example the BaseColor, Metallic, Roughness, and AO nodes are PBS Maps):



- The Shader node defines the shading of the PBS texture maps generated by the Substance engine.
- In a PBS workflow, some Physically Based shader settings or PBS map settings override the Material node settings. In this case, specular and shine settings in the Material menu do not have any effect. If transparency is not needed, the Material node is not required, and can be removed.

Substance Textures with Inputs

Some semi-transparent presets, such as those with water or glass, may require one or more media inputs to be added manually. This is done so that the Substance node and attached maps can correctly create a plausible effect. When you load a Substance Texture preset that requires media input, one or more input nodes are attached to the Substance node in the schematic.

NOTE If you accessed Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.

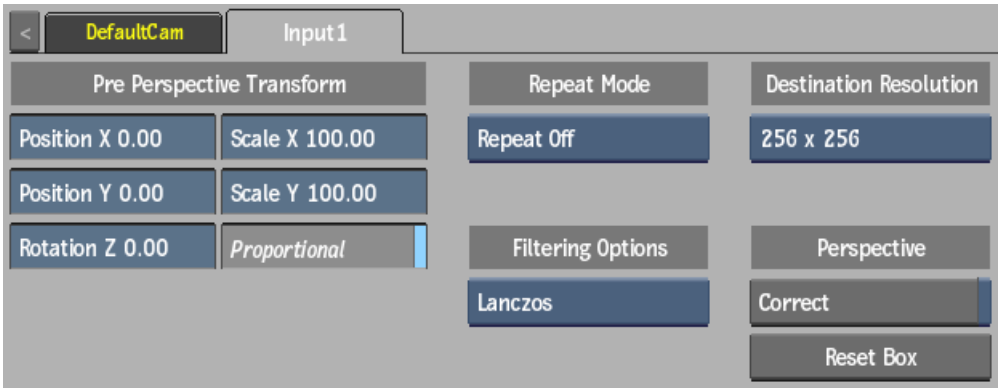
To specify media as the input source:

1. Select the Input node in the schematic.
2. Select the appropriate media in the Media list.
3. Click Apply.

   NOTE If media is selected in the Media list prior to adding a Substance Texture preset that requires an input, the media is automatically added to the input. You can change the input by following the steps above. For presets with more than one input, only the first input is added automatically.

4. Double-click the Input node in the schematic to access the Input menu.

Input Menu Settings



Position X field Displays the X-axis offset applied to the clip. Editable.

Position Y field Displays the Y-axis offset applied to the clip. Editable.

Rotation field Displays the rotation offset applied to the clip. Editable.

Scale X field Displays the X-axis scaling offsets applied to the clip. Editable.

Scale Y field Displays the Y-axis scaling offsets applied to the clip. Editable.

Proportional button Enable to affect the Scale fields proportionally.

Repeat Mode box Select how the input media is repeated (after Transform settings are applied).

Filter box Select the type of filtering to apply to the input media.

Destination Resolution box Select the square destination resolution of the input media.

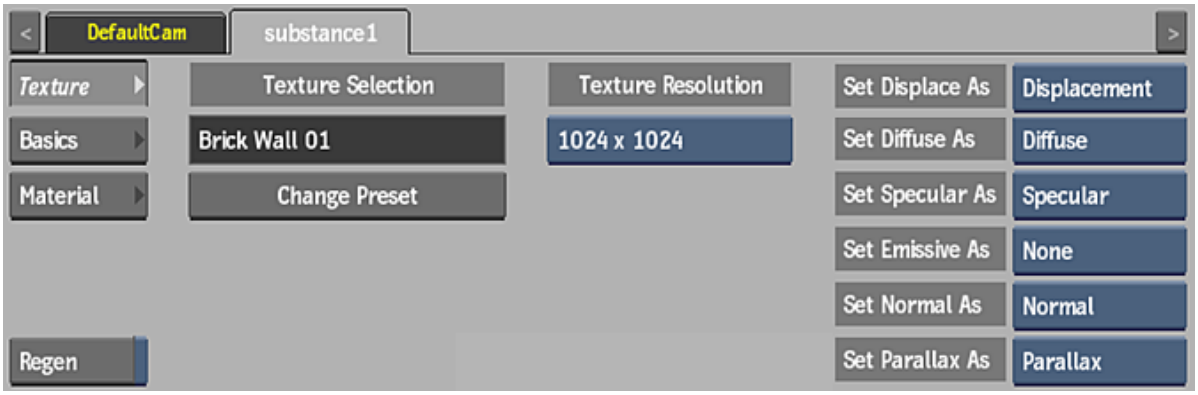
Perspective Correct box Enable to apply perspective transformations to the input. Switch to Object view (F8), and interactively align the perspective of what area you want to include in the input image.

Reset Box button Click to reset the Perspective box in the Object view.

Substance Menu Settings

Double-click the Substance node to access the Substance menu. The Substance menu is divided into three tabs.

The Texture tab contains settings related to the selection and resolution of the texture.



Preset Name field This locked field displays the name of the current preset.

Change Preset button Click to open the file browser to select a different preset.

Texture Resolution box Select the resolution of the pattern.

Lock Width and Height button Not shown. Enable to lock the texture resolution width and height. Disable to be able to select different width and height values. Available when using third-party Substance Textures or the Substance Materialize node.

Texture Resolution Width box Not shown. Select the width resolution of the pattern. Available if Lock Width And Height is disabled.

Texture Resolution Height box Not shown. Select the height resolution of the pattern. Available if Lock Width And Height is disabled.

Set Displace As box Select a map to be used as the Displace pass. This is useful if you want to rewire one of the maps in the substance (such as the Bump map), that might be invisible, to be seen explicitly.

Set Diffuse As box Select a map to be used as the Diffuse pass. This is useful if you want to rewire one of the maps in the substance (such as the Bump map), that might be invisible, to be seen explicitly.

Set Specular As box Select a map to be used as the Specular pass. This is useful if you want to rewire one of the maps in the preset (such as the Bump map), that might be invisible, to be seen explicitly.

Set Emissive As box Select a map to be used as the Emissive pass. This is useful if you want to rewire one of the maps in the preset (such as the Bump map), that might be invisible, to be seen explicitly.

Set Normal As box Select a map to be used as the Normal pass. This is useful if you want to rewire one of the maps in the preset (such as the Bump map), that might be invisible, to be seen explicitly.

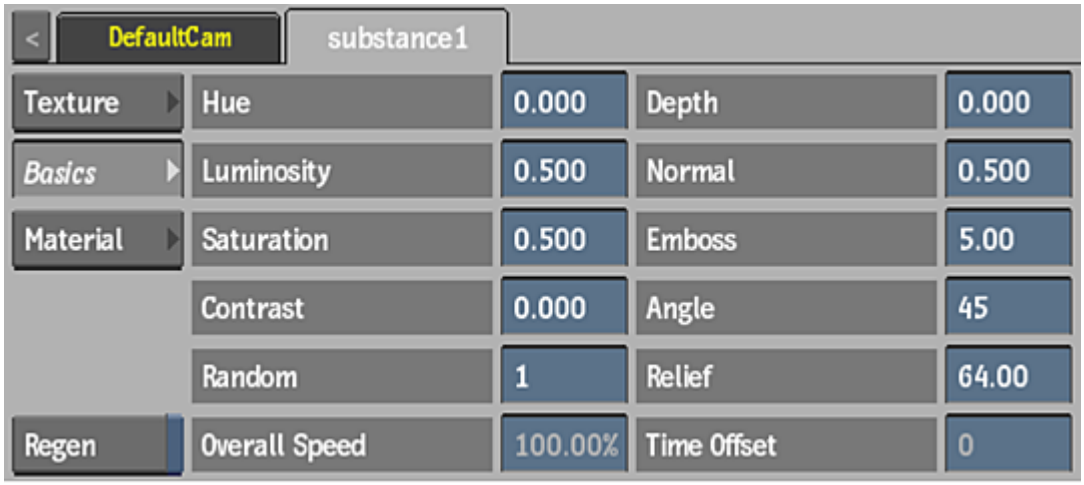
Set Parallax As box Select a map to be used as the Parallax pass. This is useful if you want to rewire one of the maps in the preset (such as the Bump map), that might be invisible, to be seen explicitly.

TIP Use the Output Rewire boxes to define how the images making up the Subtance are wired to each texture map. If a texture map is not used by the selected Substance, the Output Rewire box is set to None. You can also rewire a texture map to None to disable it and remove it from the Action Schematic (the map is not deleted, just disabled).

Regen button Enable to dynamically refresh the image as changes are made to the settings. If you notice a slowdown in interactivity, disable Regen.

NOTE The Regen button is available from all Substance menu tabs.

The Basics tab contains settings common to all Substance presets. Settings that are not applicable to a specific preset are greyed out.



Hue field Displays the colour range of the texture. Editable.

Luminosity field Displays the brightness level of the texture. Editable.

Saturation field Displays the level of colour purity of the texture. Editable.

Contrast field Displays the gradations between the light and dark areas of the texture. Editable.

Random field Displays the random seed value of the generated texture. Editable.

Depth field Displays the attenuation of depth of the texture. Editable.

Normal field Displays the attenuation of the normals of the texture. Editable.

Emboss field Displays the level of enhanced details of the texture. Editable.

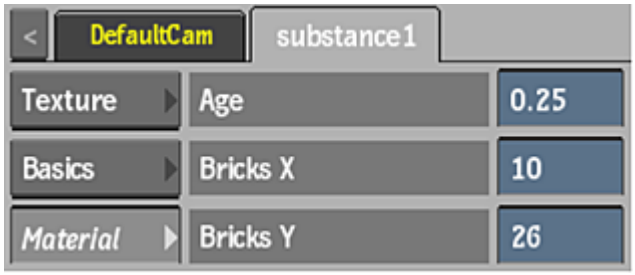
Angle field Displays the angle of diffuse in relation to the level of Emboss applied to the texture. Editable.

Relief field Displays the frequency of surface detail between attached Parallax and Displacement maps. Editable.

Overall Speed field Displays the rate at which the animation plays. Editable.

Time Offset field Displays the start point of the animation. With a value of 0, the animation starts at frame 1. With a value of 100, the animation begins as if it has been generating for 99 frames. You cannot animate this field. Editable.

The settings in the Material tab vary depending on the preset chosen.



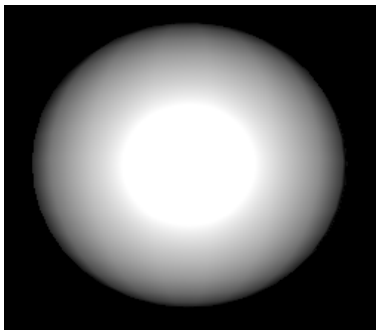
For example, the Material settings for the BrickWall01 preset allow you to age the bricks, and alter the number of the bricks on the X and Y axes.

Some presets include a Flow field in the Material settings. Animate the Flow field to create a motion behaviour specific to the preset.

Substance Materialize

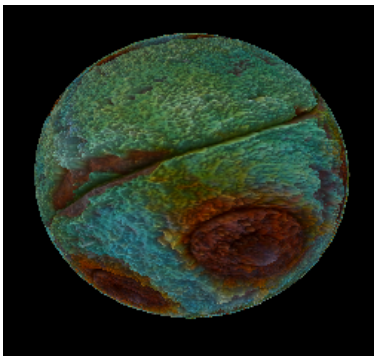
The Substance Materialize node allows you to easily create realistic material textures based on your own image inputs. When you add a Substance Materialize node to the Action schematic, you'll notice the familiar Substance tree of maps and shaders. Simply specify your input image, as well as optional Details, Grunge, and Relief maps, and use the menu settings to fine-tune the results.

Substance Materialize is compatible with *Allegorithmic's Bitmap2Material (B2M) version 2.2*.



Initial sphere geometry

Input image: Single-frame .tif photo of rusty screws in a wall.



Result after applying inputs and various Substance settings.

NOTE If you access Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.

Substance Materialize Usage Tips

- A **Bitmap_Diffuse** map is added to the schematic tree when creating a Substance Materialize. By default, this uses the same input as your input image, but you can change it.
- Three other optional inputs are available in the schematic tree for a Substance Materialize:
  - **Bitmap_Details_Optional**: an input image file allowing you to specify fine details in your result.
  - **Bitmap_Relief_Optional**: an optional grayscale image file you can use to enhance elevation. Black represents the lowest elevation on the map, while white represents the highest.
  - **Bitmap_Grunge_Optional**: allows you to alter the look of your material, mainly by adding some dust and cracks on it. You can either chose one the 15 grunge maps embedded into the Substance Materialize node (see the Grunge parameter settings in the Substance node menu), or use your own image in this input. You can see the 15 grunge maps on the *B2M Help site*.

    NOTE Make sure that you set a Diffuse Opacity other than 0 in the Grunge settings to be able to see the effect of the grunge map.

- For each input, you also can use the Input menu (page 767) (as with Substance Textures with inputs), and the Object view (`F8`). The Object view lets you visualize changes that you make to the Input menu settings. In the Object view, you can also interactively align the perspective of the area you want to include in the input image.
- The Substance menu is somewhat different than if you added a Substance Texture preset. The Texture tab is similar, but the Parameters drop-down list allows you apply settings in a number of different areas.
  - Specular
  - Shape Recognition
  - Relief

- Lighting
- Grunge
- Global
- Diffuse
- Detail
- AO

See the tooltips (or the *B2M Help site*) for help with each setting.

Adding a Substance Materialize Node

To add a Substance Materialize node:

1. In the schematic, select the surface or 3D geometry to which you want to apply the Substance Materialize.
2. In the Media list, select the image that you want to use as the input. This image is also used as the default Bitmap Diffuse input.
3. Do one of the following:
   - Drag the Substance Materialize node from the node bin and place it in the schematic.
   - Double-click the Substance Materialize node.

   The Substance tree, with applicable maps, shaders, and inputs appears in the Action schematic.
4. To specify different media as the input source, select the input node in the schematic, select the media in the Media menu, then click Apply.
5. Optional: If you have Details, Grunge, or Relief maps, you can also use them to enhance your Substance material. Simply apply them from the Media list in the same manner as with the Input image.
6. Double-click any of the nodes in the schematic to access its menu.

Using the Shader Node

You can add a shader to your scene to apply to all objects in the scene. You also have the option of selectively applying a specific shading algorithm, or turning shading off completely. You can use multiple shaders in an Action scene, but only one shader can be applied to each object. Even if no shaders are present in the schematic, Action uses an implicit default shader that applies to the whole scene.

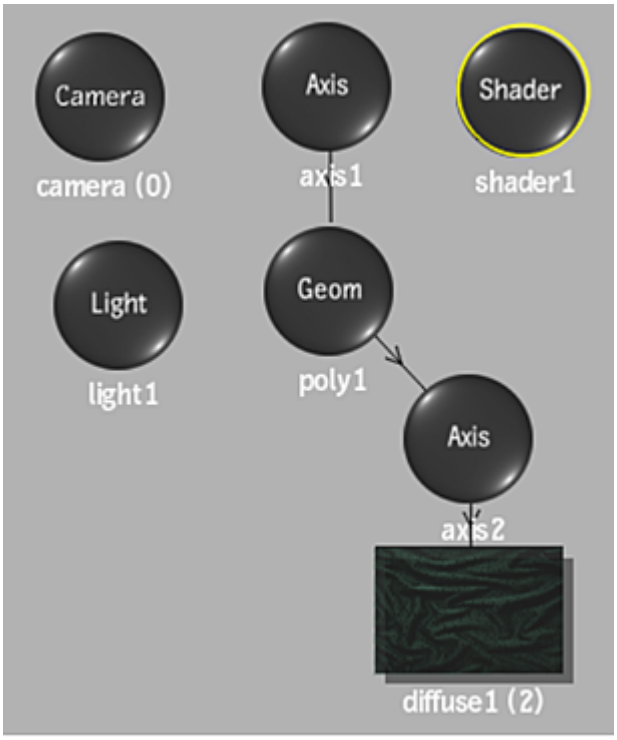
When working with Substance (page 762) nodes, a preset schematic, with an attached Shader node, is automatically created.

NOTE The shader node is not a typical Action object — its appearance in the schematic represents the ability to selectively include or exclude shading from objects in the scene.

To add a shader to the scene:

1. Do one of the following:
   - Drag the Shader node from the node bin and place it in the schematic.
   - Drag the Shader node from the node bin and place it in Result view.
   - Double-click the Shader node. You do not need to be in Schematic view to add a node in this manner.

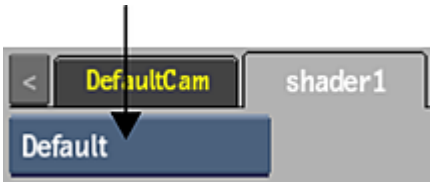
A shader is added to the scene. If you select a surface or 3D geometry in the scene before adding the Shader node, a shader inclusion link is automatically applied. You can also apply inclusion and exclusion links manually. See Applying a Selective Shader (page 772).



Unlike many objects, a shader is not a confined object in the perspective space, and therefore is added without an axis.

2 To display the Shader menu, double-click the selected shader in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).

3 From the Shader Type box, select a shading algorithm, or turn shading off.



See Shader Types and Settings (page 775) for more information on the shader algorithms, and their settings.

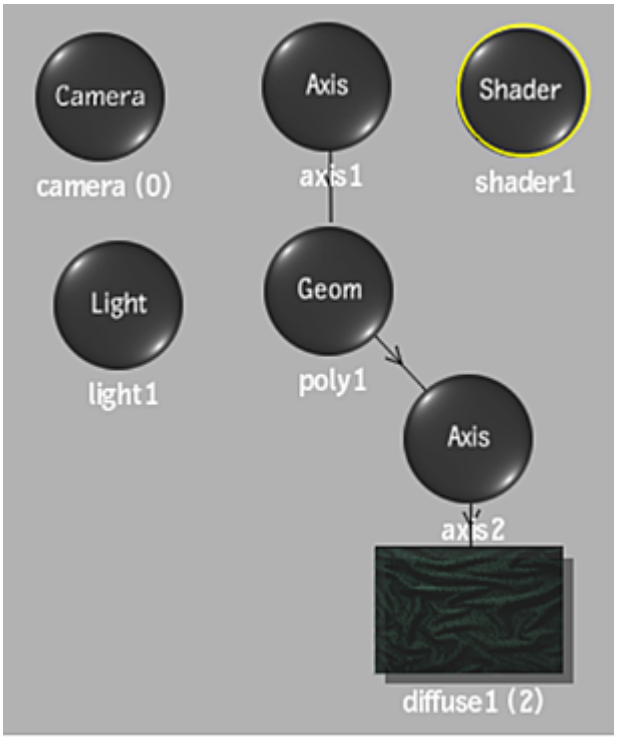
Applying a Selective Shader

If you select a surface or 3D geometry in the scene before adding a Shader node, a shader inclusion link is automatically applied; otherwise, the shader is applied to all objects. You can also apply inclusion and exclusion links manually. You may want a shader to only affect an individual or specific group of objects, or prevent a shader from affecting an individual or specific group of objects.

To apply selective shading:

1 Add a shader to the scene.

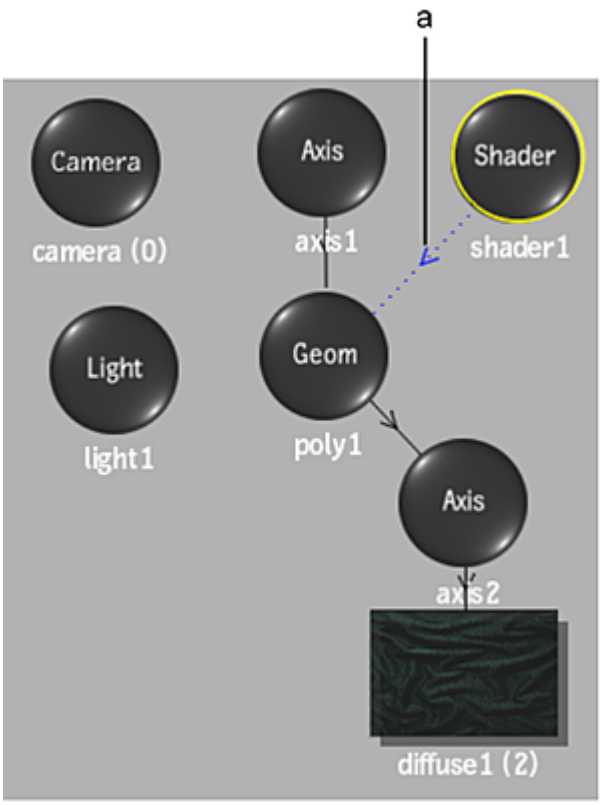
All objects are affected.



2 Select Light Link from the Tools box.

3 To affect only a selected object, click the Shader node, and drag it to an object you want affected.

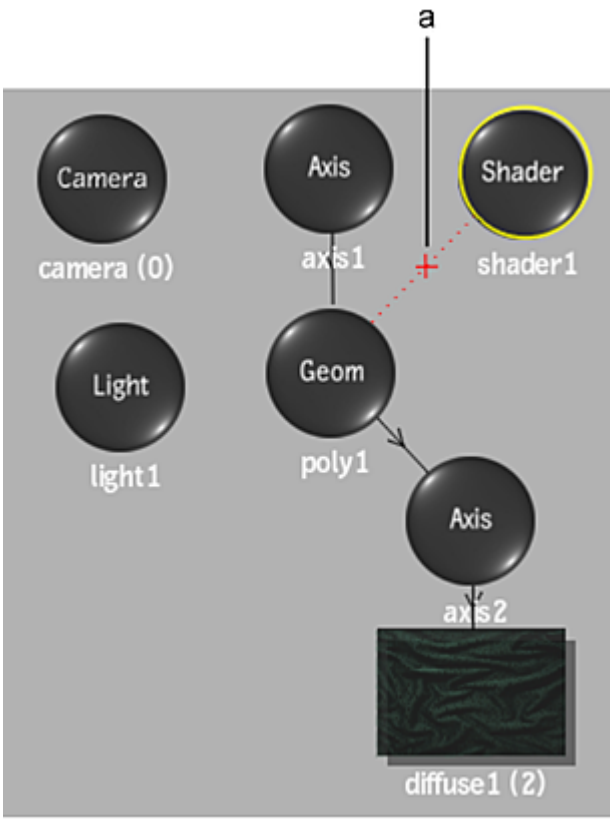
The selected object is connected to the shader by a blue dotted line with an arrow, and only the selected objects are affected.



(a) Shader inclusion link

4 To exclude an object, hold the `Alt` key while clicking and dragging from the shader to the object you do not want affected.

Excluded surfaces are connected to the shader by a red dotted line with an "X", and they are not affected by the shader.



(a) Shader exclusion link

NOTE To remove the inclusion or exclusion link, click and drag over the line that connects the Shader node to the object (while in Light Link mode).

Shader Types and Settings

Depending on the look you are trying to apply to an object or the scene, you can select from among different shader algorithms. The differences between the shader types are sometimes subtle, as they build upon the same algorithms, such as Fresnel or the Oren-Nayar diffuse model.

Some of the settings in the Shader menu change depending on the shader type chosen. See the following sections for the specific settings for each shader type.

Shader Type box Select a shading algorithm, or turn shading off.

Physically Based Shader

Use the Physically Based shader (PBS) to accurately represent real-world materials. In a PBS workflow, some Physically Based shader settings override the Geom, Material, or Surface node settings. For example, specular and shine settings in the Material or Geom menu do not have any effect.

NOTE Some of these same settings are available as shader types from the PBS Map node (page 813) (by using the PBS Map node, you can affect the shader on a per pixel basis).

Roughness field Displays how the light reflects from the surface, based on the roughness of the material. Use a lower value for pure specular, or a higher value for high dispersion. Editable.

Metallic field Displays the relative weight between the dielectric and metallic BRDF model, where 0 is pure dielectric and 1 is metallic. Editable.

SubSurface field Displays the diffuse part to approximate a subsurface scattering effect for the dielectric model (has no effect on the metallic model). Editable.

Specular Level field Displays the Fresnel F0 specular reflectance at normal incidence for the dielectric model (for the metallic model the F0 follows the Base Color of the material). Editable.

Specular Tint field Displays the contribution of the Base Color (Diffuse) to tint the specular for the dielectric model (has no effect on the metallic model). Editable.

Anisotropic field Displays the distortion of the specular lobe, according to the tangent and bi-normal angles. Editable.

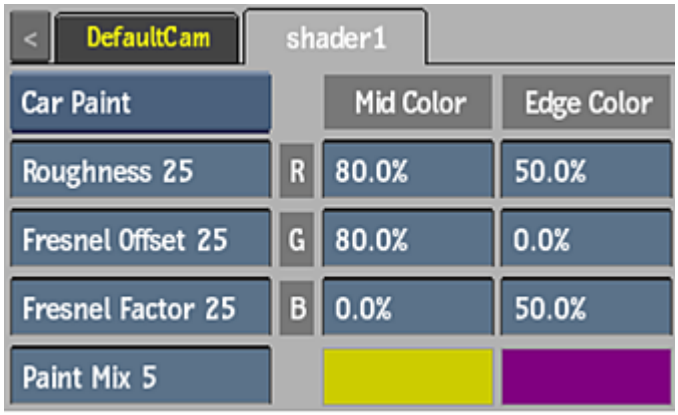
Anisotropic Angle field Displays the rotation of the distortion of the anisotropic specular lobe. Editable.

Emissive Intensity field Displays the intensity of the light emitted by the material. Editable.

Intensity Colour pot Displays the colour of the emitted light by material. Click to change the colour.

Car Paint Shader

Use the Car Paint shader to blend between two colour tones, based on the viewing angle and the normal of the object. This shader includes Cook-Torrance shaders and Fresnel controls for the specularity.



Roughness field Displays the shape of specularity of the shader.

Fresnel Offset field Displays the total amount of specular light.

Fresnel Factor field Displays the amount of specular light at grazing angles.

TIP To cancel out the Fresnel effect altogether, set the Fresnel Offset to 0 and the Fresnel Factor to 100.

Paint Mix field Displays the viewing angle of the normal that occurs between parallel (mid colour) and perpendicular (edge colour). Editable.

Red Mid Colour field Displays the red mid colour value. Editable.

Green Mid Colour field Displays the green mid colour value. Editable.

Blue Mid Colour field Displays the blue mid colour value. Editable.

Mid colour pot Diplays the mid colour. Editable.

Red Edge Colour field Displays the red edge colour value. Editable.

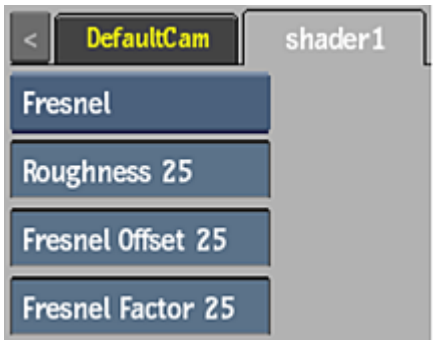
Green Edge Colour field Displays the green edge colour value. Editable.

Blue Edge Colour field Displays the blue edge colour value. Editable.

Edge colour pot Displays the edge colour. Editable.

Fresnel Shader

The Fresnel shader contains only Fresnel controls for the specularity.



Roughness field Displays the shape of specularity of the shader.

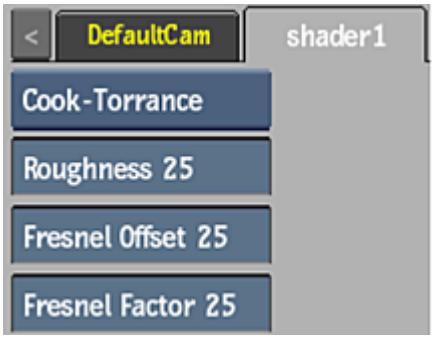
Fresnel Offset field Displays the total amount of specular light.

Fresnel Factor field Displays the amount of specular light at grazing angles.

TIP To cancel out the Fresnel effect altogether, set the Fresnel Offset to 0 and the Fresnel Factor to 100.

Cook-Torrance Shader

Use the Cook-Torrance shader for high specularity materials, such as metals or shiny plastics. This shader includes Fresnel controls for specularity.



Roughness field Displays the shape of specularity of the shader.

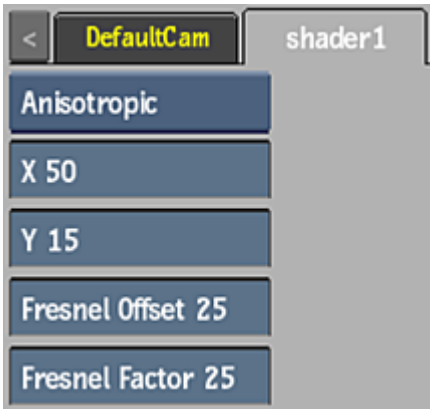
Fresnel Offset field Displays the total amount of specular light.

Fresnel Factor field Displays the amount of specular light at grazing angles.

TIP To cancel out the Fresnel effect altogether, set the Fresnel Offset to 0 and the Fresnel Factor to 100.

Anisotropic Shader

Use the anisotropic shader to control the specular effect of the highlights.



X Roughness field Displays the shape of specularity of the shader along the X axis. Editable.

Y Roughness field Displays the shape of specularity of the shader along the Y axis. Editable.

Fresnel Offset field Displays the total amount of specular light.

Fresnel Factor field Displays the amount of specular light at grazing angles.

TIP To cancel out the Fresnel effect altogether, set the Fresnel Offset to 0 and the Fresnel Factor to 100.

Displacement Mapping

Use displacement mapping to create a 3D model from a 2D surface. The values of a selected colour channel in the displacement source clip are used to create a displacement map. Displacement mapping uses the media's matte clip, so you can turn the matte on or off to get the desired effect.

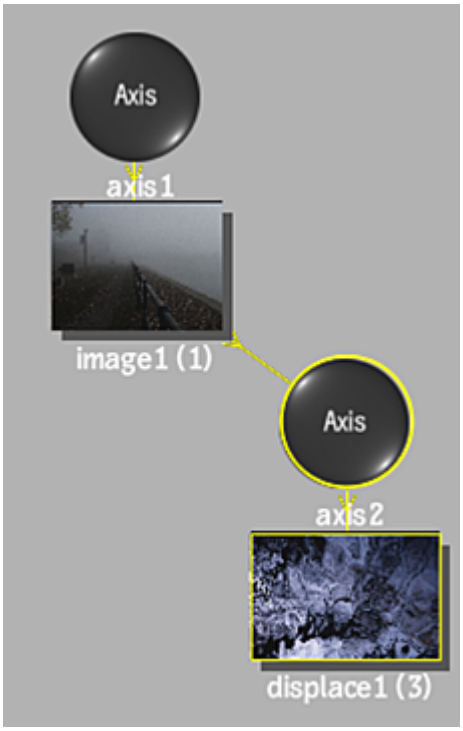
You have the choice of applying hardware or software displacement mapping. Hardware displacement mapping is GPU-accelerated and allows you to create a normal map if none exists. Software displacement mapping displaces the pixels of the surface along the positive or negative X, Y, and/or Z axes. Hardware Displacement is faster than software displacement, especially when using low resolution values in an image surface (high polygon count).

NOTE If the Displace node is attached to an imported 3D Geometry, you may need to select a UV Mapping mode other than Default in the Geometry menu for the displace pattern to have an effect on the geometry. See NO LABEL .

To add a displacement map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the displacement.
2. Click Media.
3. In the Media menu, select the media you want to use for the displacement.
4. Do one of the following:
   - Drag the Displace Map node from the node bin and place it in the schematic.
   - Drag the Displace Map node from the node bin and place it where you want it in Result view.
   - Double-click the Displace Map node. You do not need to be in Schematic view to add a node in this manner.

   The displace object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the displace node indicates the media used for the displacement.

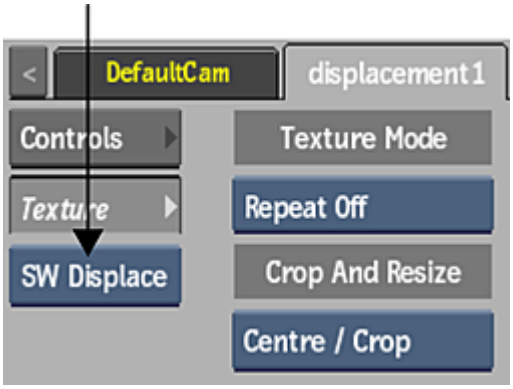


To specify different media as the source, select the media in the Media menu, then click Apply, or use the Read File tab to manage the texture media (see Working With Textures in Map Nodes (page 762)).

NOTE If you accessed Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.

5 Double-click the Displace node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).

The Displacement menu appears. You can choose between Hardware Displacement or Software Displacement using the Displacement Type box.



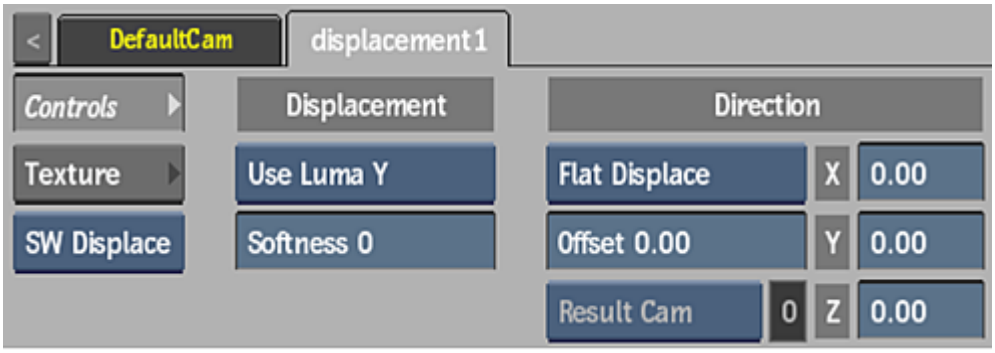
Software Displacement Menu Settings

Displacement Type box Select whether to use hardware or software displacement mapping.

Regen button Enable to dynamically refresh the image as changes are made to the menu settings.

The Software Displacement menu is divided into two tabbed sections: Controls and Texture.

Controls Tab



Channel box Select a colour channel to calculate the displacement map.

Softness field Displays the level of rounding off, or softening of the spikes that result from colour values in the image that vary from pixel to pixel in the displacement map. Editable.

Softness rounds the edges of the displacement. The larger the softness, the smoother the displacement. Softness also affects rendering; the larger the softness, the longer it takes to render.

Displace Direction box Select the direction in which a displace occurs when a displacement map is attached to a surface. Geometries parented to a displacement map always use Normal Displace as the direction.

Select:	To Displace:
Flat Displace	In the X, Y, and Z directions.
Normal Displace	Bilinear and bicubic surfaces according to their normals.
Camera Displace	In the direction of the camera selected in the Displace Camera box.

Offset field Displays the offset to the displacement of X and Y. Editable.

Displace Camera box Active only when Camera Displace is selected in the Displace Direction box. Select which camera to take into account when using camera displacement.

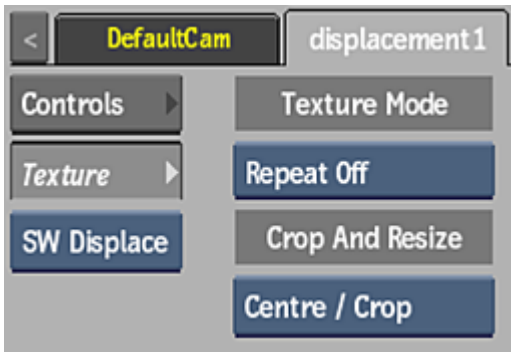
Displace Camera field Displays the active displace camera number. Non-editable.

Displacement X field Displays the amount of displacement in pixel units along the X axis. Editable.

Displacement Y field Displays the amount of displacement in pixel units along the Y axis. Editable.

Displacement Z field Displays the amount of displacement in pixel units along the Z axis. Editable.

Texture Tab



Repeat mode box Select how the map pattern is repeated on the surface.

Fit Method box Select a fit method option to be applied to the map.

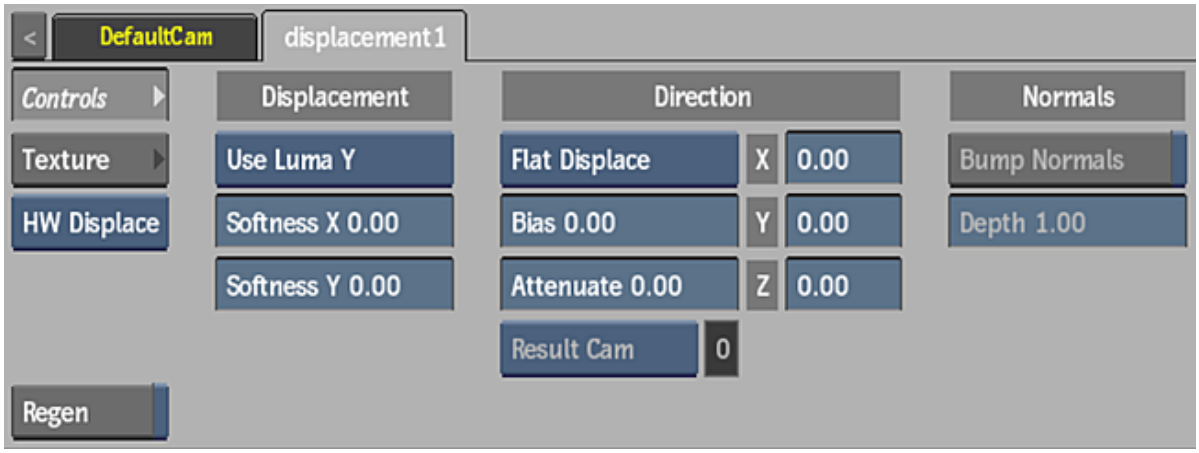
Hardware Displacement Menu Settings

Displacement Type box Select whether to use hardware or software displacement mapping.

Regen button Enable to dynamically refresh the image as changes are made to the menu settings.

The Hardware Displacement menu is divided into two tabbed sections: Controls and Texture.

Controls Tab



Channel box Select a colour channel to calculate the displacement map.

Softness X field Displays the amount of X-axis blur applied to the map. Editable.

Softness Y field Displays the amount of Y-axis blur applied to the map. Editable.

Displace Direction box Select the direction in which a displace occurs when a displacement map is attached to a surface. Geometries parented to a displacement map always use Normal Displace as the direction.

Select:	To Displace:
Flat Displace	In the X, Y, and Z directions.
Normal Displace	Bilinear and bicubic surfaces according to their normals.
Camera Displace	In the direction of the camera selected in the Displace Camera box.

Bias field Displays the distance between the real surface of the object and the perceptual ground (zero level) of the texture. Editable.

Attenuate field Displays the level of amplitude of the effect caused by the displacement map texture. Editable.

Displace Camera box Active only when Camera Displace is selected in the Displace Direction box. Select which camera to take into account when using camera displacement.

Displace Camera field Displays the active displace camera number. Non-editable.

Displacement X field Displays the amount of displacement in pixel units along the X axis. Editable.

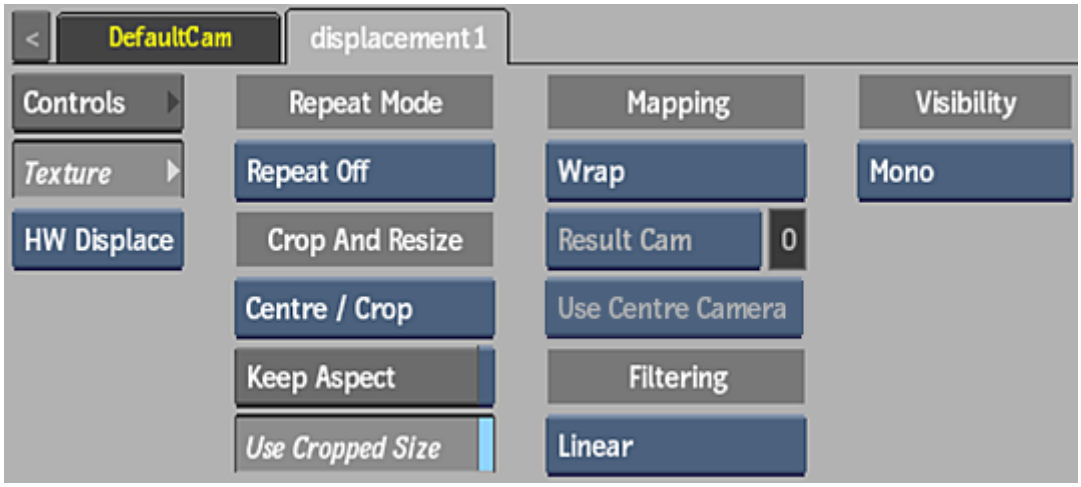
Displacement Y field Displays the amount of displacement in pixel units along the Y axis. Editable.

Displacement Z field Displays the amount of displacement in pixel units along the Z axis. Editable.

Bump Normals button Enable to allow the map to create a normal map for enhanced lighting effects. Used only if no normal map exists for the object using the displacement texture. The availability of this setting is dependant on your graphics card.

Depth field Displays the amount of Z scale applied to the generated normals. Higher values attenuate the effect toward the normals of the parent surface. Editable.

Texture Tab



Repeat mode box Select how the map pattern is repeated on the surface.

Fit Method box Select a fit method option to be applied to the map.

Keep Aspect button Enable to preserve the aspect ratio of non-square pixels (not available for the Fill fit method).

Use Cropped Size button Enable to replace the map with the cropped size of the media. Disable to use the cropped media as is.

Mapping box Select the type of texture mapping.

Wrap Wrap mapping completely envelops the 3D model with the texture map according to the object's texture coordinates. To use this option, you must import a model that has its own texture coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu.

Plane Planar mapping applies the map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the texture values. Planar mapping positions the lower-left corner of the map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

Camera box Specify which camera's FOV to take into account when using perspective or projection mapping.

Camera field Displays the active perspective or projection camera number.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the map.

Camera Type box Select the camera type visibility for the map.

Parallax Mapping

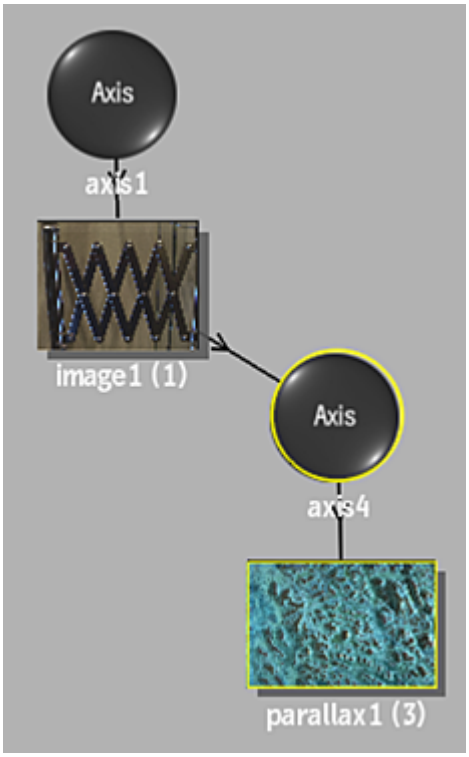
Parallax mapping can generate a visual result similar to that of a displacement map without actually displacing polygons. Parallax mapping uses the height map (similar to a displacement map) to determine how the object texture should be warped in order to simulate shading and occlusion effects, without actual polygonal displacement.

Because a parallax map does not actually change the polygons of the object, there are inherent limits to the illusion — looking at the object from the side will reveal the cheat. It is best used on surfaces and geometries where you do not see the angles or edges.

To add a parallax map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the parallax.
2. Click Media.
3. In the Media menu, select the media you want to use for the parallax.
4. Do one of the following:
   - Drag the Parallax Map node from the node bin and place it in the schematic.
   - Drag the Parallax Map node from the node bin and place it where you want it in Result view.
   - Double-click the Parallax Map node. You do not need to be in Schematic view to add a node in this manner.

   The parallax object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the Parallax node indicates the media used for the parallax.



To specify different media as the source, select the media in the Media menu, then click Apply, or use the Read File tab to manage the texture media (see Working With Textures in Map Nodes (page 762)).

NOTE If you accessed Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.

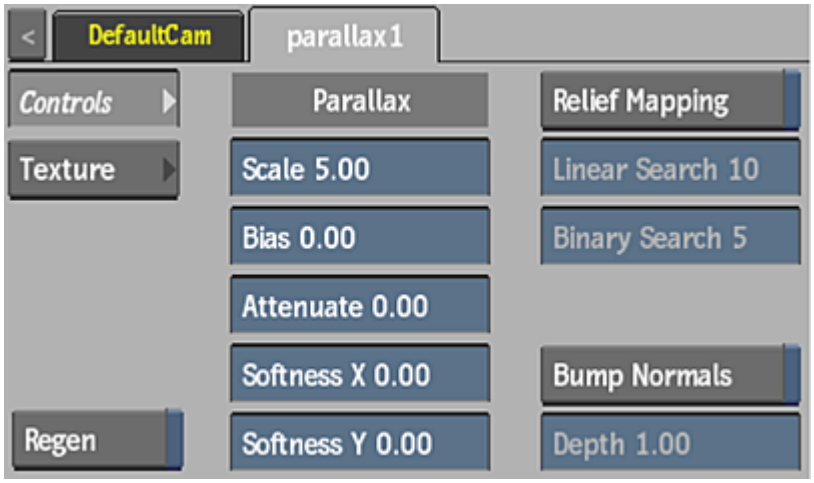
5 Double-click the Parallax node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).

The Parallax menu appears.

Parallax Menu Settings

The Parallax menu is divided into two tabbed sections: Controls and Texture.

Controls Tab



Regen button Enable to dynamically refresh the image as changes are made to the parallax settings.

Scale field Displays the perceptual height of the texture driven by the parallax map. Editable.

Bias field Displays the distance between the real surface of the object and the perceptual ground (zero level) of the texture. A value of half of the Scale field value should give the best results.

Attenuate field Displays the level of amplitude of the effect caused by the parallax map texture. Editable.

Softness X field Displays the amount of X-axis blur applied to the parallax map. Editable.

Softness Y field Displays the amount of Y-axis blur applied to the parallax map. Editable.

Relief Mapping button Enable to augment the 3D surface detail by creating occlusions where changes occur in the parallax map. Rendering is slower when enabled, but the results may be better.

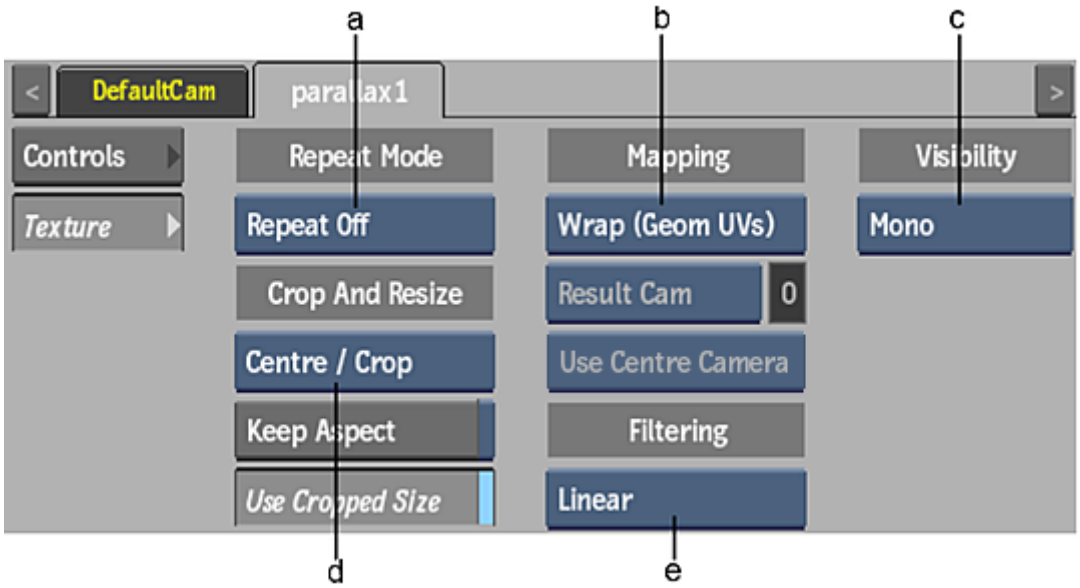
Linear Search field Displays the number of steps to search for the best depth in the relief map. Editable.

Binary Search field Displays the number of steps to refine the precision of the best depth found by the linear search. Editable.

Bump Normals button Enable to allow the parallax map to create a normal map for enhanced lighting effects. Used only if no normal map exists for object using the parallax texture. Enabling also affects the Normal output in Action.

Depth field Displays the amount of Z scale applied to the generated normals. Higher values increase the effect of normals. Editable.

Texture Tab



(a) Repeat Mode box (b) Mapping box (c) Camera Type box (d) Fit Method box (e) Filter box

Repeat Mode box Select how the parallax map pattern is repeated on the surface.

Fit Method box Select a fit method option to be applied to the parallax map.

Keep Aspect button Enable to preserve the aspect ratio of non-square pixels (not available for the Fill fit method).

Use Cropped Size button Enable to replace the parallax map with the cropped size of the parallax media. Disable to use the cropped parallax media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the texture map according to the object's texture coordinates. To use this option, you must import a model that has its own texture coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu. See NO LABEL .

Plane Planar mapping applies the map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the texture values. Planar mapping positions the lower-left corner of the map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

Perspective Camera box Specify which camera's FOV to take into account when using perspective or projection mapping.

Perspective Camera field Displays the active perspective or projection camera number. Non-editable.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the parallax map.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

TIP You can set the default filtering type in the Action Setup menu Preferences tab.

Camera Type box Select the camera type visibility for the parallax map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

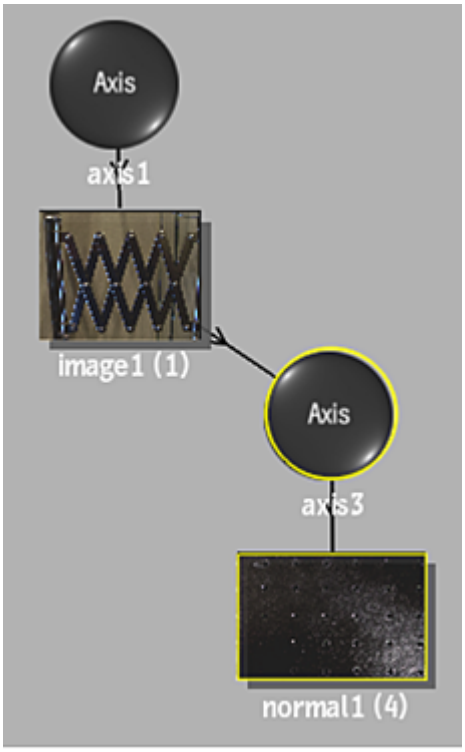
Normal Mapping

A normal map is used to simulate bumps and lighting on a surface or geometry. You can apply a normal map to modify how a surface reacts to shading. You can manipulate a surface's normals based on X, Y, and Z offsets.

To add a normal map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the normal.
2. Click Media.
3. In the Media menu, select the media you want to use for the normal.
4. Do one of the following:
   - Drag the Normal Map node from the node bin and place it in the schematic.
   - Drag the Normal Map node from the node bin and place it where you want it in Result view.
   - Double-click the Normal Map node. You do not need to be in Schematic view to add a node in this manner.

The normal object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the normal node indicates the media used for the normal.



To specify different media as the source, select the media in the Media menu, then click Apply, or use the Read File tab to manage the texture media (see Working With Textures in Map Nodes (page 762)).

NOTE If you accessed Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.

5 Double-click the Normal node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).

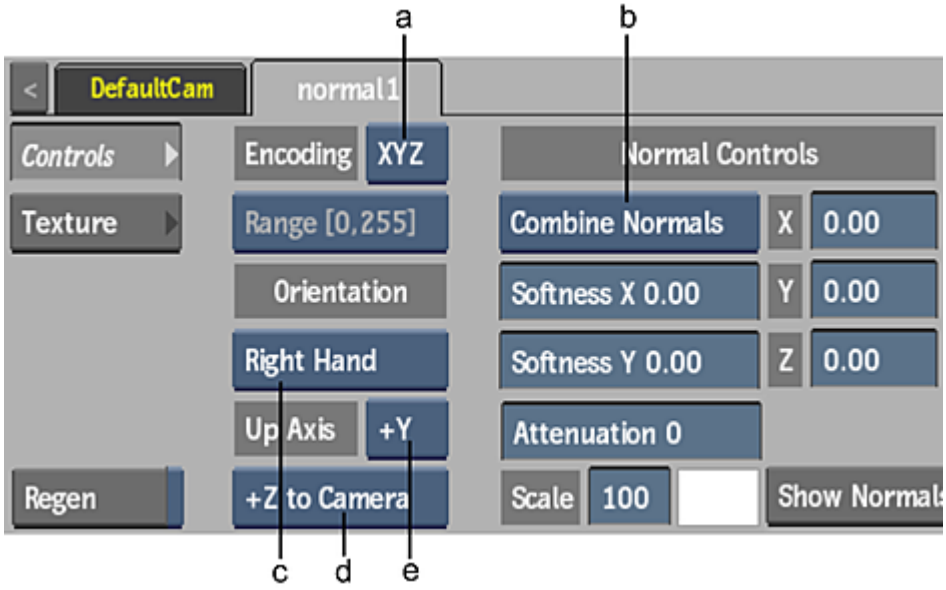
The Normal menu appears.

Normal Menu Settings

The Normal menu is divided into two tabbed sections: Controls and Texture.

Controls Tab

Use the Encoding and Orientation settings to help Action with information on how the normal map was encoded and rendered. Then use the Normal Controls to tweak the map in Action.



(a) Encoding box (b) Normals box (c) Orientation box (d) Camera box (e) Up Axis box

Encoding box Select the order that the normal map is encoded based on the interpretation of the RGB channels (XYZ or XZY).

Range box When working with floating point normal map media, select the range of the normal map media: [0, 1] or [-1, 1]. When working with 8-, 10-, or 12-bit images, the Range box displays the appropriate range for Action, but the box is greyed out.

Orientation box Select whether the orientation of the coordinate system of the map is Left Hand or Right Hand.

Up Axis box Select which axis is the up axis of the map.

Camera box Select which axis of the map corresponds to the Z axis in Action. The selection in the Up Axis box determines the available selections in this box.

Normals box Select whether to combine or replace the normal map with the surface normals.

Select:	To:
Combine Normals	Combine the normals map texture with the surface's normals.
Replace Normals	Apply only the normal map texture to the surface (ignoring the surface normal properties).

Softness X field Displays the amount of X-axis blur applied to the normal map. Editable.

Softness Y field Displays the amount of Y-axis blur applied to the normal map. Editable.

Attenuation field Displays the level of amplitude of the effect caused by the normal map texture. Editable.

X Normal axis Displays the amount of offset in pixel units along the X axis.

Y Normal axis Displays the amount of offset in pixel units along the Y axis.

Z Normal axis Displays the amount of offset in pixel units along the Z axis.

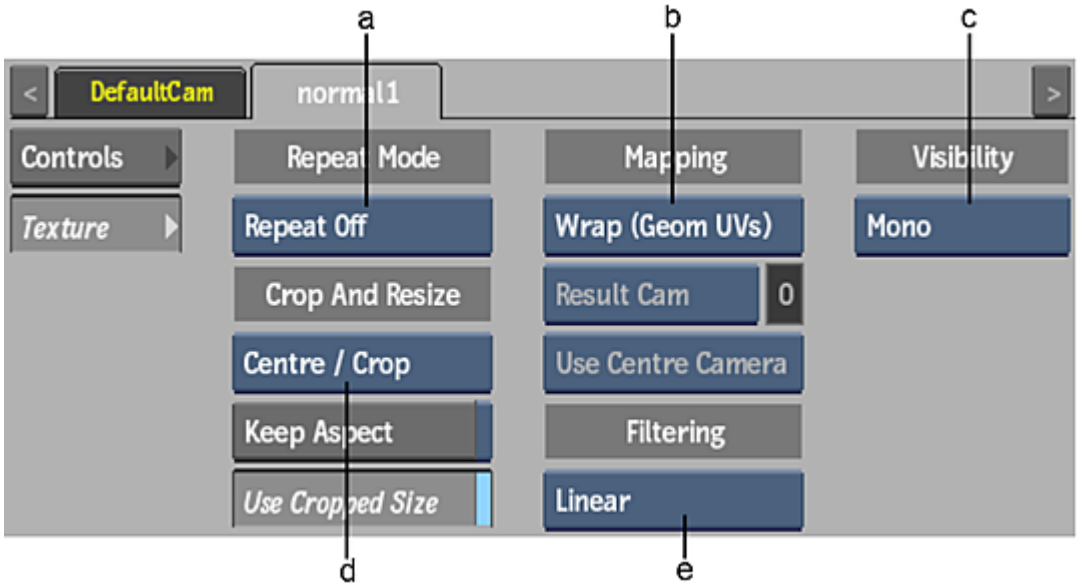
Show Normals button Enable to display normal vectors over the surface (does not affect the render).

Normals colour pot Displays the normal vectors colour. Editable.

Scale field Displays the scale of the normal vectors. Editable.

Regen button Enable to dynamically refresh the image as changes are made to the normal settings.

Texture Tab



(a) Repeat Mode box (b) Mapping box (c) Camera Type box (d) Fit Method box (e) Filter box

Repeat mode box Select how the normal map pattern is repeated on the surface.

Fit Method box Select a fit method option to be applied to the normal map.

Keep Aspect button Enable to preserve the aspect ratio of non-square pixels (not available for the Fill fit method).

Use Cropped Size button Enable to replace the normal map with the cropped size of the normal media. Disable to use the cropped normal media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the texture map according to the object's texture coordinates. To use this option, you must import a model that has its own texture coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu.

Plane Planar mapping applies the map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the texture values. Planar mapping positions the lower-left corner of the map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

Camera box Specify which camera's FOV to take into account when using perspective or projection mapping.

Camera field Displays the active perspective or projection camera number. Non-editable.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the normal map.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

TIP You can set the default filtering type in the Action Setup menu Preferences tab.

Camera Type box Select the camera type visibility for the normal map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

NOTE If the Normal node is attached to a 3D Geometry node, you may need to select a UV Mapping mode other than Default in the Geometry menu for the normal pattern to have an effect on the geometry. See NO LABEL .

Specular Mapping

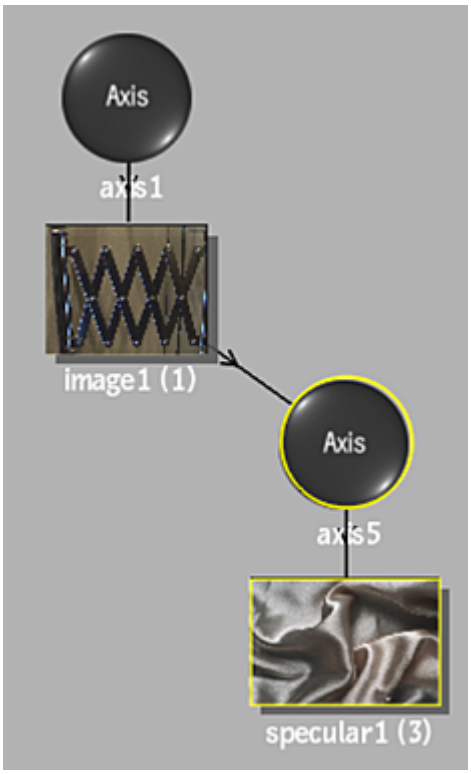
A specular map defines the shininess and highlight colour of a surface or geometry.

To add a specular map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the specular.
2. Click Media.
3. In the Media menu, select the media you want to use for the specular.
4. Do one of the following:
   - Drag the Specular Map node from the node bin and place it in the schematic.

- Drag the Specular Map node from the node bin and place it where you want it in Result view.
- Double-click the Specular Map node. You do not need to be in Schematic view to add a node in this manner.

The specular object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the Specular node indicates the media used for the specular.



To specify different media as the source, select the media in the Media menu, then click Apply, or use the Read File tab to manage the texture media (see Working With Textures in Map Nodes (page 762)).

NOTE If you accessed Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.

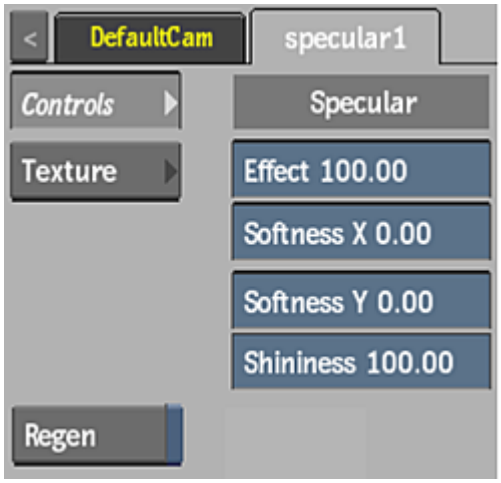
5 Double-click the Specular node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).

The Specular menu appears.

Specular Menu Settings

The Specular menu is divided into two tabbed sections: Controls and Texture.

Controls Tab



Effect field Displays the amount of specular colour. Editable.

Softness X field Displays the amount of X-axis blur applied to the specular map. Editable.

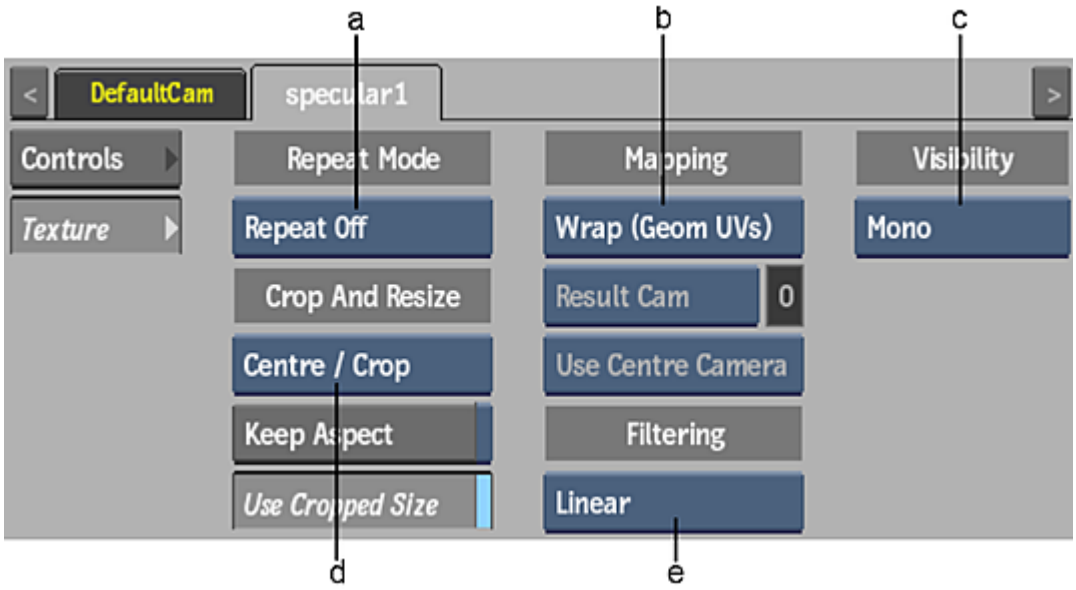
Softness Y field Displays the amount of Y-axis blur applied to the specular map. Editable.

Shininess field Displays the amount of shininess in the specular map. Editable.

NOTE To display proper results, the specular map also uses the value of the surface or geometry shine field. A shine value of 0.0 in the Surface or Geometry menu is interpreted as 1.0.

Regen button Enable to dynamically refresh the image as changes are made to the specular settings.

Texture Tab



(a) Repeat Mode box (b) Mapping box (c) Camera Type box (d) Fit Method box (e) Filter box

Repeat Mode box Select how the specular map pattern is repeated on the surface.

Fit Method box Select a fit method option to be applied to the specular map.

Keep Aspect button Enable to preserve the aspect ratio of non-square pixels (not available for the Fill fit method).

Use Cropped Size button Enable to replace the specular map with the cropped size of the specular media. Disable to use the cropped specular media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the texture map according to the object's texture coordinates. To use this option, you must import a model that has its own texture coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu. See NO LABEL .

Plane Planar mapping applies the map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the texture values. Planar mapping positions the lower-left corner of the map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

Camera box Specify which camera's FOV to take into account when using perspective or projection mapping.

Camera field Displays the active perspective or projection camera number. Non-editable.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the specular map.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).

Select:	To apply:
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

TIP You can set the default filtering type in the Action Setup menu Preferences tab.

Camera Type box Select the camera type visibility for the specular map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

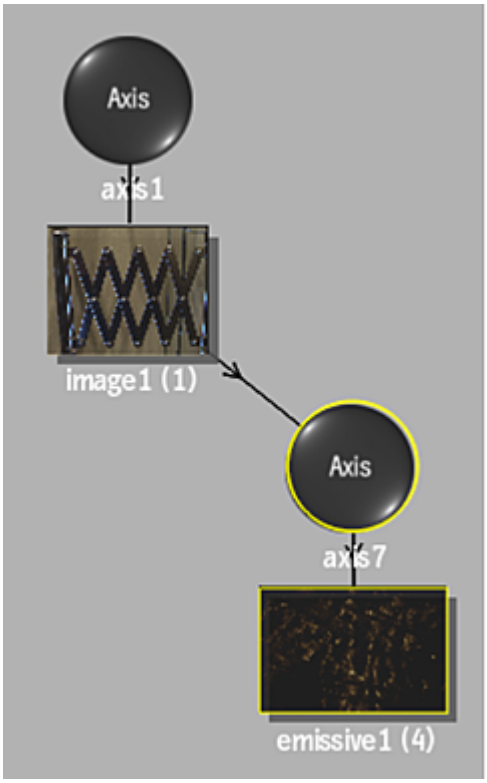
Emissive Mapping

An emissive map uses colours to simulate a glowing effect within the texture. The effect of an emissive map does not go beyond the limits of an object, and therefore cannot be used to generate glows around the rest of the scene or the surrounding objects.

To add an emissive map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the emissive.
2. Click Media.
3. In the Media menu, select the media you want to use for the emissive.
4. Do one of the following:
   - Drag the Emissive Map node from the node bin and place it in the schematic.
   - Drag the Emissive Map node from the node bin and place it where you want it in Result view.
   - Double-click the Emissive Map node. You do not need to be in Schematic view to add a node in this manner.

   The emissive object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the Emissive node indicates the media used for the emission.



To specify different media as the source, select the media in the Media menu, then click Apply, or use the Read File tab to manage the texture media (see Working With Textures in Map Nodes (page 762)).

NOTE If you accessed Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.

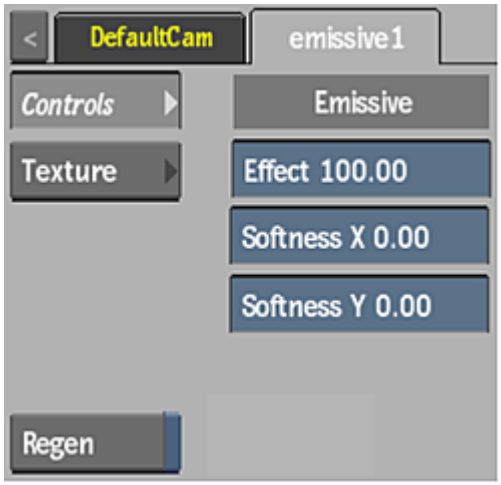
5 Double-click the Emissive node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).

The Emissive menu appears.

Emissive Menu Settings

The Emissive menu is divided into two tabbed sections: Controls and Texture.

Controls Tab



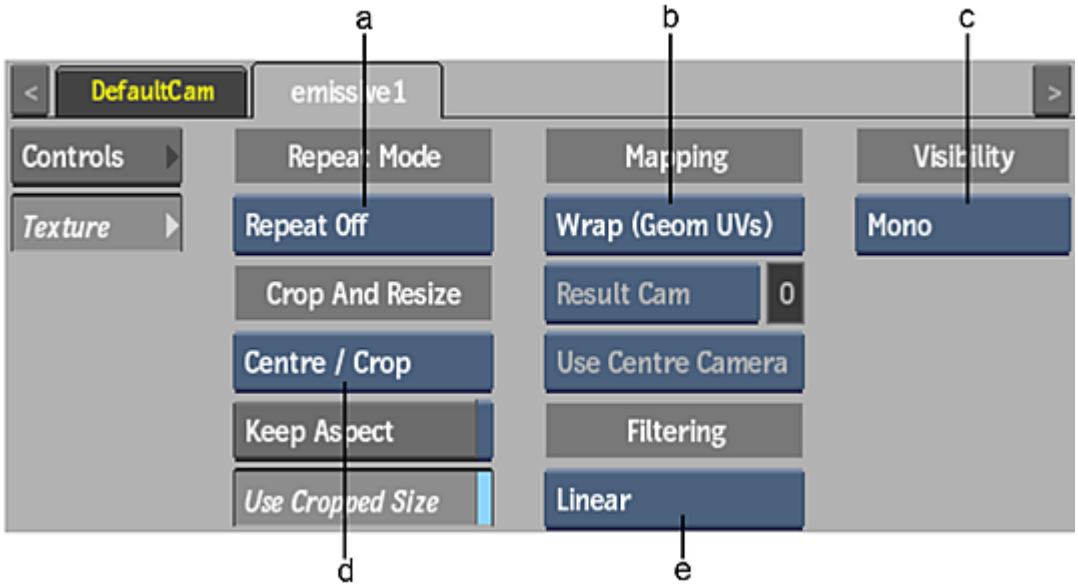
Effect field Displays the amount of emissive colour. Editable.

Softness X field Displays the amount of X-axis blur applied to the emissive map. Editable.

Softness Y field Displays the amount of Y-axis blur applied to the emissive map. Editable.

Regen button Enable to dynamically refresh the image as changes are made to the emissive settings.

Texture Tab



(a) Repeat Mode box (b) Mapping box (c) Camera Type box (d) Fit Method box (e) Filter box

Repeat Mode box Select how the emissive map pattern is repeated on the surface.

Fit Method box Select a fit method option to be applied to the emissive map.

Keep Aspect button Enable to preserve the aspect ratio of non-square pixels (not available for the Fill fit method).

Use Cropped Size button Enable to replace the emissive map with the cropped size of the emissive media. Disable to use the cropped emissive media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the texture map according to the object's texture coordinates. To use this option, you must import a model that has its own texture coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu. See NO LABEL .

Plane Planar mapping applies the map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the texture values. Planar mapping positions the lower-left corner of the map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

Camera box Specify which camera's FOV to take into account when using perspective or projection mapping.

Camera field Displays the active perspective or projection camera number. Non-editable.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the emissive map.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

TIP You can set the default filtering type in the Action Setup menu Preferences tab.

Camera Type box Select the camera type visibility for the emissive map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

Diffuse Mapping

Use a diffuse map to define the diffuse reflection and main colour of a surface, 3D model, or 3D text. Since the diffuse map and its axis are parented by the surface or geometry node, animating the parent's axis also animates the diffuse map, which has the effect of keeping the map properly in place on the model. A diffuse map uses the specular highlight, diffuse colour, and shine set by its parent.

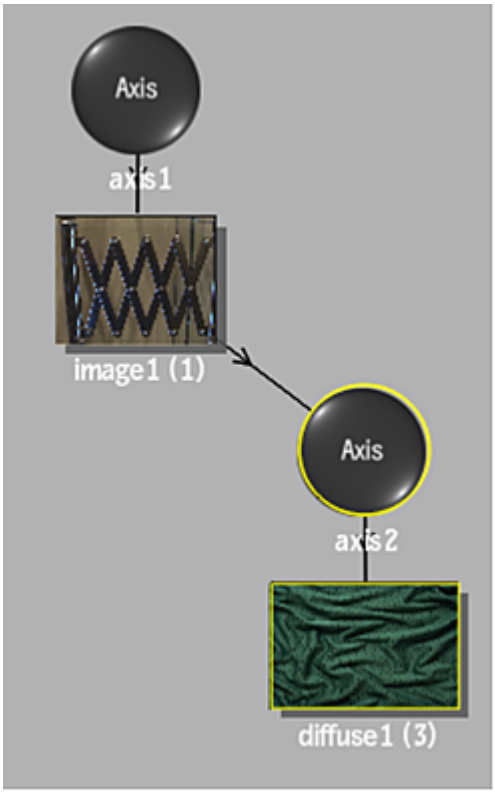
When adding a diffuse map to a shaded surface, the diffuse is used when generating the shadow. The diffuse is only used to apply the colour to the shadow, so effects such as surface displacement still reference the media associated with the surface. If a node has an applied diffuse map, it is the matte setting of the diffuse map that controls whether the object will be included in the various output mattes (scene matte, blend matte, for example).

To add a diffuse map:

1. In the schematic, select the surface, 3D model, or 3D text to which you want to apply the diffuse map.
2. Click Media.
3. In the Media menu, select the media you want to use for the diffuse map texture (you can change this texture later).

   A diffuse map uses the front and matte of the media. If you do not want to apply transparency to your diffuse map, turn its matte off.
4. Do one of the following:
   - Drag the Diffuse Map node from the node bin and place it in the schematic.
   - Drag the Diffuse Map node from the node bin and place it where you want it in Result view.
   - Double-click the Diffuse Map node. You do not need to be in Schematic view to add a node in this manner.

   The diffuse object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the Diffuse node indicates the media used for the diffuse.



To specify different media as the source, select the media in the Media menu, then click Apply, or use the Read File tab to manage the texture media (see Working With Textures in Map Nodes (page 762)).

NOTE If you accessed Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.

5 In the Rendering section of the Action Setup menu, enable or disable Shading depending on the method of diffuse mapping you are using. When Shading is enabled, normals are used. You must enable Shading when using Reflection mapping because it also uses normals.

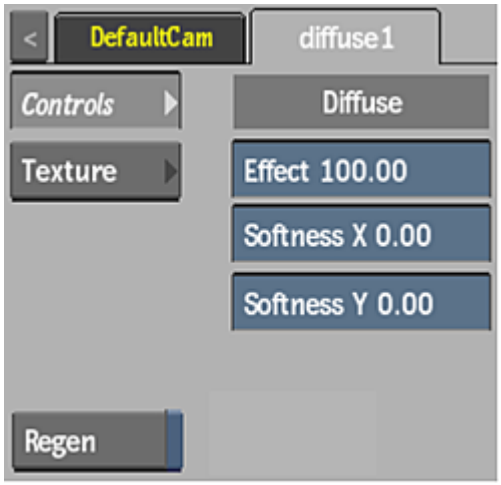
6 Double-click the Diffuse node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).

The Diffuse menu appears.

Diffuse Menu Settings

The Diffuse menu is divided into two tabbed sections: Controls and Texture.

Controls Tab



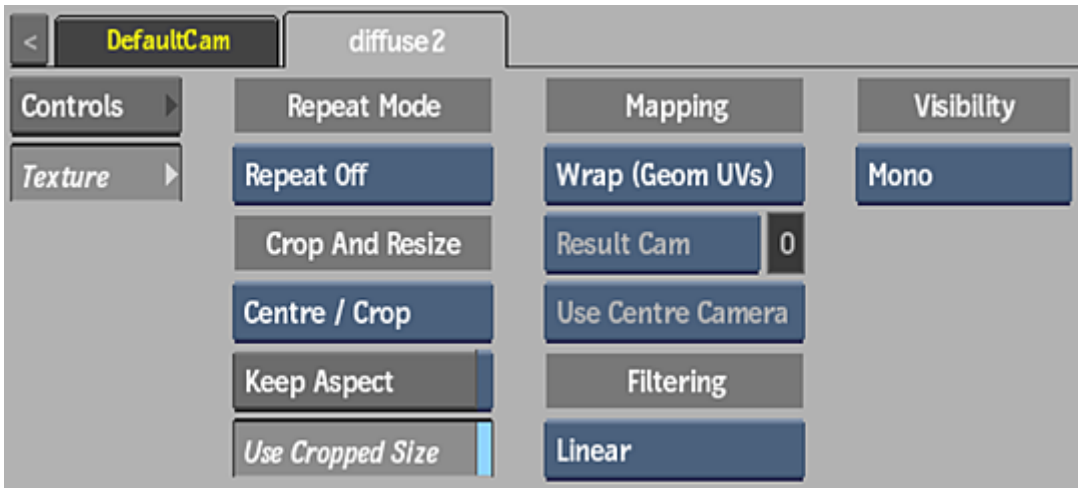
Effect field Displays the amount of diffuse colour. Editable.

Softness X field Displays the amount of X-axis blur applied to the diffuse map. Editable.

Softness Y field Displays the amount of Y-axis blur applied to the diffuse map. Editable.

Regen button Enable to dynamically refresh the image as changes are made to the settings.

Texture Tab



Repeat Mode box Select how the map pattern is repeated on the surface.

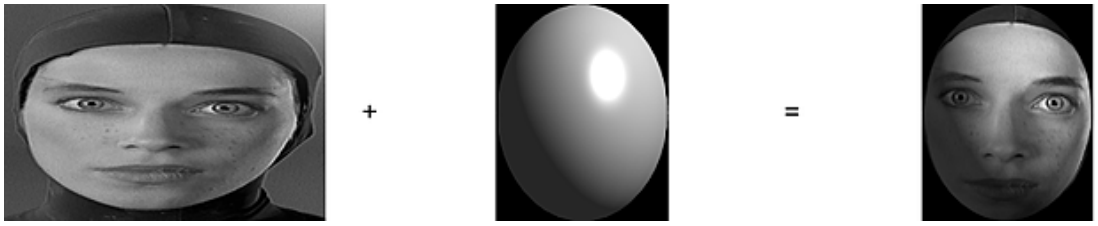
Fit Method box Select a fit method option to be applied to the diffuse map.

Keep Aspect button Enable to preserve the aspect ratio of non-square pixels (not available for the Fill fit method).

Use Cropped Size button Enable to replace the diffuse map with the cropped size of the diffuse media. Disable to use the cropped diffuse media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the diffuse map according to the object's diffuse coordinates. To use this option, you must import a model that has its own diffuse coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu. See NO LABEL .



Plane Planar mapping applies the diffuse map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the diffuse values. Planar mapping positions the lower-left corner of the diffuse map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the diffuse map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

Camera box Specify which camera's FOV to take into account when using perspective or projection mapping.

Camera field Displays the active perspective or projection camera number. Non-editable.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the diffuse map.

Select:	**To apply:**
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.

Select:	To apply:
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

TIP You can set the default filtering type in the Action Setup menu Preferences tab.

Camera Type box Select the camera type visibility for the map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

NOTE When a diffuse map is connected to a flat surface, you can enable Auto Expand in the Surface menu to automatically resize the surface when the diffuse map's Axis settings are changed.

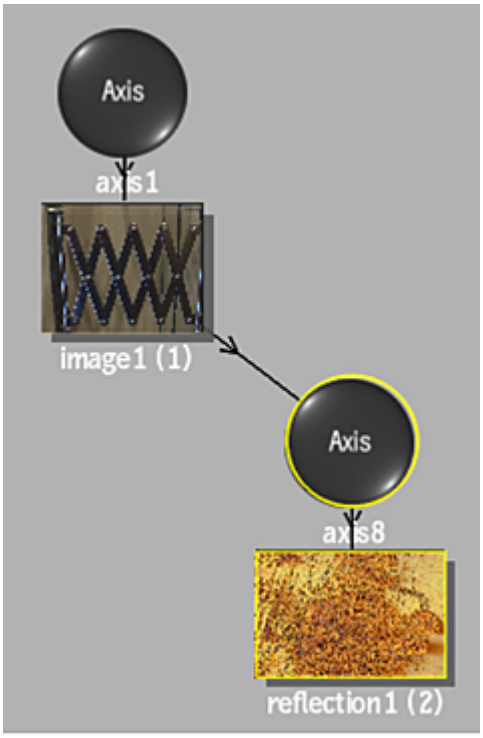
Reflection Mapping

Reflection mapping simulates a mirrored surface by using the specular reflection values in the map. You have the option of combining the result of a reflection map to a diffuse map.

To add a reflection map:

1. In the schematic, select the surface, 3D model, or 3D text to which you want to apply the diffuse map.
2. Click Media.
3. In the Media menu, select the media you want to use for the diffuse map.
4. Do one of the following:
   - Drag the Reflection Map node from the node bin and place it in the schematic.
   - Drag the Reflection Map node from the node bin and place it where you want it in Result view.
   - Double-click the Reflection Map node. You do not need to be in Schematic view to add a node in this manner.

   The reflection object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the Reflection node indicates the media used for the diffuse.



To specify different media as the source, select the media in the Media menu, then click Apply, or use the Read File tab to manage the texture media (see Working With Textures in Map Nodes (page 762)).

NOTE If you accessed Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.

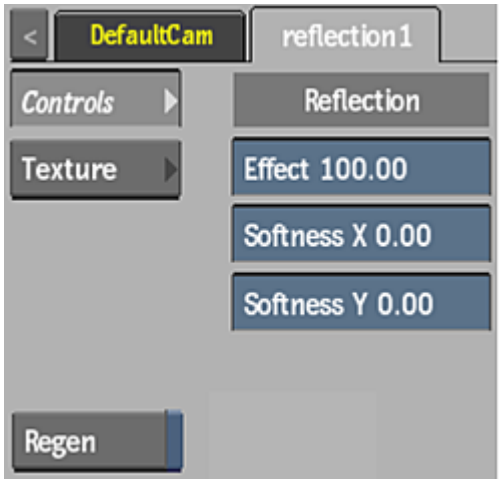
5 Double-click the Reflection node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).

The Reflection menu appears.

Reflection Menu Settings

The Reflection menu is divided into two tabbed sections: Controls and Texture.

Controls Tab



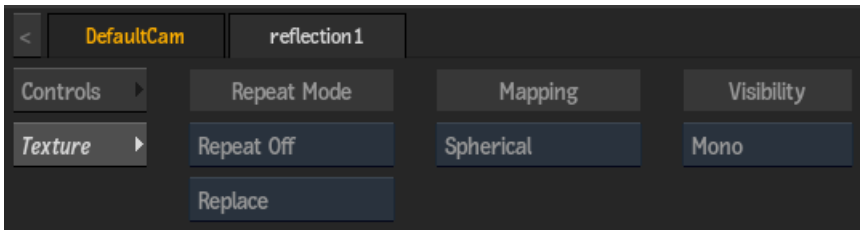
Effect field Displays the amount of diffuse colour. Editable.

Softness X field Displays the amount of X-axis blur applied to the diffuse map. Editable.

Softness Y field Displays the amount of Y-axis blur applied to the diffuse map. Editable.

Regen button Enable to dynamically refresh the image as changes are made to the settings.

Texture Tab



Repeat Mode box Select how the map pattern is repeated on the surface.

Add/Replace box Select whether to add or replace the reflection map to the diffuse colour.

Mapping Type box Select the type of texture map used, to help create the reflection effect.

Camera Type box Select the camera type visibility for the map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

Position Mapping

A position map (sometimes called a world position pass), is an image where each pixel's R, G, B colour values represent the x, y, z coordinates of the corresponding vertex, in 3D world space. In Action, a position map works as a type of 3D displacement, that is, it displaces every pixel in that image to the actual location in space of the vertex it represents, allowing you to:

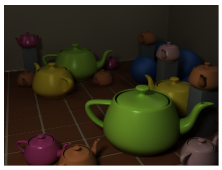
- Establish depth, position, and scaling in images, for compositing and creative purposes.
- Relight your scene.
- Create stereo objects.

■ Creatively transfer the position of one object onto another object.

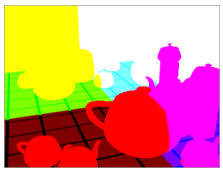
A typical workflow would be to import (from a 3D application) the beauty pass, the position pass (as a 16-bit floating point OpenEXR clip), the normal pass, and the FBX camera that rendered the scene. Then, apply the position and the normal map to the beauty pass image surface, and set the scale of the position map so that it aligns properly with the 3D camera frustum.

NOTE Make sure that your position pass rendering settings in the 3D application are sufficiently high so that the position map displays correctly in Flame. For example, when rendering with Mental Ray in Maya, set the quality level to Production or higher.

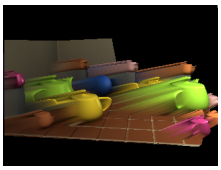
When looked from the original camera POV, your scene looks just like the rendered image. But since pixels are accurately located in space relative to the original 3D scene with the attached position map, you can correctly position 2D layers in the same perspective as objects of the 3D scene.



Beauty Pass



Position Map



Action scene with camera orbited

You can use position maps generated by 3D applications or Action itself (Position is available as an output type in the Output (page 598) menu).

To add a position map:

1. In the schematic, select the surface to which you want to apply the position map.
2. Click Media.
3. In the Media menu, select the media you want to use for the position map.
4. Do one of the following:
   - Drag the Position Map node from the node bin and place it in the schematic.
   - Drag the Position Map node from the node bin and place it where you want it in Result view.
   - Double-click the Position Map node. You do not need to be in Schematic view to add a node in this manner.

   The position object is added to the schematic with its own parent axis. The new axis is the child of the selected surface. In Schematic view, the number in brackets next to the name of the Position node indicates the media used for the position map.

   To specify different media as the position map source, select the media in the Media menu, then click Apply.

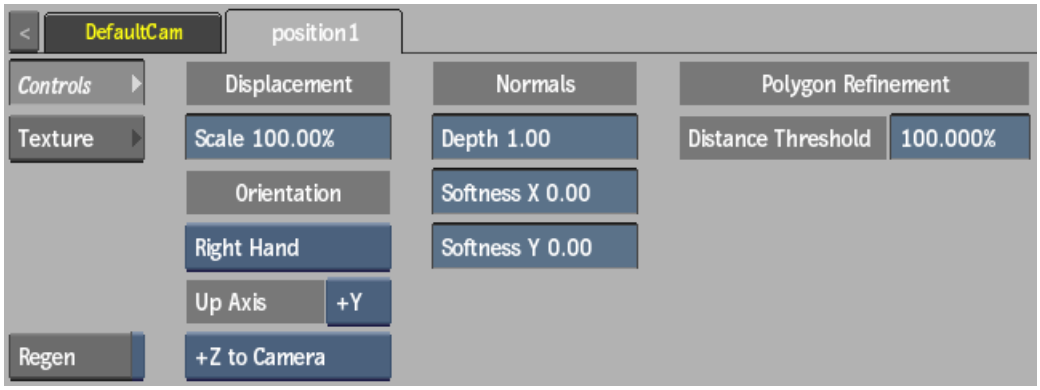
   NOTE If you accessed Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.
5. Double-click the Position node in the schematic, or follow the tab population rules for the Object menu.

   The Position menu appears.

Position Menu Settings

Controls Tab

Use the Orientation settings to help Action with information on how the position map was rendered. The Normals settings affect the generated normals present in the position map.



Scale field Displays the scale of the position map in relation to Action space. Editable.

Orientation box Select whether the orientation of the coordinate system of the map is Left Hand or Right Hand.

Up Axis box Select which axis is the up axis of the map.

Camera box Select which axis of the map corresponds to the Z axis in Action. The selection in the Up Axis box determines the available selections in this box.

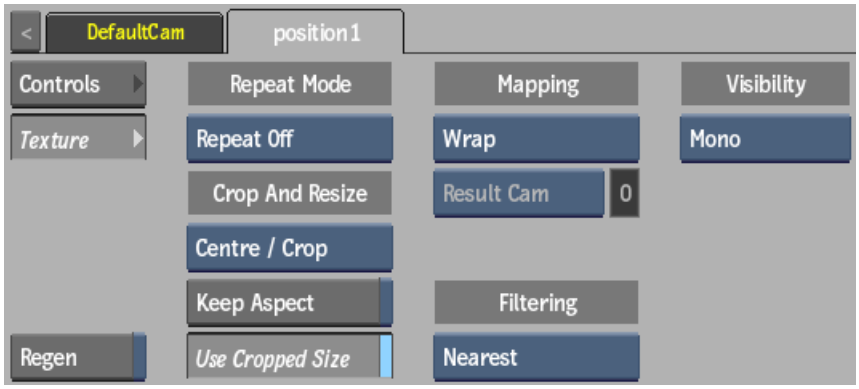
Depth field Displays the amount of Z scale applied to the generated normals. Higher values attenuate the effect toward the normals of the parent surface. Editable.

Softness X field Displays the amount of X-axis blur applied to the map. Editable.

Softness Y field Displays the amount of Y-axis blur applied to the map. Editable.

Distance Threshold field Displays the threshold at which the longest triangles producing disconnected meshes in the geometry are hidden. Editable.

Texture Tab



Repeat mode box Select how the map pattern is repeated on the surface.

Fit Method box Select a fit method option to be applied to the map.

Keep Aspect button Enable to preserve the aspect ratio of non-square pixels (not available for the Fill fit method).

Use Cropped Size button Enable to replace the map with the cropped size of the media. Disable to use the cropped media as is.

Mapping box Select the type of texture mapping.

Wrap Wrap mapping completely envelops the 3D model with the texture map according to the object's texture coordinates. To use this option, you must import a model that has its own texture coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu.

Plane Planar mapping applies the map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the texture values. Planar mapping positions the lower-left corner of the map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

Camera box Specify which camera's FOV to take into account when using perspective or projection mapping.

Camera field Displays the active perspective or projection camera number.

Filter box Select the type of filtering to apply to the map.

Camera Type box Select the camera type visibility for the map.

UV Mapping

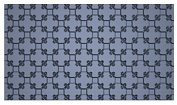
A UV map is a pass which records the way the pixels of an image should be displaced within the image plane, so that the warped texture looks like the textured geometry from the point of view of the CG camera. The UV map acts as a 2D distortion field, or a pixel look up table, recording where every pixel of an input image texture should be, had it been applied on a geometry filmed by the CG camera. It is a very effective way of faking 3D mappings, without having a single polygon. It is most effective when combined with a Normal map, which can be used to simulate the shading of the actual geometry even though everything is perfectly flat.

Applying a UV map to surface or geometry, warps its texture according to the defined UV. The U information is coded in the Red channel, while the V information is coded in the Green channel of the UV map image. For each pixel of the UV map, the red or green value corresponds to a given pixel on the unwrapped texture, and it is this pixel that gets displaced at the position of the same pixel in the attached surface or geometry.

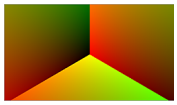
Here are a few things to keep in mind when working with UV maps:

- To avoid pixelation in your imported UV passes, make sure that the UV pass is rendered as a 16-bit floating point image from your 3D application.

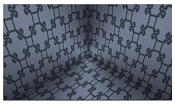
- It is a good idea to render UV passes without anti-aliasing to avoid warping on the edges of objects that the UV map is applied to. Since you can also output a UV Map from Flame, make sure that anti-aliasing is not selected in the Render Options section of the Output menu.
- You can embed object IDs in the Blue channel when performing a UV render pass in Maya, allowing you to specify which objects are displaced within the UV map in Action. In Maya, when performing a UV Render Pass, enable *Embed Object ID in Blue Channel*. When disabled, blue channel information is ignored.
- For best results, in the Texture settings of the UV Map menu in Flame, the Fit Method box should be set to Fill, and the Filter box to Nearest.
- If you accessed Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.



Flat texture



UV Map



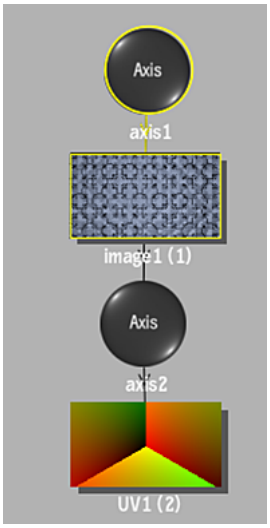
Texture with UV Map applied

Adding a UV Map

To add a UV map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the UV map.
2. Click Media.
3. In the Media menu, select the media you want to use for the UV map.
4. Do one of the following:
   - Drag the UV Map node from the node bin and place it in the schematic.
   - Drag the UV Map node from the node bin and place it where you want it in Result view.
   - Double-click the UV Map node. You do not need to be in Schematic view to add a node in this manner.

   The UV object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the UV node indicates the media used for the UV.



To specify different media as the source, select the media in the Media menu, then click Apply, or use the Read File tab to manage the texture media (see Working With Textures in Map Nodes (page 762)).

NOTE If you accessed Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.

5 Double-click the UV node in the Schematic view, or follow the tab population rules for the Object menu.

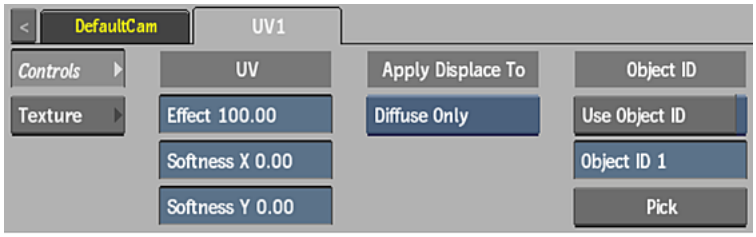
The UV menu appears.

UV Menu Settings

Regen button Enable to dynamically refresh the image as changes are made to the UV settings.

The UV menu is divided into two tabbed sections: Controls and Texture.

Controls Tab



Effect field Displays the amount of displacement applied to the UV map. Editable.

Softness X field Displays the amount of softness along the X axis of the UV map. Editable.

Softness Y field Displays the amount of softness along the Y axis of the UV map. Editable.

Apply Type box Select which attached maps are affected by the UV map.

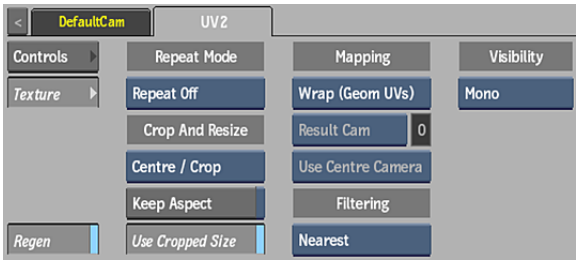
Use Object ID button Enable to use Object ID info embedded into the blue channel when using a UV map generated by Maya. When disabled, blue channel information is ignored.

The object ID allows you to control which objects within the UV map are displaced. In Maya, when performing a UV Render Pass, enable *Embed Object ID in Blue Channel*.

Object ID field Displays the specific blue value corresponding to an object ID. Editable.

Pick button Click to display the picker to select an object ID in the image through its blue value.

Texture Tab



Repeat mode box Select how the UV map pattern is repeated on the surface.

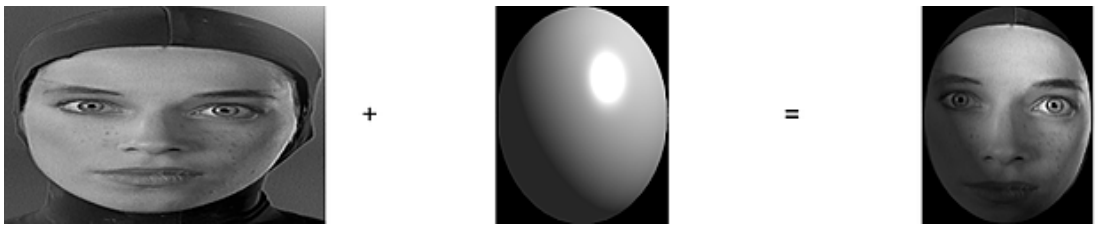
Fit Method box Select a fit method option to be applied to the UV map.

Keep Aspect button Enable to preserve the aspect ratio of non-square pixels (not available for the Fill fit method).

Use Cropped Size button Enable to replace the UV map with the cropped size of the UV media. Disable to use the cropped UV media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the diffuse map according to the object's diffuse coordinates. To use this option, you must import a model that has its own diffuse coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu. See NO LABEL .



Plane Planar mapping applies the diffuse map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the diffuse values. Planar mapping positions the lower-left corner of the diffuse map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the diffuse map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Per-

spective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

Camera box Specify which camera's FOV to take into account when using perspective or projection mapping.

Camera field Displays the active perspective or projection camera number. Non-editable.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the UV map.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

Camera Type box Select the camera type visibility for the UV map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

PBS Mapping

Use the PBS (Physically Based Shading) Map to create realistic textures based on how light interacts with surfaces. Different types of PBS textures are available from withing the PBS map. You can use the PBS Map to create your own textures, or add a Substance PBR (page 762) node (or from other online services providing PBR maps) to automatically include a number of PBS maps, depending on the preset. PBS maps also support texture maps from 3D renderers, such as a high quality AO map.

These are the available physically based texture types (similar settings are available in the Shader Node when you select Physically Based (page 775) as the Shader Type, but by using the PBS Map, you can affect the shader on a per pixel basis instead of as a global value for the whole object):

- Base Color
- Roughness

- Specular Level
- Metallic
- Anisotropic
- Subsurface
- Specular Tint
- AO
- Opacity
- Custom (Displays the custom ID corresponding to a PBS type not currently available in the PBS Type box. Use this with the Lightbox API.)

NOTE On a Mac system, you are limited to Base Color, Roughness, Metallic, AO, and Opacity shader types.

To add a PBS map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the PBS map.
2. Click Media.
3. In the Media menu, select the media you want to use for the parallax.
4. Do one of the following:
   - Drag the PBS Map node from the node bin and place it in the schematic.
   - Drag the PBS Map node from the node bin and place it where you want it in Result view.
   - Double-click the PBS Map node. You do not need to be in Schematic view to add a node in this manner.

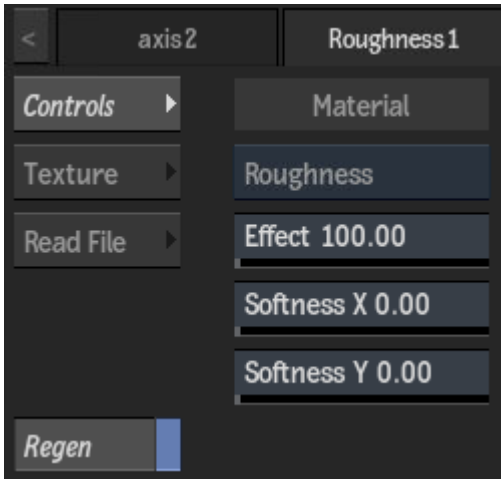
   By default, an Undefined material object is added to the schematic with its own parent axis. Once you select a shader type to apply from the PBS menu, the name under the material node updates to that type. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the node indicates the media used.

   To specify different media as the source, select the media in the Media menu, then click Apply, or use the Read File tab to manage the texture media (see Working With Textures in Map Nodes (page 762)).
5. Double-click the PBS node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).

   The PBS menu appears.

PBS Menu Settings

Controls Tab



PBS Type box Select which type of physically based shader to apply.

Effect field Displays the amount of the selected shader type to apple. Editable.

Softness X field Displays the amount of X-axis blur applied to the PBS map. Editable.

Softness Y field Displays the amount of Y-axis blur applied to the PBS map. Editable.

Custom ID field Available if Custom is selected in the PBS Type box. Displays the custom ID corresponding to a PBS type not currently available in the PBS Type box. Use this with the Lightbox API.

Regen button Enable to dynamically refresh the image as changes are made to the PBS settings.

Texture Tab

Repeat Mode box Select how the map pattern is repeated on the surface.

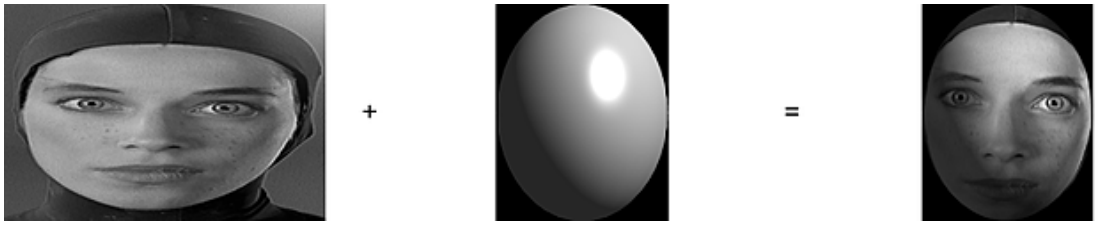
Fit Method box Select a fit method option to be applied to the PBS map.

Keep Aspect button Enable to preserve the aspect ratio of non-square pixels (not available for the Fill fit method).

Use Cropped Size button Enable to replace the PBS map with the cropped size of the PBS media. Disable to use the cropped PBS media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the PBS map according to the object's coordinates. To use this option, you must import a model that has its own texture coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu. See NO LABEL .



Plane Planar mapping applies the map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the texture values. Planar mapping positions the lower-left corner of the diffuse map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the texture map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

Camera box Specify which camera's FOV to take into account when using perspective or projection mapping.

Camera field Displays the active perspective or projection camera number. Non-editable.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the PBS map.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.

Select:	To apply:
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

TIP You can set the default filtering type in the Action Setup menu Preferences tab.

Camera Type box Select the camera type visibility for the map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

Projecting Textures

Like a slide projector, textures can be projected onto 3D models to create an effect where a texture is animated on an object in 3D space. The advantage of using a Projector node is that you can cast images on arbitrary surfaces. You can create a spotlight texture and project it onto a 3D model, so that you can see "into" it.

You can attach a look-at connection (page 579) between Action objects with axis characteristics (Axis, Camera, Light, and Projector). The connection allows the parent object to rotate to face the child look-at object, no matter where it is positioned. Look-At links can be animated; therefore you can link different objects from the parent at different frames. In the channel editor, you can see a lookAt channel for every look-at parent.

Similar to cameras in Action, you can set a Free or Target projector mode. Select Free to project the scene in the direction that you aim the projector, or Target to aim the projector at a target object in the scene based on a point of interest. Different settings are available in the Projector menu based on the choice of mode.

NOTE Some Action maps, such as Diffuse and Emissive maps, also allow you to use Projection mapping as a type of texture mapping. This method of projecting behaves as if it were projected by the selected camera.

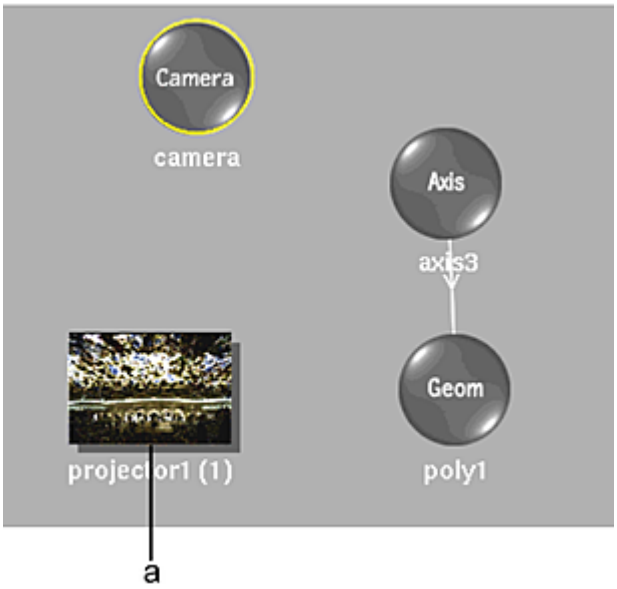
To project a texture:

1. Click Media to access the Media menu.
2. In the Media list, select the media that you want to project.

   NOTE If you accessed Action as a Timeline FX, you are limited to one front/matte media, and therefore may not get the desired result. In this case, you can access Action from Batch or Batch FX, or from the Tools tab.

3. Do one of the following:
   - Drag the Projector node from the node bin and place it in the schematic.
   - Drag the Projector node from the node bin and place it where you want it in Result view.
   - Double-click the Projector node. You do not need to be in Schematic view to add a node in this manner.

The Projector node appears in the schematic and a projector icon appears in the scene.

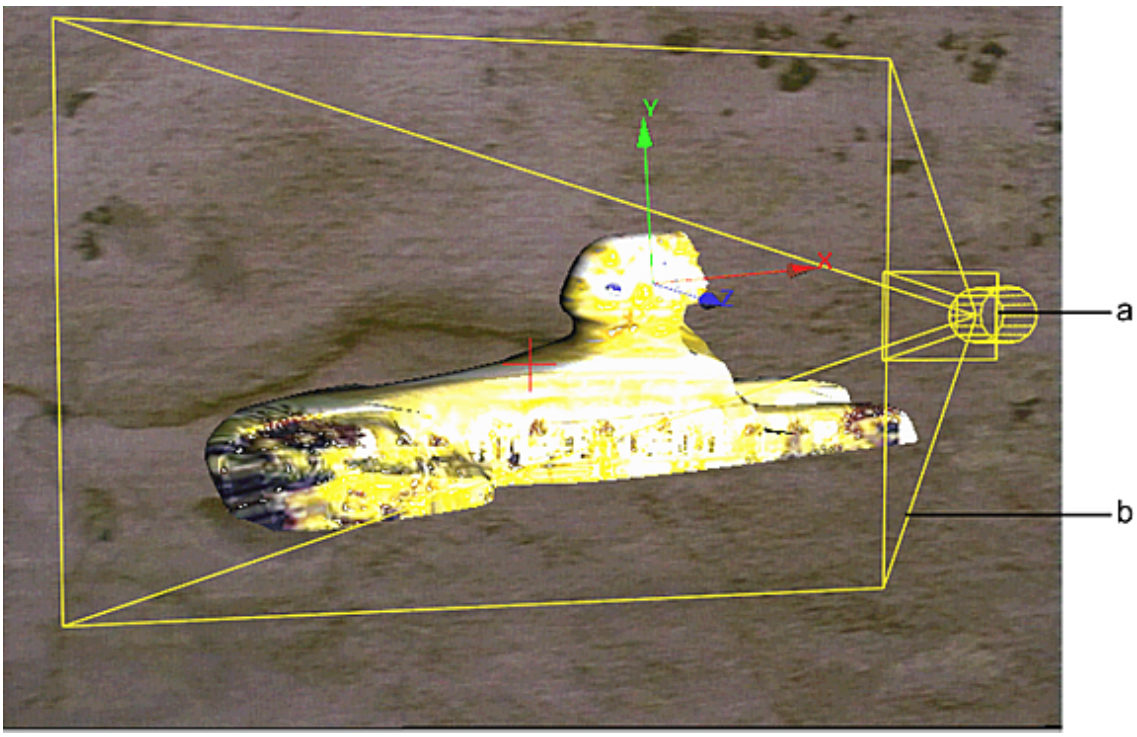


(a) Projector node

4 Double-click the Projector node to display the Projector menu, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).

The Projector is selected.

5 Switch to Camera view to manipulate the projector in the scene using the View box.



(a) Projector icon (b) Yellow outline shows the projector field of view

To add multiple projections:

1 Click Media to access the Media menu.

2 In the Media list, select the media you want to use as projectors by `Ctrl`-clicking the media.

3 Use the node bin to add a projector.

Multiple Projector nodes appear in the schematic and projector icons appear in the scene.

4 Use the Order field in the Projector menu to set the order of projectors.

The front projector is projector1. A new projector is always added to the back. Changing the order number of a projector affects the order of other projectors.

Applying Selective Projections

You can use projections selectively; either inclusively or exclusively. Similarly to the Selective Lighting feature, you can connect a projection to an image, object, or its axis and make the projection affect only that connection (inclusive). Conversely, a projection that affects every object *except* the one it is connected to is exclusive.

To use selective projections, you must have a scene with a minimum of two objects in order for the selective projection to affect one and not the other.

To create an inclusive selective projection:

1 Select the media to use as a texture for projection.

2 Add a projector using the Node bin.

The projection is added to the scene.

3 To make the projection selective, access the schematic by using the ~ key or by selecting Schematic from the View box.

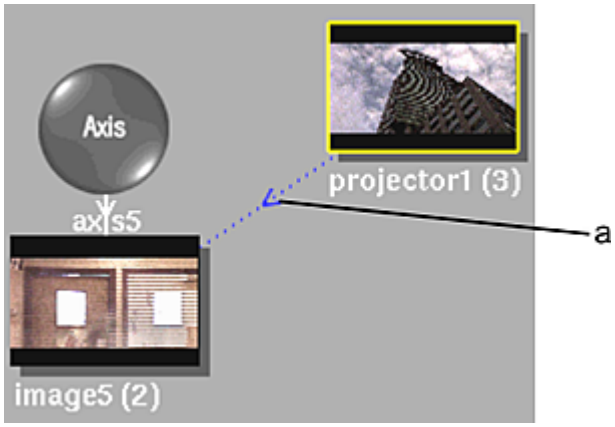
In the schematic, the projector is displayed much like an image. By default, it projects an image across the entire scene.

4 Select Light Link from the Tools box to create lighting connections.

5 Do one of the following:

- To project only a selected surface, drag a connection from the Projector node to one or several nodes in your scene. A selective projection connection is represented by a blue dotted line.

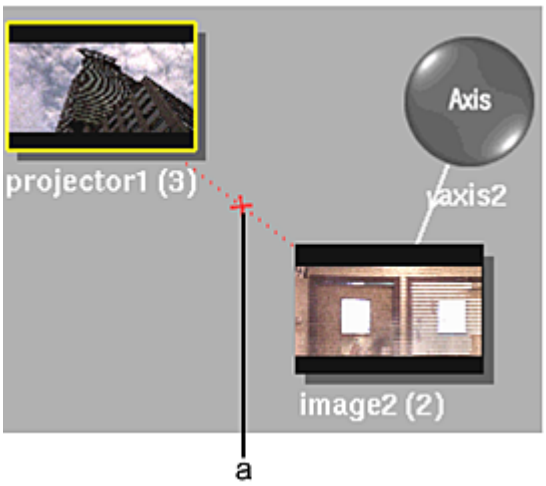
  Once a selective connection is created, the projector only affects the object to which it is connected.



(a) Inclusive projection connection

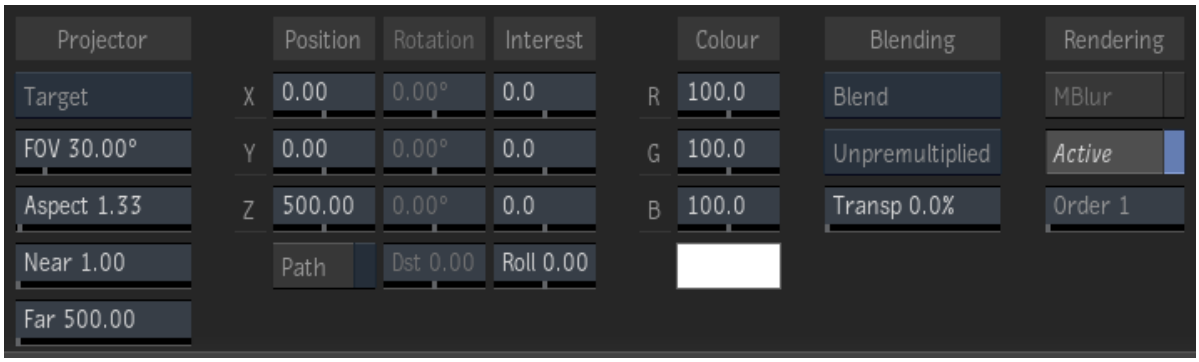
- To exclude a surface, hold the `Alt` key as you drag a connection from the Projector node to a surface node. The exclusive connection is represented by a red dotted line.

  Once an exclusive connection is created, the projector ignores the node(s) to which it is connected.



(a) Exclusive projection link

Projector Menu Settings



Projector Mode box Select Free to view the scene in the direction that you aim the projector, or Target to aim the projector at a target object in the scene based on a point of interest.

Field Of View field Displays the projector's field of view value. Editable.

Aspect Ratio field Displays the ratio of height to width for the projected image. Editable.

Far field Displays the far view of the selected projector. Editable.

Near field Displays the near view of the selected projector. Editable.

X Position field Displays the position of the projector in 3D space along the X axis.

Y Position field Displays the position of the projector in 3D space along the Y axis.

Z Position field Displays the position of the projector in 3D space along the Z axis.

TIP You can also set the texture projector position by dragging the projector in the scene.

Motion Path button Enable to use motion path animation for the projector.

X Rotation field Displays the rotation of the selected projector along the X axis. Available when Free is selected in the Projector Mode box. Editable.

Y Rotation field Displays the rotation of the selected projector along the Y axis. Available when Free is selected in the Projector mode box. Editable.

Z Rotation field Displays the rotation of the selected projector along the Z axis. Available when Free is selected in the Projector mode box. Editable.

Distance field Displays the position of the projector's focus. Available when Free is selected in the Projector Mode box.

X Point of Interest field Displays the position of the point of interest along the X axis. Available when Target is selected in the Projector Mode box. Editable.

Y Point of Interest field Displays the position of the point of interest along the Y axis. Available when Target is selected in the Projector Mode box. Editable.

Z Point of Interest field Displays the position of the point of interest along the Z axis. Available when Target is selected in the Projector Mode box. Editable.

Roll field Displays the amount of projector roll. Available when Target is selected in the Projector Mode box.

Red Colour field Defines the red value of the selected projector. Editable.

Green Colour field Defines the green value of the selected projector. Editable.

Blue Colour field Defines the blue value of the selected projector. Editable.

Projector colour pot Displays the colour of the projector. Editable.

Blend Mode box Select a mode to modify how you want to blend the projected image in the scene.

Blend Premultiplication box Select whether the blend operation is premultiplied or not. You can set this as a default preference in the Action Setup menu.

Transparency field Displays the transparency level of the projected texture effect. Editable.

Motion Blur button Enable to use a motion blur effect for the selected projector (can only be used if the global Motion Blur is enabled in the Setup menu).

Active button Enable to activate the projected texture effect in the scene. When disabled, the Projector node remains in the schematic and the projector icon appears smaller in the image area.

Order field Displays the order of projectors. Editable.

Matchbox in Action

While you can use interactive Matchbox shader effects elsewhere in Flame (such as from the timeline or Batch), using Matchbox in Action presents some unique benefits:

- Matchbox effects in Action can be parented to a texture node (or multiple texture nodes). Multiple Matchbox effects can also be added to a texture node. Supported texture maps are Diffuse, Normal, UV, Displacement (HW Only), Parallax, IBL, Specular, Emissive, Lens Flare textures, and Substance textures. These Matchbox effects occur in texture space, after Media list effects but before Action objects, and can be seen as pre-processing effects.
- You can also connect a Matchbox shader to an Action camera, to create a Camera FX that is applied to the whole scene. Camera FX are scene-based post-processing effects.

Keep in mind that the Matchbox XML schema contains tags to identify if the shader is designed as an Action node, a Timeline FX, or a Transition. The tags *ShaderType* and *SoftwareVersion* are added automatically to the XML by the shader_builder script.

The Action Priority Editor (page 565) has a Matchbox Only mode to view and change the drawing order of Matchbox effects.

See the sections below for information on how to add, and things to watch for, when working with each type of Action Matchbox effect.

Matchbox Shaders Connected to Texture Nodes

To add a Matchbox node in Action connected to a Texture node:

1. Do one of the following:
   - With a supported texture node selected in the schematic (Diffuse, Normal, UV, Displacement (HW Only), Parallax, IBL, Specular, Emissive, Lens Flare texture, or Substance texture), double-click the Matchbox node in the All Nodes bin.

     The file browser opens to allow you to select a Matchbox effect.
   - With an Action surface or 3D Geometry node selected in the schematic, double-click a Matchbox preset node from the dynamic Matchbox node bin (page 550).

     The Matchbox node with a parented Diffuse node (with its own parent axis) are added to the scene as children of the selected surface or geometry.

TIP If you add a Matchbox node to the schematic without a supported texture node present, you can parent the Matchbox to an image, and a Diffuse Map node (with accompanying Axis) is automatically added to create a proper Matchbox connection. Matchbox nodes are not supported on Stereo objects.

Matchbox nodes and their links are displayed as black in the schematic to help you identify them. If you added the Matchbox from the dynamic node bin, the name of the effect also appears on the node in the schematic.

2. Double-click the Matchbox node in the schematic to display its menu.

   The Matchbox node populates the user interface dynamically, based on the GLSL (and XML) code, but there are a few UI elements that are constant, located in the Shader tab.

3. If you added a Matchbox effect from the All Nodes bin, you can change the effect by clicking Change Shader in the Shader tab of the Matchbox menu.

4. Use the Action Object Solo (F8) view on the parent texture to cycle through three Matchbox views: Context, Result, and Result Matte. The F8 view also allows you to interact directly in the image window with icons, if they are present in the shader.

TIP Instead of using a source node for a simple front/matte transform, consider using the Front Matte Offset Matchbox (available as a preset, along with many other useful Matchbox effects).

Matchbox Shaders Connected to Cameras (Camera FX)

You can connect Matchbox shaders to the Action camera and have the result of the shader applied to the result of the Action scene. Similar to using Texture Matchbox shaders, the priority editor is used to handle the pipeline order.

NOTE Post-processing (Camera FX, Lens Flares, and Rays) effects are only visible in the Live Preview view. Camera FX can have 2D and 3D widgets visible in the Live Preview view. Only the widget(s) of the currently selected Matchbox are shown. Lens Flares and Rays are rendered at the end of the Camera FX pipeline, when enabled in the Comp output.

GMasks can be connected to a Camera to limit their effect to the post-processing pipeline exclusively. GMask output passes have an option to only show GMasks connected to the current camera, so you can set GMask to not interact with the scene, and only be created to be used with the post-processing pipeline. You can also use a GMask input Matchbox shaders (if they have been implemented to use this input).

The Action post-processing pipeline can connect the Matchbox shader automatically to the appropriate Action output. To support this, shader developers need to set the *InputType* tag properly in the shader XML (each Action output has its equivalent *InputType* tag). This includes RGBA shader rendering. The following input types are supported:

- Front (this defines the Primary Output render layer selection).
- Back
- Matte
- 3D Motion Vectors
- Albedo
- Ambient Occlusion
- Comp (this defines the Action Comp output of the render layer, in case the Primary is another type).
- Emissive
- GMask
- Lens Flare
- Motion Vectors
- Normals
- Occluder
- Position
- Reflection

- Roughness
- Shadows
- Specularity
- UV
- Z-Depth
- Z-Depth HQ (32-bit Z-Depth output packed in two 16-bit channels; you must use the Matchbox API to leverage the data).

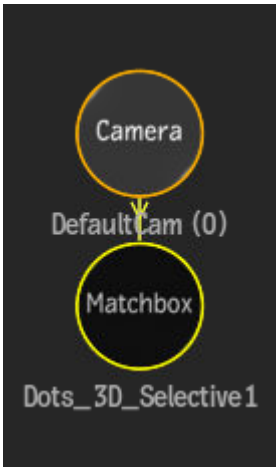
Matchbox Camera FX also have access to the Acton output list in their patch panel (in the Shader tab on the Matchbox menu), so you can manually set the proper connection if the Matchbox shader hasn't been tagged properly in the XML.

To add a Matchbox node in Action connected to a Camera:

1. Do one of the following:
   - With a Camera node selected in the schematic, double-click the Matchbox node in the All Nodes bin.

     The file browser opens to allow you to select a Matchbox effect. Some Matchbox shaders supplied with Flame are already optimized for use as Camera FX; navigate to the ACTION_CAMERA_FX sub-folder.
   - With a Camera node selected in the schematic, double-click a Matchbox preset node from the dynamic Matchbox node bin (page 550).

     The Matchbox node is added to the scene as a child of the selected camera. Once a Matchbox node is parented to a camera, it becomes a Camera FX, and you are not able to connect the Matchbox node to any other node in the schematic (other than other camera nodes).

     Matchbox nodes and their links are displayed as black in the schematic to help you identify them. If you added the Matchbox from the dynamic node bin, the name of the effect also appears on the node in the schematic.

     NOTE Since a Matchbox connected to a Camera node (Camera FX) is considered a post-processing effect in Action, make sure that your image window is set to Live Preview mode to be able to see the results of the shader.

2. Double-click the Matchbox node in the schematic to display its menu.

   The Matchbox node populates the user interface dynamically, based on the GLSL (and XML) code, but there are a few UI elements that are constant, located in the Shader tab.

3 If you added a Matchbox effect from the All Nodes bin, you can change the effect by clicking Change Shader in the Shader tab of the Matchbox menu.

Rendering Camera FX

When rendering a Camera FX pipeline, Action uses Multi-Render Targets (MRT) to allow outputs to be much more efficient. Depending on the Rendering settings, the scene is rendered only once or twice instead of as many times as there are output passes, leading to a significant speed increase in the case of Camera FX requiring many input connections. The biggest speed improvement can be found when the Accumulation Rendering (page 603) box is set to Accumulation On or Accumulation Off, since all passes can then be done using a single MRT. In the case of Accumulation RGBA, two MRTs are needed, which reduces the speed improvement compared to the other Accumulation settings, but still offers a significant boost compared to not using MRT at all.

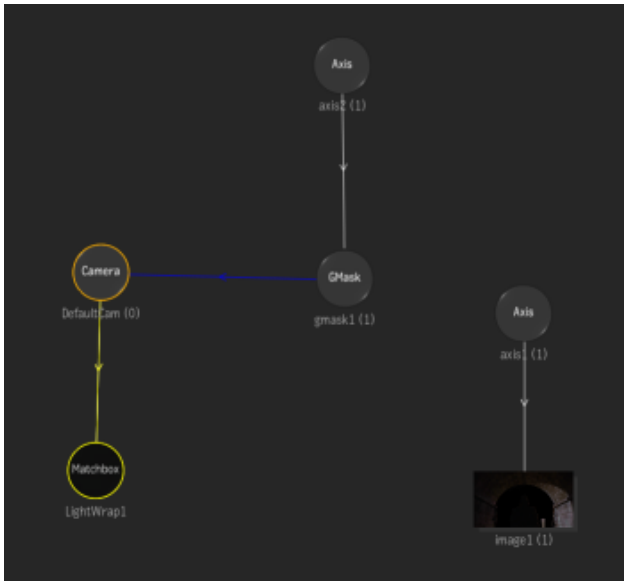
Matchbox in Action Examples

The following examples can help you to get a feel for the workflow of using different types of Matchbox effects in Action.

TIP Post-processing (Camera FX, Lens Flares, and Rays, Blooming, and Ambient Occlusion) effects are only visible in the Live Preview view.

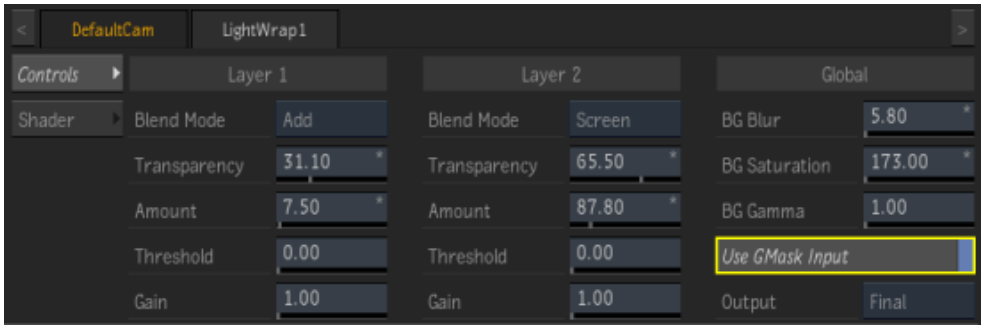
Using a GMask to Limit a Camera FX Matchbox

A GMask connected to the Camera with a GMask link can be used to limit the effect of a Camera FX Matchbox.



In this example, a Gmask is used to limit the effect of a Lightwrap effect that works well for the roof, but is too strong for the character. The Gmask limits the effect and the gradient gradually softens out the Lightwrap, so the top of the head of the character still has a little bit of Lightwrap.

To be able to use a GMask in this manner, you need to used a compatible Matchbox shader and enable Use GMask iInput in the menu. GMask output passes have an option to only show GMasks connected to the current camera, so you can set GMask to not interact with the scene, and only be created to be used with the post-processing pipeline.



Without Lightwrap

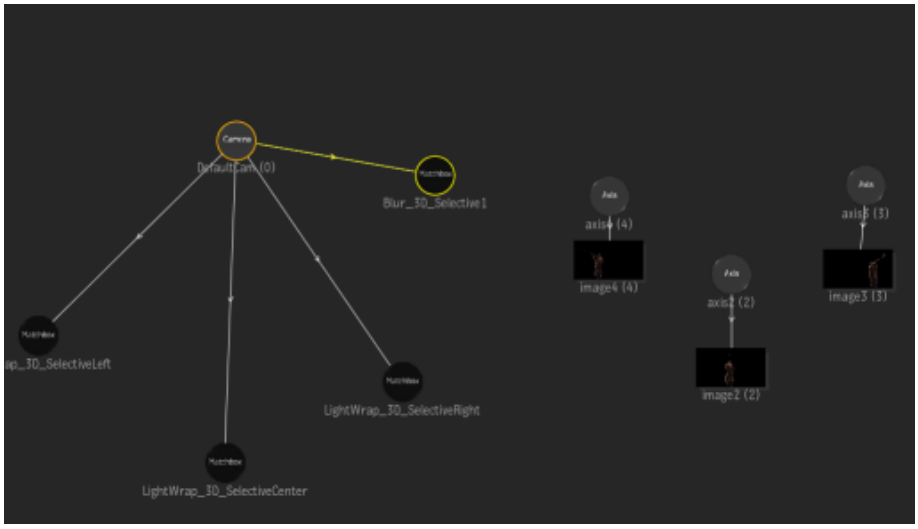
Without GMask

With GMask and Lightwrap

3D Selectives as Camera FX

Some Matchbox shaders include 3D Selective functionality, allowing you to segment the render of Action using different 3D techniques, applying the given effects to a specific portion of the image.

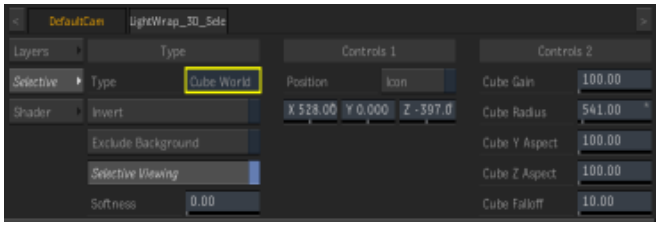
The following example shows the usage of 3D Selective with both the Lightwrap and Blur 3DSelective shader. The Blur is used to create a DepthOfField effect using the Distance mode, while three different Lightwrap3DSelective shaders are used to create independent Lightwrap effects on each of the trumpeters, as single values for all three didn't worked out in this case.



3D Selective on one trumpeter

Result

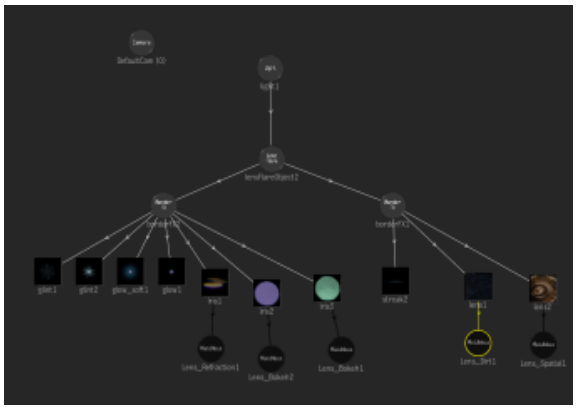
For each Lightwrap, the Cube World Selective type was used to isolate each trumpeter.



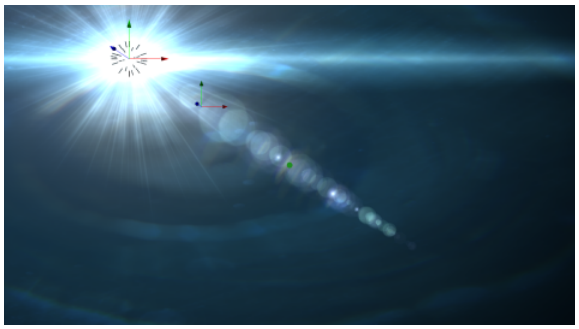
Replacing Lens Flare Elements with Matchbox Effects

You can use the Matchbox Lens texture grid shaders with Action Lens Flares. You can use the LensBokeh, LensRefraction, and LensOptic Matchbox to replace Glint Texture, Glow Texture, Iris Texture, Ring Texture

and Streak Texture nodes to create more organic Lens Flare results; and you can use LensDirt and LensSpatial Matchbox to replace the Lens Texture to create more organic lens dirt and light leak type of effects.

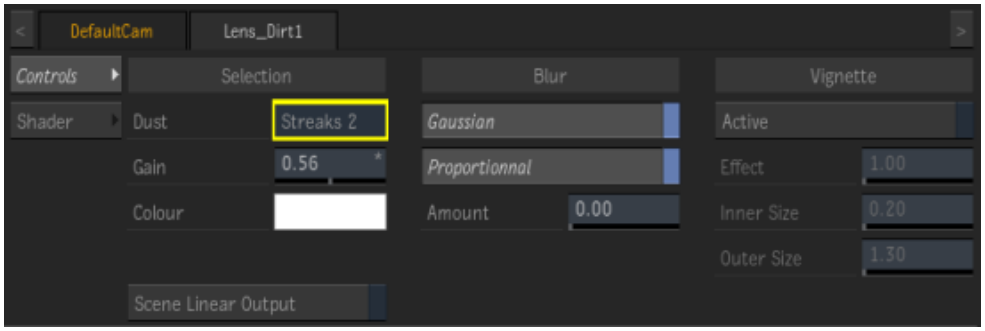


Lens Flare schematic with Matchbox replacements



Result

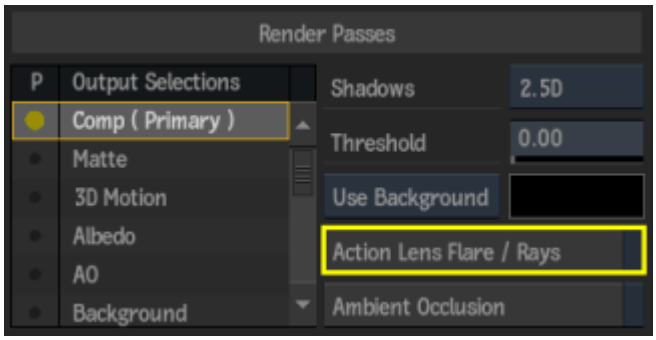
In each Matchbox Lens effect's menu, you can use the texture grid functionality to select different looks. In this example, You can select different Dust elements for the Lens Dirt effect.



Using a Matchbox Lens Flare in the Camera FX Pipeline

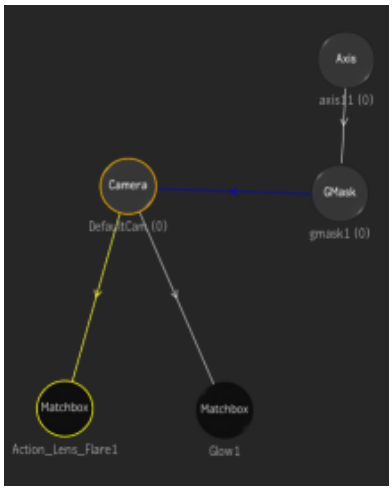
You can also use a Lens Flare Matchbox shader as part of the Camera FX pipeline. Since Lens Flares (and Rays) are rendered at the end of the Camera FX pipeline, you can use instead the ActionLensFlare Matchbox shader, which allows you to put the Lens Flare result anywhere in the Camera FX pipeline.

For this use case, make sure that you disable the Action Lens Flare / Rays button from the Comp output Render Passes menu:

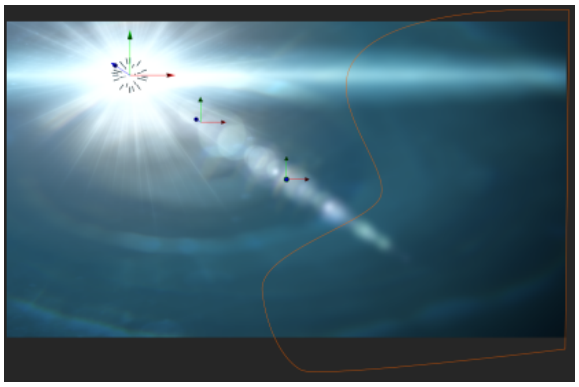


In this example, there is also a Matchbox Glow effect in the Camera FX pipeline, as well as a GMask that allows you to blur part of the Lens Flare result without affecting the Action result. The Glow can affect both

the Lens Flare and the Action result, since the ActionLensFlare Matchbox allows you to place it anywhere in the rendering pipeline (and in this case before the Glow effect).



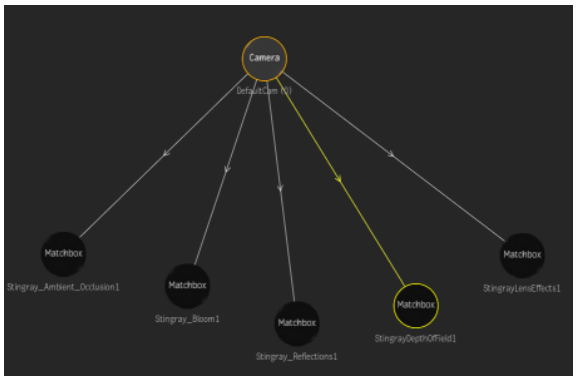
Lens Flare Matchbox as Camera FX



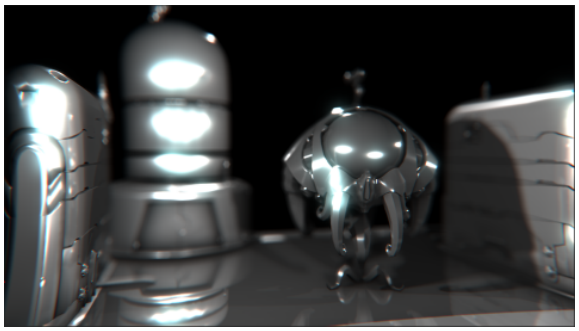
Result

Using Stingray Matchbox Shaders

Stingray Matchbox shaders are based on the Autodesk Stingray game-engine technology, allowing you to produce visually stunning results in Action. These shaders can be combined (as in the Camera FX example below). You can also use GMasks and 3D Selectives with the Stingray shaders to help you limit the effects to certain areas of your image.



Stingray Matchbox Shaders as Camera FX



Result

These are the available Stingray Matchbox shaders, as well as some usage tips:

Stingray Motion Blur This shader allows you to create motion blur effects, based on the motion vector provided (either by the Action Motion Vector output or the Batch Motion Analysis node). This shader can also use a Z-Depth input to help provide better results. When used as Camera FX in Action, it can automatically receive the current Camera Near/Far values, when Frustum is set to Auto, otherwise it can be set manually. The Exposure field is available to allow control on how much motion blur to create. While this tool can create quite nice small and subtle motion blur effects, it cannot create large motion blur effects due to the nature of the technique used; therefore the Exposure value is limited to 10.

Stingray Ambient Occlusion This shader allows you to create lighting occlusion, by computing how each point in a scene is exposed to ambient lighting. To do so, the shader requires the Z-Depth and Normals information of a 3D scene. You can control its impact in the scene by modifying the Intensity, Radius, and

Quality fields. The Use Z-Depth setting allows you to use a normal Z-Depth input, since, by default, the shader is expecting the Z-Depth HQ Action output for maximum quality (however a regular 16-bit Z-Depth input can be used). The Only AO setting allows you to output just the AO result without the scene result. When used as Camera FX, you can use both the Use GMask Input and the Selective options to tailor exactly where and how much ambient occlusion effect you want in your image. Multiple Stingray Ambient Occusion shaders can then be added, using different values and affecting different areas of the scene.

Stingray Bloom This shader allows you to reproduce an imaging artifact of real-world cameras, where bright areas create feathering (when overwhelming the camera capturing the scene). By default, the shader is set to use the Stingray Tone Map, which handles geometry content, so in the case of an image sequence, you may prefer using the Scene Linear Input option instead, if applicable. The first step of the Bloom shader is to create the bright pass which can be isolated using the Bright setting of the Pass option. To control how much of the image is contributing to the bright pass, you can adjust the Threshold and the Exposure settings. This bright pass becomes the source of the blooming effect to which the Blur Amount, Falloff, Offset, Bloom Colour, and Gain settings are then applied. When used as Camera FX, you can use both the Use GMask Input and the Selective options to tailor exactly where and how much Bloom effect you want in your image. Multiple Stingray Bloom shaders can then be added, using different values and affecting different areas of the scene.

Stingray Reflections This shader allows you to create single-bounce screenspace reflections, based on the Z-Depth and Normals information of a 3D scene. The shader uses the roughness information of the material through a Roughness map to control how sharp or diffuse the reflection is. To control this in Action, you can use the Shader node with either the Physically Based or the Cook-Torrance shading mode. The basic Reflection settings are the Intensity, Tint, Gain, and As Additive Ambient (which allows you to add the reflection as ambient light or blend it to the scene). The Use Z-Depth setting allows you to use a normal Z-Depth input, since, by default, the shader is expecting the Z-Depth HQ Action output for maximum quality (however a regular 16bit Z-Depth input can be used). The Only Reflections Output setting allows you to output just the Reflection result, without the scene result. You can use the Screen Edge and Surface Thickness Threshold settings to help control where the reflections are going to appear in the image. Since the Stingray Reflection shader is a single-bounce reflection, you may need to isolate which objects cast or which objects receive reflections, to get around some limitations of the technique. To do this, you can apply the shader as a Camera FX, and use the Casting and Receiving Selective settings. You can also use the GMask functionality to control the reflection further.

Stingray Depth of Field This shader allows you to create focus range blur effects. Use the Centre Z setting for focus position, and the Range setting to define the size of the focus area. Both of these settings are in Action camera-space coordinates. To control the Blur, you have independent control on the Near and Far side of the focus area. Max sets the blur size, while Distance sets how quickly you will reach the Max blur value, starting from the corresponding focus area position. You can further impact how the blur is going to behave in the defined Distance region, by adjusting the curves found in the Advanced tab section. By default it is set to linear, but it can be freely adjusted. Lastly, you can change the chromaticity of the effect by setting a colour to both the Far and Near blur area which affects the RGB channels independently.

Stingray Lens Effects This shader is a simple lens distort/correction tool with chromatic aberration capabilities. The Distortion setting distorts/corrects the image with a simple lens model, while the Fringe Intensity and Colour settings allow you to control the chromatic aberration of the lens. When used as a Camera FX, you can use the Use GMask Input option to only apply the overall effect to a portion of the image.

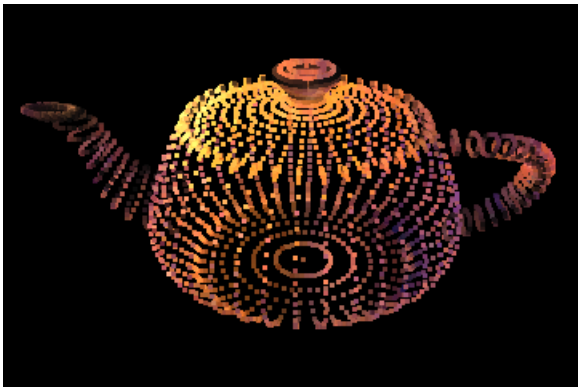
Action Atomize

When you attach an Atomize node to a surface or geometry in Action, the object becomes a configurable point cloud. Pixels at each vertex of the surface or geometry become square or round points. The Atomize points always face the camera, and are displayed at the same size regardless of the distance from the camera. Atomize points inherit any properties of the parent object, as well as any attached maps (such as transparency, blend modes, and colours).

NOTE Atomize points are displayed in Action screen space, therefore they are affected by zooming in the image window.

Depending on the vertex resolution of the parent surface or geometry, the Atomize result can become quite detailed (more vertices equals more points), allowing you to creatively incorporate point clouds into your Action compositions.

Teapot Geometry with Diffuse Map



Teapot with Atomize applied

Adding an Atomize Node

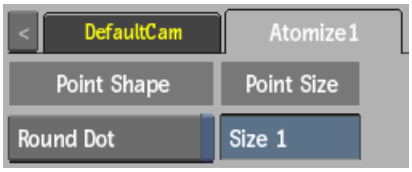
To add an Atomize node:

1. In the schematic, select the surface or 3D geometry to which you want to apply the Atomize.
2. Do one of the following:
   - Drag the Atomize node from the node bin and place it in the schematic.
   - Drag the Atomize node from the node bin and place it where you want it in Result view.
   - Double-click the Position Map node. You do not need to be in Schematic view to add a node in this manner.

   The Atomize object is added to the schematic as a child of the selected surface or geometry.
3. Double-click the Atomize node in the schematic, or follow the tab population rules for the Object menu.

   The Atomize menu appears.

Atomize Menu Settings



Round Dot button Enable to display the points as circles; disable to display as squares.

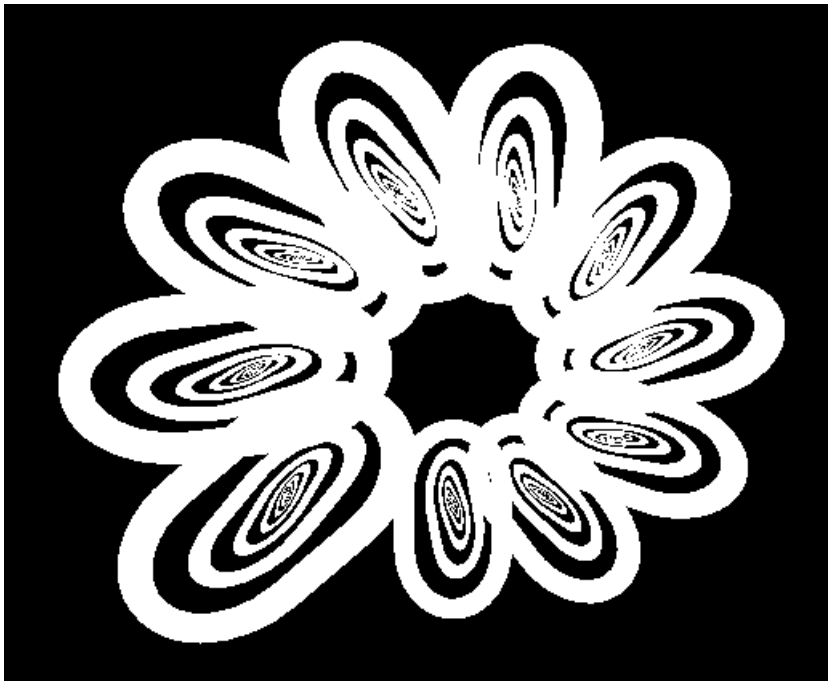
Size field Displays the size of the points. Editable.

NOTE The size range is from 0 to 100, but the actual maximum value that will produce a change is determined by your graphics card. For example, you may notice that the size doesn't increase with a value higher than 64.

Action Replica

The Replica node allows you to create multiple instances of most Action objects (including the Replica node itself), then introduce cascading animatable offsets (in position, rotation, scale, centre, and shear). Think of Replica as a macro that saves you the time of creating multiple Axis nodes with mimic links and individual offsets.

The following is a simple one-character 3D Text geometry with two Replica nodes creating the effect:

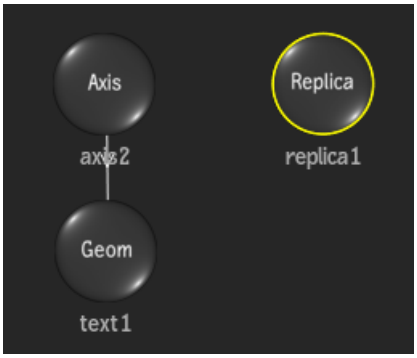


Adding a Replica Node

To add a replica node to the scene:

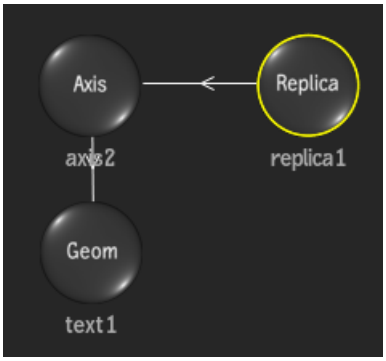
1 Do one of the following:
- Drag the replica node from the node bin and place it in the schematic.
- Drag the replica node from the node bin to Result view.
- Double-click the replica node. The node appears next to the last added object. You do not need to be in Schematic view to add a node in this manner.

An icon representing the replica is added to the schematic.



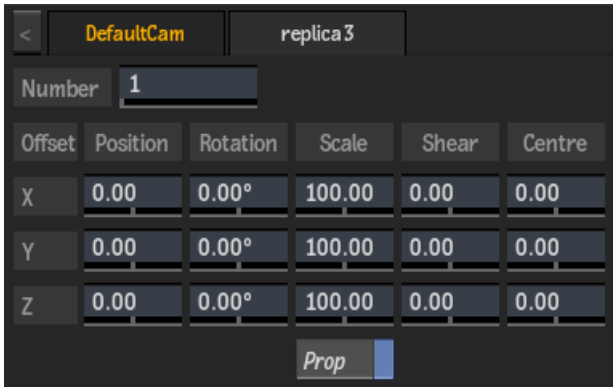
Unlike many objects, a replica node is added without an axis.

2 Parent the replica node to the object that you want to work with.



3 To display the Replica menu, double-click the selected replica node in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 571).

Replica Menu Settings



Number field Displays the number of instances of the attached object to create. Editable. Each instance creates its own entry in the Action Priority Editor, so you can change the drawing order of instances.

X Position field Displays the position offset along the X axis. Editable.

Y Position field Displays the position offset along the Y axis. Editable.

Z Position field Displays the position offset along the Z axis. Editable.

X Rotation field Displays the rotation offset along the X axis. Editable.

Y Rotation field Displays the rotation offset along the Y axis. Editable.

Z Rotation field Displays the rotation offset along the Z axis. Editable.

X Scale field Displays the scale along the X axis. Editable.

Y Scale field Displays the scale along the Y axis. Editable.

Z Scale field Displays the scale along the Z axis. Editable.

Proportional button Enable to change the fields proportionally.

X Shear field Displays the shear along the X axis. Editable.

Y Shear field Displays the shear along the Y axis. Editable.

Z Shear field Displays the shear along the Z axis. Editable.

X Centre field Displays the centre offset along the X axis. Editable.

Y Centre field Displays the centre offset along the Y axis. Editable.

Z Centre field Displays the centre offset along the Z axis. Editable.

Action 3D Paths

A 3D path is an animatable 3D spline that you attach to other Action objects, such as surfaces, geometries, 3d text, cameras, or lights. The attached objects then follow the spline based on the path normals, allowing you to create effects, such as a 3D roller coaster.

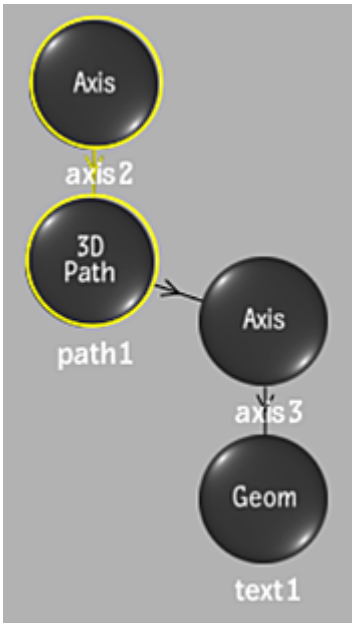
Adding a 3D Path Node

When you add a 3D Path node to your Action schematic, the node is added with an axis.

To add a 3D Path node to the scene:

1. Do one of the following:
   - Drag the 3D Path node from the node bin and place it in the schematic.
   - Drag the 3D Path node from the node bin and place it in Result view.
   - Double-click the 3D Path node. You do not need to be in Schematic view to add a node in this manner.

   A Path object (called path1, by default), with its parent axis, appears in the schematic.
2. Create mode is automatically selected in the Tools box so you can create your spline. See Creating 3D Path Splines (page 835).
3. Parent the 3D Path node to another object in your schematic, such as a Light node or a 3D Text node. The 3D path becomes part of the transformation hierarchy of the attached object.



3D Path node parented to a 3D Text node

4 To open the 3D Path menu, double-click the 3D Path node in the schematic, or follow the tab population rules for the Object menu.

See Populating Menu Tabs of Selected Objects (page 571).

Creating 3D Path Splines

You can draw open or closed splines to use as your 3D path.

To create a spline:

1 Make sure that the Tools box is in Draw Shape mode (this is the default when you first add a 3D Path node).

2 In the image window, click to add vertices.

TIP `Shift`-drag to add freehand segments to the spline. Vertices are added where you drag, and appear when you release `Shift`. After closing or finishing the spline, you can use the Lasso Fit field to increase or decrease the number of vertices that define the freehand segments of the spline.

3 To complete your spline, do one of the following:

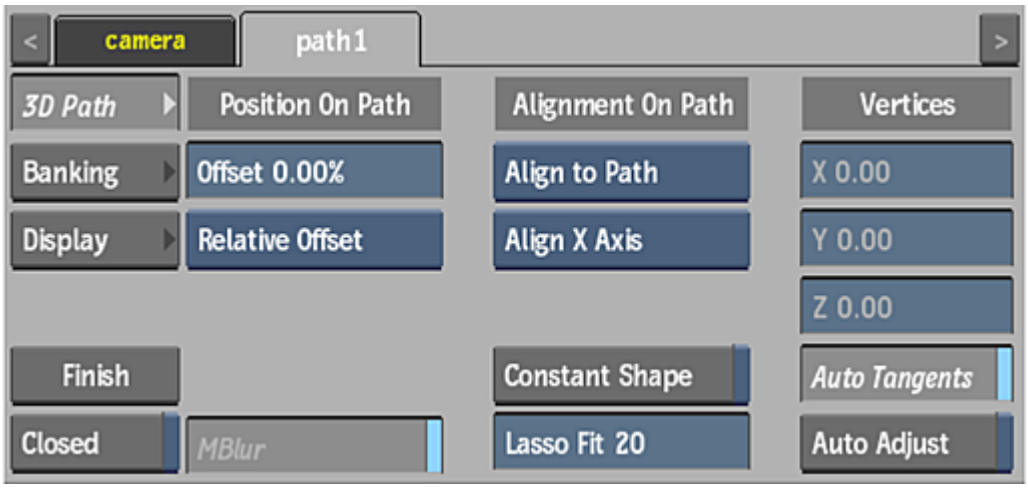
- Click the first vertex to close the spline.
- Click Finish in the 3D Path menu to leave the spline open. If you decide later that you want to close the spline, enable Closed in the 3D Path menu.

4 Make sure that the Tools box is in Select mode, so that you do not add more vertices by mistake.

When the spline is closed or finished, its vertices and tangents can then be edited.

Editing Splines

Use the settings in the 3D Path tab to work with the spline you created. You edit splines in the same way as you edit garbage masks, working with their vertices and tangents. See Manipulating Vertices and Tangents (page 1111).



Finish button Click to finish an open spline.

Closed button Enable to close a spline from the last to first point.

Motion Blur button Enable to use a motion blur effect for the selected path (can only be used if the global Motion Blur is enabled in the Action Setup menu).

Constant Shape button Enable to modify the spline's shape without setting keyframes. This forces all animatable parameters to be set for the whole clip rather than for only the current frame. It also removes any existing keyframes and applies the shape of the current frame to the rest of the clip.

Lasso Fit field Displays the number of points in the segments of the spline that are drawn freehand. Use a lower number to simplify the curve by removing vertices and tangents, resulting in a smoother curve. Editable.

X Vertex field Displays the position of the selected vertices on the X axis. Editable.

Y Vertex field Displays the position of the selected vertices on the Y axis. Editable.

Z Vertex field Displays the position of the selected vertices on the Z axis. Editable.

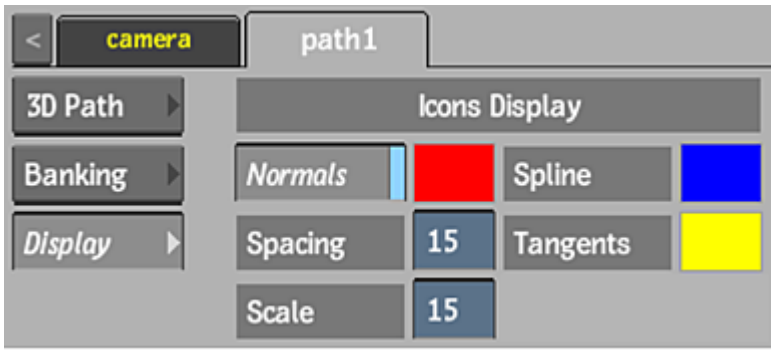
Auto Tangents button Enable to position a tangent for each vertex set, to create a smooth curve between the vertices. When enabled, it is possible to create a spline with both straight and curved segments.

When Auto Tangents is disabled, the tangents are positioned under the vertex, resulting in straight lines between vertices. When you draw freehand segments in a spline with Auto Tangents off, vertices are added with broken tangents, allowing the spline to follow your cursor movement.

Auto Adjust button Enable to automatically adjust the tangent handles of the two adjacent vertices when moving vertices, to create smooth curves between the vertices.

Changing Spline Display Properties

You can change spline display properties in the Display tab. For example, you can change the colour of tangents on the splines you draw. This is useful to better contrast the spline's tangents from the clip so that they are easier to work with.



Normals button Enable to display normals along the 3D path.

Normals colour pot Displays the normals colour. Editable.

Spacing field Displays the space between the displayed normals, in pixels. Also used to calculate the position of the object on the path. A lower value may result in better positioning and smoother movement of the object, but rendering may be slower. Editable.

Scale field Displays the scale of the displayed normals, in pixels. Editable.

Spline colour pot Displays the colour for the display of splines. Editable.

Tangents colour pot Displays the colour for the display of tangents. Editable.

Positioning Objects on the Path

Use the Position On Path settings in the 3D Path or Banking tabs to offset the position of attached objects along the path and beyond. For open paths, if you offset past the first or last points on the path, the position is extrapolated accordingly. For closed paths, if you offset past the first or last points on the path, the attached object continues on the path with a tangent interpolated from the first and last normals.

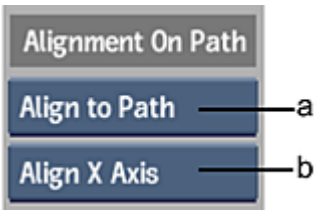
Offset field Displays the amount of offset to apply to the object on the 3D path. Use to animate the attached object along the path. Editable.

Offset box Select whether to offset the object from the path in a relative (expressed as a percentage of the path) or absolute mode (expressed in pixels).

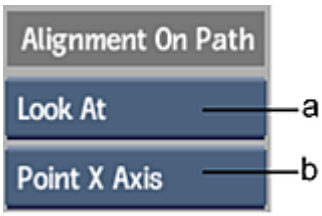
NOTE These settings are repeated in the 3D Path and Banking tabs to make it easier for you not to have to switch tabs to change the settings. The same settings are reflected in both tabs.

Aligning Objects on the Path

Use the Alignment to Path section of the 3D Path tab to set orientation behaviour.

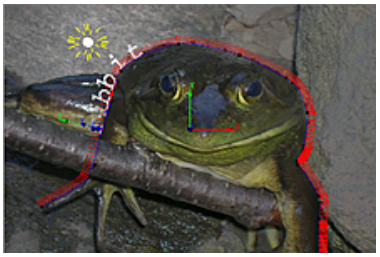


(a) Alignment option box (b) Orientation Axis box



(a) Alignment option box (b) Point Axis box

Alignment option box Select how the object connected to the 3D Path node aligns to the path.

Select:	To:	Example:
Align Off	Not align the attached object to the path. In this case, the Orientation Axis box and banking controls are unavailable. This can be useful for vertical text effects.	
Align to Path	Align the attached object to the 3D path. You can then select which axis is aligned to the path in the Orientation Axis box, and use the banking controls.	
Look At	Point the attached object to a look-at object, such as a light. You can then select which axis is pointed to the look-at object in the Point Axis box, and use the banking controls. See Applying a Look-At Connection (page 839).	

Orientation Axis box Select which axis is aligned to the path. Available when Align to Path is selected in the Alignment option box.

Point Axis box Select which axis is pointed to the attached look-at object. Available when Look At is selected in the Alignment option box.

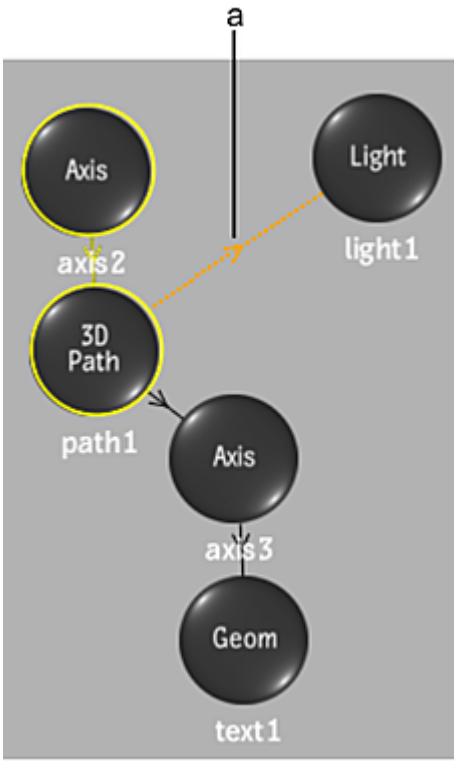
Applying a Look-At Connection

You can create interesting 3D path effects by attaching a look-at connection between the path and another object in your scene. The attached object on the path then rotates to face the look-at object, no matter where it is positioned. You attach a look-at connection in the schematic between the 3D Path node and any object with axis characteristics (Axis, Camera, Light, Projector, Particle Animator).

To apply a look-at connection:

1. Do one of the following:
   - Select Look At in the Alignment option box.
   - Select Look At in the Tools box.
2. In the schematic, drag from the 3D Path node to an object with axis characteristics.

   The selected object is connected to the 3D Path node by an orange dotted line with an arrow.

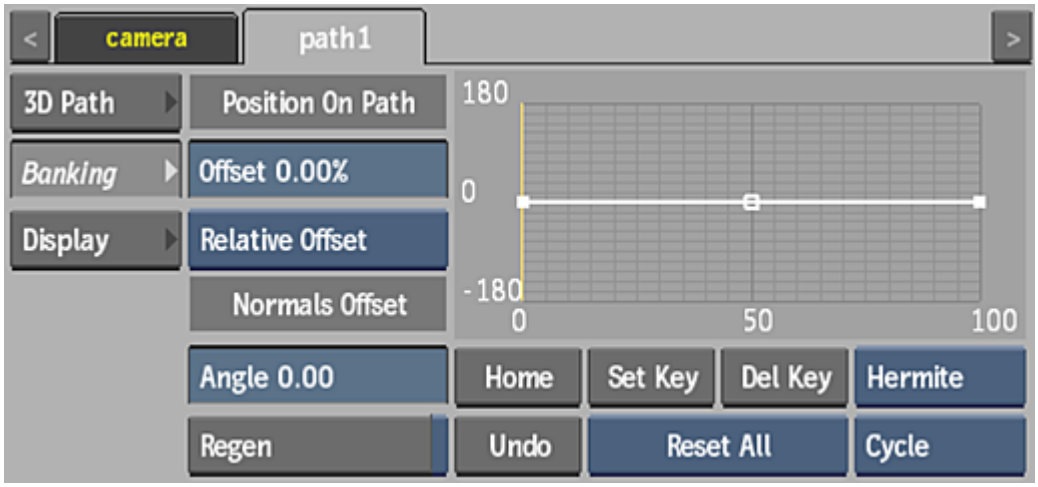


(a) Look-at connection

3. Select which axis looks at the attached object in the Point Axis box.
4. Optional: Use the banking curve to orient the attached objects.

Using the Banking Curve

When Align to Path or Look At is selected in the Alignment option box, the Banking tab becomes available. Banking uses the normals of the path to orient objects attached to it. Use the banking curve and settings to control the torsion effect of the object as it travels along the 3D path.



The Position on Path settings are the same as those in the 3D Path tab. See Positioning Objects on the Path (page 837).

Banking Curve Adds twists and torsion to the normals of the 3D path. Use the options in the Tools box to add, select, delete, or move keyframes on the Banking curve. The horizontal axis represents the length of the path, and the vertical axis displays the orientation, expressed in degrees.

Normals Offset Angle field Displays the angle of rotation of all normals, applied to the entire banking curve. Editable.

NOTE Changes made to the banking curve and Angle field are cumulative.

Regen button Enable to dynamically refresh the image as changes are made to the banking curve.

Home button Resets the Banking curve viewer to show the whole curve.

Undo button Undoes Banking curve operations.

Set Key button Sets the current values for the banking curve in the current frame (when Auto Key is disabled).

Del Key button Deletes the selected banking curve keyframes.

Reset Selection box Select whether to reset all of the banking settings (Reset All) or just the banking curve (Reset Key).

Interpolation box Select the default interpolation type for the Banking curve.

Extrapolation box Select the default extrapolation type for the Banking curve.

Action Cameras

The scene is what you see through the camera lens. Typically, you work with the camera to frame and animate the view to achieve the effect you want. In Action, you have the choice of using the automatic camera or the manual camera, whose F-Stop, film size, and focal length you can set yourself. You can also animate specific camera properties.

Adding a Camera

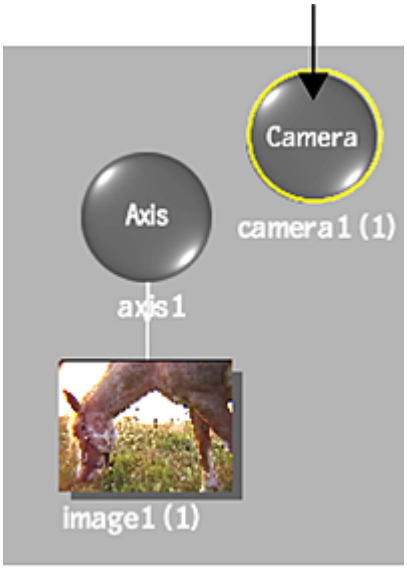
By default, a camera exists in the Action scene (you may need to pan in the schematic to see the camera node). The Z position of this default camera is placed relative to the resolution of your Action scene. You

can add multiple cameras to change point of view or depth of field from one camera to another. Add and animate multiple cameras when creating compositions. You can also switch from one camera to another at any point.

To add a camera:

1 Do one of the following:
- Drag the camera node from the node bin and place it in the schematic.
- Drag the camera node from the node bin to Result view, so you can see its effect on the scene before placing it exactly where you want.
- Double-click the camera node. You do not need to be in Schematic view to add a node in this manner.
- With the cursor in the viewport, press `Space-C` to add a new camera node that uses the current view settings.

A new camera is added to the scene. An icon representing the camera is added to the schematic.



2 To display the Camera menu, double-click the selected camera in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 571).

The result camera always appears as the first tab on the right side of the Object menu. To allow you to easily access the camera without losing your place in the scene, this special camera tab (appearing in orange) does not follow the tab population rules. If a camera node is selected in the schematic, the special Camera tab does not appear, and the normal tab population rules apply.

TIP You can attach a look-at connection (page 579) between Action objects with axis characteristics (Axis, Camera, Light, and Projector). The connection allows the parent object to rotate to face the child look-at object, no matter where it is positioned. Look-At links can be animated; therefore you can link different objects from the parent at different frames. In the channel editor, you can see a lookAt channel for every look-at parent.

Camera Menu Settings

When accessing Action as a Timeline FX, you have access to a quick menu with some of these Camera settings. To see the full Camera menu, click the Editor button to enter Action.

Camera Tab

Camera Type box Select Free (to view the scene in the direction that you aim the camera), or Target (to aim the camera at a target object in the scene based on a point of interest).

Free cameras are easy to use because you do not have to manipulate the point of interest. You can simply animate the camera rotation or camera tilt as though it were on a tripod. Use the Distance field in conjunction with Free Camera.

Field of View field Displays the camera field of view value, measured in degrees. Editable.

When Physical Camera is inactive (in the Fog /DOF tab), use the field of view angle in the Y direction to adjust the width of the camera frustum. When Physical Camera is active, use the focal length for the same purpose.

Near field Displays the near view of the selected camera. Editable.

Far field Displays the far view of the selected camera. Editable.

See Moving the Clipping Planes (page 845).

X Position field Displays the position of the camera eye on the X axis. Editable

Y Position field Displays the position of the camera eye on the Y axis. Editable

Z Position field Displays the position of the camera eye on the Z axis. Editable

Motion Path button Enable to animate the camera eye on a motion path. See Moving the Camera Eye and Point of Interest (page 845).

X Rotation field Displays the level of camera rotation along the X axis. Available when Free is selected in the Camera Type box. Editable.

Y Rotation field Displays the level of camera rotation along the Y axis. Available when Free is selected in the Camera Type box. Editable.

Z Rotation field Displays the level of camera rotation along the Z axis. Available when Free is selected in the Camera Type box. Editable.

Distance field Displays the position of the camera's focus. Available when Free is selected in the Camera Type box. This setting affects the depth of field (when enabled in the Action Setup menu). Editable.

X Point of Interest field Displays the position of the point of interest on the X axis. Available when Target is selected in the Camera Type box. Editable.

Y Point of Interest field Displays the position of the point of interest on the Y axis. Available when Target is selected in the Camera Type box. Editable.

Z Point of Interest field Displays the position of the point of interest on the Z axis. Available when Target is selected in the Camera Type box. Editable.

Roll field Displays the amount of camera roll (available with Target Camera). Available when Target is selected in the Camera Type box. Editable.

Parenting Offset box Select an offset option for viewing an image when parenting a camera node. Origin sets the image to the camera origin; Target sets the image to the default viewplane distance relative to the camera; and Live Target sets the image to the current viewplane distance based on the FOV.

Parenting Offset field Displays the offset value, as computed from the default camera field of view and the default image size. This value does not change even if other camera parameters are changed. Non-editable.

Result Camera box Specify which camera is active. The active camera is the one that will be used when processing/rendering your scene.

Result Camera field Displays the active camera number. Non-editable.

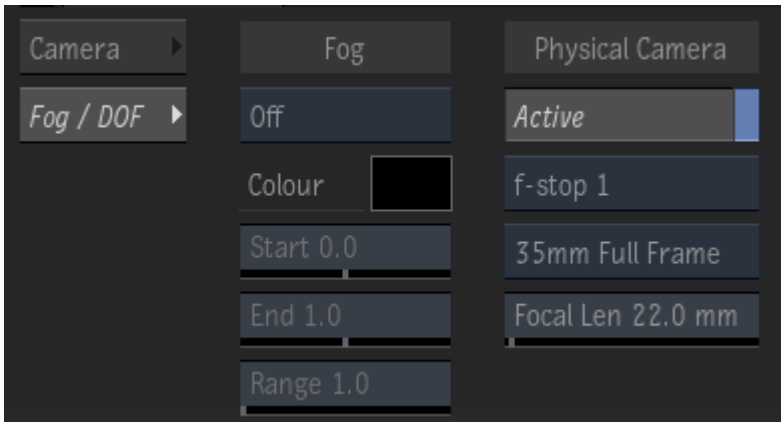
Import Camera button Opens the Import Camera file browser to import a camera. See Importing and Exporting Cameras (page 846).

Export Camera button Opens the Export Camera file browser to save a camera. Exported data includes Position X, Y, and Z; Poi X, Y, and Z; FOV; Roll, Target or Free Camera; motion path and explicit keyframe camera animation.

See Importing and Exporting Cameras (page 846).

Reset Camera button Resets the Camera menu to its default settings.

Fog / DOF tab



Fog box Select a fog type to cause objects to fade as their distance from the camera increases. See Applying Fog to the Scene (page 855).

Fog colour pot Displays the fog colour. Editable.

Fog Start field Displays the distance at which the fog should start. Editable.

Fog End field Displays the distance at which the fog should end. Editable.

Fog Range field Displays the range or distance from the camera for exponential type fog. Editable.

Physical Camera button Enable to activate the physical camera, which simulates a manual camera. Use the physical camera fields to adjust the field of view for the camera.

f-stop box Select a depth of field value. F-stop is the ratio between the focal length of the lens and the diameter of the aperture. For example, the F-stop value for a 100mm lens with a 50mm full aperture is 2 (or f/2).

Large f-stop values correspond to smaller apertures. The depth of field for your physical camera increases as you increase the f-stop value.

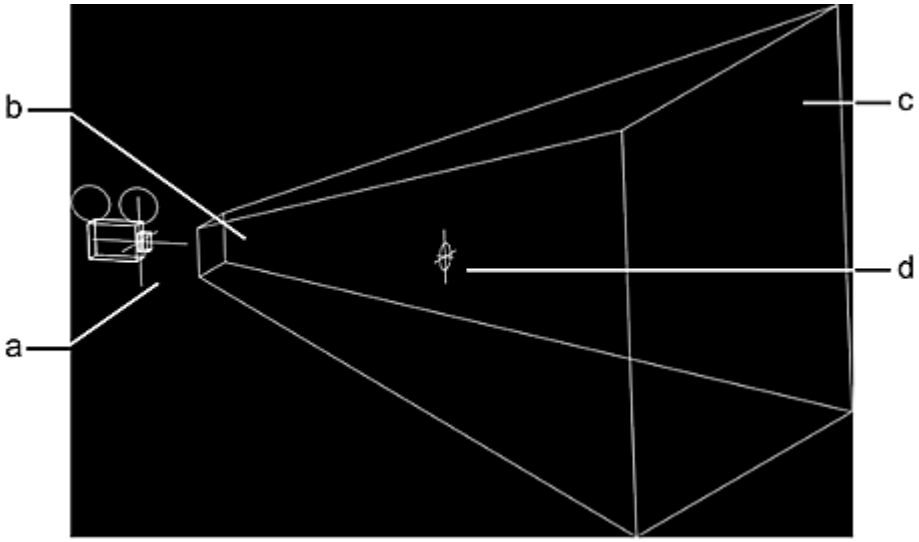
Film Size box Select a standard film size. As the film size increases, the focal length also increases and the focal point moves further away from the camera eye. Changing the film size also increases the focal length.

Focal Length field Displays the field of view. Changes to the Focal Length automatically update the field of view (based on the film size) and vice versa. Editable.

As the focal length increases, the field of view decreases. As the focal length decreases, the field of view increases.

About the Frustum

The volume of space viewed by the camera eye is called the frustum. The frustum is in effect a viewing pyramid. The camera eye is located at the apex of the pyramid, and the far clipping plane forms the base. The pyramid may be truncated by the near clipping plane.



(a) Camera eye (b) Near clipping plane (c) Far clipping plane (d) Camera interest point or look-at point

If you place a surface within the frustum, it is visible in the final animation. If the surface is located outside the scope of the frustum, it is not visible at that frame in the animation.

Viewing the Camera and Frustum

To see the camera and frustum:

1. From the View box, select Side.
2. In the image window controls, click

   to zoom out from the scene.
3. Select Pan in the Tools box and pan around the scene until you see the camera eye icon. Alternatively, use Orbit mode to pan around the scene in circular motion.
4. Go to the Camera menu and drag the Roll field until you see the four sides of the frustum.

5 To modify the frustum, do one of the following:

- Change the position of the near clipping planes to alter the depth of the frustum. See Moving the Clipping Planes (page 845).
- Change the position of either the camera eye or the camera's point of interest to alter the orientation of the frustum. See Moving the Camera Eye and Point of Interest (page 845).
- If the Physical Camera button is disabled, enter a value in the FOV (field of view) field to adjust the width of the camera frustum.
- If the Physical Cam button is enabled, enter a value in the Focal Length field to narrow or widen the frustum. You can also alter the depth of the frustum using only the near clipping plane.

Moving the Clipping Planes

The camera frustum is determined by six clipping planes: the left, right, top, bottom, near, and far clipping planes. The depth of the frustum is affected by the near and far clipping planes. The values for these channels are expressed in units relative to the position of the camera eye.

To move the clipping planes:

1 From the View box, select Side, Front or Top depending on how your camera is positioned. Ideally, you will want a view that profiles the camera so the near and far planes are visible.

2 In the Camera menu, enter a value in the Near field to edit the near clipping plane's position.

The value in the Near field corresponds to the position of the near clipping plane. The default value is 1. Any object between the camera eye and the near clipping plane is outside the camera frustum and does not get processed in the final result.

3 Enter a value in the Far field to edit the far clipping plane's position.

The value in the Far field corresponds to the position of the far clipping plane. The default value is 10000. Any object positioned behind the far clipping plane is outside the camera frustum and does not get processed in the final result.

4 Narrow or widen the frustum by modifying the camera's FOV (field of view). Increasing the FOV narrows the frustum and field of view. Decreasing the value widens the frustum and field of view.

Moving the Camera Eye and Point of Interest

Objects in the scene can be recorded from an arbitrary position as determined by the orientation of the camera eye in world space.

To change the position of the camera eye:

1 In the Camera menu, modify the Position X, Y, and Z fields.

You can also animate the camera eye on a motion path. Enable the Motion Path button and drag the camera icon while viewing the scene in Top, Side, or Front view.

The camera point of interest is the point in world space at which the camera eye is directed. The point of interest is always at the centre of the camera's frustum. Changing the position of the point of interest causes the orientation of the frustum to change. You can take advantage of the relationship between the point of interest and the frustum to make the camera follow a moving object. To do this, animate the point of interest while keeping the camera eye in a fixed position.

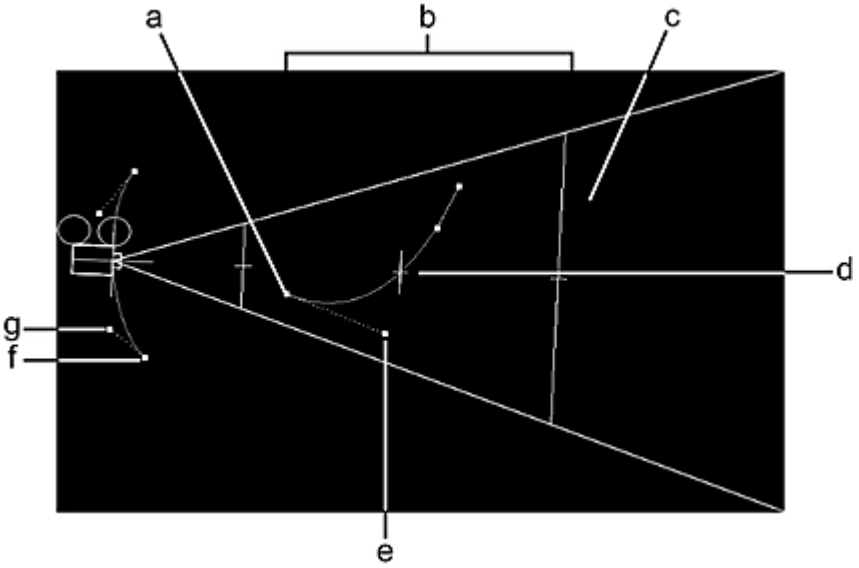
To change the point of interest:

1. In the Camera menu's Camera Type box, select Target.
2. Do one of the following:
   - Modify the Interest X, Y, and Z fields.
   - Move the point of interest by dragging while viewing the scene in Top, Side, or Front view.

TIP You can animate the point of interest using a motion path by enabling the Motion Path button.

Adjusting the Depth of Field

Using depth of field, you can blur objects that are outside the camera's focal range. The further an object is from the focal range, the more it is blurred.



(a) Near sharpness plane (b) Focal range (c) Far sharpness plane (d) Motion path for point of interest (e) Camera point of interest (f) Motion path for camera eye (g) Camera eye

The Target camera uses depth of field and clearly indicates its focal range when in Top, Side, or Front view. With the Free camera, you can change the field of view by `Alt`-dragging the near sharpness plane. This changes the camera only; what you are viewing remains the same size.

To adjust the depth of field:

1. In the Action menu, click Setup.
2. In the Accumulation Settings section, enable Depth of Field. Objects are blurred on either side of the point of interest.
3. Change the amount of blur using the Depth of Field Softness field.

Importing and Exporting Cameras

You can either import a camera you created and edited in Action, or import one from another 3D application. Once a camera is imported, you can edit its parameters, change its animation, and then export it back to the application it came from.

FBX Cameras

Flame supports the import and export of 3D data saved in the FBX 3D format. This format provides a means for exchanging 3D data for scene compositions—such as cameras—between tools and packages developed by different manufacturers.

To export a camera to FBX format:

1. Select the camera you want to export.
2. In the Camera menu, click Export.
   The Export Camera file browser appears.
3. From the Export Type box, select FBX.
4. Set the FBX Scale and Export Options, as needed.
5. Navigate to the location where you want to export the camera animation.
6. Enter a name for your exported camera in the file field.
7. Click Save.

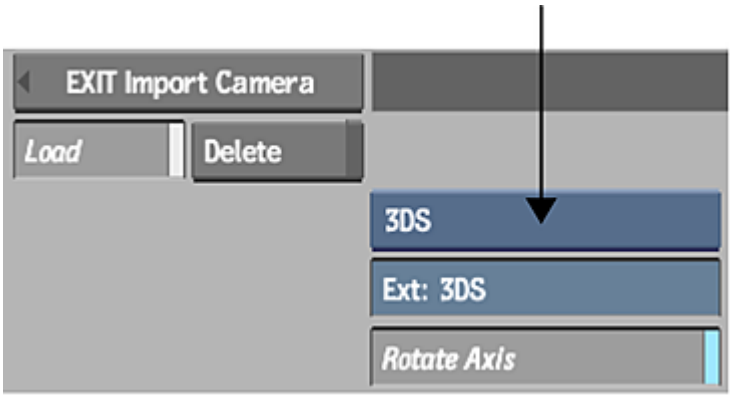
3ds Max Cameras

An alternative to animating a camera's position is to import a camera animation from 3ds Max. 3ds Max camera animation can be saved as a . 3DS or .ase (ASCII Scene Export) file in your scene. While the .ase format contains only a camera's positional data, the .3DS format contains the camera's position, point of interest, roll and field of view values.

Once the camera is imported, you can edit any of the camera's values and, if necessary, export it back to 3ds Max in its native 3DS format using the Camera menu.

To import a 3ds Max camera:

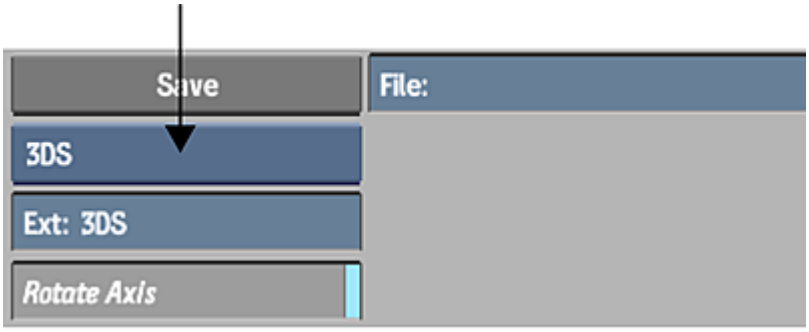
1. In the Camera menu, click Import.
   The Import Camera file browser appears.
2. From the Import Type box, select either the .3DS or .ase format.



3. If needed, enable Rotate Axis to rotate the imported camera by 90° on the X-axis so that it is compatible with Action's coordinate system.
4. Navigate to the location where a 3ds Max camera setup was exported, and select the file.
   You return to Action and the imported camera is applied to your scene.

To export a camera to 3ds Max format:

1. Select the camera you want to export.
2. In the Camera menu, click Export.
   The Export Camera file browser appears.
3. From the Export Type box, select 3DS.



4. If needed, enable Rotate Axis to rotate the exported camera by -90° on the X-axis so that it is compatible with the coordinate system of the 3D application.
5. Navigate to the location where you want to export the camera animation.
6. Enter a name for your exported camera in the file field.
7. Click Save.

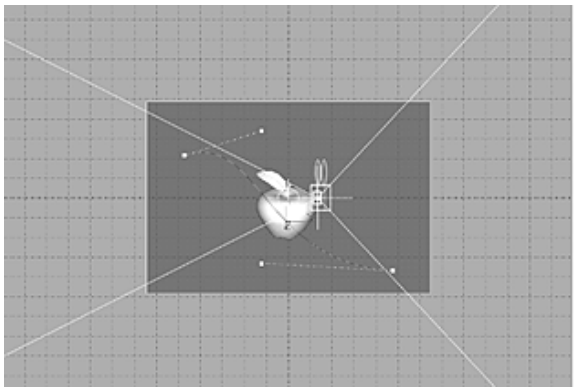
TIP When exporting from Flame, save the .3DS file in a directory that is readable by a 3ds Max system.

Camera, Working, and Orthographic Views

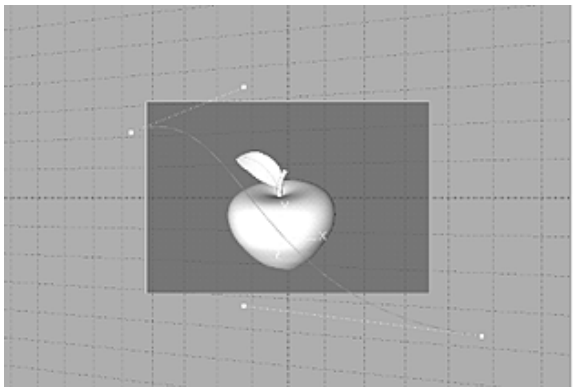
You can view the scene from various angles and display multiple views of these angles simultaneously. This is helpful in setting light sources, camera angles, stereo parameters, and animation keyframes more accurately.

You can view the scene from Camera view, the Working view, and three orthographic views. In Camera view, an object becomes smaller as it moves farther away from the camera. Working view is very similar to Camera view, except that no camera settings are affected. Working view is useful for trying out different settings and positions without actually making changes that can affect your cameras. In Orthographic view, an object remains the same size, regardless of its distance from the camera. Orthographic views are more helpful for aligning objects.

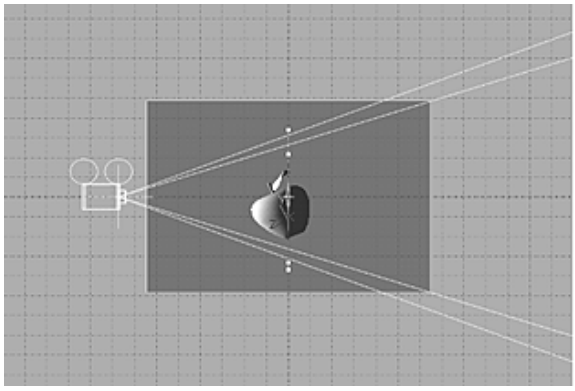
The following figures illustrate the different angles by which the scene can be viewed. The scene in this example contains a grey back clip and the 3D model of an apple.



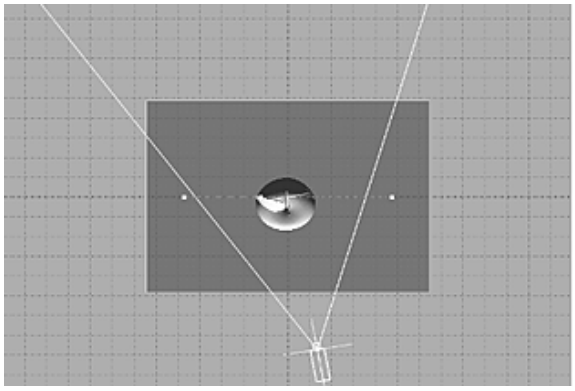
Front view



Camera view



Side view



Top view

Setting Camera, Working, and Orthographic Views

To set camera, working, and orthographic views:

1 From the View box, select Camera, Working, or an orthographic view (listed under Action 3D Views).

Select:	To:
Camera	View the scene in Camera view. This is the scene as viewed by the camera eye. In other words, your field of vision in world space is equivalent to the viewing frustum of the camera. The size of objects depends on their distance from the camera eye.
Top	View the scene as if you are positioned on the positive Y-axis. This is an orthographic view; there is no perspective deformation.
Side	View the scene from the side, as if you are positioned on the positive X-axis. This is an orthographic view; there is no perspective deformation.

Select:	To:
Front	View the scene as if your line of vision is directed into the camera eye. This is an orthographic view; there is no perspective deformation.
Working	View the scene just as in Camera view. Try out different positions and adjustments without affecting the camera settings.

2 If you have multiple cameras in the scene, use the Camera box that appears (below the left-side Action menu buttons) to define which camera is used for the Camera view in the image window. The camera selected in this box is not necessarily the camera used to process the scene.

3 Use the Working view if you want to view the scene using different adjustments, without saving any of the settings to a camera setup. For this reason, no parameters are animatable.

NOTE Create an animatable camera node from the current Working view by pressing `Space-W`.

4 Adjust the view with the Ortho Views controls, if needed.

When using an orthographic view, you may notice that parts of the object you are viewing are getting cut off. Adjust the near and far ortho views. You gain more space to view the object, but lose some viewing precision.

In the Display section of the Action Setup menu, adjust the Near and Far fields.

NOTE The Ortho Views parameters in the Action Setup menu are only for viewing objects, and cannot be animated or saved. The Near and Far fields in the Camera menu are used to set clipping planes. See Moving the Near and Far Clipping Planes (page 865).

Modifying the Camera

You can gesturally modify the camera directly in the scene using options in the Tools box. A mode remains in effect until you select a different mode.

To gesturally modify the camera:

1 Make a selection in the Tools box.

Select:	To:	Mouse Shortcut:	Pen Shortcut:
FOV	Move the camera field of view.		
Track Camera	Move the camera lens and look-at point.	`Alt + Middle Mouse button`	`Alt + Shift + Pen c`

Select:	To:	Mouse Shortcut:	Pen Shortcut:
Tilt	Tilt the camera up and down by moving the look-at point. Also changes the camera roll. This option only modifies Target cameras.		
Roll	Rotate the camera on the Z-axis. This option only modifies Target cameras.		
Orbit	Rotate the camera lens around the look-at point.	`Alt + Left Mouse button`	`Alt + Pen down`
Dolly	Move the camera lens towards (zoom in) or away from (zoom out) the look-at point.	`Alt + Right Mouse button`	`Alt + Ctrl + Pen do`

2 Drag the cursor in the image window.
The camera is modified. Related Camera menu controls are updated to reflect the changes.

Zooming In and Out

Use the Zoom option to move the camera eye toward or away from the point of interest. While viewing the scene in Camera view, zoom in or out from the point of interest to move the camera eye closer to or farther from the point of interest. In Top, Side, or Front view, you can enlarge or reduce the scene in the image window without affecting the camera. Zooming has no effect in Schematic view.

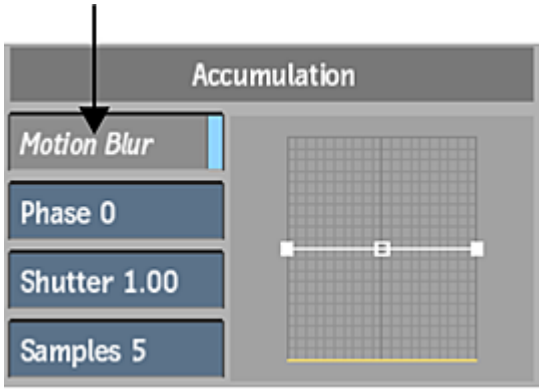
To zoom the camera:

1 From the Tools box, select Zoom.
2 Place the cursor in the image window.
The cursor changes to a magnifying glass.
3 To zoom in, drag the cursor to the left. To zoom out, drag the cursor to the right.

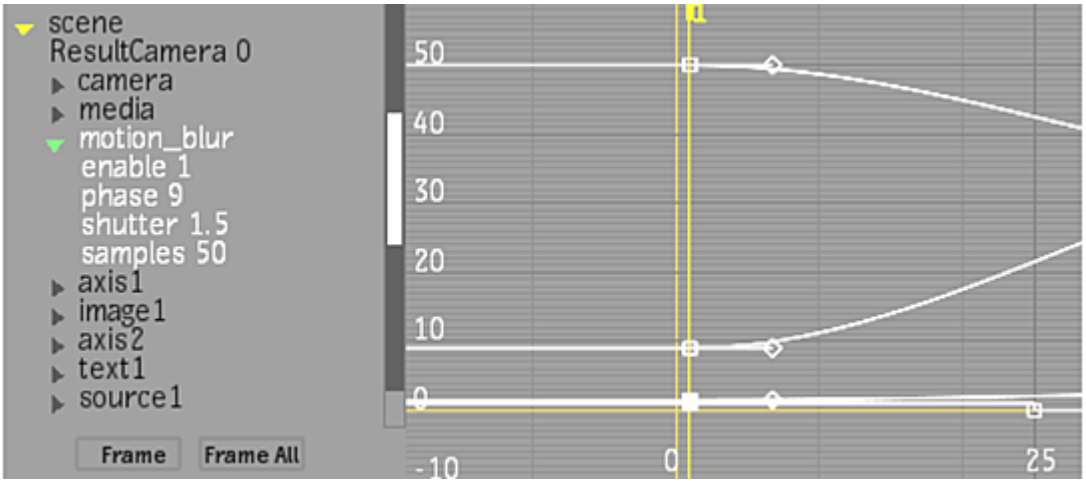
About Motion Blur

Use the Motion Blur tool to simulate the blur created by fast-moving objects. Motion Blur is used with both the normal and physical cameras.You can apply motion blur globally (to the entire scene) and then exclude objects in the scene from its effect.

To use motion blur, enable the Motion Blur button in the Action Setup menu and specify motion blur settings.



You can animate the Motion blur button, as well as the Phase, Shutter, and Samples fields. They can be found in the Channel Editor under the *motion_blur* folder.



See Accumulation Settings in Rendering Tab (page 585).

Blurring a Single Object

You can apply motion blur to an object rather than to the entire scene. To use the per object motion blur, you must apply Motion Blur globally, and then disable motion blur per object.

NOTE The motion blur curve in the Action Setup menu is global. It cannot be set per node. When motion blur is enabled, all surfaces have the same blur characteristics but not necessarily the same animation.

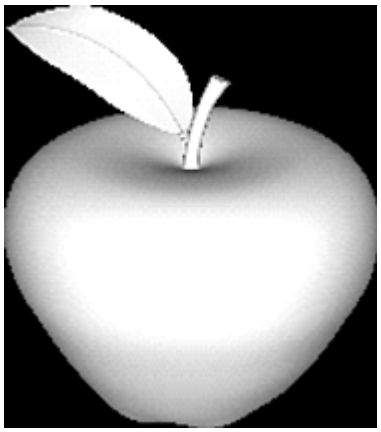
To blur a single object:

1. Enable Motion Blur in the Action Setup menu.
   This activates a global motion blur applied to everything within the scene.
2. Disable motion blur per object (for example, in the surface, axis or light menu) to exclude objects from the global motion blur.

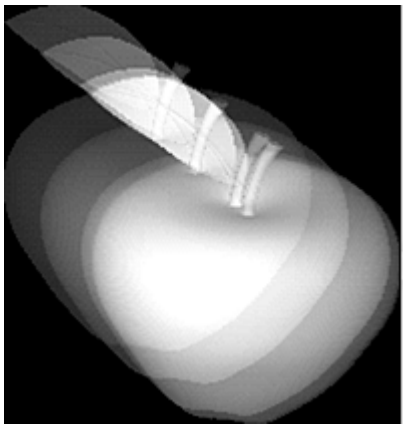
Creating Custom Motion Blurs

The motion blur curve controls the sample weight over the scope of the motion blur. The point on the left is the weight of the first sample and the point on the right is the weight of the last sample. By changing the curve, you can create custom motion blur effects such as a Gaussian blur.

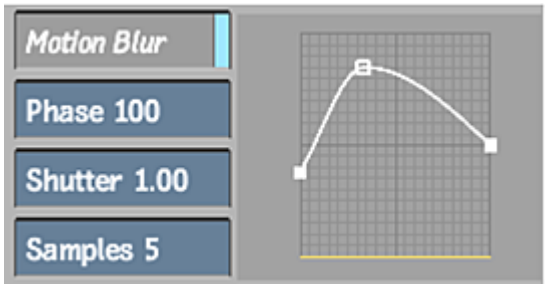
As with the Channel Editor, you can add keyframes to the motion blur curve using Add mode, move keyframes with Move mode, and modify the curve's shape using tangent handles.

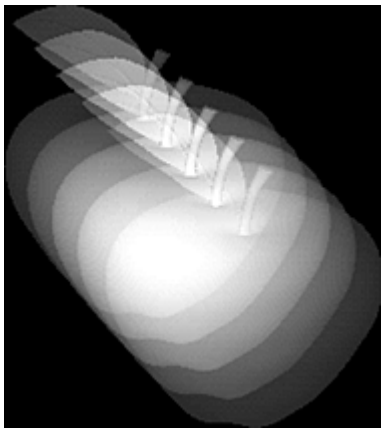


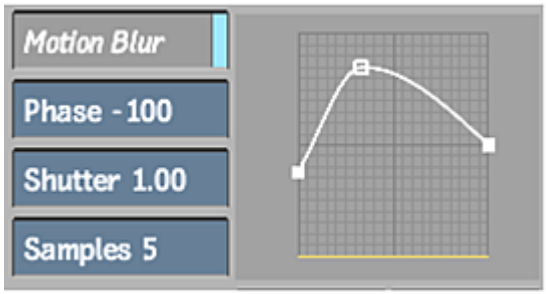
The original animation with Motion Blur disabled.



Motion Blur is enabled using a custom motion blur curve and phase set to 100.







Motion Blur is enabled using the same motion blur curve as above. Phase is set to -100.

TIP You can preview your motion blur effect by clicking Preview. This will display the rendered frame at the current frame in the timebar.

Simulating Motion Blur on a Still Object

You can add motion blur to an object that has no axis movement, by adding a second axis to simulate the motion.

To simulate motion blur on a still object:

1. From the View box, select Schematic.
2. From the Node bin, drag an Axis node to the schematic.
3. Press `Shift` and drag the original Axis node over the new Axis node.

Still object with Axis

New axis added

New axis as child of original axis

Image courtesy of Das Werk

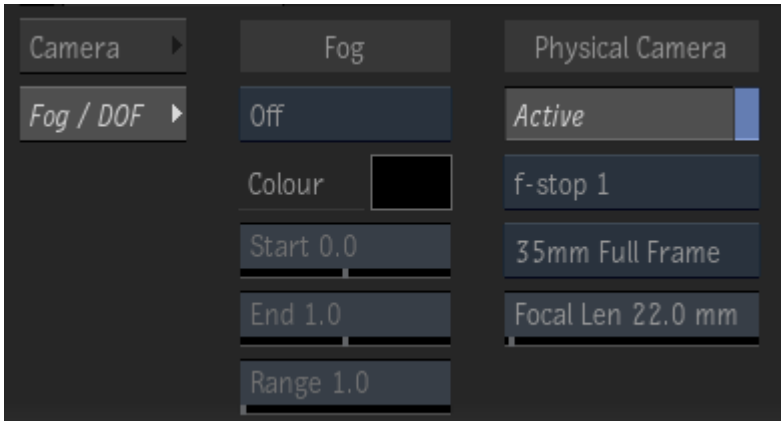
4. Enable Motion Blur and animate the new axis with the desired motion.

About Fog

Use fog in the scene to create visual effects such as mist, haze, and murky water. The fog effect causes objects to fade as their distance from the camera increases. The further an object is from the camera, the foggier the scene appears.

The fog effect is based on depth-cueing, which means the intensity of the fog varies along the Z-axis, and the specified colour gradually increases or decreases its intensity.

You control the density and colour of the fog using the fog controls in the Camera menu's Fog / DOF tab..



Try combining your fog effect with other effects such as transparency to create a more genuine look for non-solid substances such as water. You can also use expressions to improve the quality of fog in a scene so that it looks more natural. See Working with Expressions in the Channel Editor (page 1712).

Applying Fog to the Scene

To add fog to the scene:

1 In the Camera menu, select a fog type from the Fog box.

Type	Description
Linear	Specifies the distance at which fog should start and end.
Exponential	Fog = e -(density * z) where z represents the range or distance from the camera. The range should lie within the realm of the near and far clipping planes. The minimum value is 1.
Exponential2	Fog = e -(density * z)2. The exponential types provide more depth, more natural looking fog, and make the edges appear smoother.

2 For Linear fog, specify the distance at which the fog should start and end in the Start and End fields. For Exp or Exp2 fog, specify the range in the Range field.

3 To select the colour of the fog, click the colour pot to display the colour picker.

Typically, the fog colour should match the background colour of the image in the scene.

Action 3D Cameras

The 3D camera is a full-featured animatable camera in Action that allows you to build 3D compositing scenes, mixing 3D objects and stereo objects.

Typically, you work with the 3D camera to frame and animate the view to achieve the effect that you want. You can also animate specific camera properties.

Use the 3D camera in Stereo mode to create three-dimensional renders with the illusion of a three-dimensional depth-of-field. When rendering a stereoscopic scene, Action takes into account all of the stereoscopic camera attributes. Action outputs two clips: one rendered for the left camera and one for the right camera. These clips can then be viewed in stereo mode, used in other stereo clips, output to VTR, or composited by another program.

Adding a 3D Camera

By default, a 3D camera exists in the Action scene when using stereo clips, or when Action has been set up to work in stereo (you may need to pan in the schematic to see the camera node). You can add multiple 3D cameras in order to change point-of-view or depth-of-field from one camera to another. You can add and animate multiple cameras when creating compositions. You can also switch from one camera to another at any point.

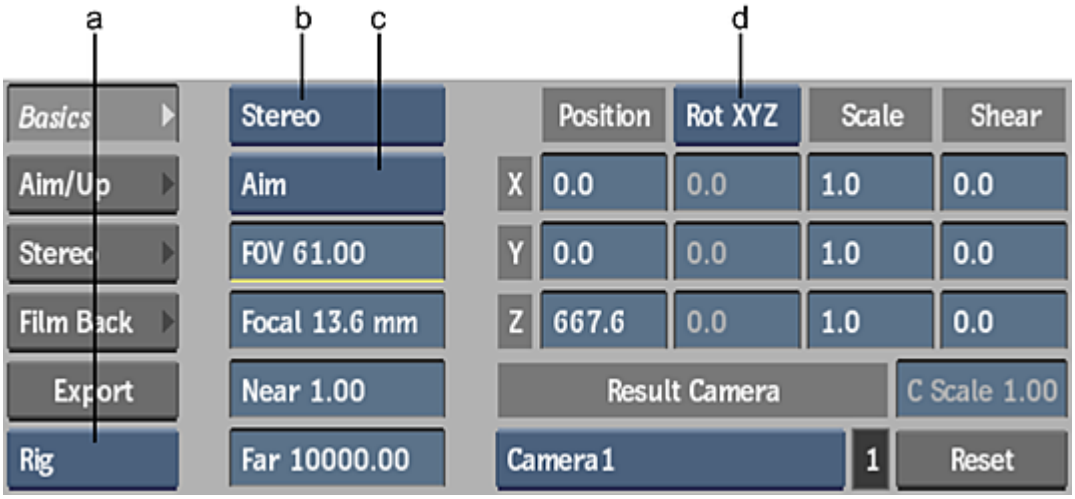
To add a stereo camera to a scene:

1. Do one of the following:
   - Drag the Camera 3D node from the node bin and place it in the schematic.
   - Drag the Camera 3D node from the node bin and place it where you want it in the Result view.
   - Double-click the Camera 3D node. You do not need to be in Schematic view to add a node in this manner.

   A new 3D camera is added to the scene. An icon representing the camera is added to the schematic.
2. Select the 3D Camera node to make it the active camera in the scene.
3. In the Object menu that appears, set the Result Camera to a numbered camera representing a 3D camera.
4. In the Output menu, set the Mode to Stereo and set the Camera to Result Cam.

3D Camera Parameters

Basics Tab



(a) Stereo Camera View Type box (b) 3D Camera Type box (c) Camera Type box (d) Rotation Order box

Export FBX Camera button Opens the Export Camera file browser to save an FBX camera.

Stereo Camera View Type box Available when Stereo is selected in the 3D Camera Type box. Select Left, Right, or Rig (for Stereo Rig).

3D Camera Type box Select whether the 3D camera is stereo or mono.

Camera Type box Select whether the camera is Free, Aim, or Aim and Up.

Select:	For:
Free	Static scenes and for simple animations (up, down, side-to-side, in and out), such as panning out of a scene. A Free camera views the scene in the direction that you aim the camera. You can simply animate the camera rotation or camera tilt as though it were on a tripod. Use the Rotation fields in conjunction with a Free camera.
Aim	Slightly more complex animations (along a path, for example), such as a camera that follows the erratic path of a bird. The Aim camera ensures the camera is specifically aimed at a target object in the scene. Use the Roll and Aim fields in conjunction with the Aim camera.
Aim and Up	Complex animations, such as a camera that travels along a looping roller coaster. Use the Aim and Up camera to specify which end of the camera must face upward. Use the Roll, Aim, and Up fields in conjunction with the Aim and Up camera.

FOV field Displays the angular field of view value, measured in degrees. Use to adjust the width of the camera frustum. Editable.

Focal Length field Displays the focal length of the camera lens, measured in millimeters. Increasing zooms the camera in and increases the size of objects. Decreasing zooms the camera out and decreases the size of objects. Editable.

Near Clipping Plane field Displays the position of the near clipping plane, in pixels, which represents the distance from the camera to the closest point within which image details are processed. Editable.

Far Clipping Plane field Displays the position of the far clipping plane, in pixels, which represents the distance from the camera to the farthest point within which image details are processed. Editable.

See Moving the Near and Far Clipping Planes (page 865).

X Position field Displays the position of the camera, in pixels, on the horizontal (X) axis. Editable.

Y Position field Displays the position of the camera, in pixels, on the vertical (Y) axis. Editable.

Z Position field Displays the position of the camera, in pixels, on the perpendicular (Z) axis. Editable.

Rotation Order box Select the order in which the camera is rotated, on the horizontal, vertical, and perpendicular (X, Y, and Z) axes.

X Rotation field Displays the level of rotation of the camera on the horizontal (X) axis, in degrees. Active when Camera Type is set to Free. Editable.

Y Rotation field Displays the level of rotation of the camera on the vertical (Y) axis, in degrees. Active when Camera Type is set to Free. Editable.

Z Rotation field Displays the level of rotation of the camera on the perpendicular (Z) axis, in degrees. Active when Camera Type is set to Free. Editable.

X Scale field Displays the scale of the camera on the horizontal (X) axis, as a percentage. Editable.

Y Scale field Displays the scale of the camera on the vertical (Y) axis, as a percentage. Editable.

Z Scale field Displays the scale of the camera on the perpendicular (Z) axis, as a percentage. Editable.

X Shear field Displays the shearing of the camera (diagonal shift) on the horizontal (X) axis, as a percentage. Editable.

Y Shear field Displays the shearing of the camera (diagonal shift) on the vertical (Y) axis, as a percentage. Editable.

Z Shear field Displays the shearing of the camera (diagonal shift) on the perpendicular (Z) axis, as a percentage. Editable.

Result Camera box Specify which camera is active. The active camera is the one that will be used when processing/rendering your scene.

Camera Scale field Displays the size of the camera relative to the scene, independently for either Left or Right camera views. For example, if Camera Scale is set to 0.5, the camera's view covers an area half as large, but objects in the camera's view are twice as large. If Focal Length is set to 35, the effective focal length for the camera would be 70. Editable.

Reset button Resets the 3D Camera menu to its default settings.

FBX Unit Scaling field Not shown. This locked field displays the scale factor of the FBX camera used within the application. Use to help set the Pixels to FBX Units field when exporting an FBX camera. Non-editable.

Aim/Up Tab

When Camera Type is set to Aim or Aim and Up, the available options are enabled in the Aim/Up tab.

3D Camera Type box Select whether the 3D camera is stereo or mono.

Camera Type box Select whether the camera is Free, Aim, or Aim and Up.

Select:	For:
Free	Static scenes and for simple animations (up, down, side-to-side, in and out), such as panning out of a scene. A Free camera views the scene in the direction that you aim the camera. You can simply animate the camera rotation or camera tilt as though it were on a tripod. Use the Rotation fields in conjunction with a Free camera.
Aim	Slightly more complex animations (along a path, for example), such as a camera that follows the erratic path of a bird. The Aim camera ensures the camera is specifically aimed at a target object in the scene. Use the Roll and Aim fields in conjunction with the Aim camera.
Aim and Up	Complex animations, such as a camera that travels along a looping roller coaster. Use the Aim and Up camera to specify which end of the camera must face upward. Use the Roll, Aim, and Up fields in conjunction with the Aim and Up camera.

Parenting Offset box Select an offset option for viewing an image when parenting a camera node. Origin sets the image to the camera origin; Target sets the image to the default viewplane distance relative to the camera; and Live Target sets the image to the current viewplane distance based on the FOV.

Parenting Offset field Displays the offset value, as computed from the default camera field of view and the default image size. This value does not change even if other camera parameters are changed. Non-editable.

X Aim field Displays the position of the aiming target of the camera on the horizontal (X) axis, in pixels. Editable.

Y Aim field Displays the position of the aiming target of the camera on the vertical (Y) axis, in pixels. Editable.

Z Aim field Displays the position of the aiming target of the camera on the perpendicular (Z) axis, in pixels. Editable.

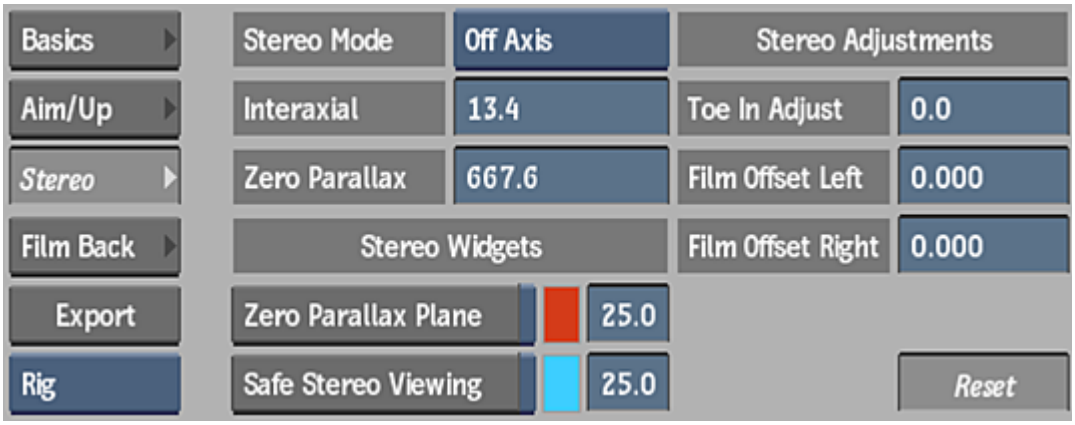
X Up field Displays the up direction on the horizontal (X) axis, in pixels. Editable.

Y Up field Displays the up direction on the vertical (Y) axis, in pixels. Editable.

Z Up field Displays the up direction on the perpendicular (Z) axis, in pixels. Editable.

Roll field Displays the amount of camera roll, in degrees. A positive value rolls the camera clockwise, where a negative value rolls it counter-clockwise. Available only with the Aim, and Aim and Up cameras. Editable.

Stereo Tab



Stereo Mode box Select the method for computing the zero parallax plane.

Select:	To:
Converged	Compute the zero parallax plane by toeing-in the cameras. You can compare this effect to our focusing on an object by rotating our pupils inwards. However, a dangerous side effect may occur where you get a keystone effect on the pairs of render images, causing visual confusion in other elements in the scene. In a rendered image, our focus tends to cascade over the entire image and we are not focusing on a single object, which is not true in real life. You should only use Converged when an object is at the center of the screen with no scene elements at the render borders on either the left or right camera frustum.
Off-axis	Compute the convergence plane by shifting the frustum using camera film back. This is the safer way to compute stereo image pairs and avoids keystone artifacts. Off-axis is the default setting.
Parallel	Create a parallel camera setup where there is effectively no convergence plane. This is useful for landscape settings where objects exist at infinite focus.

Interaxial Separation field Displays the distance between the left and right cameras, in pixels. Editable.

Zero Parallax field Displays the distance on the camera view axis, in pixels, where the zero parallax plane occurs (the point where objects appear off screen). Objects in front of the zero parallax plane have negative parallax. Objects behind the zero parallax plane have positive parallax. Editable.

NOTE In general, your object should be behind the zero parallax plane. In other words, the camera distance should be greater than the zero parallax plane value. The zero parallax value, the camera separation, and focal length are all used to determine the shift that must be applied to film back on the respective left and right cameras. The zero parallax distance is enabled only when Stereo Mode is set to Off-Axis or Toe-In.

Zero Parallax Plane button Enable to display the zero parallax plane.

Zero Parallax colour pot Displays the colour used for the zero parallax plane. Editable.

Zero Parallax Transparency field Displays the level of transparency for the zero parallax plane. Editable.

Safe Stereo Viewing Volume button Enable to display the safe viewing volume created by the intersection of the frustum of the left and right cameras.

Safe Stereo Volume colour pot Displays the colour used for the safe stereo viewing volume. Editable.

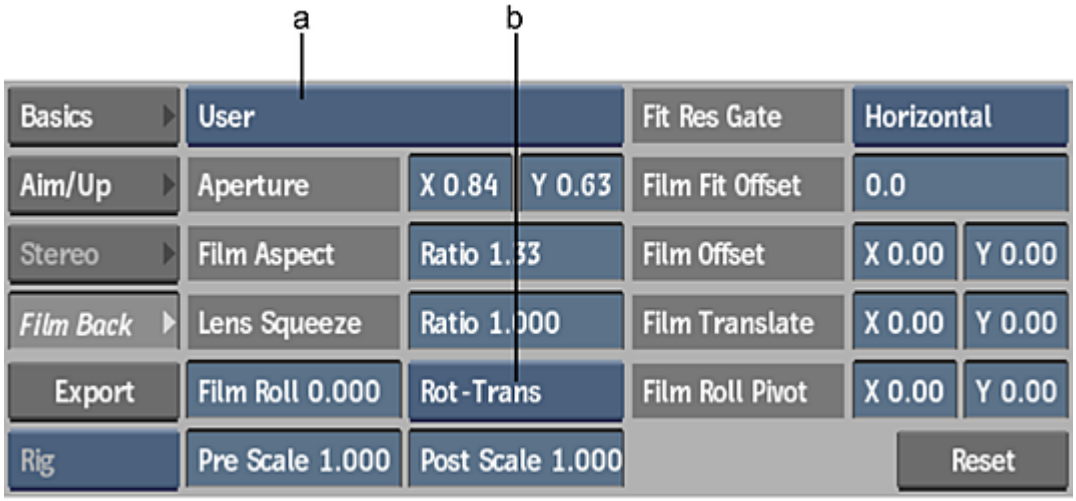
Safe Stereo Volume Transparency field Displays the level of transparency for the safe stereo viewing volume. Editable.

Toe In Adjust field Displays the offset, in degrees, applied to the computed toe-in effect when Stereo Mode is set to Converged. Editable.

Film Offset Left Cam field Displays the horizontal film offset for the left camera. Editable.

Film Offset Right Cam field Displays the horizontal film offset for the right camera. Editable.

Film Back Tab



(a) Film Gate box (b) Film Roll Rotation Order box

Film Gate box Select a preset film frame format type. This action automatically sets the corresponding Camera Aperture, Film Aspect Ratio, and Lens Squeeze Ratio. To set these attributes individually, set Film Gate to User. The default setting is User.

Camera Aperture X field Displays the width of the camera Film Gate setting, measured in inches. This setting has a direct effect on the camera's angle of view and automatically updates Film Aspect Ratio. Editable.

Camera Aperture Y field Display the height of the camera Film Gate setting, measured in inches. This setting has a direct effect on the camera's angle of view and automatically updates Film Aspect Ratio. Editable.

Film Aspect Ratio field Displays the ratio of the camera aperture width versus height. Modifying this field automatically updates the Camera Aperture fields. Editable.

Lens Squeeze Ratio field Displays the amount horizontal compression that is applied to the image. Used with some cameras (for example, anamorphic cameras), which compress the image horizontally to record a wider aspect ratio image onto a square area on film. Editable.

Film Roll Value field Displays the amount of rotation applied to the film back. The rotation occurs around the specified pivot point. Used to compute a film roll matrix, which is a component of the post-projection matrix. Editable.

Film Roll Rotation Order box Select how the roll is applied with respect to the pivot value.

Enable:	To:
Rotate-Translate	First rotate the film back, then translate it by the pivot point value.

Enable:	To:
Translate-Rotate	First translate the film back, then rotate it by the film roll value.

Pre Scale field Displays the artificial 2D camera zoom that is applied before the film roll. Used in 2D effects. Editable.

Post Scale field Displays the artificial 2D camera zoom that is applied after the film roll. Used in 2D effects. Editable.

Film Fit Resolution Gate box Select the size of the resolution gate relative to the film gate (Film fit). If the resolution gate and the film gate have the same aspect ratio, then the Film Fit setting has no effect.

Select:	To:
Fill	Fit the resolution gate within the film gate.
Horizontal	Fit the resolution gate horizontally within the film gate.
Vertical	Fit the resolution gate vertically within the film gate.
Overscan	Fit the film gate within the resolution gate.

Film Fit Offset field Displays the offsets of the resolution gate relative to the film gate either vertically (if Film Fit Resolution Gate is Horizontal) or horizontally (if Film Fit Resolution Gate is Vertical). Film Fit Offset has no effect if Film Fit Resolution Gate is Fill or Overscan. Editable.

Film X Offset field Displays the horizontal offset, in pixels, of the resolution gate and the film gate relative to the scene. Changing the Film X Offset produces a two-dimensional track. Editable.

Film Y Offset field Displays the vertical offset, in pixels, of the resolution gate and the film gate relative to the scene. Changing the Film Y Offset produces a two-dimensional track. Editable.

Enter:	To:
1	Have the view guide fill the view. The edges of the view guide may be exactly aligned with the edges of the view, in which case the view guide is not visible.
> 1	Increase the space outside the view guide. The higher the value, the more space is outside the view guide.

Film X Translate field Displays the artificial 2D horizontal camera pan. Used in 2D effects. Editable.

Film Y Translate field Displays the artificial 2D vertical camera pan. Used in 2D effects. Editable.

Film Roll X Pivot field Displays the horizontal pivot point from the center of the film back, which is used during the rotating of the film back. Used to compute the film roll matrix, which is a component of the post projection matrix. Editable.

Film Roll Y Pivot field Displays the vertical pivot point from the center of the film back, which is used during the rotating of the film back. Used to compute the film roll matrix, which is a component of the post projection matrix. Editable.

Optimizing Attributes of the 3D Camera

Here are some general guidelines for tweaking the stereo attributes of the 3D camera:

- Many parameters are relative to each other, and can be scaled and changed upon import using the Auto Fit in Scene or FBX Unit to Pixels settings.
- Tweak the Interaxial Separation to move the cameras closer to or farther away from one another.

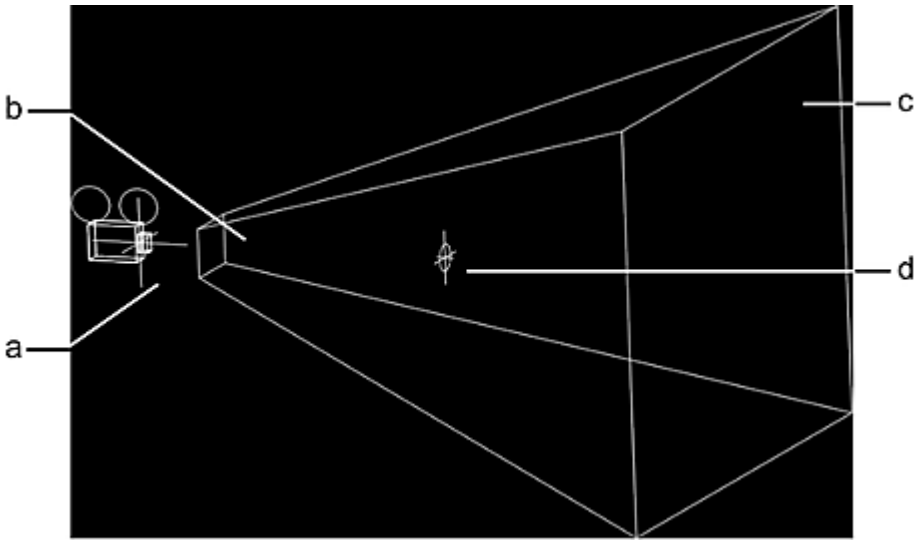
  NOTE You should re-adjust your Interaxial Separation if you change your output device, since the settings for one display method may differ from another.

- Increase the Zero Parallax to move objects further away from the camera. The 3D effect becomes less pronounced in this case. Decrease your Zero Parallax to move objects closer to the camera. You can see more depth if you do this.

  The stereoscopic effect is the most realistic when the Zero Parallax Plane is in between the two objects.
- You may need to re-adjust your camera attributes if you change the resolution of your output device.
- You can also increase the Far Clip Plane to increase the depth of the camera.
- In Anaglyph viewing mode, the red/cyan colors for objects are swapped depending on whether they are behind or in front of the parallax plane. For objects behind the zero parallax plane, they appear in cyan/red. For objects in front of the zero parallax plane, they appear in red/cyan.

Working with the Frustum

The volume of space viewed by the camera is called the frustum. The frustum is, in effect, a viewing pyramid. The camera is located at the apex of the pyramid, and the far clipping plane forms the base. The pyramid may be truncated by the near clipping plane. The point of interest, or aim, is the target at the center of the camera's view.



(a) Camera (b) Near clipping plane (c) Far clipping plane (d) Aiming target

If you place a surface within the frustum, it is visible in the final animation. If the surface is located outside the scope of the frustum, it is not visible at that frame in the animation.

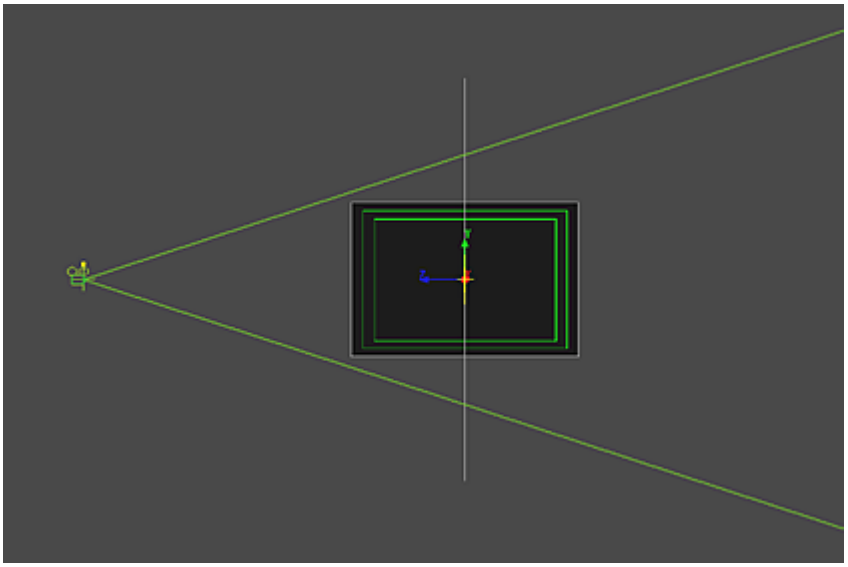
To see the camera and frustum:

1 From the View box, select Side.

2 In the image window controls, click

to zoom out from the scene.

3 Enable the Pan button, and pan around the scene until you see the camera icon.



4 From the Camera menu, click the Aim/Up tab.

5 Ensure that Camera Type is set to Aim, or Aim and Up.

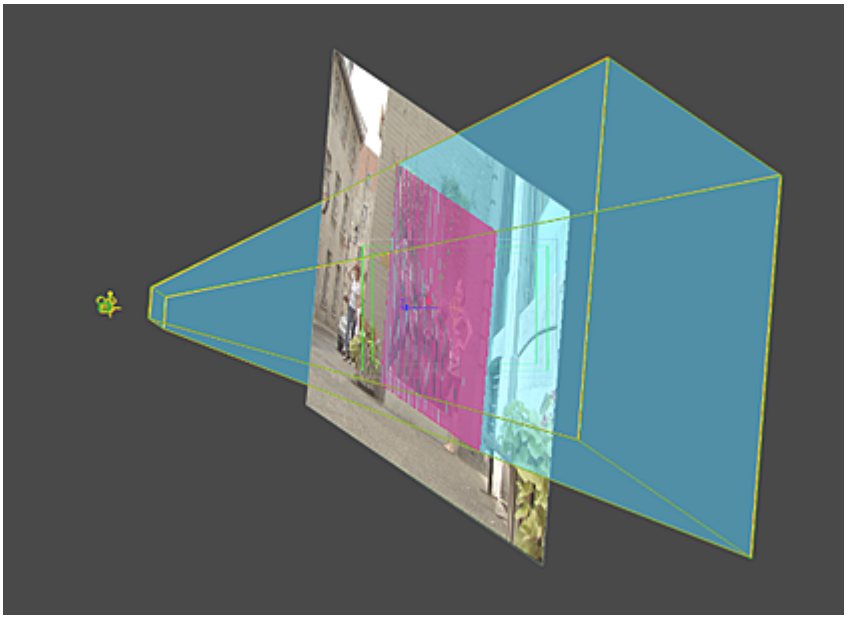
6 Drag the Roll field until you see the four sides of the frustum.

7 On the Stereo tab, enable Zero Parallax Plane and Safe Stereo Viewing.

The safe viewable volume of the frustum, and the zero parallax plane area appear and are denoted with a transparent colour. The aiming target is centered at zero parallax.

8 Change the position of the near and far clipping planes to alter the depth of the frustum. See Moving the Near and Far Clipping Planes (page 865).

9 Change the position of the camera to alter the orientation of the frustum. See Moving the 3D Camera (page 866).



10 Change the position of the camera's aim to alter the orientation of the frustum. See Adjusting the Aim of the 3D Camera (page 866).

11 Adjust the angle of view to adjust the size of the objects as viewed by the camera. See Adjusting the Field of View (page 866).

Near and Far Clipping Planes

The camera frustum is determined by six clipping planes: the left, right, top, bottom, near, and far clipping planes. The depth of the frustum is affected by the near and far. The values for these channels are expressed in pixels relative to the position of the camera.

Set the Near and Far clipping planes to the lowest and highest respective values that produce the desired result. If the distance between the near and far clipping planes is much larger than is required to contain all the objects in the scene, the image quality of some objects may be poor.

TIP Objects that you want to render are usually within a certain range from the camera. Setting the near and far clipping planes just slightly beyond the limits of the objects in the scene can help improve image quality.

The ratio of far:near clipping planes determines the depth precision. Try to keep that ratio as small as possible for better results. Since most of the depth precision is concentrated around the near clip plane, try to avoid a lot of detail on distant objects.

Moving the Near and Far Clipping Planes

To move the near and far clipping planes:

1 From the View box, select Side or Top, depending on how your camera is positioned.

Either of these views profiles the camera so that the near and far clipping planes are clearly visible.

2 From the Camera menu, on the Basics tab, enter a value in the Near field to edit the position of the near clipping plane.

The value in the Near field corresponds to the position of the near clipping plane, in pixels, from the front of the camera. The farther away that the near clipping plane is placed, the higher the value. The default value is 1. Any object between the camera and the near clipping plane is outside the camera frustum and does not get processed in the final result.

3 Enter a value in the Far field to edit the position of the far clipping plane.

The value in the Far field corresponds to the position of the far clipping plane, in pixels, from the front of the camera. The lower the value, the closer the far clipping plane is placed. The default value is 10000. Any object positioned behind the far clipping plane is outside the camera frustum and does not get processed in the final result.

Moving the 3D Camera

Objects in the scene can be recorded from an arbitrary position as determined by the orientation of the camera in world space. The position of the camera can be moved left or right, up or down, or closer or farther from the central point of interest. The values for camera position are expressed in pixels relative to the aiming target.

To change the position of the 3D camera:

1 From the View box, select Front, Side, or Top, depending on how your camera is positioned, to provide a clear view of the camera motion.

2 From the Camera menu, on the Basics tab, modify the Position X, Y, and Z fields.

You can also animate the camera using keyframes and the Channel Editor.

Adjusting the Aim of the 3D Camera

The aiming target is the point in world space at which the camera is directed. The aiming target is always at the centre of the camera's frustum. Changing the position of the camera's aim causes the orientation of the frustum to change. You can take advantage of the relationship between the aim and the frustum to make the camera follow a moving object. To do this, animate the aim while keeping the camera in a fixed position. You can adjust the aim only with an Aim, or Aim and Up camera.

To adjust the aim of the 3D camera:

1 From the View box, select Front, Side, or Top, depending on how your camera is positioned, to provide a clear view of the camera motion.

2 Do one of the following:

- From the Camera menu, on the Aim/Up tab, modify the Aim X, Y, and Z fields.
- Move the aiming target by dragging.

You can also animate the aim using keyframes and the Channel Editor.

Adjusting the Field of View

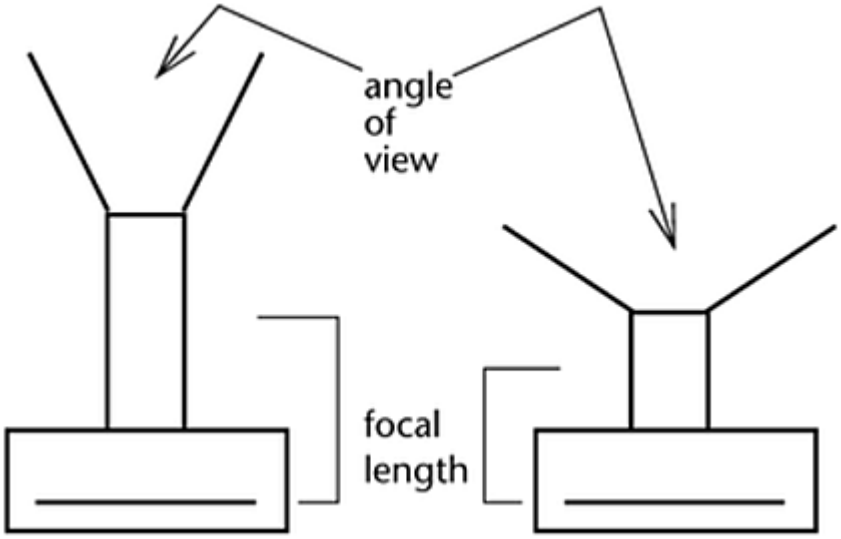
For every shot, you can decide how big an object appears in the frame, for example, whether a shot includes an entire character or just its head and shoulders. This is directly controlled by adjusting the field of view.

The field of view is the amount of a scene (measurable in terms of area) that can be viewed by the camera. The field of view can be altered by changing the angle of view or the distance between the camera and the subject. See Moving the 3D Camera (page 866).

About the Angle of View

The angle of view, or more accurately, the angular field of view, is the amount of a scene (measurable in terms of an angle in degrees, originating at the camera position) that can be viewed by the camera. The angle of view is not affected by changes in the distance between camera and subject. It can be modified directly, in the FOV field, or indirectly, via the Focal Length field.

The angle of view and the focal length are inversely proportional. As you extend the camera's focal length, the field of view gets narrower. As you shorten the focal length, the field of view gets larger.



Playing with the relationship between distance and angle of view affects the focus and perspective of objects in the scene that can be used to creative advantage.

For example, a pair of objects that are placed a distance apart from each other, but in line with the camera, can appear differently when changing these two parameters.

If these objects are viewed by a camera at long range, but using a lens with a high focal length (narrow angle of view), they will appear large in the frame and in equal focus, and will seem to be located on the same plane.

These same objects, when viewed at close range, but using a lens with a low focal length (wide angle of view), will appear to also fill the frame, but the size difference between them will be exaggerated (the foreground object will appear much bigger than the background object) and there will be a very noticeable focus difference.

Focal Length

The focal length of a lens is the distance from the center of the lens to the film plane. The shorter the focal length, the closer the focal plane is to the back of the lens. Focal length is usually expressed in millimeters.

The object's size in the frame is directly proportional to the focal length. If you double the focal length (keeping the distance from the camera to the object constant), the subject appears twice as large in the frame. The size of the object in the frame is inversely proportional to the object's distance from the camera. If you double the distance, you reduce the size of the object by half in the frame.

Adjusting the Angle of View

To adjust the angle of view:

1. From the Camera menu, ensure that the Basics tab is active.
2. If you are using degrees for angle of view, enter the value directly in the FOV field.

   Increasing the FOV value widens the frustum and decreases the Focal Length. Decreasing the FOV value narrows the frustum and increases the Focal Length.
3. If you are using different lens sizes, enter a value in the Focal Length field.

   Increasing the Focal Length narrows the frustum and decreases the angle of view. Decreasing the Focal Length widens the frustum and increases the angle of view.

Importing and Exporting 3D Cameras

Flame supports the import of Alembic 3D data, and the import and export of 3D data saved in the FBX 3D format. These formats provide a means for exchanging 3D data for scene compositions—such as cameras—between tools and packages developed by different manufacturers.

You can either import a camera you created and edited in Action, or import one from another 3D application. Once a camera is imported, you can edit its parameters, change its animation, and then export it back to the application it came from.

NOTE Some parameters in Maya are not supported in FBX, such as Shear, FilmPostScale, and CamScale. Also, many parameters that are animatable in Maya and Flame are not supported as animation curves in FBX. The 3D Camera does not support depth of field.

Export a 3D camera from Action to FBX format, which can be used later by any other FBX-compatible application.

To export a 3D camera from Action to FBX format:

1. Select the camera that you want to export.
2. In the 3D Camera menu, click Export.

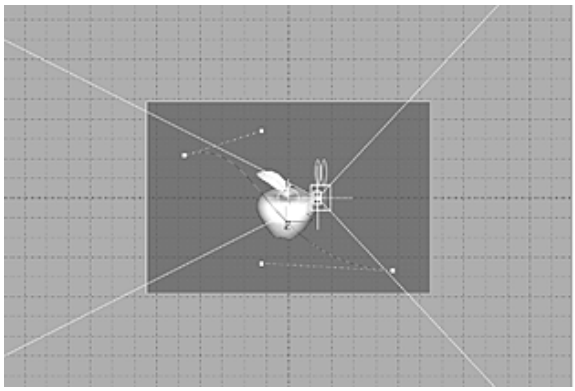
   The Export Camera file browser appears.
3. Set the FBX Scale and Export Options, as needed.
4. Navigate to the location where you want to export the camera animation.
5. Enter a name for your exported camera in the file field.
6. Click Save.

Camera, Working, and Orthographic Views

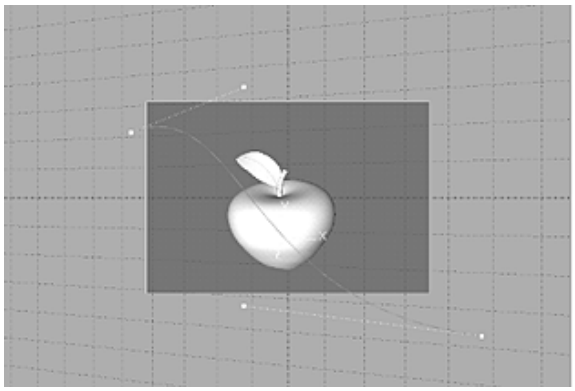
You can view the scene from various angles and display multiple views of these angles simultaneously. This is helpful in setting light sources, camera angles, stereo parameters, and animation keyframes more accurately.

You can view the scene from Camera view, the Working view, and three orthographic views. In Camera view, an object becomes smaller as it moves farther away from the camera. Working view is very similar to Camera view, except that no camera settings are affected. Working view is useful for trying out different settings and positions without actually making changes that can affect your cameras. In Orthographic view, an object remains the same size, regardless of its distance from the camera. Orthographic views are more helpful for aligning objects.

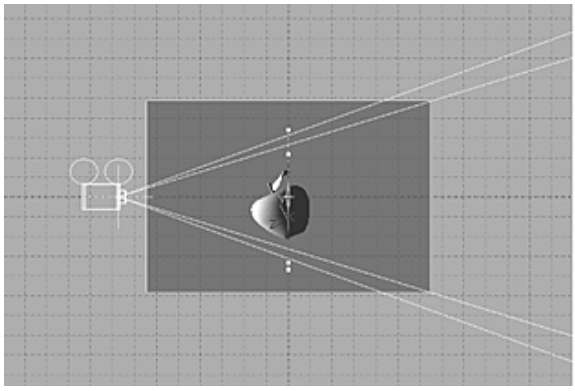
The following figures illustrate the different angles by which the scene can be viewed. The scene in this example contains a grey back clip and the 3D model of an apple.



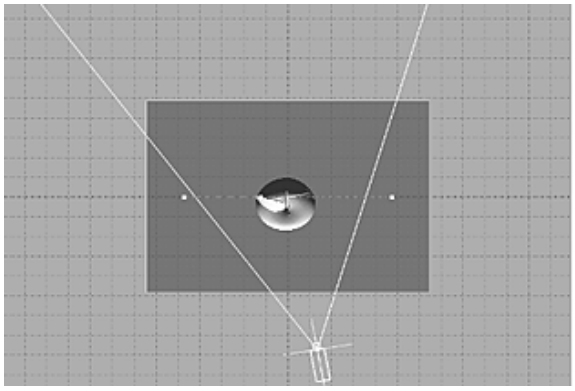
Front view



Camera view



Side view



Top view

Setting Camera, Working, and Orthographic Views

To set camera, working, and orthographic views:

1 From the View box, select Camera, Working, or an orthographic view (listed under Action 3D Views).

Select:	To:
Camera	View the scene in Camera view. This is the scene as viewed by the camera eye. In other words, your field of vision in world space is equivalent to the viewing frustum of the camera. The size of objects depends on their distance from the camera eye.
Top	View the scene as if you are positioned on the positive Y-axis. This is an orthographic view; there is no perspective deformation.
Side	View the scene from the side, as if you are positioned on the positive X-axis. This is an orthographic view; there is no perspective deformation.

Select:	To:
Front	View the scene as if your line of vision is directed into the camera eye. This is an orthographic view; there is no perspective deformation.
Working	View the scene just as in Camera view. Try out different positions and adjustments without affecting the camera settings.

2 If you have multiple cameras in the scene, use the Camera box that appears (below the left-side Action menu buttons) to define which camera is used for the Camera view in the image window. The camera selected in this box is not necessarily the camera used to process the scene.

3 Use the Working view if you want to view the scene using different adjustments, without saving any of the settings to a camera setup. For this reason, no parameters are animatable.

NOTE Create an animatable camera node from the current Working view by pressing `Space-W`.

4 Adjust the view with the Ortho Views controls, if needed.

When using an orthographic view, you may notice that parts of the object you are viewing are getting cut off. Adjust the near and far ortho views. You gain more space to view the object, but lose some viewing precision.

In the Display section of the Action Setup menu, adjust the Near and Far fields.

NOTE The Ortho Views parameters in the Action Setup menu are only for viewing objects, and cannot be animated or saved. The Near and Far fields in the Camera menu are used to set clipping planes. See Moving the Near and Far Clipping Planes (page 865).

Modifying the Camera

You can gesturally modify the camera directly in the scene using options in the Tools box. A mode remains in effect until you select a different mode.

To gesturally modify the camera:

1 Make a selection in the Tools box.

Select:	To:	Mouse Shortcut:	Pen Shortcut:
FOV	Move the camera field of view.		
Track Camera	Move the camera lens and look-at point.	`Alt + Middle Mouse button`	`Alt + Shift + Pen c`

Select:	To:	Mouse Shortcut:	Pen Shortcut:
Tilt	Tilt the camera up and down by moving the look-at point. Also changes the camera roll. This option only modifies Target cameras.		
Roll	Rotate the camera on the Z-axis. This option only modifies Target cameras.		
Orbit	Rotate the camera lens around the look-at point.	`Alt + Left Mouse button`	`Alt + Pen down`
Dolly	Move the camera lens towards (zoom in) or away from (zoom out) the look-at point.	`Alt + Right Mouse button`	`Alt + Ctrl + Pen do`

2 Drag the cursor in the image window.
The camera is modified. Related Camera menu controls are updated to reflect the changes.

Zooming In and Out

Use the Zoom option to move the camera eye toward or away from the point of interest. While viewing the scene in Camera view, zoom in or out from the point of interest to move the camera eye closer to or farther from the point of interest. In Top, Side, or Front view, you can enlarge or reduce the scene in the image window without affecting the camera. Zooming has no effect in Schematic view.

To zoom the camera:

1 From the Tools box, select Zoom.
2 Place the cursor in the image window.
The cursor changes to a magnifying glass.
3 To zoom in, drag the cursor to the left. To zoom out, drag the cursor to the right.

Action Analyzer

Use Action's Analyzer to compute the path of live-action camera and object motion in 3D space. Using the calculated position and motion of the virtual camera, you can match image sequences perfectly, placing any element in the scene. The perspective of the element you place in the scene changes with the perspective of the background as the camera moves. The virtual camera motion is intended to be identical to the motion of the actual camera that shot the scene.

Use the following workflow as a quick start guide to the Analyzer. Follow the links for more detailed information.

Step 1

Select back or front/matte media (mono or stereo) to analyze and add an Analyzer node.

Optional steps:
- Perform a lens correction.
- Adjust the four corners of the perspective grid to set the focal length.
- Add manual trackers to obtain a more predictable and consistent result.

NOTE Adding manual trackers can be initiated post-process.

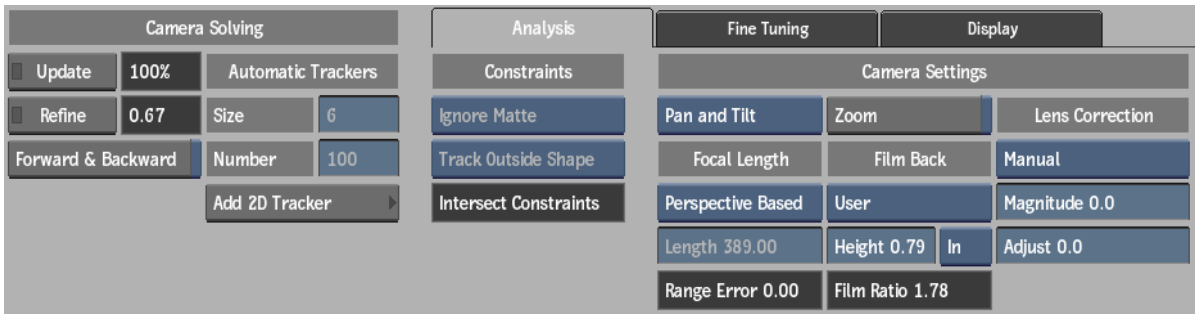
Tracking using the Analyzer Node (page 874)

Step 2

Perform Camera Tracking in the Analyzer menu.

Optional steps:

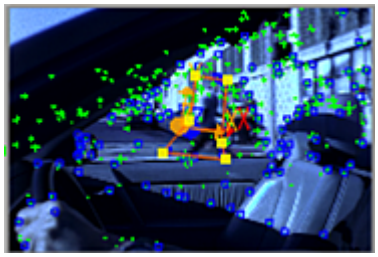
- Add mask constraints to moving areas or areas not wanted in the analysis.
- Add properties of the camera that shot the footage to be analyzed.



Camera Tracking (page 876)

Step 3

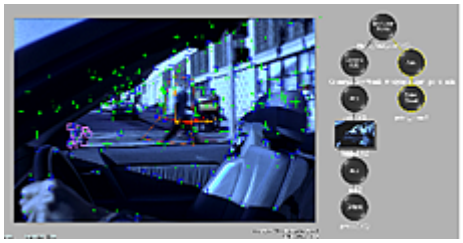
Fine-Tune and recalibrate or refine the camera tracking analysis. This step is optional depending on the results of your initial analysis.



Fine Tuning the Analysis (page 884)

Step 4

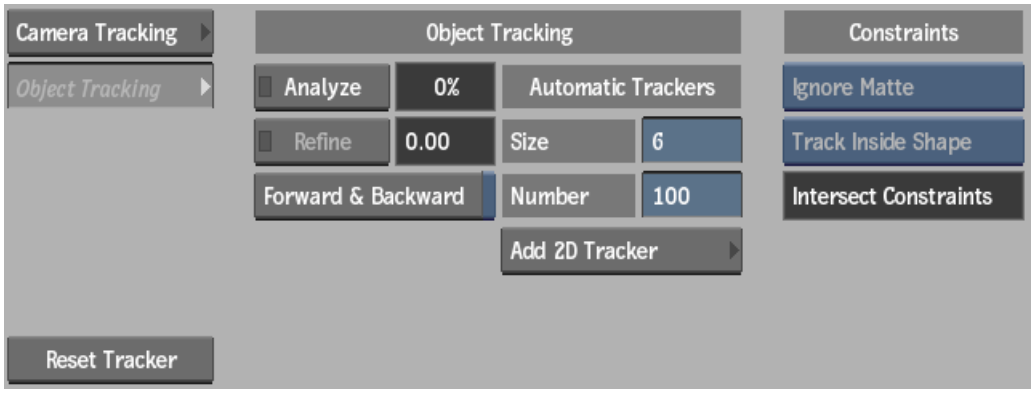
Create a Point Cloud of selected points after the analysis.



Converting the Camera Analyzer Results (page 887)

Step 5

Perform Object Tracking. If needed, after camera tracking, you can track moving objects in the scene — such as inside the masks that were not tracked in the camera tracking analysis.



Object Tracking (page 888)

Preparing to Analyze

A good analysis result is often footage dependant, therefore you may need to perform object tracking after the initial camera tracking.

Here are some things to keep in mind that can improve your analysis:

- The Analyzer works best when tracking an image sequence that has a moving camera or environment. Action GMasks or mattes should be used to isolate the background and the objects in the scene that have different motions. Masking objects cause them to be ignored in the tracking process. You should also mask any logos or watermarks in your image sequence.
- The analysis is based on point-like and corner-like image features. Balls, spheres, lines, and smooth surfaces are not considered.
- Crop out any black areas around your image (such as letterbox areas). When cropping, it is very important to keep the optical centre of the image in the centre, that is, the crop should be symmetrical in both dimensions.
- 3D tracking analyses each frame extensively and with high resolution clips, the process can be lengthy. Unlike 2D tracking, however, you are not required to analyse full-resolution clips. In many cases, analyzing proxies will produce acceptable 3D tracking results quicker than the time that would be required for the full-resolution clips.
- If you have information about the camera that shot the scene you want to track, such as the focal length of the lens and the film back size, it is recommended you specify these details to improve the results of the tracking analysis.

Obtain better tracking results by adding manual trackers

Add manual trackers to produce a more consistent track result. Each tracker consists of an inner reference box, which establishes the reference point for the tracking, and an outer tracker box, which follows the movement of the reference point. You can explicitly choose specific trackable features in the image (see How the Stabilizer Works) (page 1064).

While trackers can be added both pre- or post-processing, it is recommended to do so pre-process if there is going to be a difficult sequence to track. Add trackers post-process to improve the results of the automatic processing. If added post-process, click the Update button.

When tracking, the more track points you have in each frame, the smoother and more accurate the tracking results. As you place the trackers on the image, consider the following guidelines:

- Scatter the trackers by placing them on markings, corners, and shadows in the widest area possible. For example, avoid placing all trackers on the floor; place them on walls and other objects in the scene, as well as on the floor.
- Create a sense of depth by positioning trackers on points that lie in different planes, as well as on points located in the foreground and the background of the sequence.
- Avoid tracking points such as highlights or a point where the foreground and a background object meet, as they do not represent physical 3D points.
- Balance the number of trackers within each frame of the sequence so that as you move through the clip, some points leave the frame and other points appear in the frame. However, maintain a balance so that too many points do not leave or enter the frame at the same time.
- Avoid positioning the trackers in uniform areas or on linear edges where the track points may slide along the edge.

To add manual trackers:

1. In the Analyzer menu, click the Add 2D Tracker button.

   The Stabilizer tool opens and automatically displays the Analyzer media
2. Click Add Tracker.

   A tracker will appear on the middle of the view.
3. Place the tracker over a trackable feature.

   Add as many as you need to improve the tracking.

   Manually added trackers will be assigned a number, incrementing from the first tracker. Manual trackers can then be selected by either:

   - Selecting them directly on the screen or,
   - Selecting the corresponding Tracker button.
4. Click Analyze.
5. Click Return when done to return to the Analyzer menu.

   The trackers will now be visible in the Analyzer view in a different color from the automatically generated trackers.

NOTE Manually added trackers have persistence. They cannot be filtered, and they will not be deleted in a reset. However a tracker can be manually deleted by first selecting it, and in the Fine Tuning section, clicking Delete.

Adding an Analyzer Node

When you add a Stereo or Mono Analyzer in Action, a number of nodes are added to the schematic.

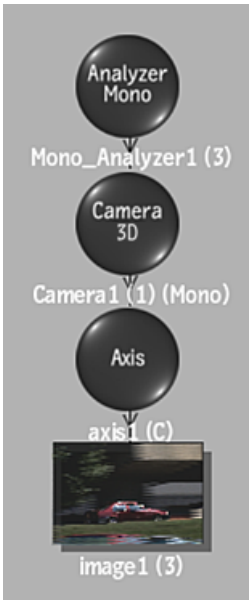
To add an analyzer to a scene:

1. From the Media list, select the back or front/matte media that you want to analyze. If you want to analyze a stereo clip, select the left clip, then press **Ctrl** and select the right clip (you cannot select Back media for a Stereo Analyzer).
2. Do one of the following:
   - Drag the Analyzer Mono or Analyzer Stereo node from the node bin and place it in the schematic.
   - Double-click the Analyzer Mono or Analyzer Stereo node. You do not need to be in Schematic view to add a node in this manner.

   NOTE If you accessed Action as a Timeline FX, Analyzer Stereo is not available.

   The image window automatically switches to an analyzer view to help you view the interactive result of your analysis.

   In the schematic view, you can see that an Analyzer node is added to the scene. The number in brackets next to the name of the Analyzer node indicates the media used for the analyzer (a zero (0) indicates Back media). The Analyzer node is parented to a 3D Camera node, which is synced to the analysis of the Analyzer node, as well as an Image (or Stereo Object) node with an axis.



   To specify different media as the analyzer source, with the Analyzer node selected in the schematic, select the new media in the Media list, then click Apply.
3. To display the Analyzer menu, double-click the Analyzer node in the schematic, or click the Analyzer button.

In the Analyzer menu, you can access Camera and Object tracking settings.

Viewing Analyzers

When you add a mono or stereo analyzer node to your scene, the image window automatically switches to an analyzer view to help you view the interactive result of your analysis. There are other views available from the View box for working with analyzers.

View box Select an option from the Action Analyzers section to set the view in the image window.

Select:	To Display:
Analyzer 2D	2D tracks after analysis. By default, these are displayed as blue squares in the image.
Analyzer 3D	3D points after a mono analysis. By default, these are displayed as green crosses in the image. The F7 keyboard shortcut is a toggle keyboard shortcut; each time you press it, the view changes from Analyzer 2D to Analyzer 3D.
Analyzer 3D Left or Analyzer 3D Right	3D points from the left or right eye perspective after a stereo analysis. The F7 keyboard shortcut is a toggle keyboard shortcut; each time you press it, the view changes from Analyzer 2D to Analyzer 3D Left to Analyzer 3D Right.
Analyzer Working	The Analyzer Working view. This view provides you with the ability to view your analyzed scene using a working camera, which you can modify to view the scene without modifying the tracked virtual camera. This can be helpful to orbit the scene in order to view the depth of 3D points, for example.

NOTE The Analyzer views also display shading in the image window, if the setting is activated in the Action Setup menu.

Analyzer View box Select which analyzer to display in the image window.

If your scene has multiple analyzers, use the Analyzer View box to select which analyzer to display in the image window. This setting appears under the Analyzer button, and is available when the image window is in Analyzer 2D or Analyzer 3D view.

Reset View button Click to reset any camera movements in the Working view.

This setting appears under the Analyzer button, and is available when the image window is in Analyzer Working view.

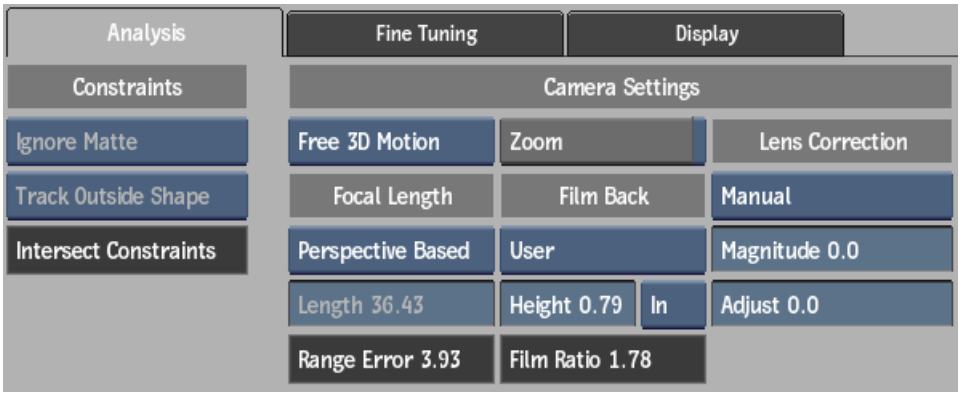
Camera Tracking

You can perform automatic 3D tracking based on the camera properties of the device used to acquire the image sequence you want to track. You can specify these properties, or let the analyzer automatically detect the best solutions for the analysis. Even if you want to perform object tracking on specific moving areas of the scene, you also start with an analysis using camera tracking to create a properly tracked camera.

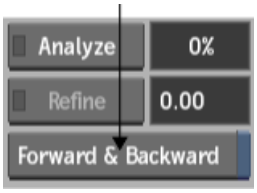
TIP To improve the results of the track, consider adding 2D manual trackers (page 874) either pre- or post-process.

To create a 3D camera track:

1. Add an Analyzer Mono or Analyzer Stereo node, as explained in Adding an Analyzer Node (page 874).
2. In the Camera Tracking menu, set Analysis options. For example, decide if you need to use a matte or GMask in the analysis. In an image sequence of a busy street, you can create a matte or mask of moving elements (such as cars and people) to isolate this area from the analysis. See Analysis Settings (page 882) for details of each Lens Correction, Constraints, and Camera setting.

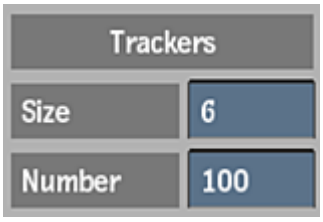


3 Enable Forward & Backward to track the image sequence backward after the forward tracking has completed. This option takes longer, but you may get better results.



4 If needed, adjust the size and number of trackers.

Smaller trackers can speed up the calculation, while larger trackers make the analysis more robust with regard to image noise and variations. A general rule is to increase the size of the trackers when tracking high-resolution footage (2K or larger) that contains more noise.



5 Click Analyze.

You can see a progress indicator beside the Analyze button. You can interrupt the analysis and resume it by clicking Analyze again.

Most of the time, tracking occurs in the background, allowing you to continue working while tracking. In Batch or Batch FX, you can use connected input and matte clips as the media to be tracked. In this case, tracking becomes a foreground process.

After tracking has completed and you press Confirm, the Analyze button changes to Update, and you can see the 2D tracks (the blue squares in the following example) and 3D points (green crosses) in your image, provided that you are in Analyzer 3D view.

Image courtesy of Behavior Communications Inc.

The 3D camera connected to the analyzer node synchronizes to the results of your 3D tracking, and any further changes you make to the 3D track are reflected in this camera.

6 If you are satisfied with the results of the tracking analysis, proceed to Converting the Camera Analyzer Results (page 887). If you want to tweak your track results, see Fine Tuning the Analysis (page 884).

NOTE As a general rule, a Refine field value between 1 and 1.5 is considered a good quality track.

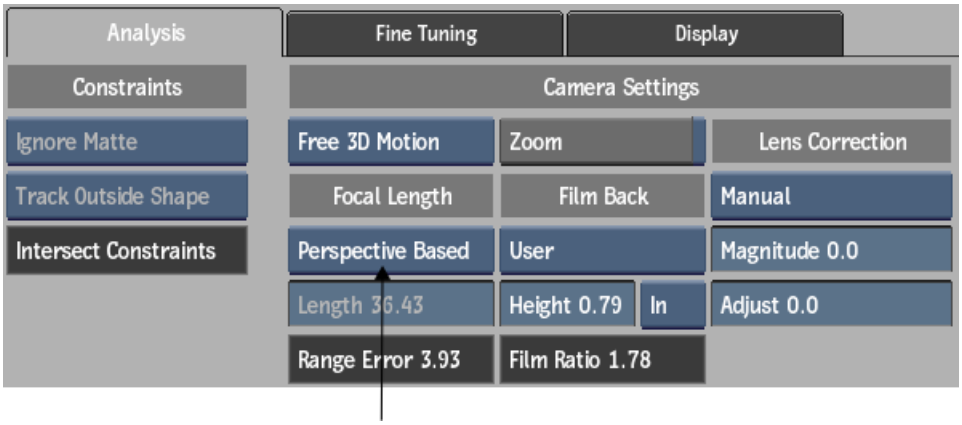
Set the focal length by adjusting the perspective grid

The tracking analysis algorithm can make a best estimate of the focal length for the camera without any user input (see Analysis Settings (page 882)). In order to refine the algorithm's analysis you can use a perspective-based grid to define the focal length. The focal length determined through the perspective grid is then used as part of the analysis. If you do not use this perspective grid or manually enter a value for the focal length, the algorithm guesses at an estimate of the focal length.

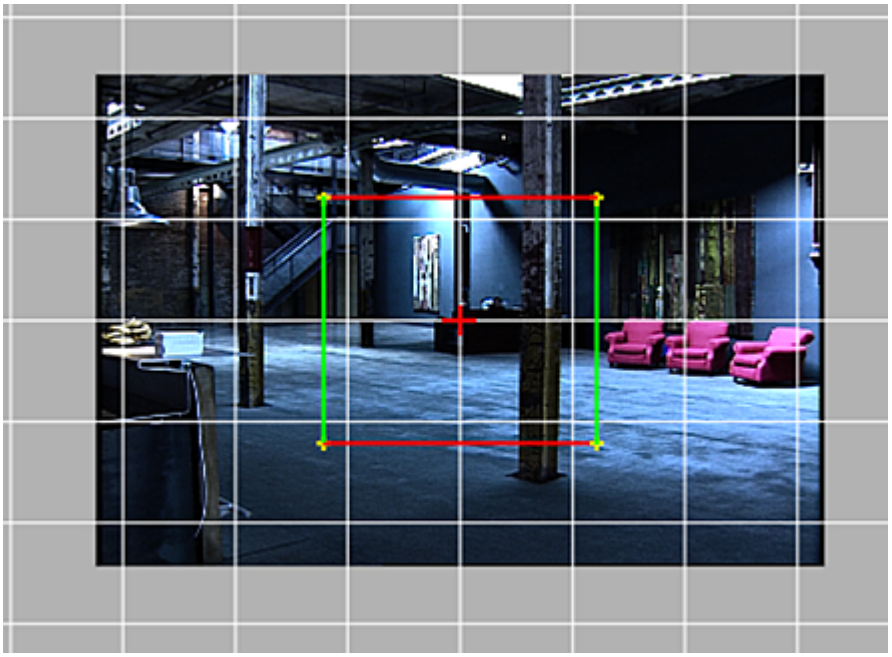
To set the focal length:

1 With the Analyzer object selected in the schematic, select the Analyzer 3D view from the View box.

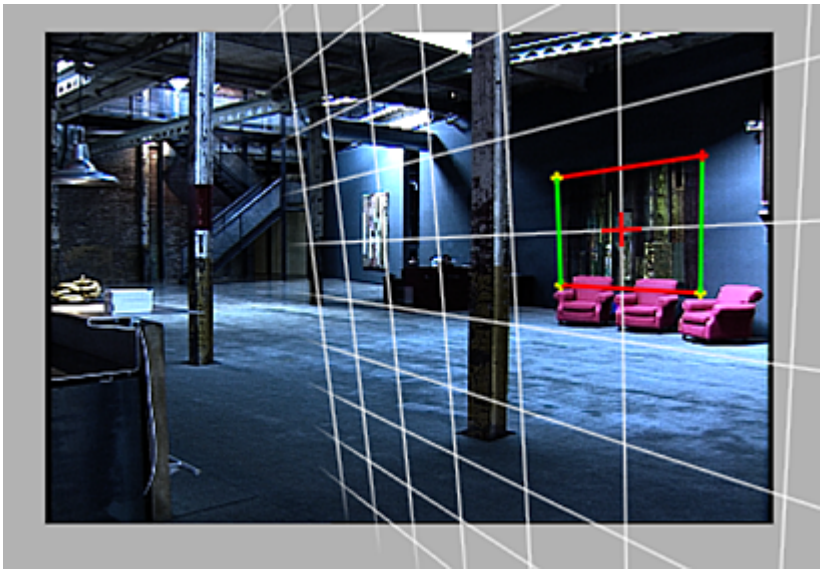
2 From the Analysis tab in the Analyzer, select Perspective Based from the Focal Length box.



A perspective grid appears in the view with the selected media.



3 While in Analyzer 3D view, drag each of the four corners of the rectangle to the desired location to form a rectangular grid. The rectangle automatically aligns to the new perspective. The Range Error, which displays the error estimate for the focal length, should be small relative to the focal length (should be less than 10% of the focal length).



NOTE When dragging one of the corner points of the rectangle, if your desired location cannot be computed (for example, if a point goes past another point on the plane), the location is remembered with a dotted line and red circle. Once you move other corners, your original location may now become viable, and the rectangle and grid align properly.

(See Perspective Grid (page 653) for specific details on placing the corner points.)

4 If you want to add manual trackers pre-process, do so now.

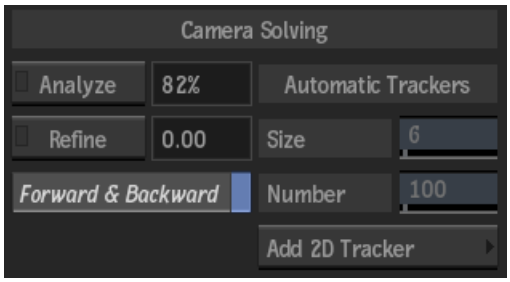
5 Click the Analyze button to start the tracking.

6 When the tracking analysis is complete, click Confirm.

Camera Tracking Settings

Camera Tracking Settings

Use the settings in the Camera Solving section to perform an automatic 3D track.



Camera Tracking button (Not shown) Displays the Camera Tracking menu, where you can track based on the settings of the camera.

Analyze button Click to analyze the image (when complete, Analyze button changes to Update).

Update button (Not shown) Click to delete all previous 3D points, and start tracking based on new information.

Analyze Progress field Displays the progress of analyzing or updating. Non-editable.

Refine button Click to use the current results of the track analysis as a starting point, and refine from this point. Click Refine again to stop the process.

Pixel Error Value field Displays a representation of the distance of 2D tracks from repositioned 3D points. Non-editable.

Forward & Backward button Enable to track the image sequence backward after the forward tracking has completed.

Size field Displays the size of the trackers. Editable.

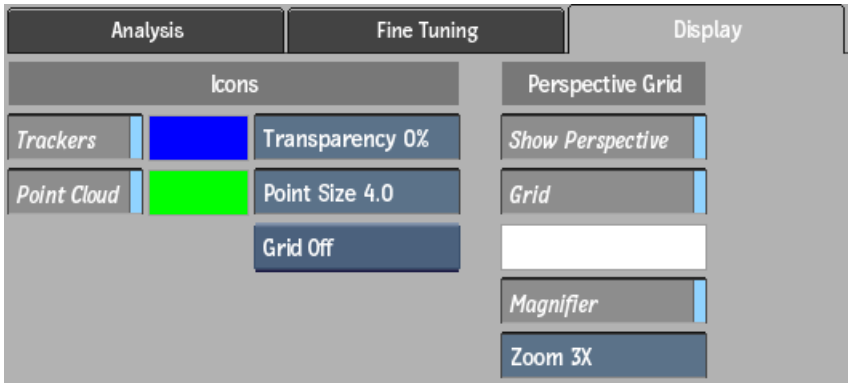
Number field Displays the number of 2D tracks created by the analysis. Editable.

Add 2D Tracker button Click to enter the Stabilizer to add manual trackers.

Reset Tracker button (Not shown) Resets all tracker data in the Analyzer menu.

Tracker Display Settings

The settings help you view your trackers in the image window.



Tracker Display button Enable to display 2D tracks in the image.

Tracker Display colour pot Displays the colour of the trackers. Editable.

Point Cloud Display button Enable to display 3D tracker points in your image.

Points Display colour pot Displays the colour of the points. Editable.

Transparency field Displays the transparency level of the trackers and points in the image. Editable.

Point Size field Displays the size of the 3D points in the image. Editable.

Grid box Select the type of grid to display in the scene. Use to position objects in the scene more accurately.

Select:	To:
Grid Off	Disable the grid.
Grid XY	Use a grid constructed on the X and Y planes.
Grid XZ	Use a grid constructed on the X and Z planes. The XZ grid is visible only when the camera is moved from its default position.
Grid YZ	Use a grid constructed on the Y and Z planes. The YZ grid is visible only when the camera is moved from its default position.

NOTE The Action grid is independent from the global grid in the Grids and Guides menu.

You can change the display of the perspective grid with the following settings (available when Perspective Based is selected in the Focal Length box in the Analysis tab):

Show Perspective button Enable to display the grid corners in the Analyzer view.

Grid button Enable to display the perspective grid in the Analyzer view.

Grid Colour pot Displays the colour of the grid lines. Click to change the colour.

Magnifier button Enable to display the magnifier while dragging a corner of the perspective grid rectangle.

Zoom field Displays the zoom factor of the magnifier. Editable up to 5x.

Analysis Settings

Use the Analysis settings before clicking Analyze.

Constraints Settings

Use the constraint settings to pass information to the analyzer about any mattes or masks in the scene.

Matte Media box Select whether the Action matte is used for tracking. Not available if Back media is being analyzed.

Select:	To:
Ignore Matte	Not use a matte.
Track Inside Matte	Use a matte to delimit the tracking results. White areas of the matte are considered for calculating the solution, and black areas are ignored.
Track Outside Matte	Use an inverted matte to delimit the tracking results. Black areas of the matte are considered for calculating the solution, and white areas are ignored.

GMask Option box Select whether to track inside or outside of an attached GMask shape.

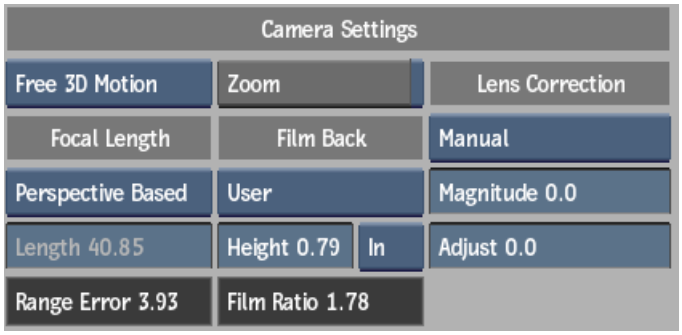
While in an Analyzer view, you can add a GMask node and instantly draw the constraining mask. If added while in Analyzer view, the GMask and Axis nodes are automatically connected to the image as part of the Analyzer tree. If you want to use an existing GMask node, make sure that you parent the Surface (mono analyzer) or Stereo Object (stereo analyzer) to the Axis of the GMask in the schematic.

If the area to be constrained is moving, you can perform planar tracking from within the GMask menu. See Tracking a Mask (page 909).

Intersect Constraints field Displays that the intersection of the two constraints above are used as the tracking region. Non-editable.

Camera Settings

Before analysing the 3D motion, define your camera properties. The camera represents the device used to acquire the sequence of images you are tracking. For each camera parameter, you can choose to let the auto 3D tracker calculate the value automatically, you can specify the value yourself, or you can set a perspective grid (page 878) to estimate the focal length. This information can help the automatic 3D camera tracker calculate better results.



NOTE You can set the camera properties before initial tracking, or after initial tracking when calibrating or refining the track.

Camera Type box Select the type of camera motion in the tracking shot.

Select:	To:
Free 3D Motion	Track a clip shot with a free-moving camera.
Pan and Tilt	Track a rotating clip shot with a camera on a tripod.
Auto Detect Motion	Automatically detect the camera type and track accordingly (default value).

Focal Length box Select Auto to let the 3D tracker calculate the camera values automatically. Select Manual to specify the values yourself. Select Perspective Based to use the perspective rectangle in the image to automatically determine the focal length of the camera.

Length field Displays the focal length of the camera, in millimeters. Editable.

Range Error field Displays the error of the estimate for the focal length. This should be a small number relative to the focal length. Non-editable.

Zoom button Enable to calculate the zoom value of the reconstructed camera for each frame.

Film Back box Select the film back size of the camera that shot the scene.

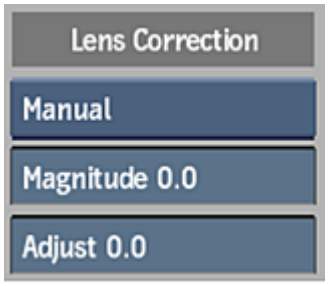
Height field Displays the height of the film back. Editable.

Film Back Units box Select the unit of measurement (inches or millimeters) for the film back height.

Film Ratio field Displays the film aspect ratio of the camera. Non-editable.

Lens Correction Settings

You can apply lens correction before or after your initial analysis. If you make any changes after analysis, you'll need to update or refine the track. While not necessarily required, performing a lens correction may give better tracking results.



Lens Correction Mode box Select Manual and set the Magnitude and Adjust fields if you want to apply lens correction during analysis.

Magnitude field Displays the magnitude of lens correction. Editable.

Adjust field Displays the level of secondary adjustment of lens correction. Editable.

Stereo Analysis Settings

The Stereo Analysis settings display information about the 3D camera after a stereo analysis is performed. These are locked fields, and are meant as informational only.

Stereo Mode field Displays the stereo mode of the 3D camera after a stereo analysis. Non-editable.

Interaxial Separation field Displays the interaxial distance between the left and right cameras of the 3D camera after a stereo analysis. Non-editable.

Zero Parallax field Displays the distance on the camera view axis (where the zero parallax place occurs) of the 3D camera after a stereo analysis. Non-editable.

Fine Tuning the Analysis

If the initial camera track analysis does not give desired results, you can use some or all of the Fine Tuning options to update and refine your track analysis. These procedures are not necessarily required, but depending on your image and the initial tracking, may give better tracking results.

Defining Real World Measurements

You can use the Real World Measures settings to select two points in your analyzed scene, and enter a measurement that represents the distance between the two points. Establishing real world units is helpful when exporting the point cloud through the 3D camera for use in a 3D application.

To define real world units in your analyzed scene:

1. In the Real World Measures section of the Fine Tuning tab, enable Define.
2. Select a point in the image, then press `Alt` and select a second point.

   A red line is drawn between the two points.
3. Disable Define.
4. In the Length field, enter a measurement of the length between the two points, in the unit of your choice (centimeters, inches, etc.). The Action scene scales to the distance you enter.

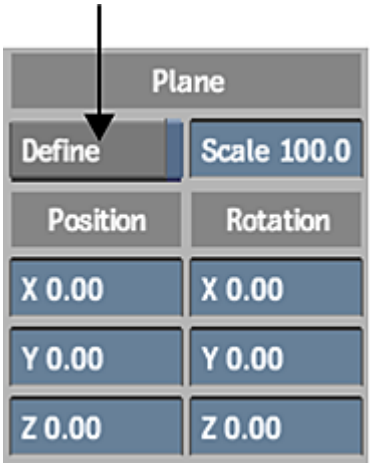
   If you export the point cloud through the 3D camera, you'll be able to specify the unit of measurement in the Export Camera menu (page 868).

Setting the Orientation

Although it is not mandatory to define a ground plane in your image, it helps orient the reconstruction of the cameras.

To define the camera track ground plane:

1. In the Plane section, enable Define.



   NOTE If you selected Pan and Tilt as the camera type (or if the Auto Detect switched to Pan and Tilt) the Plane Define button is not available. The rest of the settings in this group are still available, but only for orientation purposes.
2. Select a minimum of three points in the image that represent the plane of the X/Z axes, such as the ground, a table, or any flat surface.

   The selected points appear as red squares with white crosses.

   TIP You may want to disable the tracker temporarily and point display options (or raise the transparency level) to help you find and select the plane points.
3. Use the Position and Rotation controls to define the orientation of the ground plane.
4. Use the Scale field to specify the scale of the scene.
5. Disable Define.

Filtering Trackers to Fine Tune the Track Analysis

Use the Filter settings to delete lower quality trackers.

To fine-tune the track analysis:

1 Use the Quality slider to adjust the number of good trackers kept. The higher the quality setting, more low quality trackers are selected, such as trackers that drift off their initial reference point. Click Delete to delete the selected trackers.

Trackers of lower quality may hinder the accuracy of the camera tracking.

NOTE After you have made a change that requires the 3D tracking analysis to be refined or updated, notice that the LED next to the Refine and Update buttons turns yellow. This signifies that a Refine or Update is required, but you do not have to perform it until you have completed your tracker selections.

2 Adjust the Short slider to select short duration trackers, that is, trackers that only track a feature for a few frames. Click Delete to delete the selected trackers, leaving the longer duration trackers intact.

3 You can manually select and delete trackers from the image that you feel are not tracking properly. Do one of the following:

- To select an individual tracker, click the tracker, and then click Delete.
- To select multiple trackers in the same area, `Ctrl`-drag a selection box over a series of trackers, and then click Delete.
- With Delete mode selected in the Tools box, select trackers in the image.

Refining or Updating the Track

Once you are satisfied with your fine tuning changes, you can refine or update your Analyzer.

To refine or update the 3D track:

1 Depending on the changes you have made, you can choose to refine or update the 3D track. Do one or both of the following:

- Click Update.

NOTE The update operation deletes all previous 3D points and starts over based on the new information. Depending on your footage, and how many trackers you added, deleted, or linked, multiple updates may yield different results.

- Click Refine.

  The track analysis uses the current results as a starting point, and refines from this point.

Click Refine again to stop the process once an acceptable pixel error value is reached. The pixel error value is a representation of the distance of the 2D tracks from the computed 3D points.

TIP The refine process is footage-dependant, so your acceptable pixel error value may change depending on what is tracking. Since the refine process continues until you stop it, as a general rule, if the pixel error value does not change for a length of time (for example, 30 seconds), you can stop the refine process. The lower the pixel error value, the more accurate the reconstructed track is.

Fine Tuning Settings

Real World Measures Settings

Define button Enable to define a target length between two points in the image. Select one point, then press Shift and select the second point. When defining is complete, disable Define. Establishing real world units is helpful when exporting the point cloud through the 3D camera for use in a 3D application.

Length field Displays the measurement of the length between the two points, in the unit of your choice (centimeters, inches, etc.). Editable.

Plane Settings

Plane Define button Enable to define the ground plane in the image (when defining is complete, disable Define).

Plane Scale field Displays the scale of the ground plane. Editable.

X Position field Displays the position of the ground plane on the X axis. Editable.

Y Position field Displays the position of the ground plane on the Y axis. Editable.

Z Position field Displays the position of the ground plane on the Z axis. Editable.

X Rotation field Displays the rotation of the ground plane on the X axis. Editable.

Y Rotation field Displays the rotation of the ground plane on the Y axis. Editable.

Z Rotation field Displays the rotation of the ground plane on the Z axis. Editable.

Filter Settings

Quality field Displays the number of good trackers that are kept, as a percentage. Editable.

Short field Displays the percentage of short duration trackers (trackers that only track a feature for a few frames). Editable.

Delete button Deletes the selected trackers.

Converting the Camera Analyzer Results

When you are satisfied with the results of the 3D camera tracking analysis, you can convert the selected reconstructed points to a point locators object or actual axes in your scene. The point locators object is useful because you can easily snap objects to it.

To create a point locators object or axes from the 3D camera tracking results:

1. In the Analyzer view, select the points in the image that you want converted. Selected points are displayed as green squares with red crosses.

TIP Press `ctrl` and draw a rectangle over multiple points to select them. If any points are already selected, they become unselected.

2 Enable Fixed Camera if you want to generate a fixed camera and a moving 3D point cloud.

Typically, 3D tracking results in a moving camera and fixed points, but you also can set the camera to be fixed and the points to move.

3 Do one of the following:

- Click Create Point Locators.

  Selected point are converted to a point locators object with a parent axis. Double-click the newly created point locators object to access its menu, where you can change display settings and enable snapping. See Using the Point Locators Object (page 892).

- Click Create Axis.

  Selected points are converted to axes with a parent axis. The axes synchronize to the results of your 3D camera tracking, and any further changes you make to the 3D track are reflected in these axes.

  NOTE If you want to apply settings manually to a synchronized axis, change its name so that it does not update automatically when Analyzer settings are changed. If you reset the Analyzer settings, the synchronized axes become regular Action axes.

4 Use the created point locators or axes to view the reconstructed scene geometry or camera motion.

You can attach objects such as surfaces, 3D text, and 3D models to the new point locators or axes to help position them in 3D space.

Output Settings

Fixed Camera button Enable to generate a fixed camera and a moving 3D point cloud.

Create Axis button Click to convert selected points to axes.

Create Point Locators button Click to convert selected points to a point locators object. Double-click the new point locators node in the schematic to open the Point Locators menu.

Object Tracking

If the image sequence you are tracking has multiple moving objects, you can perform object tracking to track these objects relative to the same camera. Since you perform camera tracking first, you can use the camera data generated from the camera tracking analysis. For example, you can perform a camera tracking analysis on the complete scene, then use masks or mattes to perform multiple object tracking passes focusing on various moving objects in the scene. Each result can be converted to separate point clouds or axes, but all results conform to the 3D camera synced to the original camera tracking.

NOTE To improve the results of the track, consider adding 2D manual trackers (page 874) either pre- or post-process.

To create an 3D track analysis based on object properties:

1 Create a camera track analysis, as explained in Camera Tracking (page 876).

2 Click Object Tracking to switch to the Object Tracking menu.

3 From the Object Motion box, make a selection about the object you are tracking in relation to the camera.

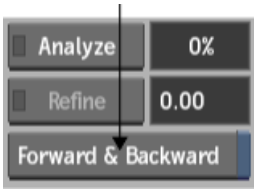
Select:	To:
Free 3D Motion	Track an object moving independently from the camera.
Orbit Around Cam	Track an object rotating around the camera, or far away from the camera.
Auto Detect Motion	Automatically detect the motion type of the object and track accordingly. For small objects, Auto Detect may not be able to establish the proper motion. In this case, select Free 3D Motion or Orbit Around Cam.

4 Set any needed constraints:

- Use the Matte Media box to select whether to use the Action matte for object tracking. If you select to track inside the matte, white areas of the matte are considered for calculating the solution, and black areas are ignored (vice-versa for tracking outside the matte).
- Use the GMask Option box to select whether to track inside or outside of an attached GMask node. This option is only available if a GMask node is attached to Analyzer tree. If you used a GMask constraint while camera tracking, the opposite option is automatically selected for object tracking.

The tracking analysis uses an intersection of the constraints, so you may choose to hide or disconnect gmasks if you want to perform a separate object track for each one.

5 In the Object Tracking section, enable Forward and Backward to track the image sequence backward after the forward tracking has completed. This option takes longer, but you may get better results.



6 Set the size of the automatic trackers in the size field.

7 Click Analyze.

Most of the time, tracking occurs in the background, allowing you to continue working while tracking. In Batch or Batch FX, you can use connected input and matte clips as the media to be tracked. If this case, tracking becomes a foreground process.

You can see a progress indicator beside the Track button. You can interrupt the analysis and resume it by clicking Track again. After tracking has completed and you press Confirm, the Track button changes to Update, and you can see the 2D tracks and 3D points in your image.

Filtering Trackers to Fine Tune the Track Analysis

Use the Filter settings to delete lower quality trackers.

To fine-tune the track analysis:

1. Use the Quality slider to adjust the number of good trackers kept. The higher the quality setting, more low quality trackers are selected, such as trackers that drift off their initial reference point. Click Delete to delete the selected trackers.

Trackers of lower quality may hinder the accuracy of the camera tracking.

NOTE After you have made a change that requires the 3D tracking analysis to be refined or updated, notice that the LED next to the Refine and Update buttons turns yellow. This signifies that a Refine or Update is required, but you do not have to perform it until you have completed your tracker selections.

2. Adjust the Short slider to select short duration trackers, that is, trackers that only track a feature for a few frames. Click Delete to delete the selected trackers, leaving the longer duration trackers intact.
3. You can manually select and delete trackers from the image that you feel are not tracking properly. Do one of the following:
   - To select an individual tracker, click the tracker, and then click Delete.
   - To select multiple trackers in the same area, `Ctrl`-drag a selection box over a series of trackers, and then click Delete.
   - With Delete mode selected in the Tools box, select trackers in the image.

Setting the Scale

After tracking has occurred, you can set the scale of the tracked object. Since you are tracking a specific object as part of an image, setting the relative scale of the object in relation to the image helps you to position objects in the reconstructed scene when you convert the 3D points into a point cloud or axes.

Refining or Updating the Track

Once you are satisfied with your fine tuning changes, you can refine or update your Analyzer.

To refine or update the 3D track:

1 Depending on the changes you have made, you can choose to refine or update the 3D track. Do one or both of the following:

- Click Update.

  NOTE The update operation deletes all previous 3D points and starts over based on the new information. Depending on your footage, and how many trackers you added, deleted, or linked, multiple updates may yield different results.

- Click Refine.

  The track analysis uses the current results as a starting point, and refines from this point.

Click Refine again to stop the process once an acceptable pixel error value is reached. The pixel error value is a representation of the distance of the 2D tracks from the computed 3D points.

TIP The refine process is footage-dependant, so your acceptable pixel error value may change depending on what is tracking. Since the refine process continues until you stop it, as a general rule, if the pixel error value does not change for a length of time (for example, 30 seconds), you can stop the refine process. The lower the pixel error value, the more accurate the reconstructed track is.

Converting Object Tracking Results

When you are satisfied with the results of the 3D object tracking analysis, you can convert the selected reconstructed points to a point locators object or actual axes in your scene. The point locators object is useful because you can easily snap objects to the locators. An image that does not deform is the best candidate for the point locators.

To create a point locators object or axes from the 3D object tracking results:

1 In the Analyzer view, select the points in the image that you want converted. Selected points are displayed as green squares with red crosses.

TIP Press `ctrl` and draw a rectangle over multiple points to select them. If any points are already selected, they become unselected.

2 Do one of the following:

- Click Create Point Locators.

  Selected point are converted to a point locators object with a parent axis. Double-click the newly created point locators object to access its menu, where you can change display settings and enable snapping. See Using the Point Locators Object (page 892).

- Click Create Axis.

  Selected points are converted to axes with a parent axis. The axes synchronize to the results of your 3D camera tracking, and any further changes you make to the 3D track are reflected in these axes.

  NOTE If you want to apply settings manually to a synchronized axis, change its name so that it does not update automatically when Analyzer settings are changed. If you reset the Analyzer settings, the synchronized axes become regular Action axes.

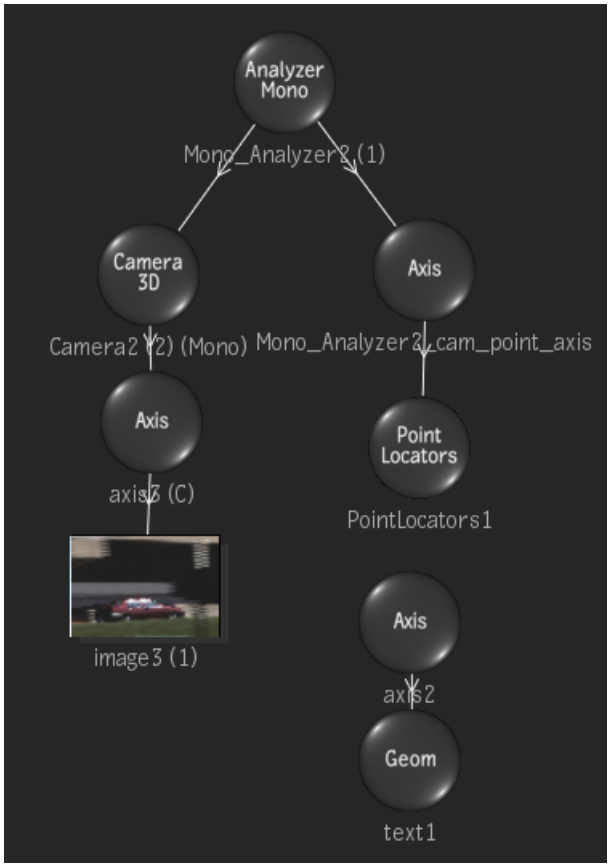
3 Use the created point locators or axes to view the reconstructed scene geometry or camera motion.

You can attach objects such as surfaces, 3D text, and 3D models to the new point locators or axes to help position them in 3D space.

Using the Point Locators Object

Once you have created one or more point locators objects from your camera or object tracking analysis, you can use the Point Locators menu to help you snap objects to the points in 3D space and have them perfectly follow the movement in the scene.

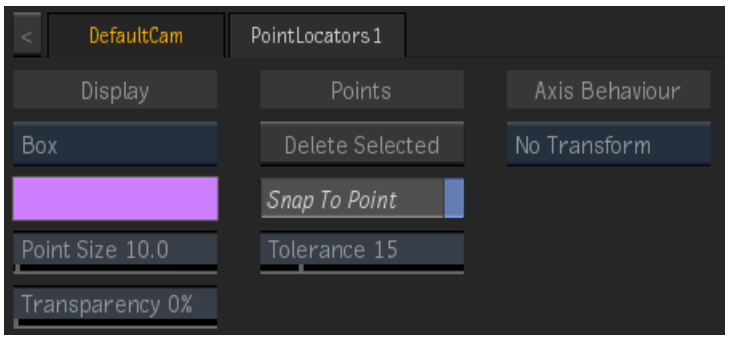
In the schematic, you can see that the axis of a point locators object is automatically attached to the Analyzer node. You can then add objects (such as a 3D text geometry), and use the Point Locators menu settings to help you position the object into your scene.



NOTE If the point locators object is moving (that is, you performed object tracking), you'll need to parent the point locators object to the object you want to position, so that it can properly follow the movement of the point locators.

To access the Point Locators menu, double-click the point locators object in the schematic.

Point Locators Menu Settings



Point Locators Display box Select how the point locators are displayed in the image. You can choose between spheres, boxes, and crosses.

Point Locators Colour pot Displays the colour of the point locators. Editable.

Point Size field Displays the size of the point locators. Editable.

Transparency field Displays the transparency level of the point locators. Editable.

Delete Selected button Click to delete selected points.

Snap To Point button Enable to snap an object automatically to a point in the point locators object. Use Shift and drag the object towards the point locator.

Snap Tolerance field Displays the minimum distance between the object and the point locator for snapping to occur. Editable.

Transform box Select how an attached object behaves based on the movement of the point locators.

Select:	To:
No Transform	Apply no transformation on the attached object.
Plane Transform	Align an object based on the average plane of the maximum number of points.
Centroid Transform	Align an object to the centre of the point locators.

Action Source Nodes

Use source nodes in Action to create more advanced techniques such as transforming or replacing a front or matte clip or applying motion blur. Source nodes can be used to separate front and matte clips in media and then apply separate transformations to each clip. For example, if you apply media to a surface and you want to create an effect where the matte moves into the scene, add a source that isolates the matte clip. A matte source lets you animate the matte's position separately from the front. You can also use source nodes to apply several matte clips to a single front clip.

Source nodes are applied before any Media list effects, such as the Modular Keyer, and may lead to unwanted effects. For example, a Modular Keyer effect applied on a media layer is also applied to the Source node. If you do not want to apply the Modular Keyer effect on a Source node, you need to create a new media layer for the source node output.

When working with Source nodes, global Scene Ambient lighting has no effect. Therefore, when you add a Source node, a Light node parented to a Camera node is automatically attached to the Source node. This light is an ambient type, and appears at half the distance of the source camera.

TIP Instead of using a source node for a simple front/matte transform, consider using the Front Matte Offset Matchbox preset.

Creating a Matte or Front Source

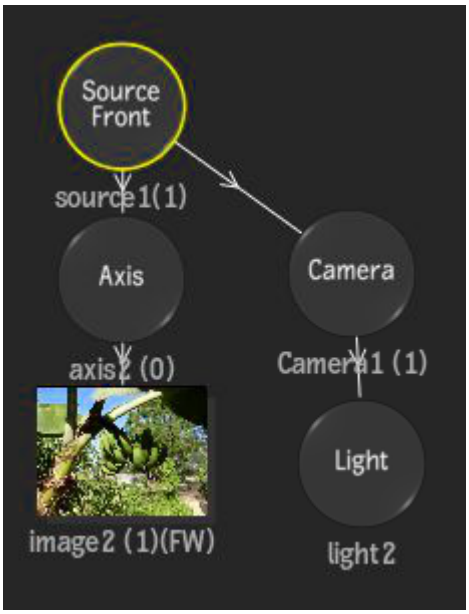
When you create a matte or front source, a new branch appears in the schematic showing the matte or front clip is separated from its media, ultimately replacing the matte or front with itself. This method of using source nodes lets you animate the media's matte or front separately.

To create a matte or front source:

1. In Action, click Media.
2. In the Media list, select the matte or front media that you want to replace.
3. Do one of the following:
   - Drag the Source Matte or Source Front node from the node bin and place it in the schematic.

- Double-click the Source Matte or Source Front node. You do not need to be in Schematic view to add a node in this manner.
- Drag the Source Matte or Source Front node from the node bin directly to the image window. The view switches to Source view automatically.

A source node branch is created in Schematic view that parents an axis and a surface, as well as a camera and a light. This light is an ambient type, and appears at half the distance of the source camera.



To view the contents of the source node in the image window, see Viewing a Source Node (page 898).

NOTE You can change a source from a front to a matte source, or vice versa, from the Source Type box in the Source menu.

4 Select the axis or surface parented by the source and create the animation.

The changes applied to the axis or surface connected to a source are applied to the media's matte or front.

For example, on a matte source, if you change the axis scale to 80%, the media's matte is scaled 80% when used with the media's front. In addition, if you replace the surface by a bilinear or bicubic, you can create complex animations where the matte behaves like a page turn leading into the scene.

Accessing the Source Menu

Once you add source nodes to your scene, you can access the Source menu to apply various settings.

To access the Source menu:

1 Do one of the following:

- Select a Source Matte or Source Front node in the schematic, and then click the Source menu button.

  NOTE If no source node is selected when first accessing the Source menu, or if there are no source nodes present in the scene, all of the options in the menu are disabled.

- Double-click a Matte or Front Source node in the schematic.

Parenting and Redrawing Source Nodes

A source cannot be parented by other objects in the schematic. If you attempt to parent a source, the connection is refused. You can, however, parent objects inside a source node and they appear in your scene based on the Source Type settings in the Object Image menu.

NOTE Be careful when you unparent or delete a source. Surfaces or other objects that were parented by the source are added to the scene.

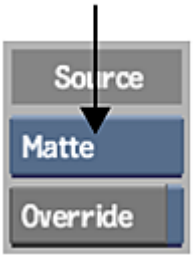
See Connecting Action Nodes (page 577).

Replacing the Front or Matte Clip

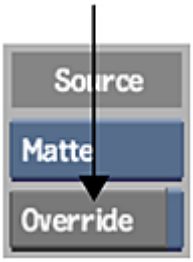
When working with source nodes, you can replace the media's front and matte clips with the front, matte, or a combination of the front and matte clips from other media. These changes can be made in the Source menu, or from the Sources tab in the Object Image menu of the image attached to the source node.

To replace the media's front or matte:

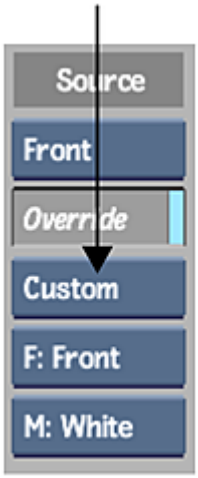
1. Double-click the source node in the schematic.
2. In the Source menu, change the source node from Matte to Front or vice versa.



3. Enable Override.



4. Select the type of source from the Source Type box.



You can also change the source type in the Object Image menu for an image parented by a source node. The source type set in the Object Image menu is independent of the source type set in the Source menu for the parent source node or nodes.

5 If you selected Custom, select the clip to use as the front and matte from the image parented by the source. You can select Front, Matte, or White for each.

For example, if you select F: Matte and M: Matte, only the matte of the source's child image is rendered and used. A summary of your choice can be seen under the image node in the schematic. In this case you will see (MM).

NOTE You can combine multiple parent sources each with their own override setting.

Source Settings

Source box Select a Front or Matte source.

Override button Enable to override a source node selection on the parent source.

Source Type box Select a source type. When selecting Custom, you can also designate a Front and Matte type in the boxes below.

Select:	To:
Front only	Set the source to use only the front of the selected surface; the matte will be replaced with white (displayed as FW under the selected surface in the schematic).
Matte only	Set the source to use only the matte of the selected surface; the front will be replaced with white (displayed as WM under the selected surface in the schematic).
Custom	Set the source to have all the options of replacing the front and the matte of the selected surface by White, original Front or original Matte.

Front Type box Select the clip to use as the front from the image parented by the source.

Matte Type box Select the clip to use as the matte from the image parented by the source.

Source View box Select which source to view in the image window. See Viewing a Source Node (page 898).

Viewing a Source Node

There are different ways to view a source node while working in Action. For example, a 2-up view with Schematic and Source views allows you to view the complete scene and the interactive result of your source node.

From the View box (on the viewing/layout toolbar under the viewports), select Result, Front, and Matte views (grouped under Action Sources). Press `F6` to cycle through these three views.

You can also the Sources Working view to provide you with the ability to view your source scene using a working camera, through which you can experiment without saving anything while preserving your camera setups. Each source node has its own working camera apart from the regular camera.

If you have multiple source nodes in your scene, you can use the Source View box (located at the bottom of the Action menu buttons) to select which source to view in the image window.

Blurring with Source Nodes

Motion blur and anti-aliasing can be set for either the front or matte clip of the media. Motion blur simulates the blur created by fast-moving objects by blurring the motion of the front or matte media.

Similar settings for anti-aliasing and motion blur can be found in the Action Setup menu. You have the option of using the same settings from the Setup menu for source nodes, or you can create specific source node settings.

Rendering Settings

Z-Buffer Mode box Select an option to determine whether the distance from the camera eye is considered.

Select:	To:
Z-Buffer	Arrange sources according to their distance from the camera eye.
Z-Buffer Off	Not consider the distance from the camera eye when arranging sources.
Shadow Mix	To render each shadow in the correct Z order with its corresponding source.

Camera Selector box Select the child camera under a parent source node.

Shadow Type Settings

Source Shadow Type box Select the type of shadow cast for the source node. This setting is available in the Shadow Cast menu only if the shadow cast node is a child of a source node, and is repeated in the Source menu.

Anti-Aliasing Settings

Anti-Aliasing Sample box Select a software anti-aliasing sampling level. To set an arbitrary amount of animatable samples, select AA N Samples, then choose the number of samples in the Samples field that appears.

Softness field Displays the softness of the anti-aliasing sample. Editable.

Samples field Available when AA N Samples is selected in the Anti-Aliasing Sample box. Displays the number of anti-aliasing samples to consider. Editable and animatable.

Accumulation Settings

Motion Blur button Enable to use a motion blur effect for the selected source node (can only be used if the global Motion Blur is enabled in the Action Setup menu).

Phase field Displays the frame that motion blur is based on (before or after the current frame). Editable.

Shutter field Displays the duration of motion blur at each frame. Editable.

Samples field Displays the quality level of motion blur and the depth of field produced by the number of samples taken at each frame. Editable.

Increasing the number of samples causes the processing time to increase linearly. The number of motion blur samples is multiplied by the number of anti-aliasing samples. To reduce the total number of passes made for each frame, reduce the level of anti-aliasing when Motion Blur is enabled.

TIP You can animate the Motion Blur button, as well as the Phase, Shutter, and Samples fields in the Channel Editor under the *source* > *motion_blur* folder.

Depth of Field button Enable to use the camera's depth of field.

Softness field Displays the softness of the depth of field. A low value yields a sharp falloff between focused and unfocused regions. Editable

Motion Blur Curve Displays the sample weight over the scope of the motion blur.

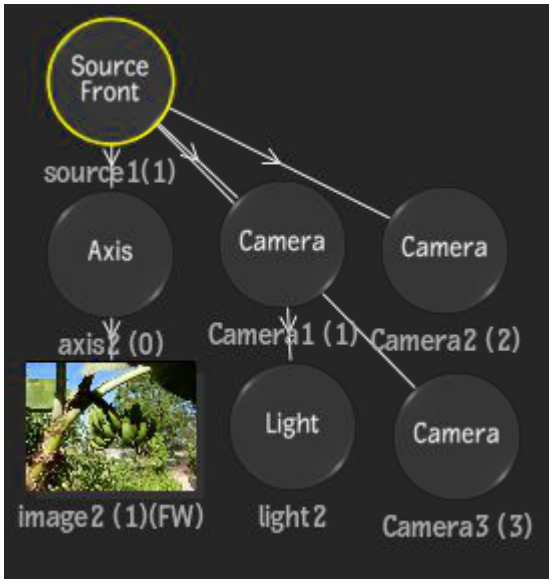
Import from Setup button Click to use the anti-aliasing and motion blur values from the Setup menu.

Home button Resets the curve viewer to show the whole curve.

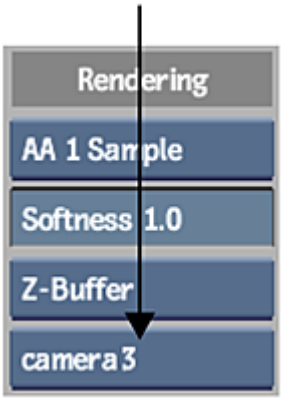
Reset button Resets the anti-aliasing and motion blur settings to their default values.

Adding Cameras to Source Nodes

You can add multiple cameras under a parent source node to change the point of view (by orbiting to a different orientation, for example). The first camera (parented to an ambient light) is automatically added to the source branch.



If you have multiple child cameras under a selected parent source node, you can switch from one camera to another from the Camera Selector box.



Action GMasks

In Action, a GMask object is an animatable spline-based object that you draw directly in an Action scene that allows you to apply transparency effects to the scene. A mask can be applied to all media by default, or limited to selected media. Depending on what you want to accomplish, you can create a transparency that reveals the background image, or hide a portion of the matte source or a front source. You can move the mask in the 3D environment using the Axis object.

When creating a mask for a stereo object, you can apply it to both eyes simultaneously. You can change the camera visibility options to apply a mask to only the Left Eye or Right Eye. Use the Visibility box to set the camera visibility options. To set up stereo objects in Action, see Starting a Stereoscopic Session (page 647).

Adding a GMask Node

When you add a GMask node to the schematic, it is added with an Axis node.

To add a GMask node:

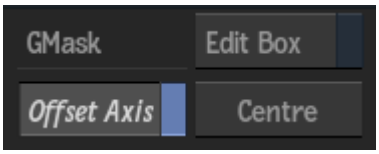
1 Do one of the following (depending on the method, the initial position of the parent Axis is different, although it can be offset afterwards) :

- Drag the GMask node from the node bin and place it in the Schematic view (the parent Axis is placed at the centre of the screen).
- Drag the GMask node from the node bin and place it in Result view (the parent Axis is placed where it is dropped).
- Double-click the GMask node (the parent Axis is placed at the centre of the GMask shape).
- Press **Alt+N** to automatically switch to Draw Shape mode, then once you add your first point, a GMask node is automatically added to the schematic (you do not need to access the node bin with this method).

A GMask object (called gmask1, by default), with its parent Axis, appears in the schematic. Once you have added the object, you can create the mask. See Creating a Mask (page 902).

TIP You can also create a mask using a preset shape. In this case, instead of the GMask node, select a GMask Plane, GMask Ellipse, or GMask Rectangle node.

2 After the mask is created, if needed, enable the Offset Axis button (at the bottom right of the image window) to be able to drag and position the parent Axis independently of the GMask shape. Click Centre to set the Axis position to the centre of the GMask shape.



Here are some considerations to keep in mind when using Offset Axis:

- Offset Axis can be used with any axis in the hierarchy, not just the axis located directly above the GMask.
- Offset Axis should be used with an axis that contains no keyframes, linked expressions, or look-at links, and ideally with Auto Key turned off. If the target channel contains a single keyframe, you need to be located at that keyframe for it to be updated properly, even if Auto Key is off.
- You cannot use Offset Axis if the target GMask vertex channel (User or Tracking) contains two keyframes or more. In this case, the Offset Axis option is greyed out.
- If you have two or more keyframes in the User channel and have not done any tracking in the GMask, you can still use Offset Axis with the Tracking channel of the GMask, but you first need to enable Edit Track (in the GMask Vertices menu) before offsetting the axis.
- While in Offset Axis mode, modifications to the camera associated to the GMask may lead to unwanted modifications to the GMask shape.

Applying Selective GMasks

Adding a GMask node to the schematic applies the mask to the entire scene. You can link the GMask object to specific media or lights to apply a mask selectively. You can also apply a link to exclude a mask.

To apply selective masks:

1 Add a GMask object to the scene. All media is affected.

2 Select GMask Link from the Tools box.

3 Do one of the following:
   - To apply a mask to an image or light, click the GMask node, and drag it to an image you want masked.

     The selected image is connected to the mask by a blue line with an arrow, and only the selected surfaces or lights are masked.
   - To exclude a mask from an image, hold the `Alt` key while clicking and dragging from the GMask node to the image you do not want masked.

     Excluded images are connected to the mask by an aqua dotted line with an "X", and they are not masked.

     NOTE GMask exclude links are not supported for lights.

Creating and Editing a Mask

After you have added a GMask object to the scene, you can now create the mask itself.

To create a mask:

1 In the Tools box, select Draw Shape (this is set by default when you first add the GMask node).
2 Do any of the following to draw the mask:
   - Click to add vertices.
   - Shift-click to hide vertices and tangents as you add them.
   - `Shift`-drag to draw the mask freehand. Vertices and tangents are hidden as you add them. After closing the mask, you can use the Lasso Fit field (in the Vertices tab of the GMask menu) to increase or decrease the number of vertices that define the freehand segments of the mask.
   - Click and drag out the tangents to adjust the curvature of the spline at the current vertex.
3 Complete the mask by:
   - Closing the mask shape while drawing (by clicking on the first created vertex). You can also enable the Closed button (in the GMask menu Mask tab) to connect a spline from the last point in the mask to the first control point (it is enabled automatically if you close the mask gesturally).
   - Leaving the mask as an open spline, and clicking Finish at the bottom right of the image window.

You can edit the mask after it is closed or finished.

Editing the Mask

Editing operations, such as adding and removing vertices, can be effected whether or not the mask is completed. You can edit the mask by gestural action on a single vertex or by using the Edit Box on multiple vertices, or by modifying the numerical settings in the GMask Vertices menu tab.

To add points to a mask:

1 In the Tools box, select Add Points.
2 Do one of the following:
   - Click on a spline between two existing vertices to add a new vertex.
   - Click on the first or last point of an open spline mask and click where you want to add a new point to continue drawing the mask.

To delete points from the mask:

1. In the Tools box, select Delete.
2. Click on a vertex to delete it.

   If the vertex is in a mask that is a closed shape, the splines that connected it to adjacent vertices will be replaced by a single spline.

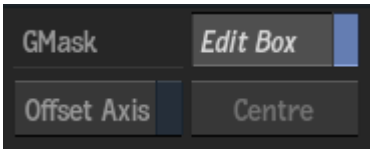
To adjust the mask (one vertex at a time):

1. Do one of the following:
   - Click and drag any of the points to adjust the mask.
   - Select a vertex and use the available settings in the Vertices menu tab.

   In either method, you can see the mask and the menu settings update accordingly.

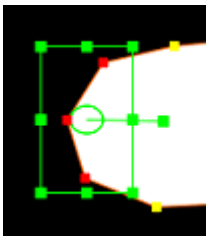
To edit multiple vertices with the Edit Box:

1. Click the Edit Box button at the bottom right of the image window (or press `Ctrl+E`).



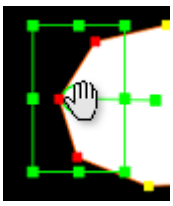
2. Press `Ctrl` and select multiple vertices that you want to edit.

   An edit box appears around the selected vertices with a centred pivot point.

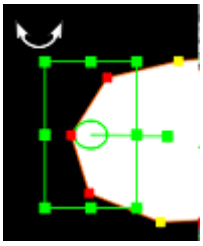


   TIP You can also use the GMask: Select All Vertices keyboard shortcut when using the Edit Box. This shortcut is based on the default User Profile for the application, and may change if you select a different profile.

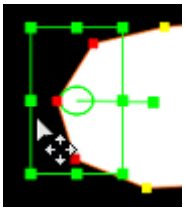
3. Use the available settings in the Vertices menu tab to apply transformations to the selection, or perform the following gestural actions, as needed:
   - Hover over the pivot point to enable to pivot widget, and move the pivot point.



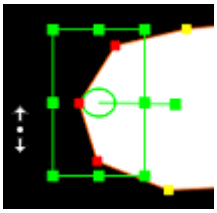
     Press `Ctrl+Alt` and click the pivot point to re-centre it.
   - Hover over the pivot arm or outside of a corner point of the Edit Box to enable the rotate widget, and drag left or right to rotate.



- Hover over an Edit Box point to enable the scale widget.



  - Press `Alt` and drag the Edit Box to scale symmetrically.
  - Use `Shift + Alt` to drag the Edit Box proportionately from the centre.
- Hover just outside the Edit Box to enable the shear widget, and drag to shear.



TIP You can also click and drag inside the Edit Box to move (translate) the complete Edit Box. Press `Shift` and drag to constrain the translation to the X and Y axes. There are also a number of other keyboard shortcuts to work with the Edit Box (including for making minor adjustments). Search the keyboard shortcut editor for Edit Box to see the shortcuts for your chosen profile.

4 Click the Edit Box button again, to deselect the vertices after you have completed your edit.

Mask Settings

Finishing and Editing Masks

These mask settings can be found at the bottom right of the image window.

Finish button Complete the creation of the mask without closing it. Once you click Finish, or a mask is closed, this button disappears.

Edit Box button Enable to adjust the mask after it has been drawn. Press Ctrl and select multiple mask vertices to form the Edit Box.

Offset Axis button Enable to be able to offset the axis position independently of the GMask. You can use the Axis menu controls or the axis icon in the image window.

Centre button Click to centre the axis within the GMask shape. Available when Offset Axis is enabled.

Customizing Mask Settings

You can customize a mask's properties by changing settings in the node's Mask tab. Some settings may be different depending on how the GMask is accessed (such as through the GMask Tracer node).

GMask Tracer Colour field Available when using a GMask through the GMask Tracer node. Displays the blend between the outgoing and incoming images inside the mask.

Transparency field Displays the percentage of transparency of the mask. Editable.

Intensity field Displays the blend between the outgoing and incoming image inside the mask. Editable.

Smoothing field Displays the amount of smoothing applied to gradient regions. Editable.

Blur field Displays the amount of blur applied to the GMask (with or without gradients). Editable.

Camera box Specify which camera to take into account when tracking or viewing in Action Object view (F8). Select Auto Frame when the GMask is co-planar with the image.

Camera field Displays the active camera number. Non-editable.

Closed button Enable to connect a spline from the last point in the mask to the first control point (enabled automatically if you close the mask gesturally).

Hole button Enable to define the attached 3D Shape spline as a hole. Disable to keep the spline filled. Available only if a 3D Shape node is parented to a GMask.

Invert button Enable to apply settings to the area outside of the mask.

Lock button Enable to lock the relative position of vertices.

Animation box Select Shape Animation to group all keyframes into a single channel called shape; select Vertex Animation to locate all keyframes on their respective channels.

Shape Animation Type box Select the interpolation type for the shape animation curve.

Tolerance field Displays the segmentation value of the spline. Use to smoothen or wrinkle the spline. Editable.

Blend Mode box Select a blending mode for the mask when it overlaps with another mask. Locked when a master blend option is selected in the Output menu.

Mask Tracer Blend Mode box Available when using a GMask through the Mask Tracer node. Select a blending mode for the mask images.

Correlation box Select how overlapping gradients are blending. This box is displayed when an Autodesk Maya Composite blending mode is selected in the Blend Mode box.

Post Processing box Select whether to use the mask as a mask or occluder when encountering post processing effects. If you attach a Gmask to a specific post-processing node, for example a lens flare, and choose Use As Occluder, only the effect will be blocked by the occluder. Use As Occluder affects occlusion of Action Lens Flares, as well as defining the result of the Occluder output passes, which can be used as an Action output as well as an input for any Camera FX.

Active button Enable to activate the GMask for rendering. Disable to use the GMask for tracking purposes only.

Motion Blur button Enable to apply a blur effect to the selected mask. (Motion blur must be enabled in the Setup menu.)

Visibility box Select to apply the mask to the Right, Left, or Mono camera in a stereo scene.

Mask Display Properties

You can change the display properties of your mask in the Display tab.

Shape Reference button Enable to display an overlay of the reference frame when tracking in Object view (F8).

Shape Current button Enable to display an overlay of the current frame when tracking in Object view (F8).

Spline colour pot Displays the colour of mask splines. Editable.

Vertices colour pot When the mask is active, displays the colour of the unselected vertices. Editable.

Gradient colour pot When the gradient is active, displays the colour of the gradient spline. Editable.

Tangents Display option box Select to show all, selected, or no tangents on any mask that is selected.

Show Icons button Enable to display tracer icons in the image window.

Foreground colour pot Displays the colour of the tracer foreground colour picker. Editable.

Background colour pot Displays the colour of the tracer background colour picker. Editable.

Manipulating Vertices and Tangents

You can manipulate vertices and tangents during and after mask creation. Edit vertices and tangents gesturally when the Tools box mode is set to Select or Break Tangents. Alternatively, edit them using the Vertices tab.

Vertices selected as a group can be transformed using the Edit Box (page 903). The angle and length of tangent handles can also be manipulated as a group.

Selecting Vertices and Tangents

You can select vertices and tangent handles using the Select option of the Tools box:

- To select an individual vertex or tangent handle, click the vertex or tangent handle.
- To select multiple vertices, `Ctrl`-drag a selection box over a series of vertices.
- To select multiple vertices using the pen, press the pen button and drag a selection box over a series of vertices.
- To add more vertices to an existing selection, press `Shift+Ctrl` and drag a selection box over the additional vertices.
- To deselect all vertices, click anywhere outside the mask.

TIP You can also use the GMask: Select All Vertices keyboard shortcut. This shortcut is based on the default User Profile for the application, and may change if you select a different profile.

Moving Vertices and Tangents

You can move vertices and tangents using the Select option of the Tools box:

- To move a vertex or tangent in any direction, click the vertex or tangent and drag.
- To move selected vertices in any direction, click one of the selected vertices and drag.
- To ensure better continuity on a garbage mask, automatically adjust tangents as you move vertices or scale the mask. Enable Auto Adjust and drag a vertex. To toggle temporarily enable or disable Auto Adjust, press and hold `G` and then drag a vertex. Auto Tangents must be enabled to use the Auto Adjust feature (you can toggle Auto Tangent on-the-fly using the GMask: Auto Tangent Toggle Mode keyboard shortcut).

- To keep tangent handles an equal length, `Ctrl+Alt`-drag one vertex handle to automatically change the length of the other tangent handle at the same time.

You can use the Vertices tab to move vertex and tangent positions.

The Selection fields display the current position of the vertex. If multiple vertices are selected, the position displayed is of the vertex that is closest to the initial vertex created in the mask.

Breaking, Removing, and Adding Tangents

You can separate two tangent handles (break the tangent) and move them separately option from the Tools box:

- To break and move a tangent handle, select Break Tangent and click the tangent handle. The tangent is displayed as a dashed line, indicating it is "broken."
- To reconnect two broken tangent handles, click the vertex or select Auto Tangents and click either of the two tangent handles. The tangent is displayed as a solid line (you can toggle Auto Tangent on-the-fly using the GMask: Auto Tangent Toggle Mode keyboard shortcut).
- To change the position of an individual tangent handle after releasing the cursor, use the Select option in the Tools box.

In the Vertices tab, use the fields in the Tangents section to change the length and angle for the incoming and outgoing tangent handles. If the tangents do not combine to form a straight line, the tangents will be broken and displayed as a broken line.

You can also use the Break option to remove tangents from vertices by clicking the vertex while in Break mode. When you remove a tangent, the interpolation of the spline between the two vertices is modified.

The shape of the border line differs depending on whether adjacent vertices are broken:

- If adjacent vertices are unbroken, the border line curves as it approaches the vertices.
- If adjacent vertices are broken, the border line is straight as it approaches the vertices.

You can create a garbage mask composed entirely of straight edges by removing the tangents from all the vertices:

- To remove the tangent of a vertex, select Break Tangent and click the vertex.
- To add a tangent back to a vertex, select Auto Tangent and click the vertex. Alternatively, using the Break Tangent option, click the vertex and drag the tangent out again.

Vertices and Tangents Settings

Vertices Settings

Settings are also updated when you gesturally update a GMask vertex selection in the image window. Settings are available based on the selection in the mask.

X Selection field Displays the position on the X axis of the selected vertex. Editable.

Y Selection field Displays the position on the Y axis of the selected vertex. Editable.

X Translate field Displays the position on the X axis of the selected vertex. Editable.

Y Translate field Displays the position on the Y axis of the selected vertex. Editable.

X Scale field Displays the scale value on the X axis of the selected vertices. Editable.

Y Scale field Displays the scale value on the Y axis of the selected vertices. Editable.

Proportional button Enable to change the scale fields proportionally.

X Shear field Displays the shear value on the X axis of the selected vertices. Editable.

Y Shear field Displays the shear value on the Y axis of the selected vertices. Editable.

Z Rotation field Displays the rotation value on the Z axis of the selected vertices. Editable.

Tracking Channel Settings

Edit Track button Enable to edit tracking data directly from the image window without having to switch to the Animation channel. Make sure the Auto Key is enabled, so as not to lose any keyframe modifications.

Tangents Settings

Settings are also updated when you gesturally update a GMask tangent selection in the image window.

In Angle field Displays the angle of the incoming tangent handle. Editable.

In Distance field Displays the length of the incoming tangent handle. Editable.

Out Angle field Displays the angle of the outgoing tangent handle. Editable.

Out Distance field Displays the length of the outgoing tangent handle. Editable.

Auto Tangents button Enable to create tangents automatically. When enabled, the Tracker will affect tangents.

Auto Adjust button Enable to modify tangents on either side of the selected vertex automatically. Press G to temporarily enable or disable during a gestural operation.

Equal Length button Enable to set tangents on either side of vertices to be the same length (based on the proximity to the next vertex).

Lasso Fit field Displays the tolerance level for the number of points in the segments of the mask, if it was drawn freehand. This field is unavailable once a mask is manipulated, so if needed, set as soon as the mask is drawn. Editable.

Adding a Gradient to a Mask

The gradient allows you to create a region of softness around the mask edge. It applies a gradient according to the distance of the borders from the mask spline. You can create a uniform gradient or customize it at different parts of the mask. The gradient can have two softness borders, one inside and one outside the mask edge.

To add a gradient to a mask:

1. From the Tools box, change the mode to Add Points.
2. Hold down `Shift` and drag away from the spline.
3. Click on the mask spline.
4. (Optional) Do one of the following:
   - `Shift`-click on the spline again and drag in the same direction as the existing gradient to add a point in the gradient at a different position and distance from the mask.
   - `Shift`-click on the spline and drag away from it in the other direction to create a second gradient.

   TIP To delete a gradient point, change the Tools mode to Delete, then press `Shift` and click the gradient handle on the mask spline.

The position and distance of a gradient point from the mask spline can be controlled gesturally in Select mode, or in the Gradient tab.

Gradient Settings

Position field Displays the relative position of the gradient control point in relation to the vertices. For example, if the control point is between the third and fourth vertices, its value is 3.5. Editable.

Offset field Displays the distance from the gradient spline to the mask spline. Editable.

Gradient Curve Displays the transparency distribution of the gradient. The y axis represents the percentage of transparency and the x axis represents the percentage of distance from the vertex to the spline. This curve can be animated. You can edit, add and delete points from the curve.

Set Key button Sets the current values for the gradient curve in the current frame(when Autokey is disabled).

Delete Key button Deletes the selected gradient curve keyframes.

Reset Selection box Select whether to reset all of the gradient settings (Reset All) or just the gradient curve (Reset Key).

Home button Resets the gradient curve viewer to show the whole curve.

Undo button Undoes the gradient curve operations.

Tracking a Mask

You can track a mask in your scene from the GMask object's Tracking tab. You can track the mask vertices in the Tracker, and track splines using the GMask object's own planar tracker.

Tracking Mask Vertices

To track mask vertices:

1. From the Action GMask menu's Tracking tab, select the media to be tracked from the Use Media field (to track the Back media, select 0).
2. In the image window, select the vertices (page 906) you want to track.
3. Select a tracking mode:

   Two types of vertex tracking are available:

   - Each Vertex: Trackers are available for each selected vertex of the GMask.
   - Cluster: Tracker1 represents the selected vertices, and Tracker2 represents rotation and scale (if selected).

   If you select Cluster, enable Rotation, Scale or both, as needed.
4. Adjust any other Tracking Settings (page 910), as needed.
5. Click Enter Tracker.

   The Tracker menu is displayed.
6. In the Tracker menu, enable Context to use the Action scene as the reference.
7. Gang the trackers.
8. Enable Auto Update Reference.
9. Click Analyze to generate the tracking data.

NOTE You can fine-tune the tracking data if necessary. See Tracking Difficult Shots and Correcting Errors (page 1091).

10 After the analysis is complete, click Return.

The GMask menu in Action reappears. The tracking data is automatically applied to the GMask media.

11 Enable Edit Track in the GMask Vertices menu to edit tracking data directly from the image window without having to switch to the Animation channel. The vertices are displayed in blue when Edit Track is enabled.

TIP If you want to add new vertices to your tracked GMask, make sure to enable Edit Track before selecting Add Points, so that the new points also follow the tracking data.

Planar Tracking

Planar tracking in Action uses tracking algorithms along with combinations of settings to perform the track. Results can vary depending on the footage, as well as settings applied. See Getting Better Results with the Planar Tracker (page 656).

To apply planar tracking to a mask:

1 Select the mask you want to track.

2 In the GMask menu's Tracking tab, set the Use Media field to assign the back or front/matte media (select 0 if you want to track the Back media).

3 Adjust any Tracking Settings (page 910), as needed (such as Rotation and Scale).

4 Select an option from the Direction box to determine if the frames before or after the current frame will be used for analysis.

TIP When tracking a shape that cuts off during analysis (for example, a person walking into or out of the frame), begin shape tracking when the whole shape is apparent, and track in the direction of the frames at which the shape begins to disappear.

5 Set any Analysis Constraints, if needed.

6 Click the Analyze button to perform the planar tracking analysis.

7 To perform additional analysis in a different direction from the first frame of the analysis, toggle the Direction box and click Analyze again.

TIP The tracking data is stored as keyframes in the *track_shape* channel of the Channel Editor. You also have access to a *shape* channel that you can use to fine tune or add additional deformation to the existing tracking animation, without affecting the tracking keyframes. The *shape* channel also holds any keyframe user data prior to the tracking, so if you reset your *track_shape* channel, you can recover your state before the tracking had been performed.

Tracking Settings

Tracking Tab

Common Settings

Tracking Mode box Select which tracking mode to apply.

Enter Tracker button Click to open the Tracker menu to apply tracking data for the selected tracking mode.

Rotation button Enable to apply rotation data to the track.

Skew button Enable to apply skew data to the track.

Scale button Enable to apply scale data to the track.

Perspective button Enable to apply perspective data to the track.

Camera box Specify which camera to take into account when tracking or viewing in Action Object view (F8).

Use Media field Displays the number of the media associated with the analysis. To track the Back media, select 0.

Tracking View box When the media associated to the analysis is a Source node, select to track the media in the Source Front, Source Matte, or Source Result view.

Media FX button Enable to take into account blur and colour correct settings from the Action Media list in the analysis.

Planar Tracking Settings

Analyze button Click to start the planar tracking analysis.

Direction box Select whether to track forward or backward.

Go to Reference button Click to move to the frame of the last Snap operation, or press Ctrl and click to go to the last analyzed frame.

Step button Click to advance to the next frame and analyze.

Set button Click to analyze the current frame, based on the reference frame, to realign the shape used for tracking. Press Ctrl and click Set to add a keyframe on the track without doing any analysis.

Skip button Click to advance to the next frame and delete the selected point on the tracking path (performs a combination of the Step and Delete buttons).

Delete button Click to delete the selected point on the tracking path. You can also use the Backspace key to delete the current keyframe, and move to the previous keyframe.

Auto Update Reference button Enable to update the tracking reference at the current frame after each analysis. Disable to track the movement of the reference point specified in the reference frame.

Snap button Click to redefine the tracking reference at the selected frame.

GMask Inclusive box Select whether to use the union of parented GMasks or GMask gradients as the tracking reference. Select None to use the whole image (Axis), or the four corner region (Perspective Grid).

GMask Exclusive box Select whether to exclude GMasks in the tracking analysis.

Tracking Algorithm box Select which planar tracking algorithm to use. Select Region Warping to use all the pixels defined within the reference, or Multi Feature Detection to use some points defined within the reference. Different settings appear below based on the algorithm chosen.

Sampling field Displays the size of the reference for Region Warping. A smaller reference increases performance at the expense of the quality of its results. Editable.

Lighting button Enable to estimate local lightning variations of the tracked surface.

Occlusions button Enable to automatically exclude parts of the reference to be analyzed based on their likelihood to be occluded.

Analyze RGB Channels button Enable to include all colour channels in the tracking analysis. Disable to only track luminance.

Average field Displays the blending factor to apply to the tracking analysis, based on the neighbouring frames. Use to remove jitter, for example. Note that this operation is destructive, and once applied, the value returns to 0 (you can use Undo to revert, if needed). Editable.

Size field Displays the area in pixels of each tracker. Editable.

Min Number field Displays the minimum number of trackers to use in the analysis. Editable.

Tolerance field Displays the level of acceptable changes in the pixel environment. For example, a low tolerance value ensures that the motion of an external object passing in front of the tracked object does not get tracked through the analysis. Editable.

Creating a Location-Based Key With the GMask Tracer

Use the settings in the Tracer tab to perform location-based keying operations on a GMask. The underlying algorithm allows each GMask to host a key based on its associated media. You can use the result of the tracer to then apply other effects, such as adding a Lightbox effect only to the sky in an image.

The GMask Tracer is also accessible from the GMask Tracer node (page 1099). One thing to consider is how you access the GMask (and Tracer). Action GMasks are primarily used to punch holes in 3D objects, while GMask Tracer GMasks are used for matte creation. As such, the mask shape may need to be inverted in Action to take full advantage of the Tracer capabilities.

To use the GMask Tracer:

1. Add a GMask to your scene.

   TIP You can also use a 3D Shape node, since a GMask node is automatically added to its branch.

2. Switch to F8 Object view and draw your GMask.
3. From the Use Media field in the GMask menu's Tracer tab, select the media to use with the tracer (to use the Back media, select 0).
4. Identify which frame you would like to create your first key shape on and click Add.

   Doing so creates two colour picker boxes in the centre of the GMask shape; one to identify the Background colour (green box) that you would like to key out, and the other the Foreground colour that you would like to preserve (red box).

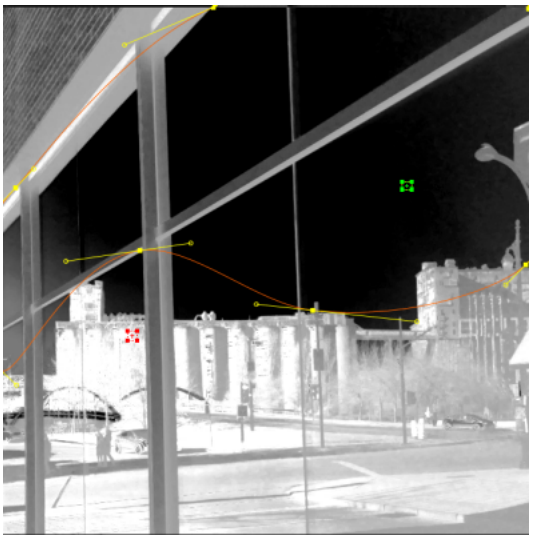
5. Resize and position the colour picker boxes, as needed.

   TIP If colour picker boxes are overlapping, press Q to toggle between selections.

6 If needed, use the Add Background and Add Foreground buttons to add and position more colour pickers at the same frame.

7 Use other settings in the Tracer menu, such as to add softness, invert the colour pickers, apply to the GMask gradient only, or add post-processing refinements.

TIP Click F8 again to see the result of the keying operation in context.



Click F4 to see the GMask only.

8 There may be colour or lighting variations in the clip. If a good key cannot be performed for the whole duration of the clip, you can move to another frame where you would like to create a new key shape, and click Add once more, then follow from Step 5. Tracer settings are interpolated between keyframes.

Here are some tips for working with the GMask Tracer:

- Multiple GMasks can be used, each providing a Tracer analysis for the clip. This can be useful, for example, with a model on a green screen. You can use different GMasks to isolate different parts of the hair.
- Use open or closed splines, with or without gradients. Experiment to find your best result.
- You can quickly enable or disable the Tracer in the Rendering section of the GMask Tracer tab; giving you the choice of having the Tracer analysis included or excluded in a tracking operation, for example.
- Use a GMask Link or Light Link (in Action) to apply a Tracer result selectively to what you want it to affect.
- While the analyses are done at specific frames, the softness and post-processing settings can be animated independently from these analysis checkpoints.

Tracer Settings

Analysis Settings

Add button Click to add a tracer at a specific frame.

Delete button Click to delete an existing tracer.

Previous button Click to jump to the previous tracer.

Next button Click to jump to the next tracer.

Media Settings

Use Media field Displays the number of the media associated with the analysis. To track the Back media, select 0.

Tracking View box When the media associated to the analysis is a Source node, select to track the media in the Source Front, Source Matte, or Source Result view.

Media FX button Enable to take into account blur and colour correct settings from the Action Media list in the analysis.

Colour Pickers Settings

Add Background button Click to add more background colour sampling boxes to define the background 3D colour volume.

Add Foreground button Click to add more foreground colour sampling boxes to define the foreground 3D colour volume.

Delete Selected button Click to delete the current colour sampling box selection, and remove it from either the foreground or background 3D colour volume.

Rendering Settings

Enable button Click to activate or deactivate the Tracer rendering result.

Invert button Click to invert the Tracer result without affecting the GMask result. Note: Inverting the whole GMask will also invert the Tracer result.

Gradient Only button Click to confine the Tracer result to the gradient section of the GMask.

Softness Settings

Background Softness field Displays the amount of 3D colour volume around the background colour range. Editable.

Foreground Softness field Displays the amount of 3D colour volume around the foreground colour range. Editable.

Post-Processing Settings

Shrink field Use to reduce the spatial extent of the Tracer result. Editable.

Dilate field Use to increase the spatial extent of the Tracer result. Editable.

Blur field Use to blur the Tracer result. Editable.

Rendering the Mask

In the Output menu, you can set up preferences to render the mask. See Processing Multiple Outputs (page 598).

To make the mask visible in the output, enable Use GMask in the Render Options panel.

The mask output in the render is dependent on the settings of the Type box In the Output Options panel. Do any of the following:

- Set the Type box to Comp to reveal the background in the masked area in the render.
- Set the Type box to Matte or Media Matte to render a matte output of the scene or the selected object in the scene that reveals the colour, intensity and smoothing in the masked area.

Action Particles

Use Action's 3D particle system to create a variety of effects such as snow, rain, fog, and tornadoes, as well as fireworks, sparks, and explosions.

Particles are 3D objects that originate from a surface, light, or 3D object. Particles can be spheres, cones, squares, points, lines, or any image, 3D text, or 3D object. Particles are generated in the direction of the normals of a surface, light, or 3D object—a normal is a line perpendicular to the surface of an object.

In Action, the main components for generating particles are generators, manipulators, and bouncers. You can also use the particle system to explode a 3D object or an image.

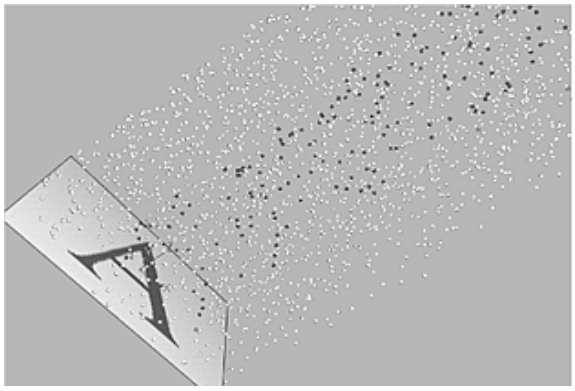
NOTE Rendering particles is faster if Z-Buffer On is selected in the Rendering section of the Action Setup menu.

Generators

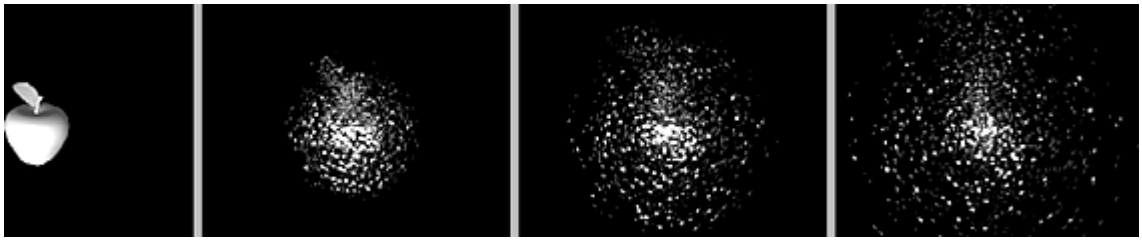
To create a particle stream, start by adding a particle generator that creates a stream of particles when attached to a light source, 3D object, or surface. You can use three different generators: light source, surface, and 3D object.

Particle generators have two settings:

- The Generate setting creates a stream of particles using the shape of a surface, a 3D object, or the spread of a light source. See Generating Particles (page 948).

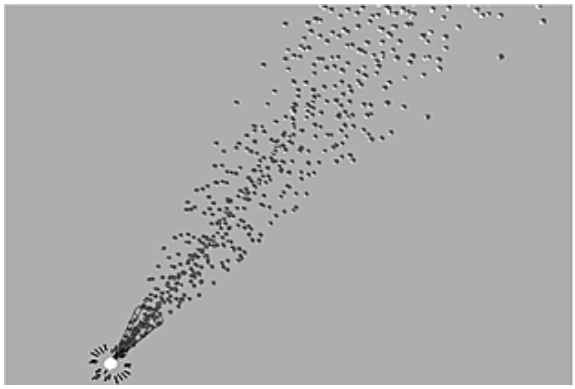


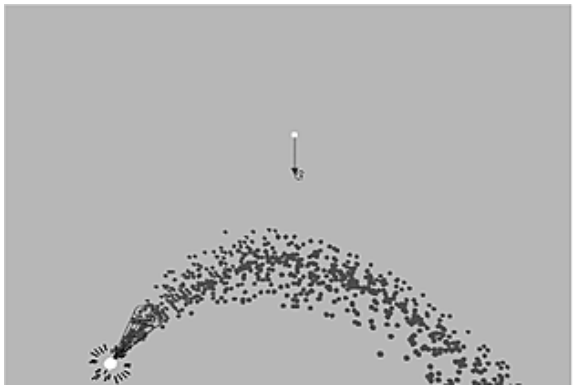
- The Explode setting breaks up a surface or 3D object. See Exploding Objects and Surfaces (page 961).



Manipulators

Particle manipulators are used to influence either the position or the speed of particles. Manipulators can simulate the effects of a vortex, gravitational pull, or the pull toward a point, line, or plane. Manipulators are parented by the particle stream that they influence.

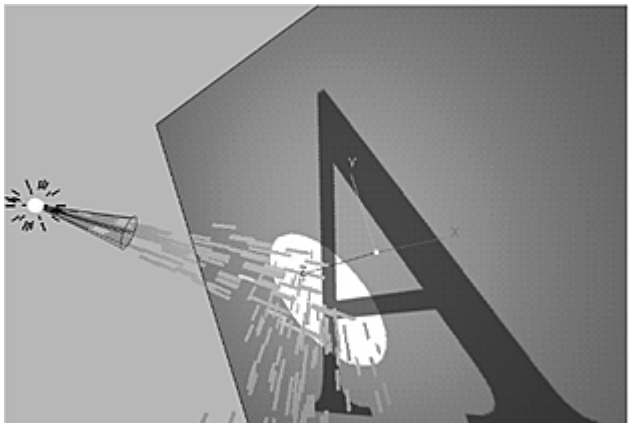




Action offers eight manipulators. Instructions on how to use each manipulator are provided in Manipulating Particles (page 964).

Bouncers

Particle bouncers are used to influence the behaviour of particles when they come into contact with a surface. You can parent an axis or light source to a particle bouncer. See Bouncing Particles (page 982).



NOTE Action setups created in a pre-9.0 version of Flame that use a Variation value for Number, LifeTime, Speed, and TrailSize can be restored in the current version, but will render slightly differently.

Particles Workflow

The following table shows a possible workflow when working with particles in Action.

Step:	Refer to:
1. Add a particle generator.	Generating Particles (page 948).

Step:	Refer to:
2. Add a particle manipulator.	Manipulating Particles (page 964).
3. Set the particle path.	Forming a Particle Path (page 973).
4. Add bouncers.	Bouncing Particles (page 982).
5. Modify the particle stream.	Setting Stream Properties (page 952).
6. Apply texture to particles.	Applying Textures to Particles (page 960).
7. Add gravity.	Simulating Gravity (page 968).
8. Render particles.	Setting Particle Rendering Properties (page 954).

Using Particle Presets

A number of particle presets are included in Action, such as a fog effect to add to your scene. These presets can also help you learn how the particle system works. The presets are divided into a number of categories and are easily added to your scene from the Action node bin.

To add a particle preset:

1. Do one of the following:
   - Drag the Presets node from the node bin and place it in the schematic.
   - Double-click the Presets node. You do not need to be in Schematic view to add a node in this manner.

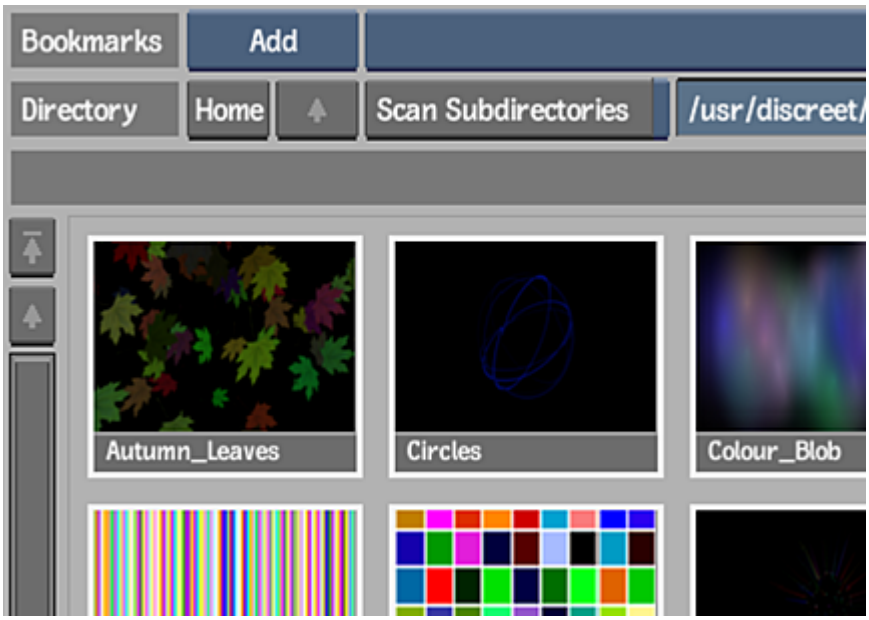
     The file browser opens.
2. From the Preset Type box, select Particles.

   The Particle Preset file browser appears, pointing to the default location of the presets: *opt/Autodesk/<product home>/particle_presets*.
3. Optional: Enable Scale to Action Resolution to load the preset in the current Action resolution.
4. Optional: Select which rendering settings to enable or disable in the preset (Z-Buffer, Shading, Polygon Resolution, and Colour Clamping).

   NOTE These settings are enabled by default, and by disabling any of them, you may not see the intended results in the preset.

5 Navigate through the subfolders to select the particle preset you want to load. Hold `Ctrl` and click to select multiple presets.

TIP Switch to Proxies view to see a visual representation of the presets.



6 Click Load.

The particle preset is then appended to your Action scene.

NOTE Some presets have *.psd* textures associated with them, and a Batch reel is created to house these textures. It is closed by default. For this reason, Particle Presets are not available when accessing Action as a Timeline FX.

Presets Browser Settings

Preset Type box Select the category of presets to display in the browser.

Scale to Action Resolution button Enable to load the preset in the current Action resolution.

Z-Buffer button Enable to load the Z-buffer rendering settings of the preset.

Shading button Enable to load the shading rendering settings of the preset.

Polygon Resolution button Enable to load the rendering resolution settings of the preset.

Colour Clamping button Enable to load the resolution colour clamping settings of the preset.

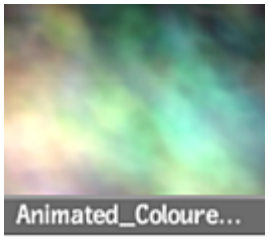
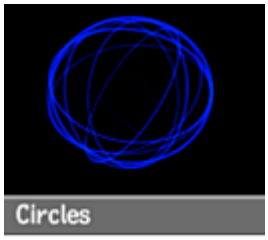
Particle Preset Reference

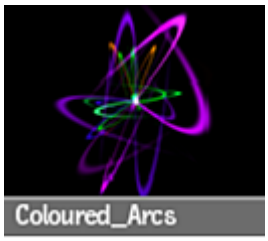
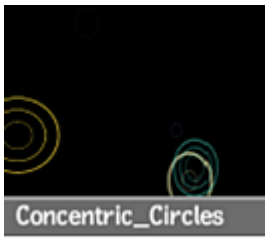
A number of particle presets are included in Action, such as pyrotechnic or liquid effects to add to your scene. These presets can also help you learn how the particle system works. The presets are divided into a number of categories and are easily added to your scene from the Action node bar.

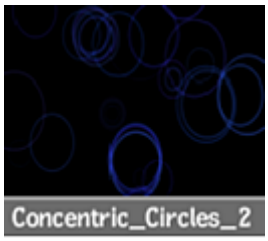
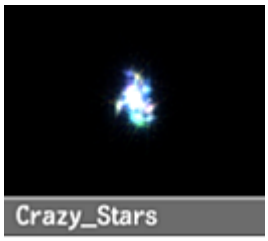
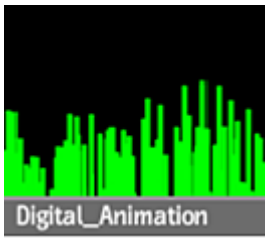
Use the following tables to get a quick overview of the particle presets, along with best use comments, if applicable. For more information on particles in general, see About Action's Particle System (page 915).

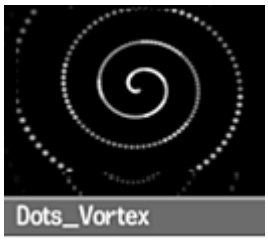
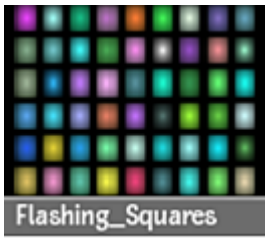
Abstract/Backdrops

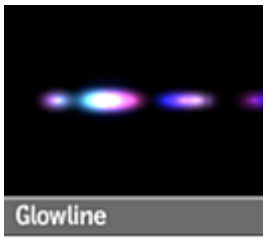
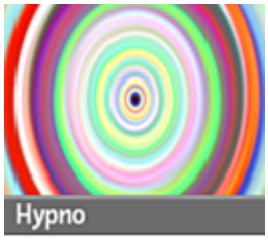
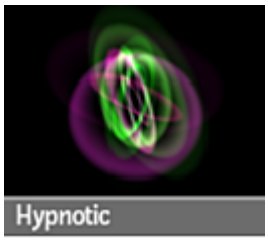
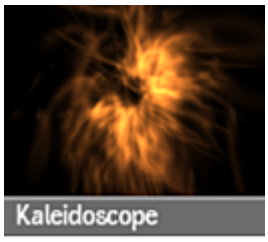
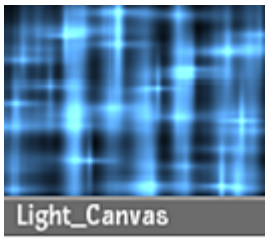
Use these presets as backdrops or to create abstract art particle effects in your scene.

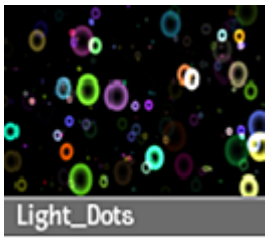
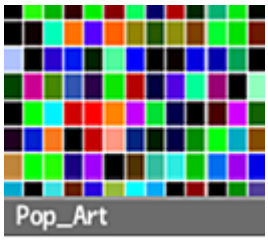
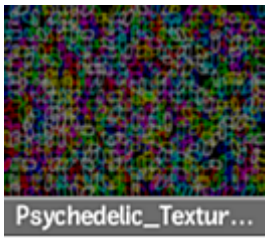
Proxy	Name/Description/Comments
	**Animated_Coloured_Background** Use Number and Lifetime to slow down or speed up the animation. Try with Z Rotation and Spin settings.
	**Art_Coloured_Lines** Speed sets the amount of stripes. Try with Colour variance in motif_generator menu.
	**Autumn_Leaves** Use Number, Size, and Speed settings. Try with the Rotation and Spin settings.
	**Burning_Text** Particle Generator and cascade effect to burn the text and make it disappear.
	**Circles** Use Number, Lifetime, and Size settings. Try with the Spin setting.

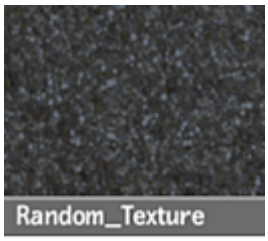
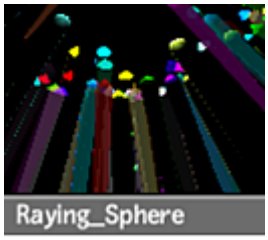
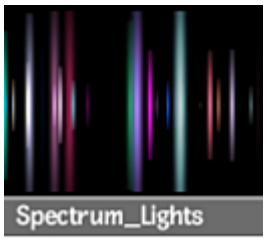
Proxy	Name/Description/Comments
	**Colour_Blob** Animated coloured gradients. Use Number, Lifetime, and Size settings. Try with Blending modes and Lighting.
	**Colour_Cloud** Animated. Use Number and Size settings. Try with Z Rotation and Lighting.
	**Colour_Vortex** Animated. Play with circ_gradient_ax (especially Y Scaling). Try with Z Spin setting. For a nice spiral effect, try Timesteps = 2 and Trail values up to 50.
	**Coloured_Arcs** Arcs rotating towards the camera. Try text_ax (posX=140, RotZ=200, ScaleX=30).
	**Concentric_Circles** Animated. Use circles_emitter Number and Lifetime fields to set the amount of circles.

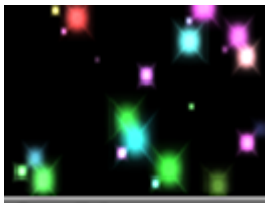
Proxy	Name/Description/Comments
	**Concentric_Circles_2** Use Number and Size. Speed up or slow down animation with Lifetime.
	**Crazy_Serpentine** Random circular patterns. Try with the size manipulator (Magnitude). Modify the Power animation in vortex manipulator.
	**Crazy_Stars** Pulsing coloured stars. Use Number and Size settings.
	**Digital_Animation** Animated rectangles. Use Number, Lifetime, Size, and Speed settings. Change colour using light_colour.
	**Dots_Spiral** Animate Size setting. Try with Magnitude in size manipulator. Change colour using the Geometry settings of dots_generator.

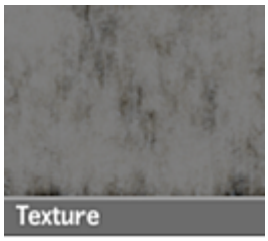
Proxy	Name/Description/Comments
	**Dots_Vortex** Use the Size setting. Try with Magnitude in the size manipulator and with Power in the vortex manipulator (try Power = 5). Move the camera and see the effect of gravity.
	**Fairy_Text** 3D text made of coloured stars. Try star_text_ax (Scale=22) and generator (Number=95, Number_V=25, Lifetime=75, Size=12, Size_V=20, Speed_V=0,1).
	**Flashing_Squares** Influence the colour using the light's colour settings.
	**Glow_Scanline** Vertical and horizontal moving glows. Use Number, Lifetime, and Size to characterize horizontal & vertical glows. Use Magnitude in the V_size and H_size manipulators to set the scaling's amplitude. Use Magnitude in the V_horizontal_osc manipulator to set the horizontal motion's amplitude. Change colour with the light.
	**Glowing_Streaks** Coloured streaks coming from behind the camera to the centre of the frame. Try Number=110, Number_V=35, Lifetime=1, and Speed_V=70. Orbit the camera.

Proxy	Name/Description/Comments
	**Glowline** Moving horizontal glows. Use Number, Lifetime, and Size settings to characterize the glow. Try with Magnitude in the size manipulator to scale the glows.
	**Hypno** Influence the colour using the light's colour settings.
	**Hypnotic** Try with Rotation and Spin settings. Orbit the camera.
	**Kaleidoscope** Change the colour using the light's colour settings.
	**Light_Canvas** Grid of animated glows. Influence the colour using the light's colour settings. Characterize the vertical and horizontal glows using the circgrad_V_ax and circgrad_H_ax axes.

Proxy	Name/Description/Comments
	**Light_Dots** Speed up or slow down the effect using the Power setting in the size Animator.
	**Moving_Glows** Circular illuminated glow. Change colour by colour correcting the white media. Use Number and Size to characterize the ring. Use the pos_scale axis to set the position and rotation of the ring in the scene.
	**Path_Light** Animated glow on a path.
	**Pop_Art** Try with a close-up shot of a face as REPLACE_ME media. Zoom in to have fewer coloured squares in the frame, or zoom out to have more.
	**Psychedelic_Texture** Animated. Use Magnitude in the size manipulator to set pattern size. Try with the Trailsize field in the Generator menu.

Proxy	Name/Description/Comments
	**Random_Texture** Animated. Use Number, Lifetime, Size, and Speed settings to characterize the texture. Try with the Spin settings to set the animation.
	**Raying_Sphere** Black sphere emitting coloured streaks. Use Number and Speed settings. Try with the Tailsize field.
	**Sky** Moving cloudy sky. Use Number and Colour_V settings to characterize the sky. Use Position settings in the sky_motion manipulator to set how the clouds are moving.
	**Space_Blob** Coloured glows with a burning effect. Try the PD_ax settings, and influence colour using the Light's colour settings.
	**Spectrum_Lights** Animated vertical glows. Use Number, Lifetime, and Size to characterize the lights. Try with Magnitude in the size manipulator to control the scaling amplitude of the lights.

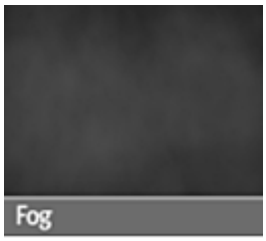
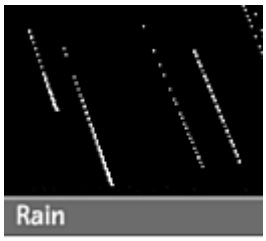
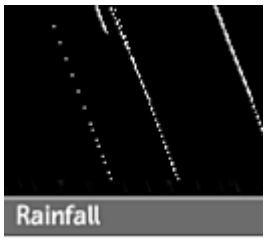
Proxy	Name/Description/Comments
	**Stagelights** Use Number, Lifetime, and Size. Try with Magnitude in the size manipulator to control the scaling amplitude of the stagelights.
	**Star_Spiral** Use Number, Lifetime, and Size. Try with Magnitude in the size manipulator to control the scaling amplitude of the stars.
	**Text_Atom** Animated 3D Text birth.
	**Text_Distort** Use Number, Lifetime, and Size. Try with Rotation_V, Spin, and Spin_V settings. Try also to change the light's colour and the glow_generator's Material settings.
	**Text_Dots** Use Number, Lifetime, and Size. Try with Rotation_V, Spin, and Spin_V settings. Try also to change the light's colour and the glow_generator's Material settings.

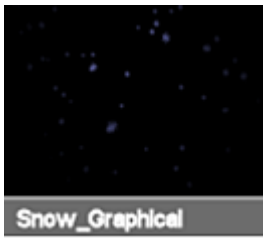
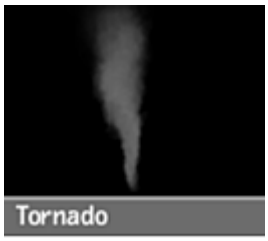
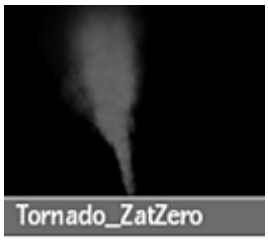
Proxy	Name/Description/Comments
	**Text_Glow** Use the Size setting. Try with Magnitude in the size manipulator.
	**Text_Particules** Animated cascade effect with the 3D Text Geom node.
	**Texture** Use the Size setting. Try with the Rotation_V setting. Play with the texture_ax settings.

Atmospheric

Use these presets to create weather-like effects, such as clouds, rain, or snow.

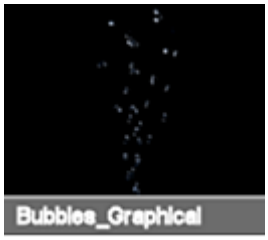
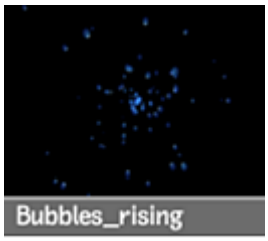
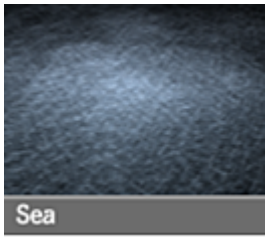
Proxy	Name/Description/Comments
	**Cloud_Generator** A one-frame cloud. Use Number and Size settings. Try with the texture's Axis settings and the geometry's Magnitude and Scaling settings.

Proxy	Name/Description/Comments
	**Fog** Use Number, Lifetime, Size, and Speed to characterize the fog.
	**Heavy_Fog** Use Number, Lifetime, Size, and Speed to characterize the fog.
	**Rain** Use Number and Speed to trigger the amount of rain. Colour correct the white media's Front, and set rain_generator Transparency to improve blending.
	**Rainfall** Use Number and Speed to trigger the amount of rain. Colour correct the white media's Front, and set rain_generator Transparency to improve blending. Use gravity's Rotation_Y to make it windier.
	**Snow** Use Number, Size, and Speed for characterizing the snow.

Proxy	Name/Description/Comments
	**Snow_Graphical** Use Number, Size, and Speed for characterizing the snow. Play with lighting.
	**Snow_Windy** Use Number, Size, and Speed for characterizing the snow. Try with wind's Magnitude and axis animation to mimic the wind.
	**Surfing_Clouds** Flying into the clouds. Use gravity's Rotation_X expression to stay above clouds, or dive into them.
	**Tornado** Use Magnitude in the size manipulator to scale the tornado. Use Magnitude and Power in the twist manipulator to alter the shape of the tornado.
	**Tornado_ZatZero** Same as Tornado, but with emitter Z=0.

Liquids

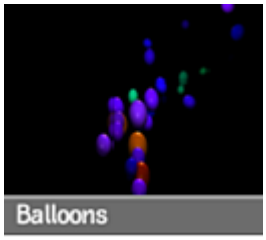
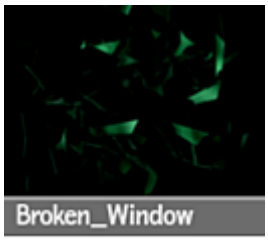
Use these presets to create fluid effects, such as bubbles.

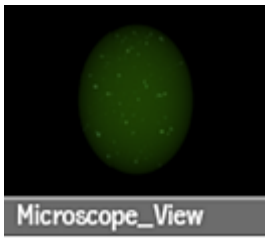
Proxy	Name/Description/Comments
	**Bubbles_Graphical** Use Number, Size, and Speed settings to characterize the bubbles. Process with Motion Blur enabled for better rendering.
	**Bubbles_Rising** Bubbles coming toward camera. Use Number, Size, and Speed settings to characterize the bubbles. Use Mass_V to randomize the bubble motion.
	**Inside_Water_Rays** Sun rays under water. Use water_generator for the water and rays_generator for the rays coming from the sun above the surface.
	**Sea** Water surface. Try the Spin_V settings to animate the sea. Customize by adding more lights to the scene.
	**Water_Bubbles** Use Number, Size, and Speed to characterize the bubbles. Try the Spin & Spin_V settings to control the flow.

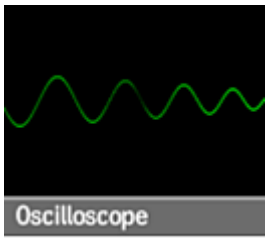
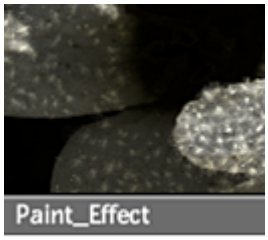
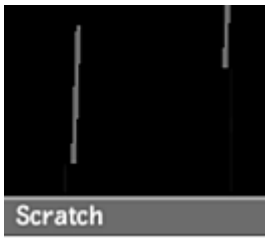
Proxy	Name/Description/Comments
	**Waterfall** Use Number, Size, and Speed to characterize the water. Try with size and gravity manipulators.

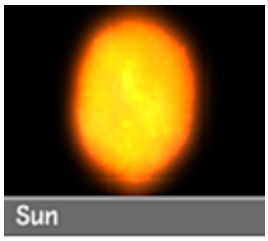
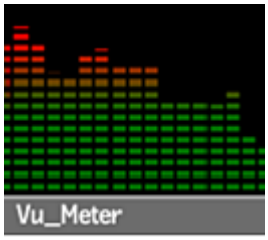
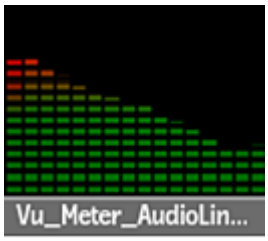
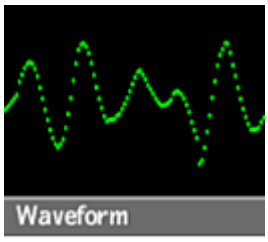
Miscellaneous

Use the miscellaneous presets to add dirt or scratches to your scene, as well as many other effects.

Proxy	Name/Description/Comments
	**Balloons** Animated. Use Number, Size, and Speed settings to characterize the balloons. Try the Spin & Spin_V settings to control the flow.
	**Blue_Thing** Moving glow following a path. Use Number, Lifetime, and Size settings to characterize the glow. Try with Rotation and Spin for motion. Change the path.
	**Broken_Window** Pieces from a broken window flying towards the camera.

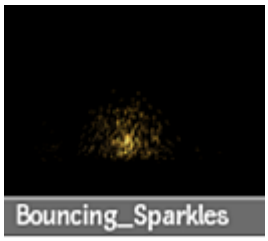
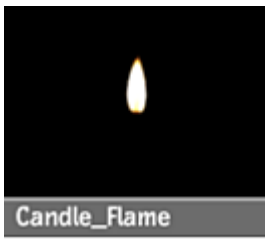
Proxy	Name/Description/Comments
	**Dirt** Use Number and Size settings. Try with variance.
	**Dust** Modify dust size using PD_ax Scale and generator Number settings.
	**Dust_Ray** Dust revealed by rays of light.
	**Hair** Use Number and Size settings. Try with variance. hair_generator4 is linked to hair_generator1, and hair_generator3 is linked to hair_generator2.
	**Microscope_View** Modify lights. Try neuronal_tex_ax settings.

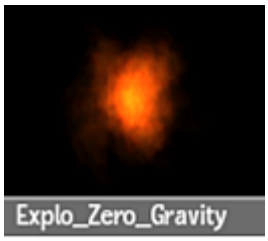
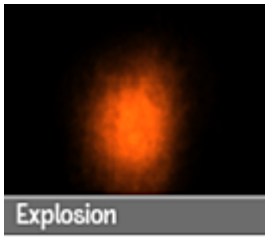
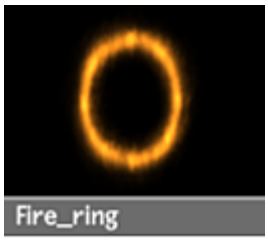
Proxy	Name/Description/Comments
	**Oscilloscope** Change the colour by using beam_generator's Diffuse setting. Use axis to control the beam's amplitude, frequency, and phase.
	**Paint_Effect** Use Number, Lifetime, and Size settings to customize the paint effect. Try changing the texture media (white square, soft cloud, etc.).
	**Scratch** Use axis to control the scratch's position, the amplitude of the horizontal motion of the scratch, and the frequency of the scratch. Hide some generators for fewer scratches, or duplicate generators if you want more scratches.
	**Shotgun** Slow-motion shotgun effect.
	**Simple_Sun** Processes quicker than Sun.

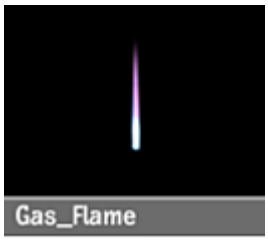
Proxy	Name/Description/Comments
	**Sun** Astronomical view of the sun. Slower to process than Simple_Sun.
	**Vu_Meter** Animated. Use beamgen Speed, PD_axis Scale_Z, and beam_emitter positioning to modify the number and width of stripes.
	**Vu_Meter_AudioLink** Similar to Vu_Meter, but you can link an audio clip to produce the animation.
	**Waveform** Animated audio signal monitor effect. Use axis to control the beam.

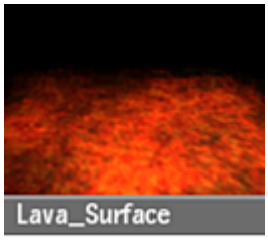
Pyrotechnic

Use the pyrotechnic presets to create fire and flame effects.

Proxy	Name/Description/Comments
	**Birthday_Sparkles** Customize the sparkles with Number, Lifetime, and Speed.
	**Bouncing_Sparkles** Sparkles bouncing on a plane. Use white_ax settings to place the bouncer plane in the scene.
	**Camp_Fire** Adapt the source of fire with emitter_ax. Characterize the flame with Number, Lifetime, and Size.
	**Camp_Fire_Smoke** Adapt the source of fire with emitter_ax. Characterize the flame with Number, Lifetime, and Size. Adjust the manipulators.
	**Candle_Flame** Characterize the flame with Number, Lifetime, and Size.

Proxy	Name/Description/Comments
	**Explo_Zero_Gravity** Explosion in a zero gravity environment.
	**Explosion** Exploding towards the camera.
	**Fire_Ring** Characterize the fire with Number, Lifetime, and Size. Adjust the camera.
	**Fire_Turbulence** Fire agitated by the wind.
	**Fireworks** Animated.

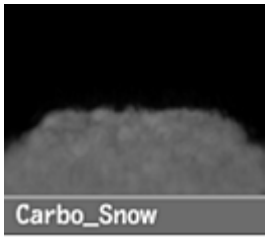
Proxy	Name/Description/Comments
	**Fireworks_BigBlue** Animated.
	**Fireworks_BigRed** Animated.
	**Fireworks_Cartoon** Animated.Process with Motion Blur enabled.
	**Fireworks_on_Path** Adjust the path.
	**Gas_Flame** Characterize the flame with Number, Lifetime, and Size.

Proxy	Name/Description/Comments
	**Lava** Lava flowing from the top of a volcano. Change the flow using the path manipulator.
	**Lava_Surface** Speed up or slow down the lava using the transform manipulator position settings.
	**Match_Flame** Characterize the flame with Number, Lifetime, and Size.
	**Meteor** Meteor burning in space. Adjust the camera.
	**Moving_Sparkles** Animate sparkles on a path. Change the path.

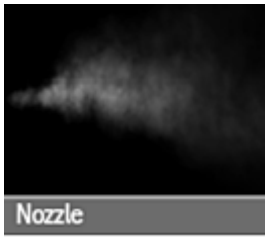
Proxy	Name/Description/Comments
	**Rocket_Propulsion** Rocket engine output.
	**Text_Plasma** Customize with Number, Lifetime, and Speed.

Smoke

These presets allow you to create various smoke effects.

Proxy	Name/Description/Comments
	**Carbo_Snow** Snow coming toward camera. Use Size to characterize the snow. Use the Magnitude setting of the gravity manipulator to set the snow's motion.
	**Dark_Smoke** Use on a light background. Use Number, Lifetime, Size, and Speed to characterize the smoke. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.

Proxy	Name/Description/Comments
	**Fire_Smoke** Characterize the smoke with Number, Lifetime, Size, and Speed settings. Use the Magnitude setting of the gravity manipulator to set the smoke's motion. Modify the fire colour using the fire_light_1 and fire_light_2 colour settings.
	**Fire_Smoke_ZatZero** Similar to Fire_Smoke, but with emitter Z=0.
	**Gun_Smoke_MB** Smoke produced by a shotgun. Process with Motion Blur enabled.
	**Heavy_Smoke** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.
	**Heavy_Smoke_ZatZero** Similar to Heavy_Smoke, but with emitter Z=0.

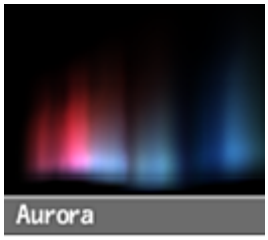
Proxy	Name/Description/Comments
	**Jet_Engine_Smoke** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.
	**Nozzle** Characterize the smoke with Number, Lifetime, Size, and Speed settings. Use vortex settings to set the smoke's motion. Change the bouncer. Not scalable.
	**Pyroclastic_Cloud** Change the path. Characterize the smoke with Number, Lifetime, Size, and Speed settings. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.
	**Smoke** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.
	**Smoke_Light** Characterize the smoke with Number, Lifetime, Size, and Speed settings.

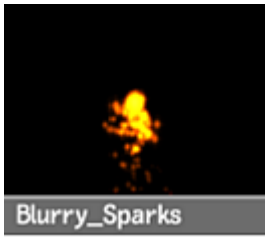
Proxy	Name/Description/Comments
	**Smoke_Lookat** Smoke pointing to a target. Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.
	**Smoke_Track** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.
	**Smoke_ZatZero** Similar to Smoke, but with emitter Z=0.
	**Train_Smoke** Smoke produced by a steam engine in motion. Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude setting of the train_speed manipulator to control the train's speed and its effect on the smoke.
	**Train_Smoke_ZatZero** Similar to Train_Smoke, but with emitter Z=0.

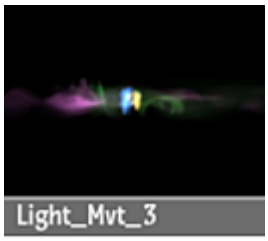
Proxy	Name/Description/Comments
	**Vapour** Vapour rising from the ground. Use Number, Lifetime, Size, and Speed settings to characterize the vapour. Try the Rotation_V and Spin_V settings to control the vapour animation
	**Wheel_Burning_MB** Animated. Process with Motion Blur enabled.

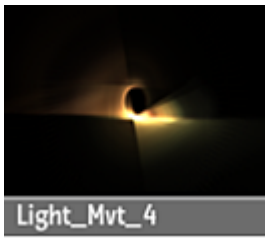
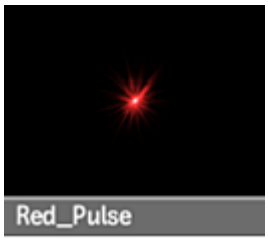
Space

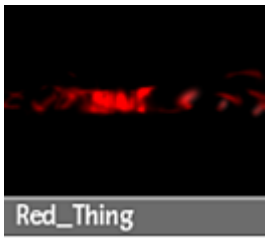
Use these presets to create astronomy effects.

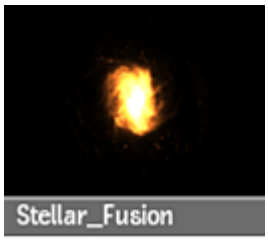
Proxy	Name/Description/Comments
	**Aurora** Aurora borealis effect. Use Number and Size to characterize the aurora. Use Lifetime to speed up or slow down the animation. Change colours by modifying the left_light & right_light settings.
	**Blue_Star** Slow moving star. Use Number, Lifetime, Size, and TrailSize settings.

Proxy	Name/Description/Comments
	**Blue_Star_2** Slow moving star. Similar to Blue_Star, but with a bigger hotspot in the middle. Use Number, Lifetime, Size, and TrailSize settings.
	**Blurry_Sparks** Use Number, Lifetime, Size, and Speed settings to set how the sparks are emitted. Use the transparency, gravity, and size manipulators to specify the behaviour of the falling sparks.
	**Cosmic_Light** Try with the PD_ax settings.
	**Cosmic_Marguerite_Blue** Pulsating blue light. Try with galaxy_tex_ax settings (posX=0, posY=0, scaleX=14, scaleY=125).
	**Cosmic_Marguerite_Green** Pulsating and rotating green light. Orbit the camera.

Proxy	Name/Description/Comments
	**Eye_Strike** Moving glows. Try with the circ_gradient1_ax and circ_gradient2_ax settings.
	**Glowing_Star** Try with the circgrad_tex_ax settings. Orbit the camera.
	**Light_Mvt_1** Try with the PD_ax settings. Process with Motion Blur enabled.
	**Light_Mvt_2** Orbit the camera.
	**Light_Mvt_3** Orbit the camera (Orbit 90°).

Proxy	Name/Description/Comments
	**Light_Mvt_4** Change the Rotation and Spin settings.
	**Light_Speed** Use Number, Speed, and TrailSize settings. Use the speed manipulator's power to set the Z acceleration.
	**Moving_Star_Field** Flying into space. Use Number and Speed settings.
	**Pulsar** Pulsating glow. Use Number, Lifetime, Size, Speed, and TrailSize settings.
	**Red_Pulse** Pulsating red star. Customize the star with Number and Speed settings. Set the star's intensity with transparency magnitude.

Proxy	Name/Description/Comments
	**Red_Thing** Glowing red light. Change the colour with the Light Colour settings.
	**Space_Dust** Reset vortex (Rot_Z) and try with vortex (Rot_Y). Try vortex (Rot_Y=-90). Change the camera settings.
	**Space_Odyssey** Change the space_tex_ax settings (animated) and emitter_ax1 Rot_X setting.
	**Star** Glowing animated star. Use Number, Size, and Speed settings to characterize the star. Change colour using the light's Colour settings.
	**Star_Field** Flickering stars in the sky. Modify Number at frame 0 to control the amount of stars in the sky.

Proxy	Name/Description/Comments
	**Star_Glow** Glowing aggregate of stars. Customize the generator with the Rotation and Spin settings.
	**Stellar** Orbit the camera and change the stellar_tex_ax settings.
	**Stellar_Fusion** Rotating celestial bodies. Orbit the camera.

Generating Particles

You can create a particle stream that behaves in different ways depending on the type of generator you use. When creating a particle stream, you can use three different generators: light source, surface, and 3D object.

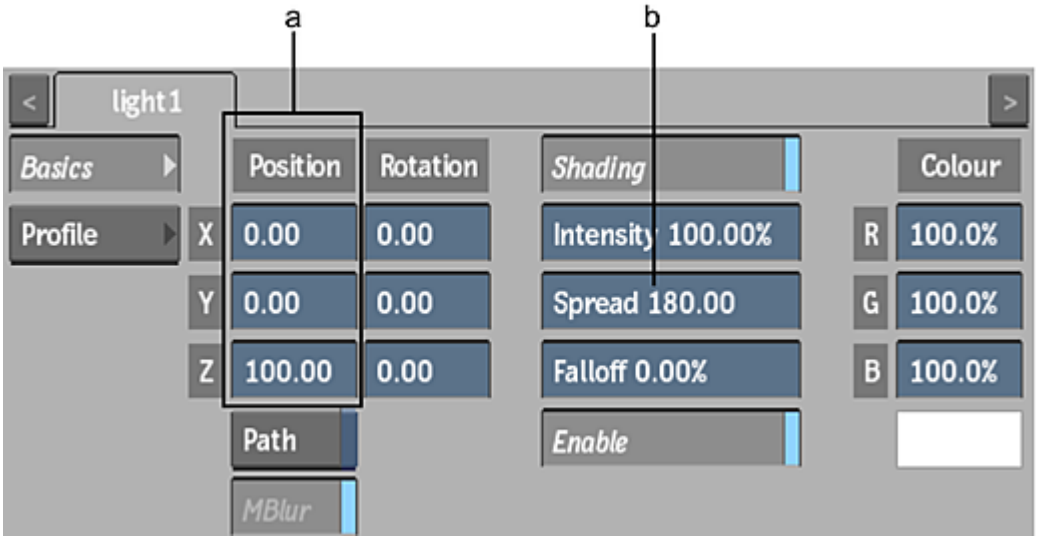
Use as a Generator:	To:
light source	Create a stream of particles starting from a single point.
surface	Generate particles based on the area, shape, and colour of a surface.
3D object	Generate particles from the centre of each polygon.

Using a Light Source

When a particle generator is attached to a light source, it uses the position and rotation of the light source to move and rotate the particle generator.

To add a particle stream using a light source:

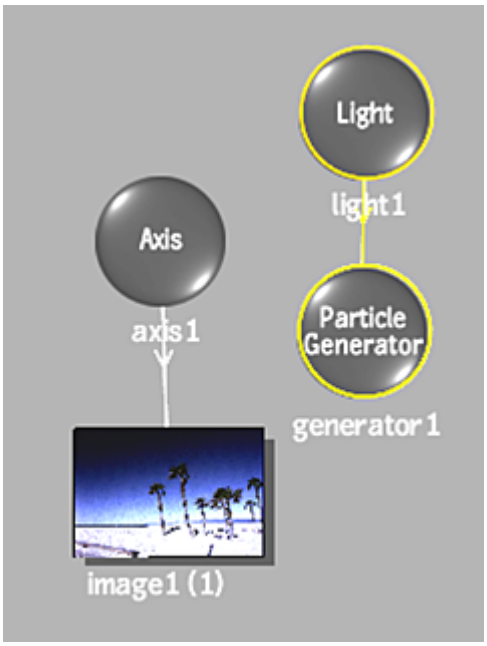
1. Add a Light node to your scene.
2. Double-click the new light in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).
3. Adjust the spread and position of the light source in the Light menu. Use the Spread field to set the spread of the particle stream. The Intensity, Falloff, and Colour fields are not used by the particle generator.



(a) Position fields (b) Spread field

4. With the light node selected in the schematic, do one of the following:
   - Drag the particle generator node from the node bin and place it in the schematic.
   - Double-click the particle generator node. You do not need to be in Schematic view to add a node in this manner.

   A particle generator object is added and automatically connected to the selected light source.



5 Set the particle generator's properties. See Customizing the Particle Stream (page 951).

6 If you do not want the light source to illuminate the scene, deactivate the light by clicking the Enable button in the Light menu. When Enable is deactivated, the light source is used as a particle generator only.

With a light source, you can animate the position of the particle stream by moving the light, changing the spread, and rotating the light. Because the generator is a light source, you can animate the position of the particle generator using a motion path.

Using a Surface

Use a surface as a particle generator to generate particles based on the area, shape, and colour of a surface. You can use a surface's matte to restrict where particles are generated. When the matte is off, the particle stream is generated from the entire surface. When the matte is on, particles are only generated where the matte is white or grey.

You can animate the shape of the surface using its tangent handles. The starting area of the particle stream matches the surface's shape animation. See Changing the Shape of a Surface (page 633).

To add a particle stream using a surface:

1 Select or add a surface that you want to use as a particle generator.

2 Adjust the surface's shape, position, and other attributes. See Modifying Surfaces (page 624).

3 With the surface node selected in the schematic, do one of the following:

- Drag the particle generator node from the node bin and place it in the schematic.
- Double-click the particle generator node. You do not need to be in Schematic view to add a node in this manner.

A particle generator object is added and automatically connected to the selected surface.

4 Set the particle generator's properties. See Customizing the Particle Stream (page 951).

5 If you do not want the surface to appear in the scene, you can hide the surface in Schematic view. A particle stream is generated from the hidden surface.

6 To make the generated particles change as the surface's clip changes, enable Indirect in the Particle Generator menu.

Using a 3D Object

You can use a 3D object as a particle generator. Particles are generated from the centre of each polygon.

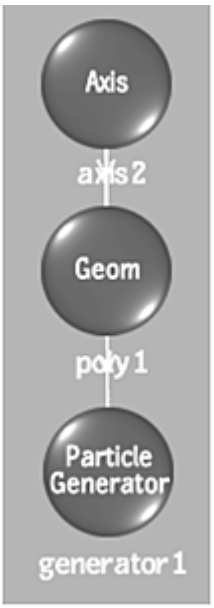
To add a particle stream using a 3D object:

1 Select or add the 3D object that you want to use as a particle generator.

2 With the 3D object node selected in the schematic, do one of the following:

- Drag the particle generator node from the node bin and place it in the schematic.
- Double-click the particle generator node. You do not need to be in Schematic view to add a node in this manner.

A particle generator object is added and automatically connected to the selected 3D object.



3 Set the particle generator's properties. See Customizing the Particle Stream (page 951).

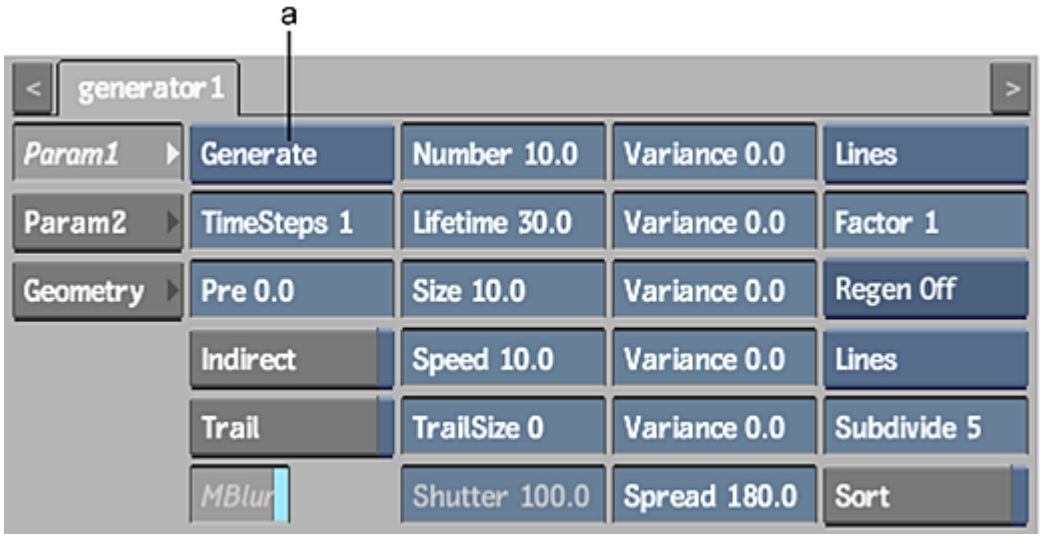
4 If you do not want the 3D object to appear in the scene, hide the model in Schematic view. A particle stream is generated from the hidden model.

Customizing the Particle Stream

Whether you use a light source, 3D object, or surface, the particle stream uses the same properties, allowing you to create custom particles and apply textures to particles. To access these properties, double-click the particle generator in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 571).

Setting Stream Properties

The following properties in the Particle Generator Param1 menu affect the particle stream.



(a) Stream Type box

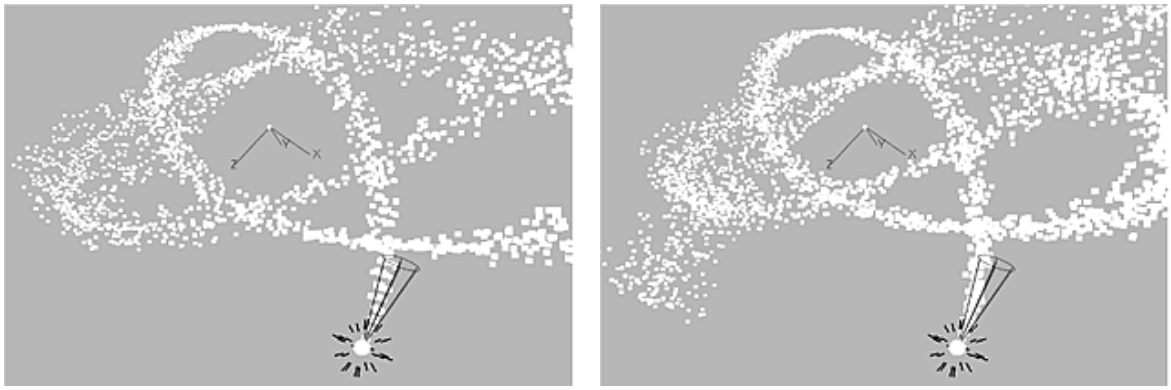
Stream Type box Select whether the particle stream will Generate or Explode.

TimeSteps field Displays the time simulation of the Generator per frame. By default, Timestep is set to 1, which means that time simulation is once per frame. Editable.

When Timestep is set to 2, the position of each particle is calculated once between frames. Any change in speed or position caused by manipulators or bouncers is applied between frames. When Timestep is set to 3, the position of each particle is calculated twice between frames. The greater the timestep value, the longer it takes to process.

Typically, you use Timestep with a particle manipulator or bouncer on a particle stream. In the following example, the same particle settings and manipulator (AccPoint) are used. Only the timestep values are different. The particle stream on the left uses a timestep of 1 while the stream on the right uses a timestep of 6.

In the following figure, notice the particle stream on the right is more accurate.



Pre field Displays the start point of the particle stream. With a value of 0.0, the particle stream starts creating particles at frame 1. With a value of 100, the clip begins as if the particle stream has been generating for 99 frames. Editable.

Indirect button Enable to allow the part of the surface used by the particle to change with each frame. For example, if the media applied to the surface changes from red to blue over 10 frames, each particle changes from red to blue as the surface changes. Disable to lock the part of the surface used by the particle to the frame where the particle was generated. For example, if the media applied to the surface changes from red to blue over 10 frames and a particle is generated at the first frame, when the surface was red, the particle stays red.

Trail button Enable to draw each particle with a trail. Set the length of the trail in the TrailSize field.

Motion Blur button Enable to use a motion blur effect for particles emitted from the selected Generator (can only be used if the global Motion Blur is enabled in the Action Setup menu).

Shutter field Displays the length of the tail for lines, cones, and quads. Shutter is expressed as a percentage. At 100%, the tail's length is the same as the head (Size field). At 400%, the tail's length is four times the size of the head. Editable.

Shutter at 100%

Shutter at 400%

Spread field Displays the spread for the second particle generator when two particle generators are parented. When the first particle stream dies, the second particle stream begins. Editable.

See Combining Particle Streams (page 985). This also sets the spread of a particle stream that bounces off surfaces. See Bouncing Particles (page 982).

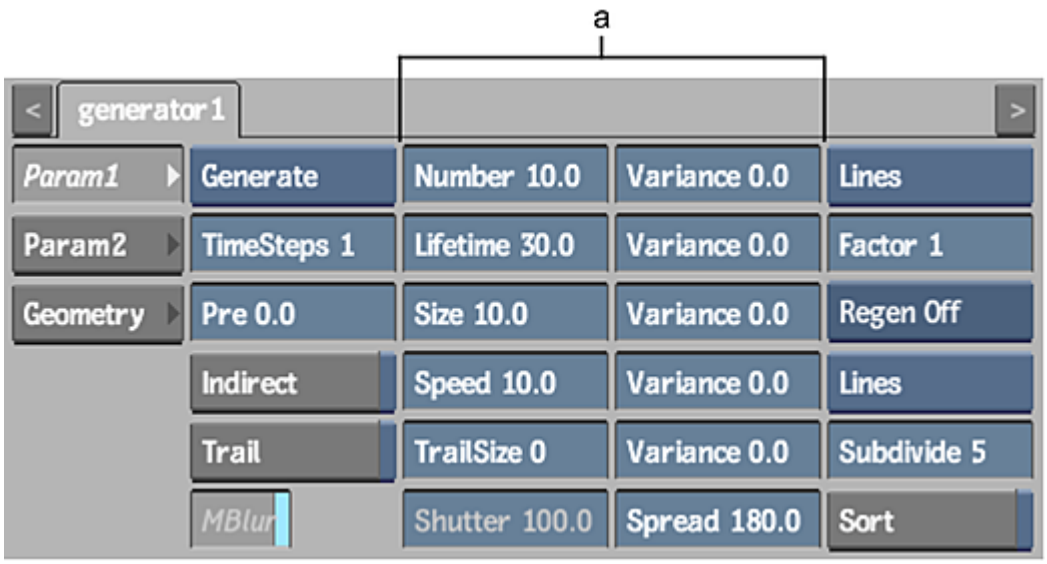
Setting Particle Properties

In the middle of the Generator Param1 menu, a table lists the different particle generation properties.

Each parameter has a variance value used to specify how much the parameter varies each time particles are generated. Exactly what gets varied depends on the parameter.

For example, if you set the Number parameter to 10 and its Variance parameter to 5, the number of particles generated at each frame is randomly selected between 5 and 15.

The unit of measurement used by each Variance field matches the parameter it is varying. For example, if the Number parameter is expressed in particles per frame, so is its Variance field. The Speed parameter is in pixels per frame and so is its Variance field.



(a) Particle generation properties

Number field Displays the number of particles generated per frame. Editable.

If you specify a number less than 1 (from 0.1 to 0.9), particles are generated randomly. For example, if you set the number to 0.1, a particle is generated every 10 frames. The exact frame within those 10 frames when the particle is generated is randomly selected. If you set the number of particles to 0.3, a particle is randomly generated approximately every three frames.

Number Variance field Displays how much the number varies each time particles are generated. Editable.

Lifetime field Displays the number of frames each particle lasts. For example, if Lifetime is 20 frames, a particle generated at frame 15 disappears at frame 35. Editable.

Lifetime Variance field Displays how much the lifetime varies each time particles are generated. Editable.

Size field Displays the size of each particle, in pixels. This parameter only affects cones, spheres, quads, squares, and objects. Editable.

Size Variance field Displays how much the size varies each time particles are generated. Editable.

Speed field Displays the speed of each particle, in pixels per frame. Editable.

Speed Variance field Displays how much the speed varies each time particles are generated. Editable.

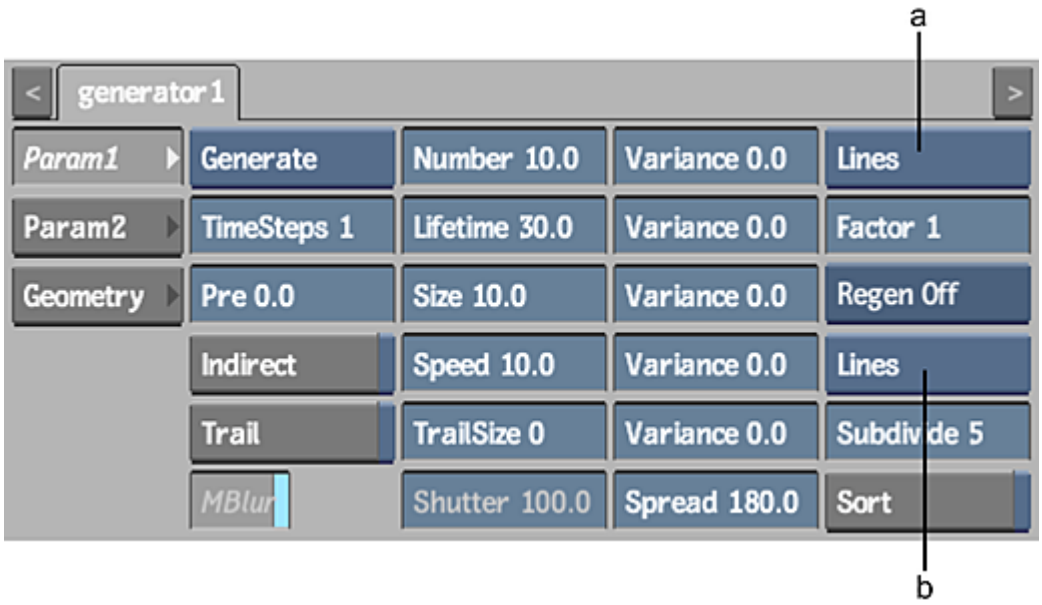
TrailSize field Displays the length of each particle's trail, in particles. A particle trail only appears when the Trail button is enabled. Editable.

TrailSize Variance field Displays how much the trail size varies each time particles are generated. Editable.

Setting Particle Rendering Properties

The following Generator properties affect the rendering and previewing of the particle stream.

NOTE Rendering particles is faster if Z-Buffer On is selected in the Rendering Section of the Action Setup menu.



(a) Interactive Type box (b) Render Type box

Interactive Type box Select the type of particle to display in the scene. For example, as you move between frames, the particle stream is redrawn. If you are using a custom 3D object for each particle, redrawing the scene may take a while. By changing the interactive type to a point or line, your interaction with Action is much quicker.

Factor field Displays the number of particles shown in the scene while working in Action (use to speed up work - does not affect the rendered result). Editable.

For example, if you generate 1000 particles with manipulators and bouncers, interacting with Action may become slow because calculations have to be made for each particle. If you specify a factor of 10, the number of particles is reduced to 10%, or 100 particles. Using Action is then faster, yet when you click Preview or Process, 1000 particles are created.

Regenerate box Select how you want to update the scene when you change particle generator parameters. Regen All updates the particle and the object generating the particle, but may cause performance slowdowns. Regen Anim only updates the particle in the scene.

Select:	**To:**
Regen All	Automatically update the particle and the object generating the particle (such as a surface) when you change any particle generator parameter. This option allows you to see the particle with all settings, such as position and colour correction, but may cause performance slowdowns, since the generating object and particle stream are recalculated each time a parameter is changed.
Regen Anim	Automatically update the particle in the scene when you change any particle generator parameter. The object generating the particle is not automatically updated in this case, so you may not see your exact desired results. If Regen All is causing slowdowns, you can try this option to get a faster update.
Regen Off	Not update the scene automatically, unless the frame is changed.

Render Type box Select the type of particle used when rendering the scene.

Select:	To:
Node	Use your own 3D object, text, or a surface for each particle. See Using Custom Particles (page 958).
Cones	Use a cone for each particle. You can adjust the size of the cone's tail.
Spheres	Use a 3D object of a sphere for each particle.
Polygons	Explode 3D objects and surfaces. See Exploding Objects and Surfaces (page 961).
Quads	Use a square with a tail for each particle. You can adjust the size of the square's tail.
Squares	Use a square for each particle.
Lines	Use a line for each particle.
Points	Use a single pixel for each particle. Points are not affected by size or trail.

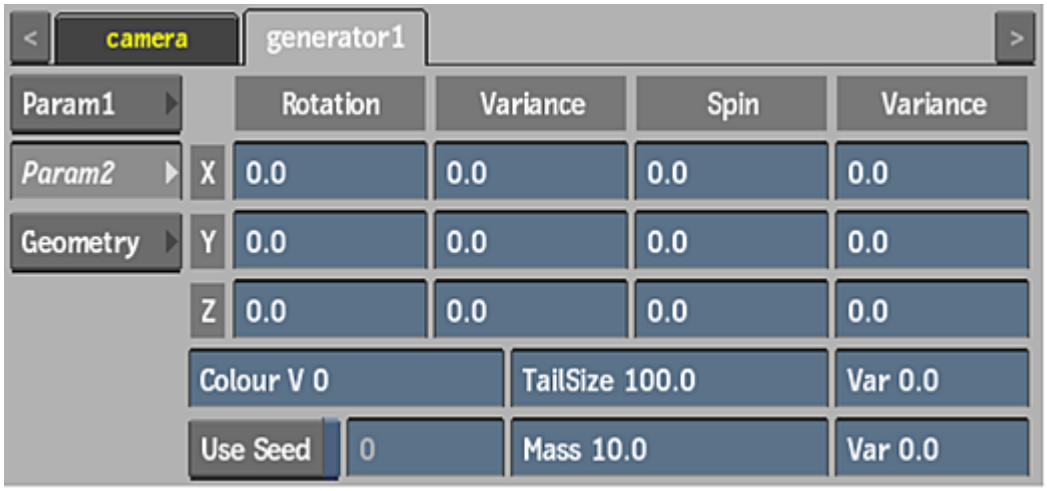
Subdivide field Displays the number of polygons used to render cones and spheres. The larger the number of subdivisions, the smoother the cones or spheres are drawn, but rendering time is increased. Editable.

If you are using an imported 3D object or text for a particle, use the Subdivide field and button in the Geometry menu to set the number of polygons. The Geometry menu's Subdivide field works differently than the Subdivide field in the Generator menu. See Subdividing a Model (page 717).

Sort button Enable to determine how the particles are drawn, according to the Z-order. If a Particle Generator node is selected, the state of the Sort button is the same in the Param1 and Geometry menus.

Setting Additional Generator Properties

Additional particle generator controls can be found in the Param2 tab.



X Rotation field Displays the X orientation for particles that have just been born (spawned). Editable.

Y Rotation field Displays the Y orientation for particles that have just been born (spawned). Editable.

Z Rotation field Displays the Z orientation for particles that have just been born (spawned). Editable.

X Rotation Variance field Displays the variation of the X rotation of a particle once spawned from the particle generator. Editable.

Y Rotation Variance field Displays the variation of the Y rotation of a particle once spawned from the particle generator. Editable.

Z Rotation Variance field Displays the variation of the Z rotation of a particle once spawned from the particle generator. Editable.

X Spin field Displays a value to act as a speed setting to an X rotation animated over time. Editable.

Y Spin field Displays a value to act as a speed setting to a Y rotation animated over time. Editable.

Z Spin field Displays a value to act as a speed setting to a Z rotation animated over time. Editable.

X Spin Variance field Displays a variation to the X Spin channel once particles are spawned from the particle generator. Editable.

Y Spin Variance field Displays a variation to the Y Spin channel once particles are spawned from the particle generator. Editable.

Z Spin Variance field Displays a variation to the Z Spin channel once particles are spawned from the particle generator. Editable.

Colour Var field Displays the variation of the colour of each particle based on the colour set in the Diffuse colour bar (Particle Geometry menu). Editable.

Use Seed button Enable to override the particle's seed (useful if you want to change the position of each particle in the particle stream).

Seed field Displays the random seed value for the particle stream properties. Editable.

TailSize field Displays the width of a particle's tail, expressed as a percentage. Works only with quad and cone particle types. Editable.

TailSize Variance field Displays the variation of the TailSize value, as a percentage. Editable.

Mass field Displays the mass of each particle (used with Damping when manipulating particles). Editable.

See Setting Manipulator Properties (page 964).

Mass Variance field Displays the variation of the mass, as a percentage. Editable.

Setting Geometry Properties

You can use the Particle Geometry menu controls to add colour, specular highlight, shine, and transparency to all particles. To access this menu, click the Geometry tab in the Generator menu.

Use the available particle geometry controls to affect the particle stream, whether it is a surface or a geometry. The controls behave in the same manner as when using with the Geometry menu for a 3D model. See 3D Geometry Menu Settings (page 711).

Setting Channel Editor Properties

The Channel Editor includes the particle properties included in the Particle menus for which you can set values and create particle animation effects. You set these properties in conjunction with those located in the Particle Generator, Particle Manipulator, and Particle Bouncer menus.

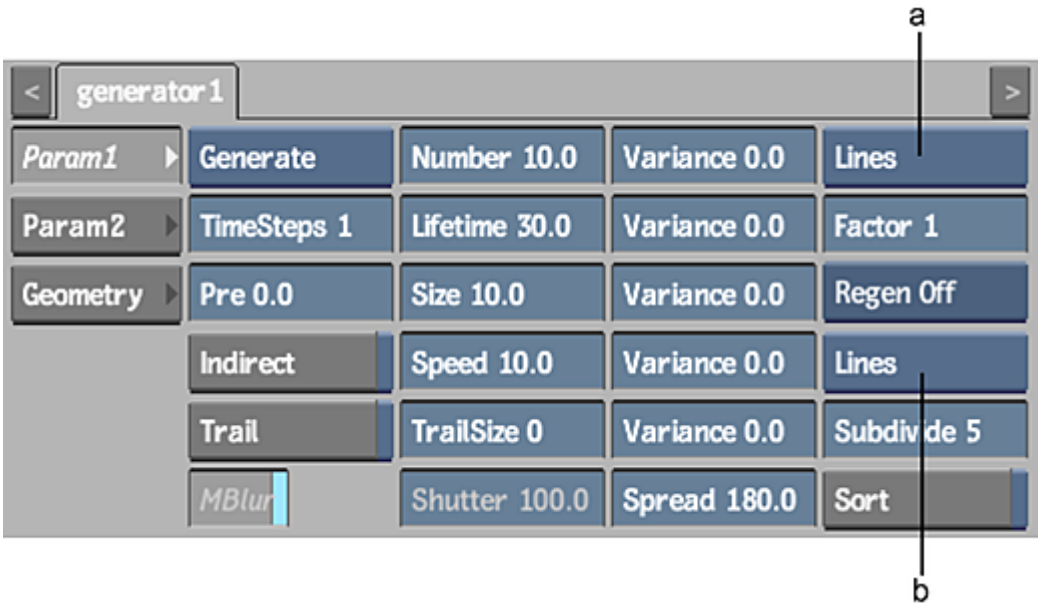
You must first add a particle generator, a particle manipulator, or a particle bouncer to the scene to view and set values for their corresponding properties in the channel hierarchy of the Channel Editor.

Using Custom Particles

You can create particle streams using your own 3D objects or text. You can also use the shape of any surface for each particle. For example, create custom particles using a light source, 3D object, or surface as a particle generator. See Generating Particles (page 948).

To use a 3D object or text as a custom particle:

1. From the node bin, add a particle generator to your scene.
2. In the schematic, double-click the Particle Generator node to access the Generator menu.



(a) Interactive Type box (b) Render Type box

3. From the Render Type box, select Node as the particle type and change the Interactive Type box to Node to view the 3D object or text as you make changes to your particle stream.
4. Import the 3D object or create the text. See Importing 3D Models (page 699) or Action 3D Text (page 723).

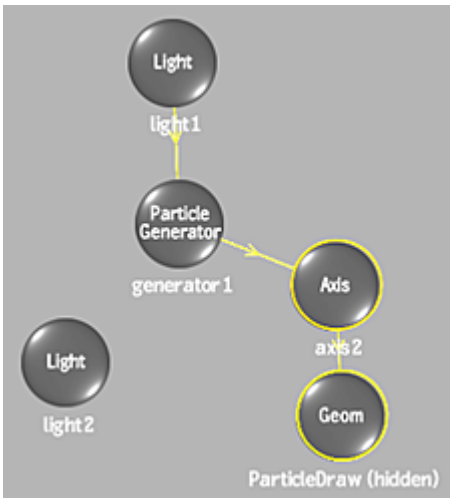
   TIP The 3D object or text used for the custom particle can also include a deformation mesh. See Deforming Models and Surfaces (page 718).

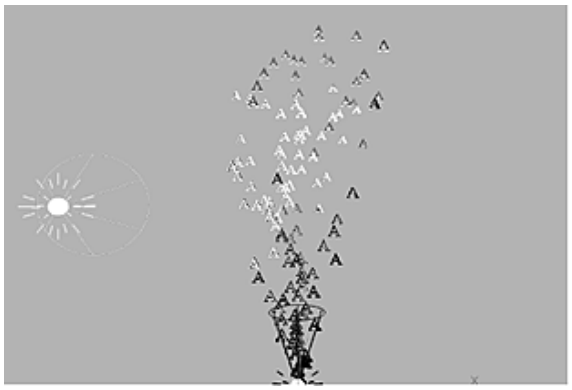
5. In Schematic view, rename your 3D object or text “ParticleDraw”. This name is case sensitive so make sure it is typed exactly as shown.
6. Parent the particle generator to the 3D object or text.

   If you want to add transformations, such as rotating all the particles, parent the particle generator to the axis of the 3D object or text as shown in the next step.

7 Hide the 3D object or text.

The particle generator uses the 3D object or text named ParticleDraw. The following figure shows the schematic and the result of using the letter A as a custom particle and a light source as the particle generator.





To use a surface as a custom particle:

1 Add a particle generator.

You can use a light source, 3D object, or surface as a particle generator. See Generating Particles (page 948).

2 Select Node as the particle type in the Render Type box. To view the surface as you make changes to your particle stream, change the Interactive Type box to Node as well. See Setting Particle Rendering Properties (page 954).

3 Add a surface to the scene. It can be a flat image, a bilinear, a perspective, or an extended bicubic surface. See Adding Surfaces (page 623).

NOTE You do not have to apply media to the surface because the particle generator only uses the surface's shape and not its media. To have an image appear on each particle, you have to apply a texture to the particle generator. See Applying Textures to Particles (page 960).

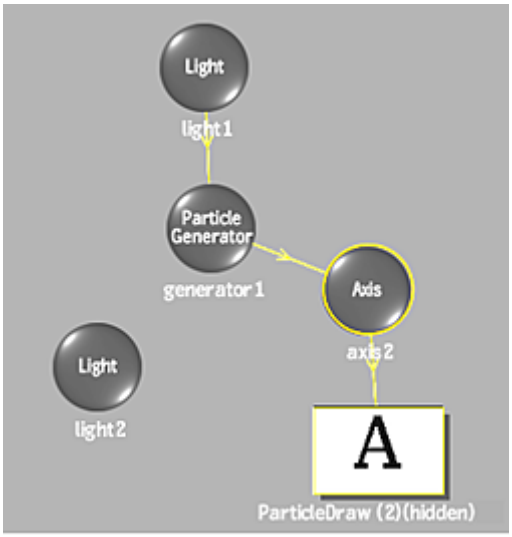
4 Rename your surface “ParticleDraw”. This name is case sensitive so make sure it is typed exactly as shown.

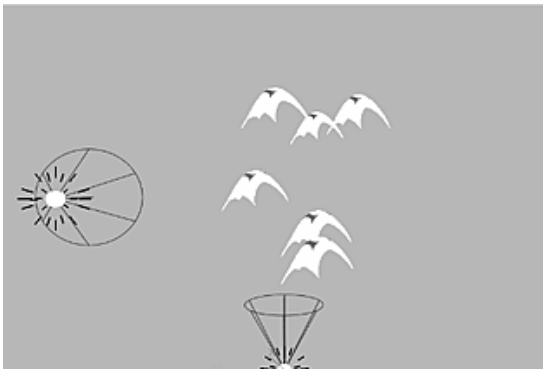
5 Parent the particle generator to the surface.

If you want to add transformations, such as rotating all the particles, parent the particle generator to the axis of the surface as shown in the next step.

6 Hide the surface.

The following figure shows the schematic and the result of using an extended bicubic surface as a custom particle and a light source as the particle generator. The shape of the surface is animated and its axis is used to rotate the particles.





NOTE You can animate the shape of a bilinear, perspective, or extended bicubic surface, and have the particles reflect the animation.

Applying Textures to Particles

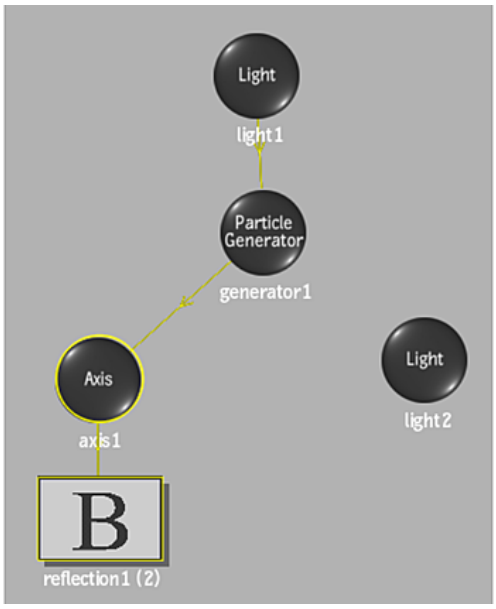
You can apply textures to particles using the same method as applying textures to 3D objects and text. You can also set its texture mapping using the Texture menu.

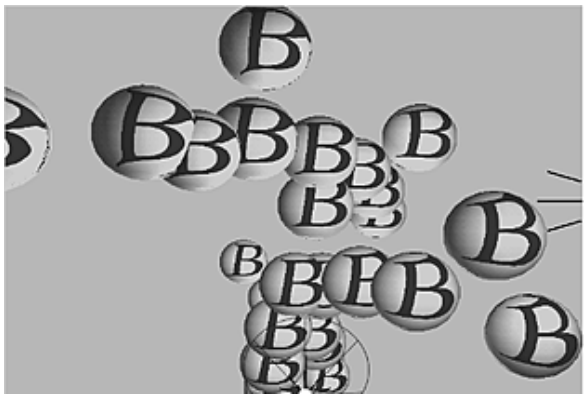
To apply a texture to a particle stream of spheres:

1. Create a particle generator.
2. From the Render Type box, select Spheres as the particle type. To view the 3D object or text as you make changes to your particle stream, change the Interactive Type box to Spheres as well. See Setting Particle Rendering Properties (page 954).
3. In the schematic, select the Particle Generator.
4. From the Node bin, double-click the a texture map node.

   A texture is added using the currently selected media and is parented by the particle generator.
5. Apply the appropriate media to the texture object and set texture menu settings.

For example, the following shows the schematic and the result of a particle stream using spheres and a reflection map, making each particle resemble a glass ball.





Exploding Objects and Surfaces

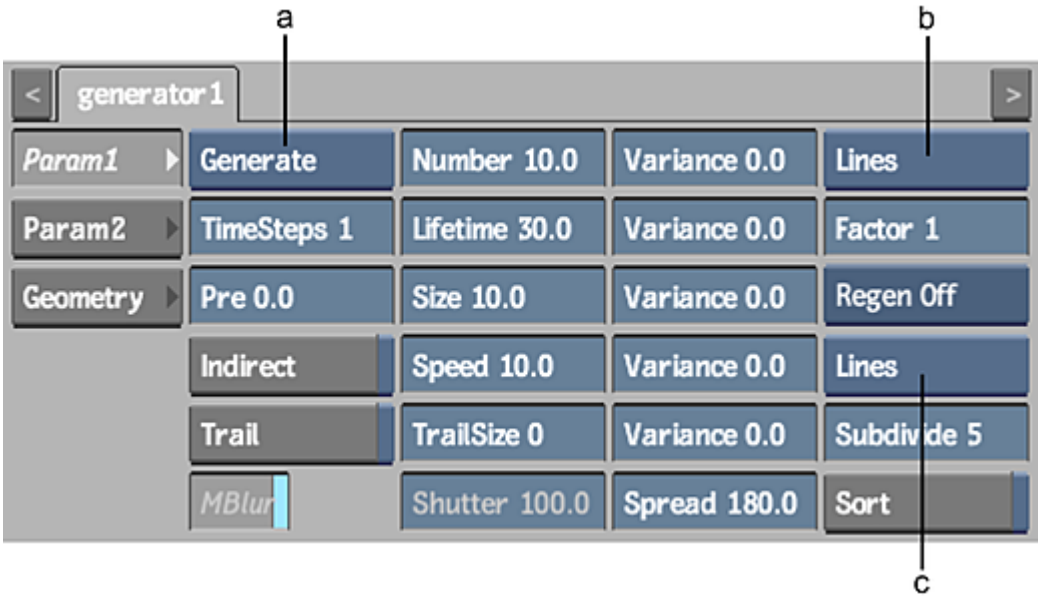
Whether you use a 3D object or a surface, exploding works much the same way. The shape of the 3D object or the surface is recreated by the particle generator, then the shape is exploded into polygons. For this to work correctly, the original 3D object or surface must be hidden.

Exploding Objects

The properties used to explode an object cannot be animated. When something explodes, it happens at a single point in time. This means that all the settings in the Particle Generator menu are used at once, at the beginning of the explosion.

To explode a 3D object:

1. Import a 3D object or create 3D text.
2. In the schematic, select the 3D object.
3. Add a particle generator. See Generating Particles (page 948).
4. From the Particle Generator menu, select Explode from the Stream Type box.



(a) Stream Type box (b) Interactive Type box (c) Render Type box

5 From the Render Type box, select Polygon as the particle type.

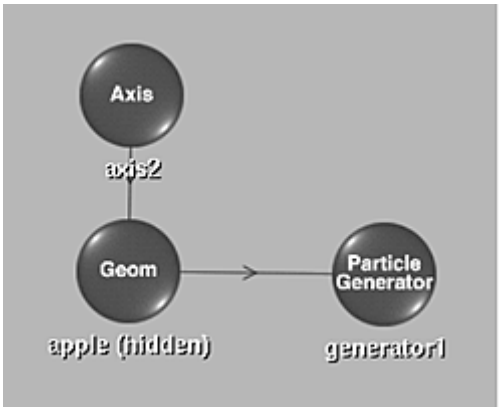
To view the explosion as you make changes to your particle stream, change the Interactive Type box to Polygon as well. See Setting Particle Rendering Properties (page 954).

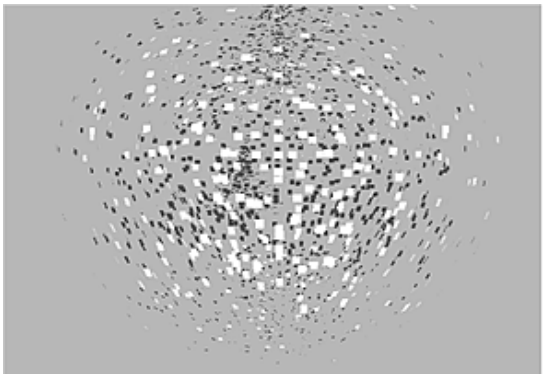
6 In the Number field, set the number of polygons to be removed from the 3D object with each pass. For example, if you specify 10 as the number of polygons per frame, then at each frame, 10 polygons are removed from the 3D object.

7 Set the other properties in the PartGen menu. See Customizing the Particle Stream (page 951).

8 Hide the 3D object or 3D text.

The following figure shows the schematic and the result of exploding a 3D object of an apple. The number of polygons is set to 200. The result shows frame 20.





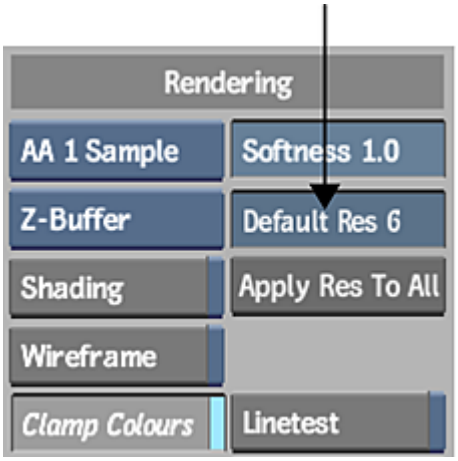
Exploding Surfaces

To make the media applied to the surface appear during the explosion, you must also apply a texture to the particle generator.

To explode a surface:

1. Select or add a surface and adjust its shape, position, rotation, and other attributes. See Adding Surfaces (page 623).
2. Select the surface in Schematic view and add a particle generator. See Generating Particles (page 948).

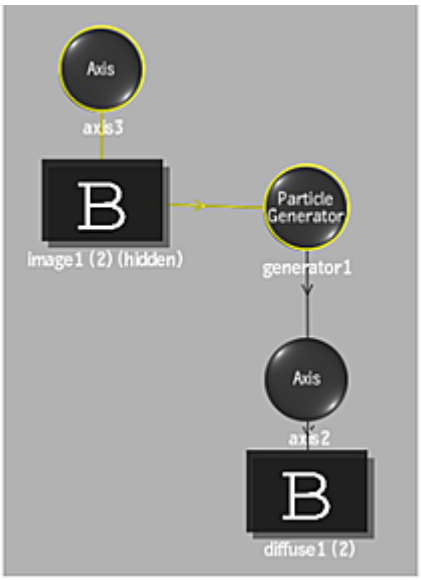
   A particle generator is automatically connected to the surface.
3. From the Particle Generator menu, select Explode as the Stream Type.
4. Select Polygon as the particle type from the Render Type box. To view the explosion as you make changes to your particle stream, change the Interactive Type box to Polygon as well. See Setting Particle Rendering Properties (page 954).
5. Set the other properties in the Particle Generator menu. See Customizing the Particle Stream (page 951).
6. In the Rendering section of the Action Setup menu, adjust the default resolution.

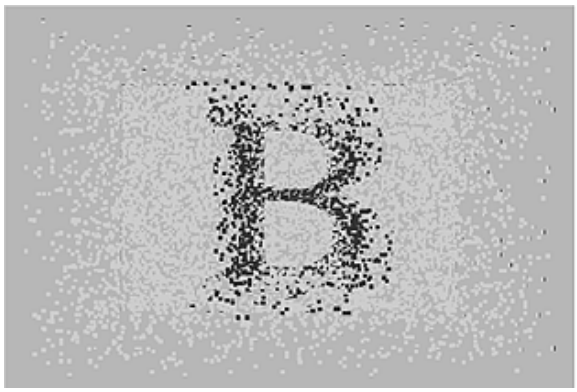


   Resolution is used to set the size of the polygons exploded from the surface. The higher the resolution, the larger the polygon.
7. Hide the surface.
8. Select the particle generator in Schematic view.
9. From the Node bin, double-click the Diffuse Map node.

   A diffuse object appears in the schematic and is parented by the particle generator. See Diffuse Mapping (page 800).
10. Double-click the diffuse object and, in the Diffuse menu, select Wrap from the Mapping box.
11. Apply the appropriate media to the Diffuse element and set any other Diffuse menu settings.

The following figure shows the schematic and the result of exploding a surface with diffuse mapping set to Wrap.





Manipulating Particles

You use manipulators to direct the particle stream and create a number of different effects, such as a swirling vortex, a swarm of objects orbiting a point, or a waterfall. A particle stream can have an unlimited number of manipulators. Manipulators only influence the particle stream that parents them.

When manipulating particles, you use the Particle Manipulator menu to set a manipulator's type, magnitude, power, and damping properties, as well as position, rotation, and scale.

Particle manipulators can be animated using motion paths. You can use the same operations on a particle manipulator that you use on an axis.

Creating a Particle Manipulator

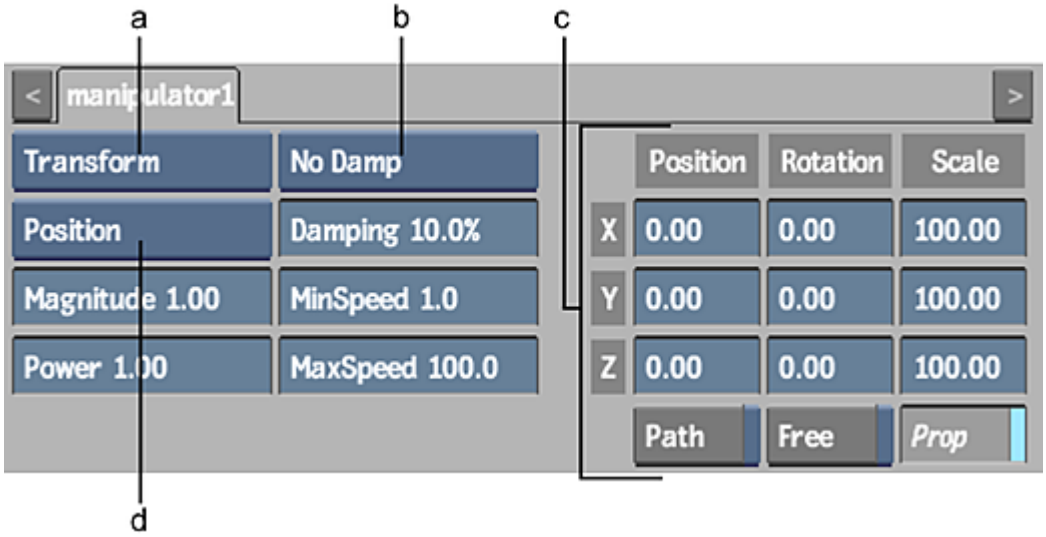
To create a particle manipulator:

1. In the schematic, select the particle generator that you want the manipulator to influence.
2. From the Node bin, double-click the Particle Animator node.

   A particle animator element is added to the scene and parented by the selected Particle Generator element.

Setting Manipulator Properties

You use the Particle Manipulator menu to select the type of manipulator, its speed and position, and the power of its falloff. To access the Particle Manipulator menu, double-click the particle animator node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).



(a) Manipulator Type box (b) Damping box (c) Axis controls (d) Influence box

The particle manipulator controls are described as follows.

Manipulator Type box Select the type of manipulator.

Select:	To:
Gravity	Simulate the effect of gravity. See Simulating Gravity (page 968).
Transform	Apply transformations from the Axis menu to the position or speed of each particle. See Applying Transformations (page 967).
Vortex	Mimic the effect of a vortex. See Creating a Vortex Effect (page 969).
Acceleration Point	Pull particles toward a point. See Using an Acceleration Point Manipulator (page 970).
Acceleration Line	Pull particles toward a line. See Using an Acceleration Line Manipulator (page 971).
Acceleration Plane	Pull particles toward a plane. See Using an Acceleration Plane Manipulator (page 972).
Path	Make the particles follow a path. See Forming a Particle Path (page 973).
Function	Enter a mathematical expression to use as a particle manipulator. See Using a Function as a Manipulator (page 974).

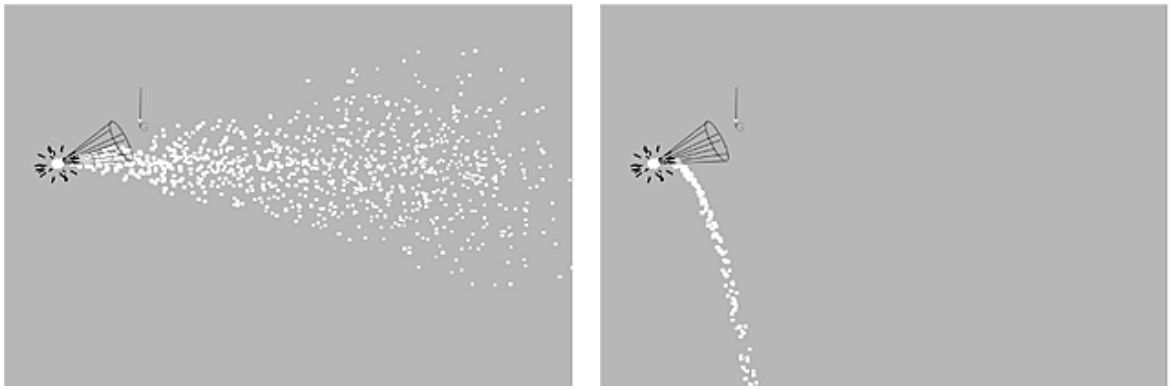
Influence box Select whether the manipulator influences each particle's position or speed.

Set the initial speed of each particle using the Speed field in the Surface Particle Generator menu.

By selecting Speed in the Influence box located on the Particle Manipulator menu, each particle's speed is changed with each pass by the selected manipulator.

When you influence by position, the position is applied only once. For example, the following figure uses a gravity manipulator to illustrate the difference between position and speed. The gravity manipulator on

the left uses the default gravity manipulator with a magnitude of 4 and with position as the influence. The gravity manipulator on the right uses exactly the same settings except the influence is set to speed.



Magnitude field Displays the unit of measurement specific to each manipulator. Editable.

Magnitude is used by each manipulator for everything from extra scale when using Transform, to a pixel per frame acceleration factor when using AccPoint, AccLine, or AccPlane.

Power field Displays the amount of falloff from the centre of a manipulator. Editable.

If you specify 0 as the power, there is no falloff; the manipulator's influence is universal. This means that a particle is affected no matter where it is located in the scene. A negative value makes particles that are farther away affected by the manipulator.

Damping box Select the type of friction to apply (damping affects the speed of particles by applying friction based on each particle's mass, size, or both).

Select:	**To:**
No Damp	Turn off damping. Particles have no friction.
Damp Mass	Activate damping based on mass. The greater a particle's mass, the slower its speed in relation to the manipulator. You set the mass using the Mass channel in the Channel Editor.
Damp Size	Activate damping based on particle size. The larger the particle, the slower it moves in relation to the manipulator. This corresponds with the Size field in the Surface Particle Generator menu.
Damp Both	Activate damping based on mass and particle size.

Damping Percentage field Displays the percent value of how much Damping affects the position or speed of the particles. Editable.

Minimum Speed field Displays the minimum speed range for particles affected by Damping. Editable.

Maximum Speed field Displays the maximum speed range for particles affected by Damping. Editable.

NOTE You can use the Min and Max Speed fields to force a minimum and maximum speed on particles without damping. To do this, select Damp Mass, Damp Size, or Damp Both. Change the Damping Percentage to 0% and change the Min and Max Speed fields accordingly. Use this feature to stop particles from being over accelerated. For example, when you use a vortex manipulator, the closer particles pass to the centre of the vortex, the more they are accelerated. Occasionally, a particle may be over accelerated and shot too far, too fast. By forcing a maximum speed, particles are not accelerated past a certain value.

Load button Not Shown; available when Function is selected as the Manipulator type. Loads a mathematical expressions to be used as a particle manipulator.

Save button Not Shown; available when Function is selected as the Manipulator type. Saves a mathematical expressions to be used as a particle manipulator.

Expression field Not Shown; available when Function is selected as the Manipulator type. Displays a mathematical expression to be used as a particle manipulator. Editable.

See Using a Function as a Manipulator (page 974).

Axis controls Use these controls to animate the position, rotation, and scale of the manipulator.

Control	Description
Position	Use the Position X, Y, and Z fields to displace each particle along the X, Y, and Z axes.
Rotation	Use the Rotation X, Y, and Z fields to rotate each particle. Each particle is not the centre of its rotation. Particles rotate around the manipulator's axis.
Scale	Use the Scale X, Y, and Z fields to increase or decrease the speed or position of each particle along the X, Y, and Z axes. For example, if speed is the selected influence, a Scale X of 100% has no effect while a Scale X of 105% increases the speed along the X axis by 5% with each pass. Enable Prop to proportionally scale the axes.

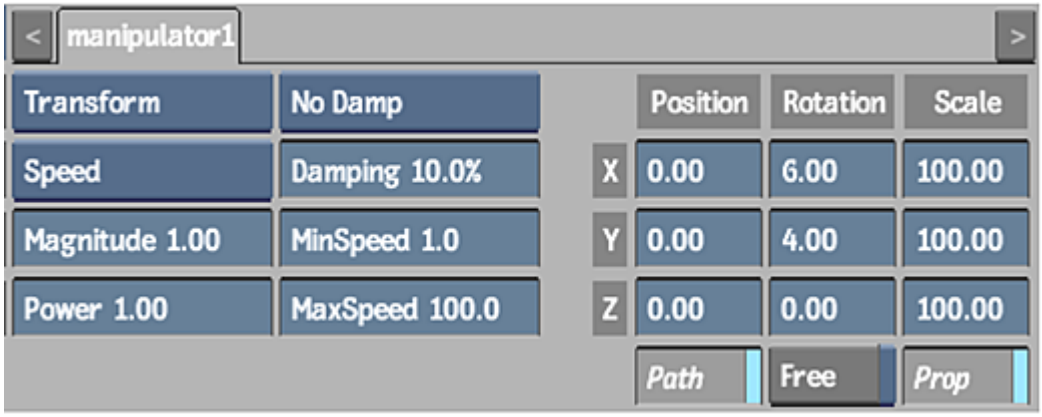
Enable the Path button to animates the position of the axis using a spline drawn in the scene. Disable Path to animate the position of the axis using explicit animation.

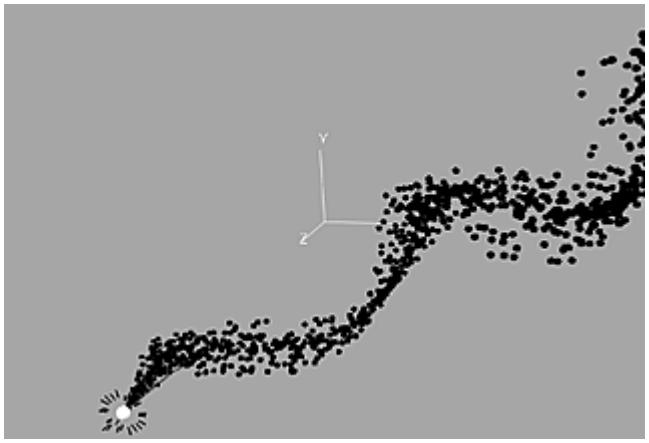
Free button Enable to ignore transformations from parent axes.

Applying Transformations

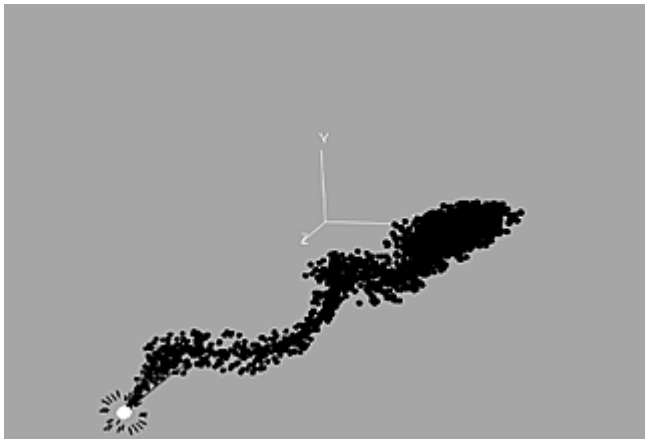
Use a transform manipulator to apply the accumulated transformations from the Axis controls to the position or speed of each particle. Unlike other manipulators, the transform manipulator is not used to position the manipulator in the scene. Transform manipulators are always placed at the centre of the particle stream they influence.

The following figure provides an example of creating a particle stream influenced by a transform manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.





In the Particle Manipulator menu, use Magnitude as an extra scaling factor whose sensitivity depends on the transformations in the Axis menu. For example, the following is a copy of the above particle stream with Magnitude set to 0.99 instead of 1.0; the particle generator and transform settings are the same.



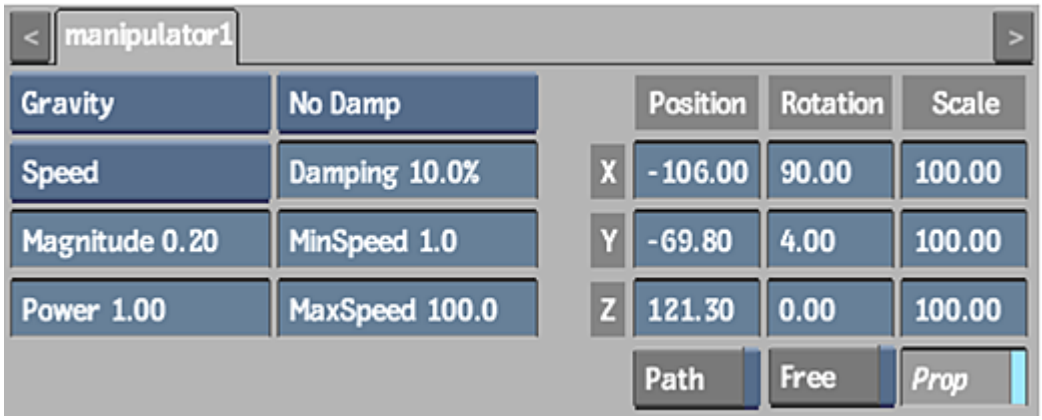
Simulating Gravity

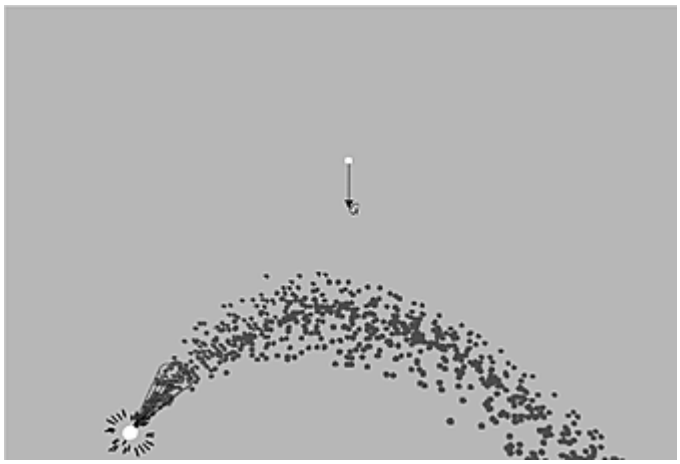
Use a gravity manipulator to simulate the effects of gravity. This manipulator works on speed or position. The Power field has no effect.

Magnitude specifies the gravitational pull in pixels per frame. A higher magnitude produces a higher gravity, which exerts a greater influence on each particle.

The icon for a gravity manipulator uses an arrow to indicate the direction of the influence. You can change the position and rotation of the arrow using the Axis menu. You can also change the scale of the gravity manipulator as another way of increasing its influence.

The following figure provides an example of creating a simple particle stream influenced by the gravity manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.



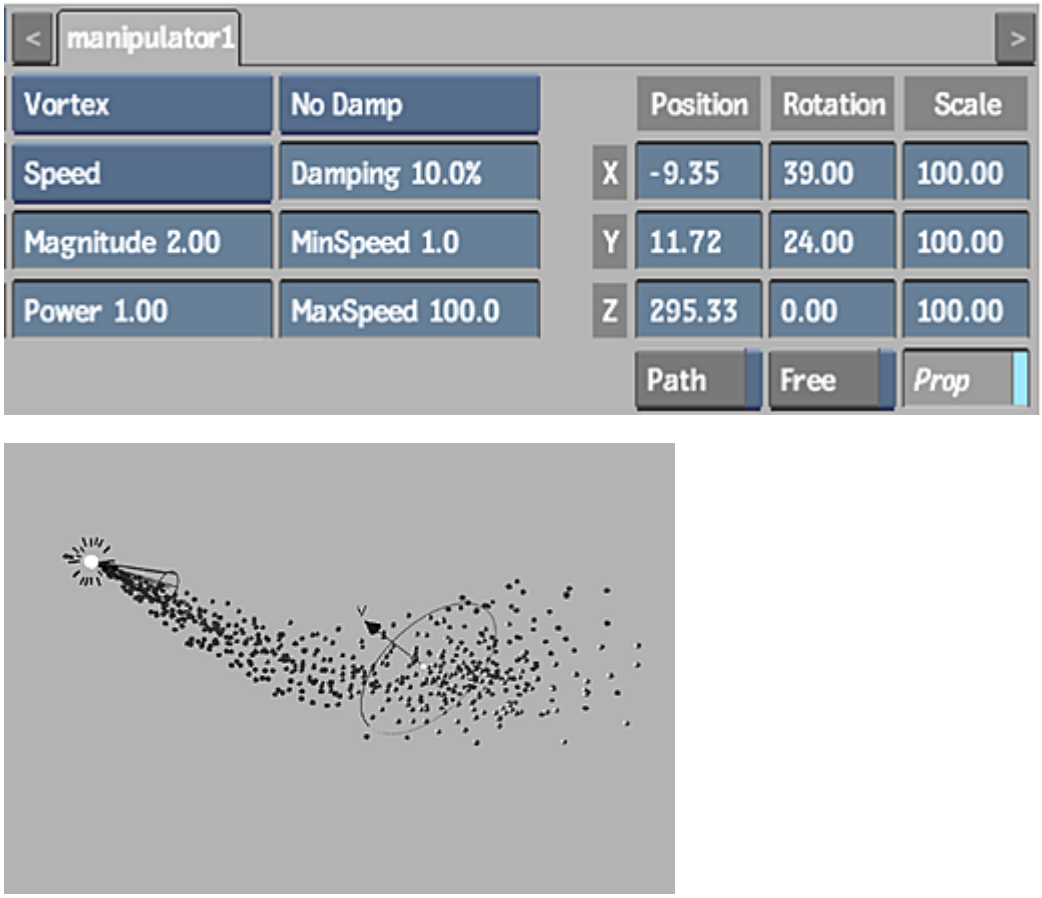


Creating a Vortex Effect

Use a vortex manipulator to influence the motion and rotation of particles. This manipulator works on speed or position. Power is the falloff from the centre of the vortex and magnitude is used for the scale of the vortex's rotation.

You can also use the Axis controls to apply additional transformations to the vortex. Scale changes the size of the vortex, and rotation affects the spin of the vortex.

The following figure provides an example of creating a simple particle stream influenced by the vortex manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.



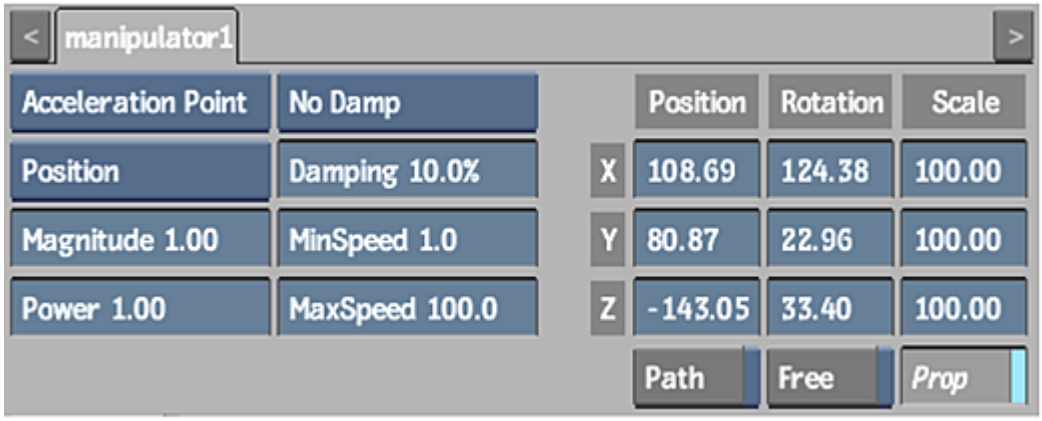
NOTE The icon used to show a vortex does not indicate the limit of the vortex. The icon is only used to represent the vortex in the scene and is not meant as an accurate depiction of the size, strength, or limit of the vortex's influence.

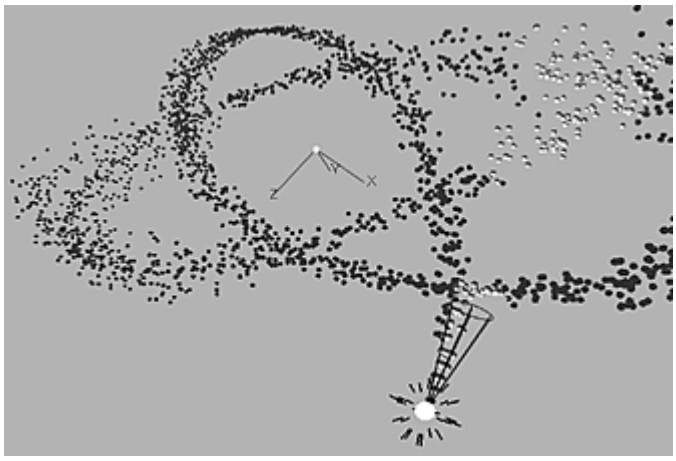
Using an Acceleration Point Manipulator

The Acceleration Point manipulator mimics the gravitational pull between bodies. Unlike the gravity manipulator, which pushes particles in a specific direction, Acceleration Point continually pulls particles toward itself on all three axes (X, Y, Z).

Magnitude is the gravitational pull in pixels per frame. A higher magnitude produces a higher gravity, which results in a greater influence. Power is the distance from the centre, or the radius, of the pull.

The following figure provides an example of creating a simple particle stream influenced by the Acceleration Point manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.





NOTE The Axis controls are used only to position the Acceleration Point in the scene. Scale and rotation have no effect.

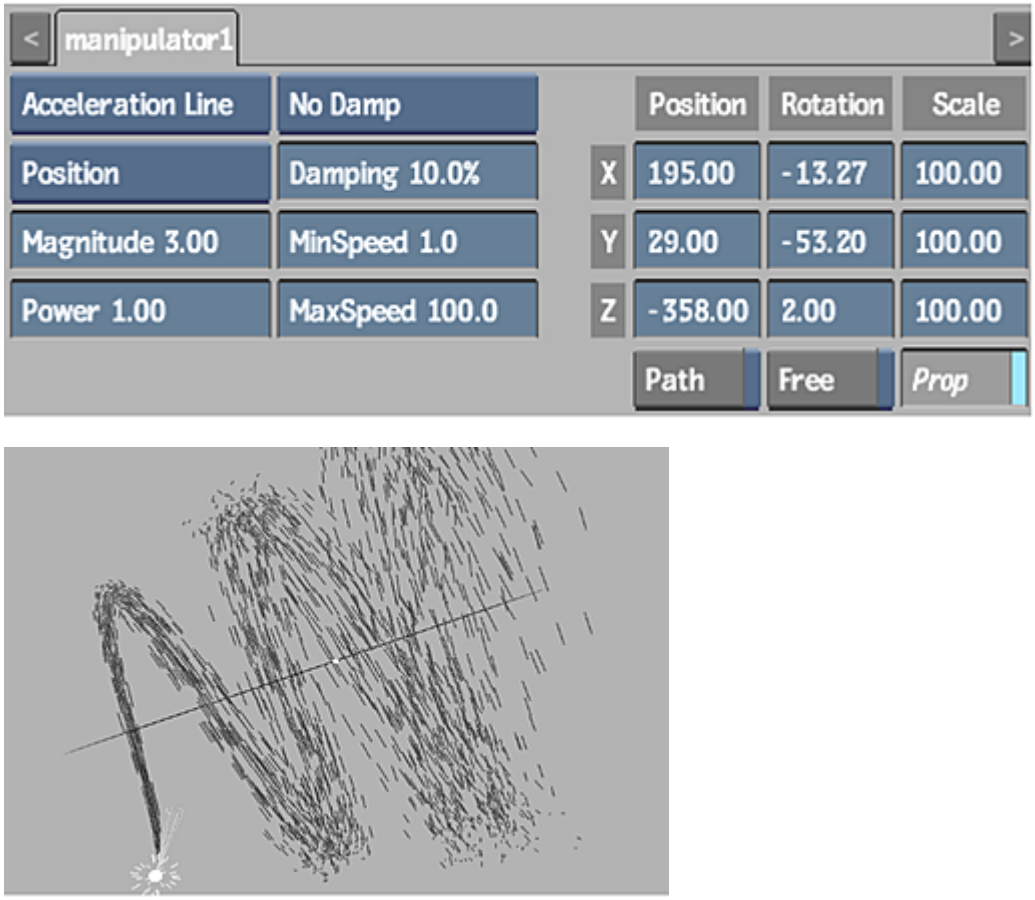
Using an Acceleration Line Manipulator

Like Acceleration Point, the Acceleration Line manipulator continually pulls particles toward itself. Acceleration Line, however, manipulates particles on two axes instead of all three.

You can set the two axes by rotating the manipulator using the Rotation fields. The Axis controls are also used to position the Acceleration Line manipulator in the scene. Scale may change the appearance of the Acceleration Line icon, but has no effect on particles.

Magnitude is the gravitational pull in pixels per frame. A higher magnitude produces a higher gravity, which results in a greater influence. Power is the falloff from the manipulator, or the radius of the pull.

The following figure provides an example of creating a simple particle stream influenced by the Acceleration Line manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.



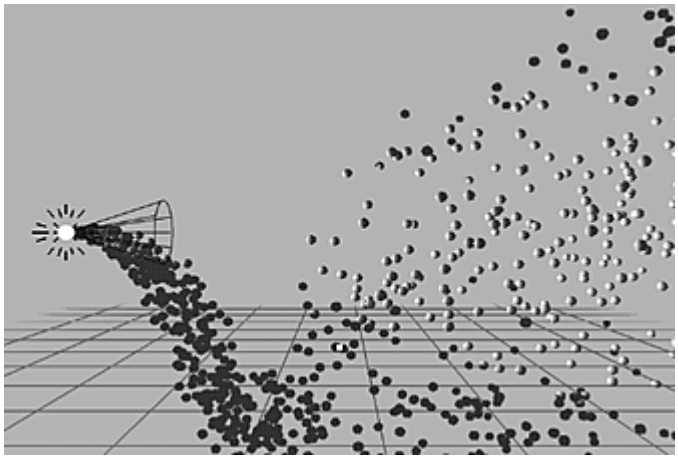
Using an Acceleration Plane Manipulator

Like Acceleration Point and Acceleration Line, the Acceleration Plane manipulator continually pulls particles toward itself. With Acceleration Plane, however, particles are manipulated on only one axis set by rotating the plane using the Rotation fields.

Use the Axis controls to position the Acceleration Plane manipulator in the scene. The Scale and fields are not used.

Magnitude is the gravitational pull in pixels per frame. A higher magnitude produces a higher gravity, which results in a greater influence. Power is the falloff from the manipulator, or the radius, of the pull.

The following figure provides an example of creating a simple particle stream influenced by the Acceleration Plane manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.



NOTE Because Acceleration Plane influences particles on only one axis, the effect may not be noticeable until you change views or orbit the camera.

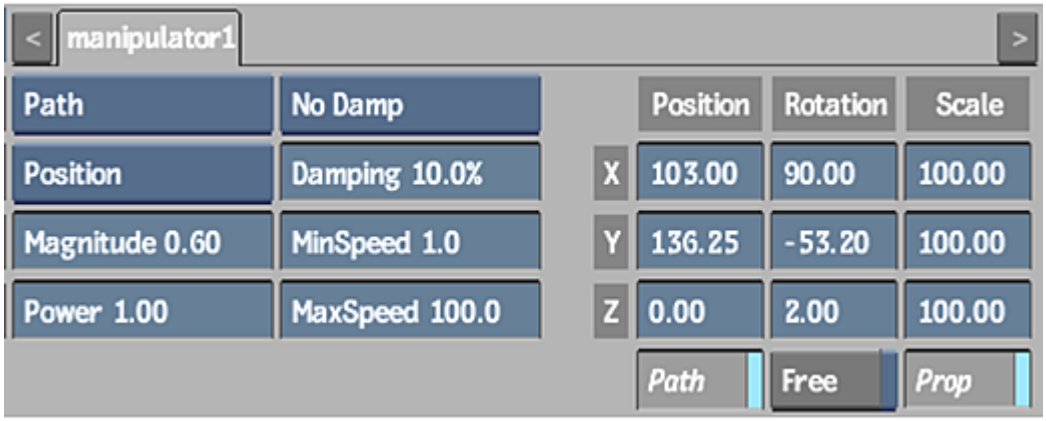
Forming a Particle Path

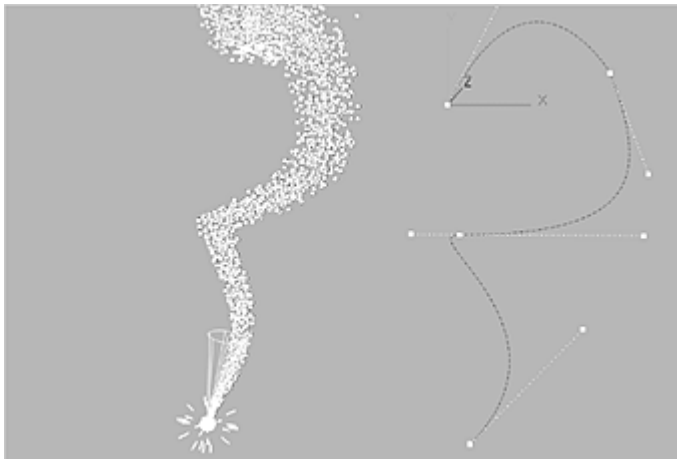
Use the path manipulator to make particles follow the motion of a path.

The motion of a path is applied to the speed or position of each particle. Note that particles do not directly follow the path. You can make the particles seem to follow the path by lowering the particle speed and magnitude, which is used as a scaling factor. Power is not used.

Use the Axis controls to animate the path. You can also use the Rotation and Scale fields to add extra transformations.

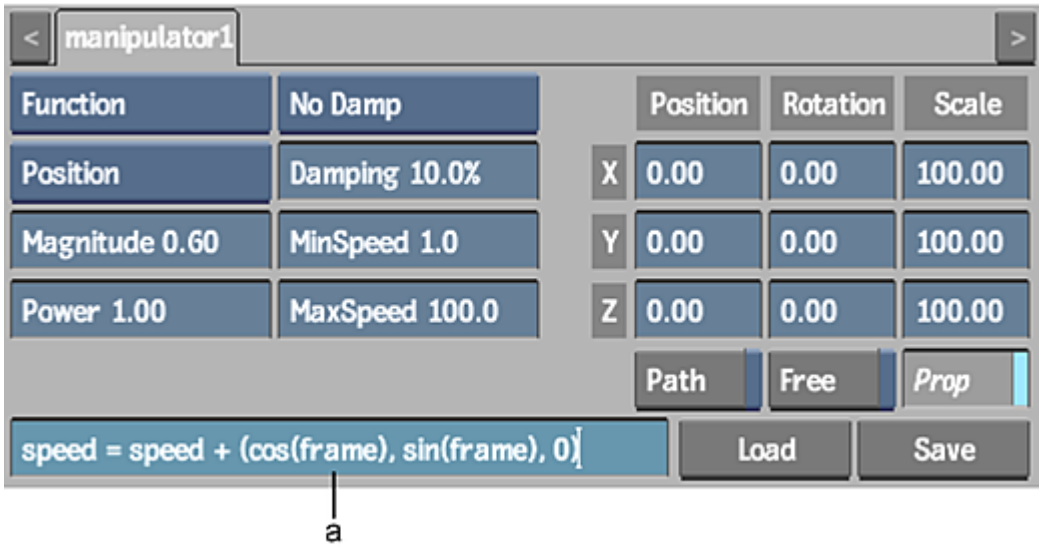
The following figure provides an example of creating a simple particle stream influenced by the path manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.





Using a Function as a Manipulator

Select Function to specify your own mathematical expressions to be used as a particle manipulator. You can use the channels from the Particle Manipulator and Particle Generator menus in your expressions, as well as arithmetic operations, mathematical conventions, functions, and constants.



(a) Expression field

The arithmetic operators, conventions, constants, and functions that you can use in your expressions are listed in the following sections. Before writing your own expressions, you should understand the following:

- A vector is a 3D coordinate written using the convention (x, y, z) where x, y, and z are separate values.

  For example, pos represents the position of each particle. If you want to increase the Y position of each particle with each pass, use the expression pos = pos + (0,1,0).

- Make sure assigned values and vector values are within an acceptable range. For example, transparency (opacity of each particle) is a value between 0 and 1. The expression transparency = size does not work unless size (particle size from the PartGen menu) falls between 0.0 and 1.0.

  To make transparency dependent on size, size must be divided by an appropriate value. For example, if size is between 1 and 10, use the expression transparency = size / 10.

- Some functions return scalars and other functions return vectors. Make sure that when you use a function, it returns the right value and that this value is within an acceptable range.

  For example, rgb is a vector of values between 0.0 and 1.0. The expression rgb = (0, 0, noise3(pos)) gives an error because noise3 returns a vector. The expression rgb = (0, 0, frame) does not give an error, but frame is the frame number and it is never less than 1. This means the blue channel is always set to full blue (1).

- You can specify more than one function by separating each with a semicolon.

  For example, to place the two expressions in the Expression field, you would type:

  speed = speed + pos; pos = pos + (0,size,0)

Expression field operators are listed in the following table.

Operator	Description
=	Equals
+	Addition
-	Subtraction

Operator	Description
*	Multiplication
/	Division
%	Percentage
(x,y,z)	Vector where *x, y, z* may also be the results of functions
==	Equivalence
!=	Not equal to
<	Less than
<=	Less than or equal to
>	Greater than
>=	Greater than or equal to

Particle and manipulator variables that you can use in the Expression field are listed in the following table.

Particle Symbol	Description
pos	Position, a vector (x,y,z).
speed	Speed, a vector (x,y,z).
rgba	Red, green, blue, and alpha colour channels for each particle, expressed as a 4D vector (r,g,b,a). Each component is a value between 0 and 1.
rgb	Red, green, and blue colour channel, expressed as a vector (r,g,b). Each component is a value between 0 and 1.
red	Red channel, a value between 0 and 1.
green	Green channel, a value between 0 and 1.
blue	Blue channel, a value between 0 and 1.
transparency	Transparency of each particle (surface or geometry), a value between 0 and 1.
lifetime	Lifetime of each particle, in frames.
lifetimeI	A value between 1.0 and 0.0 where 1.0 is when a particle is first generated and 0.0 is when it ends.

Particle Symbol	Description
mass	Mass of each particle.
size	Size of each particle, in pixels.
tailSize	Width of the particle's tail, a value between 0 and 1.

The following variables are read only. You cannot change them in the Expression field, but you can use them in your calculations.

Manipulator Symbol	Description
frame	Current frame
magnitude	Value from the Magnitude field
power	Value from the Power field
damping	Value from the Damping field
minSpeed	Value from the MinSpeed field
maxSpeed	Value from the MaxSpeed field

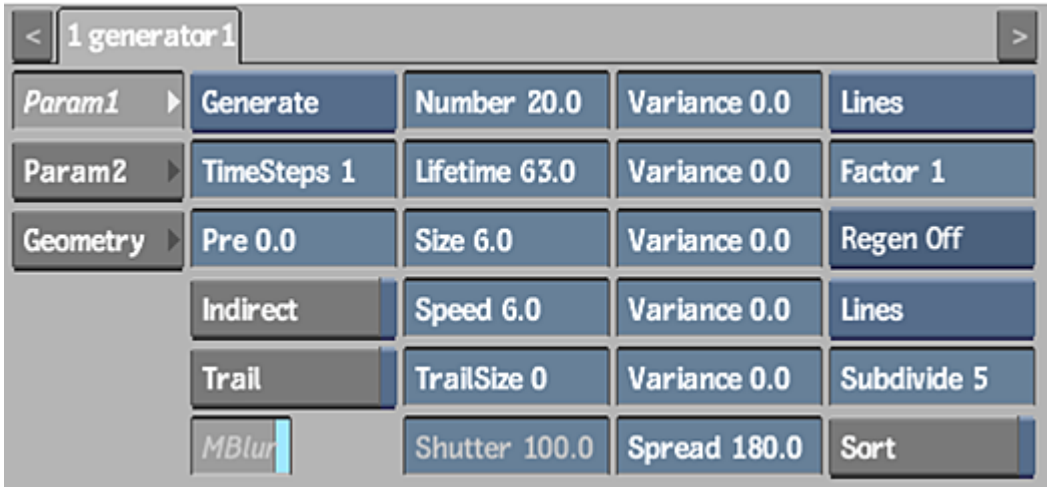
Single argument arithmetic functions are listed in the following table.

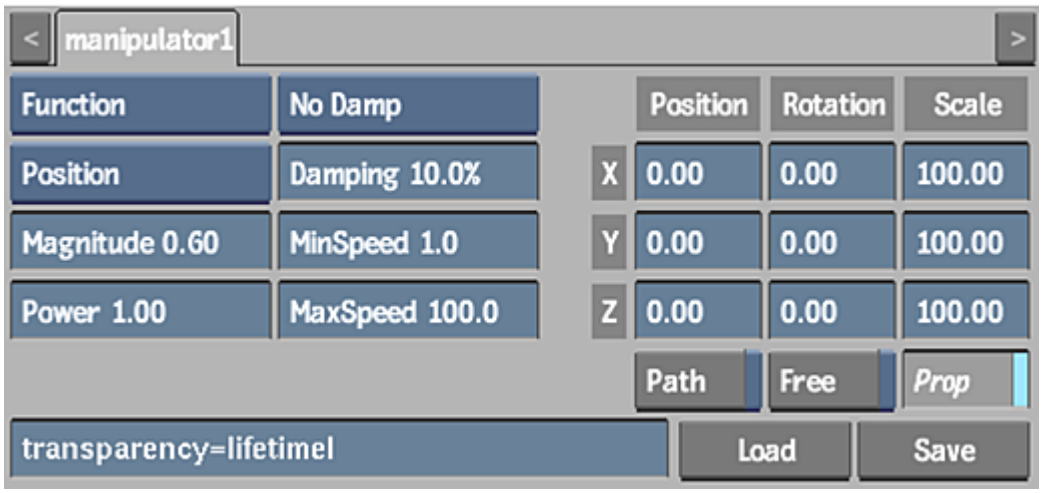
One Argument	Description
sin(*a*)	Sine of *a*
cos(*a*)	Cosine of *a*
tan(*a*)	Tangent of *a*
asin(*a*)	Arcsine of *a*
acos(*a*)	Arccosine of *a*
atan(*a*)	Arctangent of *a*
exp(*x*)	Exponential function of *x*
expm1(*x*)	Equivalent to exp(*x*)-1
log(*x*)	Natural logarithm of *x*
log10(*x*)	Base 10 logarithm of *x*

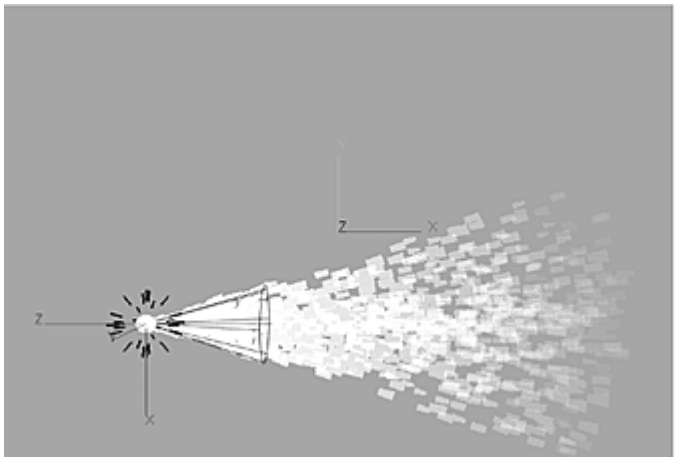
One Argument	Description
log1p(*x*)	Equivalent to log (1+ *x*)
sqrt(*x*)	Square root of *x*
abs(*x*)	Absolute value of *x*
trunc(*x*)	Integer value of *x*
floor(*x*)	Smallest integer greater than or equal to *x*
ceil(*x*)	Largest integer greater than or equal to *x*
round(*x*)	*x* rounded to the nearest integer
radians(*a*)	*a* converted to radians
degrees(*r*)	*r* converted to degrees
sign(*x*)	Returns +1 or -1 depending on the sign of *x*
length(*p*)	Euclidean length of point *p*
noise(*v*)	Noise of vector *v*, returns a float
fnoise(*v*)	Fractal noise vector *v*, returns a float
noise3(*v*)	Noise of vector *v*, returns a vector

Making Particles Transparent

The following example illustrates how to make particles become transparent as they reach the end of their lifetime using the expression transparency = lifetimeI. The settings in the Particle Manipulator menu, Particle Generator menu, and the resulting effect are shown.





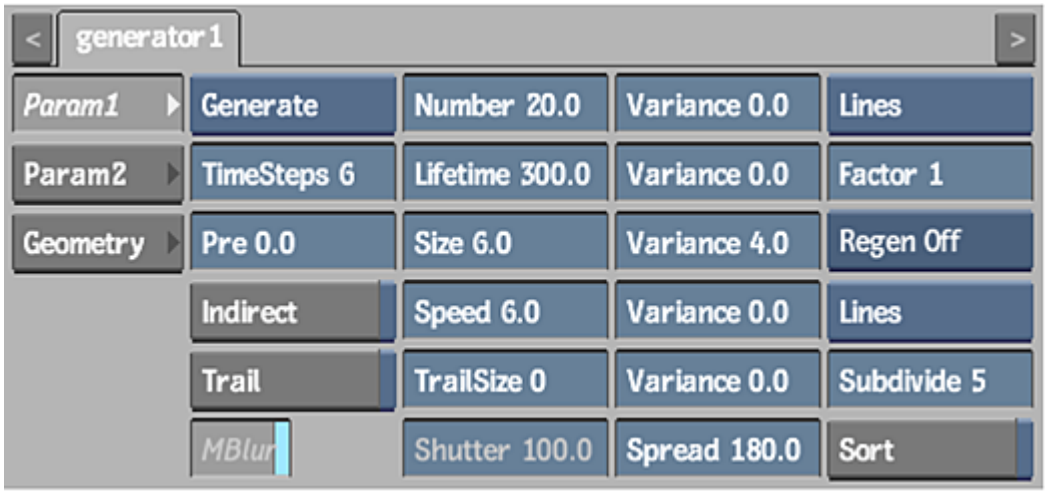


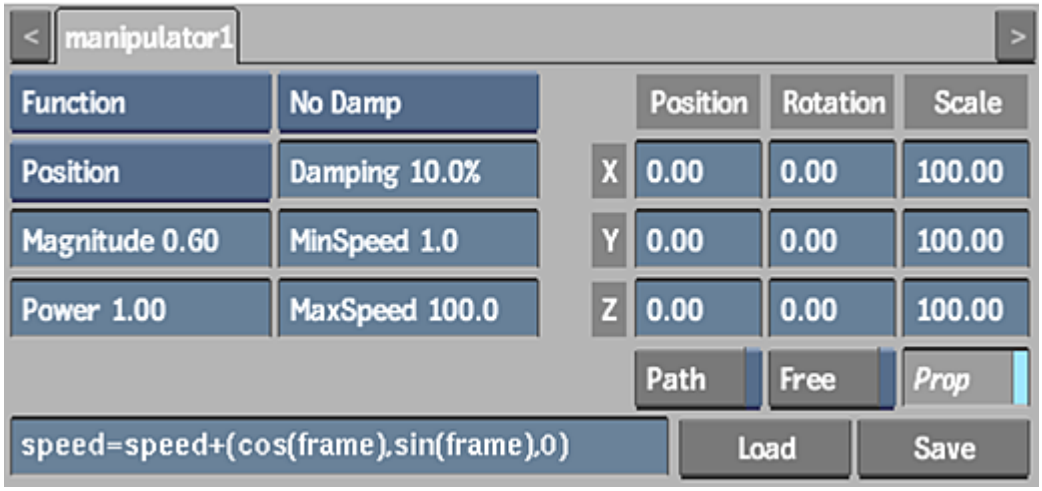
Making Particles Spin

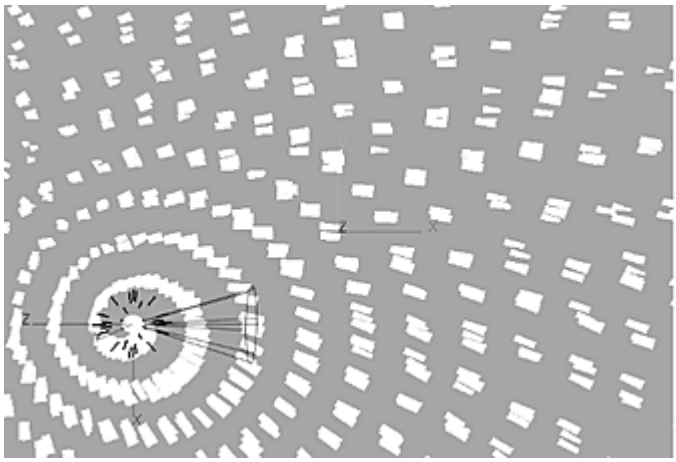
The following example illustrates how to spin particles using the expression:

speed = speed + (cos(frame),sin(frame),0)

The settings in the Particle Manipulator menu, Particle Generator menu, and the resulting effect are shown. Notice that the particle Timestep value is increased to improve the accuracy of the particle stream.





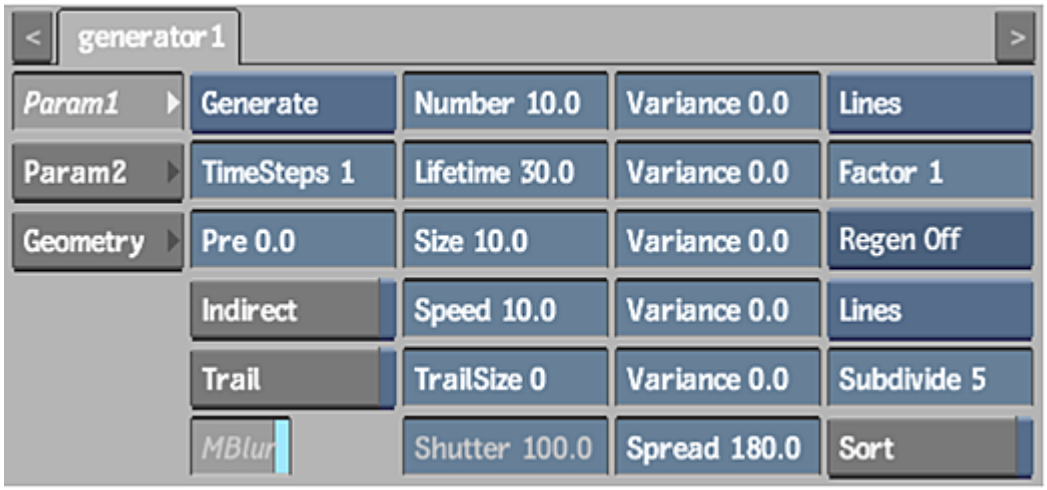


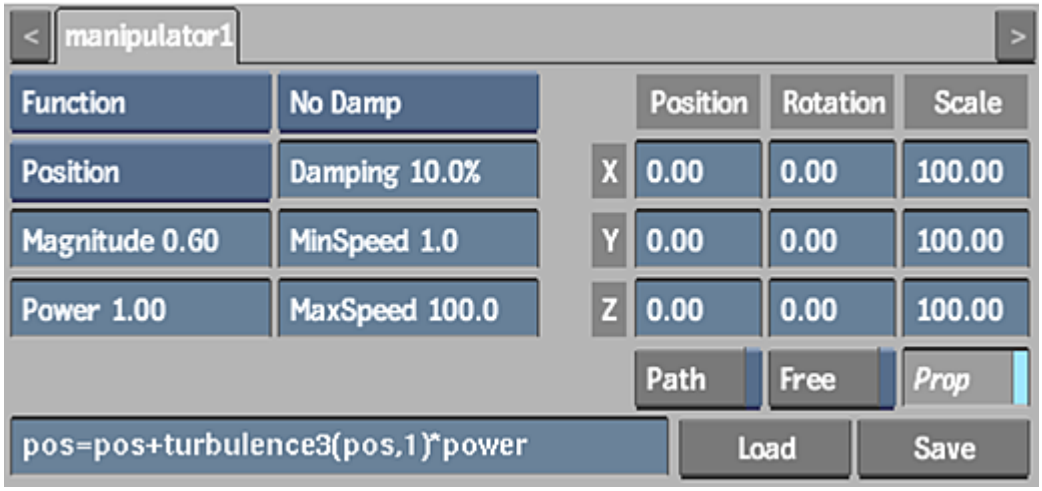
Arithmetic functions that have two or more arguments are listed in the following table.

Symbol	Description
atan(*x*,*y*)	Arctangent of *y* over *x*
pow(*x*,*y*)	*x* to the power of *y*
mod(*x*,*y*)	Returns the remainder of dividing *x* by *y*
min(*x*,*y*)	Minimum value of *x* and *y*
max(*x*,*y*)	Maximum value of *x* and *y*
step(*x*,*y*)	Returns 0 if *x* < *y*, 1 if *x* >= *y*
dot(*v1*, *v2*)	Dot product of two vectors; returns a scalar
cross(*v1*, *v2*)	Cross product of two vectors; returns a vector
turbulence(*v*, *o*)	Turbulence of vector *v* and octave *o*; returns a float
turbulence3(*v*, *o*)	Turbulence of vector *v* and octave *o*; returns a vector
smoothstep (*min,max, x*)	Returns 0 if *x* < *min*, 1if *x* >= *max*; if neither are true, returns a hermite interpolation between 0 and 1
clamp(*x, min, max*)	*x* clamped to the range [*min,max*]

The following example illustrates how to create animated turbulence using the expression:

pos = pos + turbulence3(pos, 1) * power



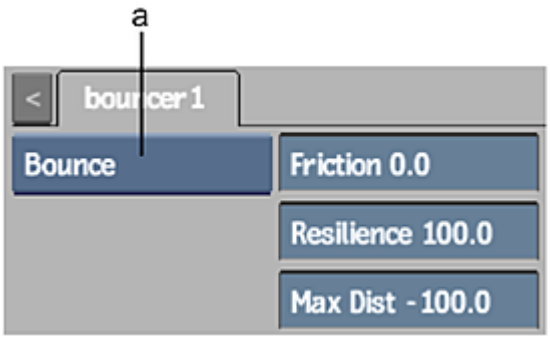


Bouncing Particles

You can bounce a particle stream off either an image or a bouncer. When particles bounce off an image, they bounce off the surface wherever its matte is white or grey. Particles only pass through the areas of the image where its matte is pure black. If a surface's matte is turned off, particles bounce off the entire surface.

When you use a bouncer, it must be the parent of an axis or a light source. A bouncer shows up as a sphere in the Action scene when you use it with a light source. Otherwise, all other types of surfaces create flat bouncers.

Whether you use a bouncer or an image, bouncing particles are controlled using the Particle Bouncer menu.



(a) Bounce Type box

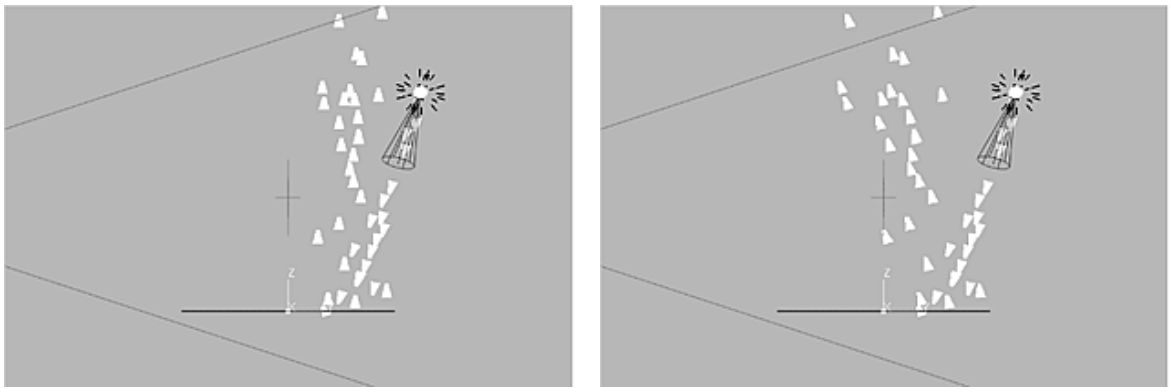
Bounce Type box Select a bounce behaviour.

Select:	To:
Bounce	Bounce particles off images or bouncers. All bounce properties work with this type of bounce behaviour.
Extinct	Make particles disappear when they reach a surface. Only the Max Dist parameter works with this type of bounce.

Select:	To:
Generator	Create an additional particle stream when particles bounce. When using this bouncing behaviour, you must add another particle generator. See Combining Particle Streams (page 985).
Generator+ Extinct	Make particles disappear and create an additional particle stream when particles reach a parented surface. Only the Max Dist parameter works with this type of bounce unless the particles end in another particle stream. See Combining Particle Streams (page 985).

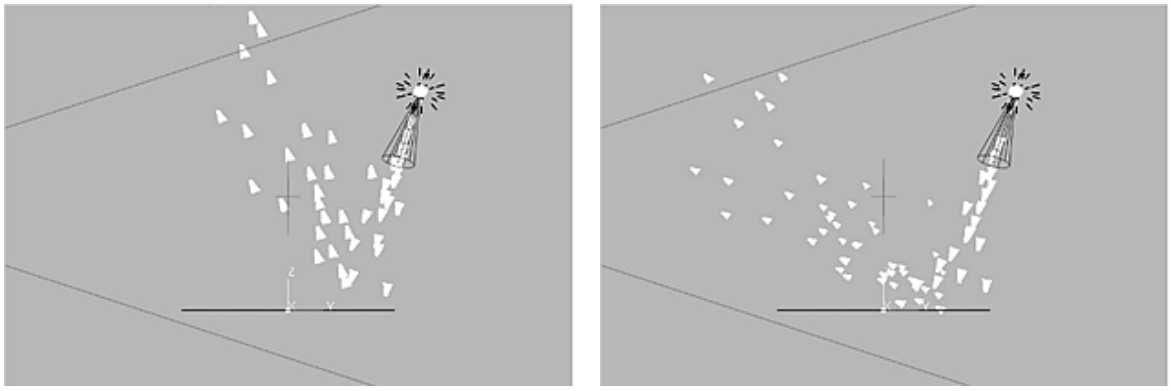
Friction field Displays a percentage to influence the angle of bouncing particles. Editable.

The greater the percentage, the more particles bounce straight. For example, in the left figure, Friction is set to 100%, which bounces particles almost perfectly straight. In the figure on the right, Friction is set to 20%, which causes the particles to deviate once they make contact (bounce).



Resilience field Displays the amount of energy lost with each bounce. Editable.

For example, at a Resilience of 100%, the figure on the left shows there is no energy lost between bounces. Each bounce is at the same height as the last bounce. When Resilience is set to 80%, 20% of energy is lost between each bounce. At 50%, the figure on the right shows the height of each bounce is reduced by 50%. Note that Friction is set to 0 for both examples.



Maximum Distance field Displays the distance a particle must travel past an image for it to still bounce or become extinct. Editable.

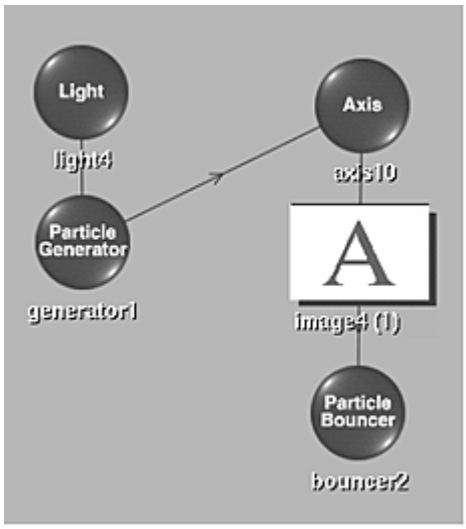
Because an image is only a pixel in width, it is unlikely that fast moving particles will hit the image directly. Max Dist acts like a buffer zone after the surface. If particles fall through the surface, set the Max Dist to -100. Particles that miss the surface by 100 pixels will still bounce.

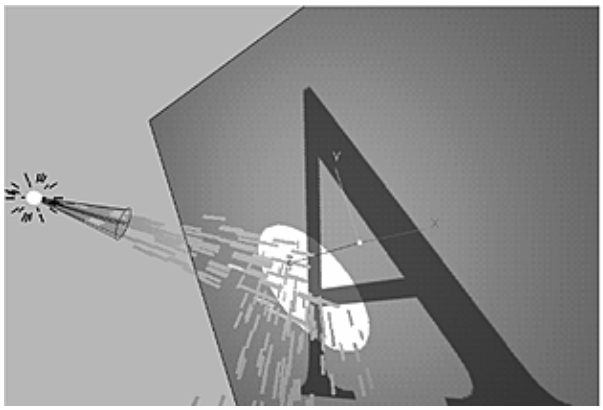
If particles still fall through the surface, increase the timestep to increase the accuracy of the particle stream.

Bouncing Particles Off an Image

You can bounce particles off an image surface only. Particles will bounce off bilinear and bicubic surfaces, but only on the original flat surface and not on any changes in shape.

To create the bouncing particle effect, you start with a particle generator. When you determine the image off which you want to bounce particles, you must parent the particle generator to the image; otherwise, the particles will ignore the image. The following figure shows the schematic and the result of bouncing particles off an image using a light source as a generator.





To bounce particles off an image:

1. Add an image off which you want to bounce particles.
2. Create a particle generator using a light source, 3D object, or surface as a particle generator. See Generating Particles (page 948).
3. Parent the particle generator to the image's axis.
4. Select the image's axis and, in the Axis menu, enable Free.

   The parent-to-child relationship between the particle generator and the image is severed, allowing you to place the image in the scene more easily. Any transformations and animations from the particle generator are not passed to the image.
5. Move the image in the scene until the particle stream passes through it. To do this, you may have to switch views or orbit the camera.
6. In the schematic, select the image.
7. From the Node bin, double-click the Particle Bouncer node.

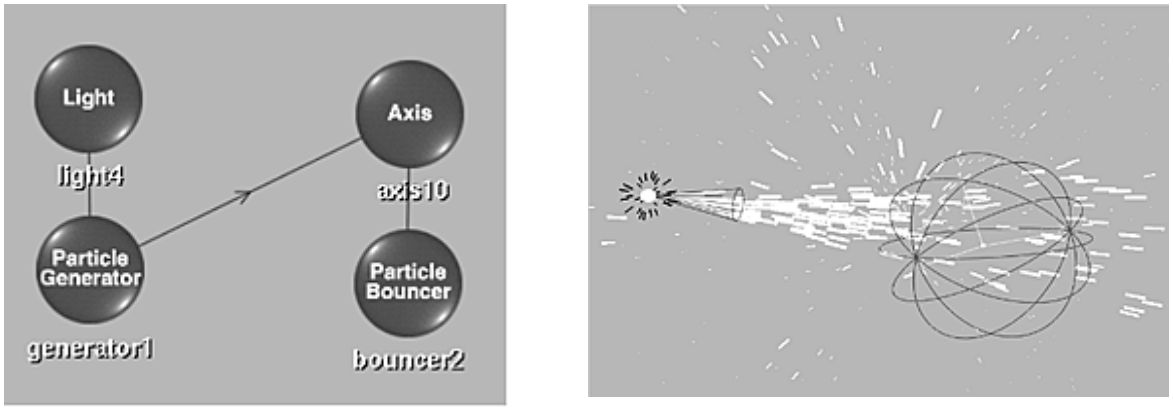
   A particle bouncer is added and parented by the image.
8. To access the Particle Bouncer menu, double-click the particle bouncer node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 571)).

Bouncing Particles Off a Bouncer

A particle bouncer element is added and automatically parented to a light source. The particle bouncer may not be parented by the right light source or axis. The bouncer must be parented by the light or axis added in the last step.

NOTE The shape of the bouncer depends on the particle generator type.

The following figure shows the schematic and the result of bouncing particles off a bouncer.



To bounce particles off a bouncer:

1. Create a particle generator using a light source, 3D object, or surface as a particle generator. See Generating Particles (page 948).
2. Add an axis or a light source to the scene.
3. From the Node bin, double-click the Particle Bouncer node.
4. If you are using an axis to set the position of the particle bouncer, select the axis and go to step 5. Otherwise, go to step 7.
5. In the Axis menu, enable Free to sever the parent-to-child relationship between the particle generator and the axis.

   Any transformations and animations from the particle generator are not passed to the axis. This makes it easier to place the particle bouncer in the scene.
6. Parent the particle generator to the particle bouncer's axis or light source.

   You must parent the particle generator to the axis or light source; otherwise, the particles will ignore the particle bouncer.
7. In the Particle Bouncer menu, select Bounce from the Bounce Type box and set the appropriate bounce properties.

NOTE If the particle bouncer is not parented correctly, unparent the particle bouncer and make the axis or light source added in the last step its parent.

Combining Particle Streams

You can combine two or more particle streams by parenting them together. This has two possible results depending on whether the first particle stream ends or bounces off an object.

NOTE Multiple particles can be parented, but only in a straight line, that is, a particle can only be the particle of one other particle. For example Particle 1 can parent Particle 2, and Particle 2 can parent Particle 3 in the same scene.

Particles Generating Another Particle Stream

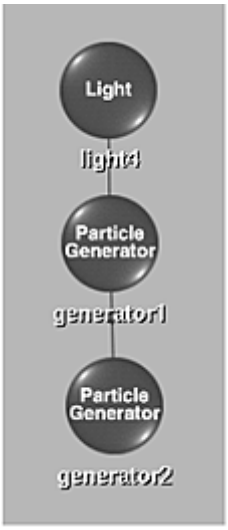
When each particle in the particle stream ends, the second particle stream begins. This can be used, for example, to create fireworks.

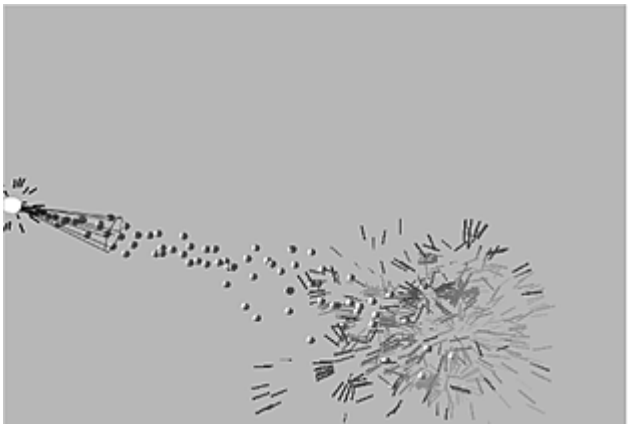
To make particles generate other particles:

1. Create a particle generator. You can use a light source, 3D object, or surface as a particle generator. See Generating Particles (page 948).
2. In the schematic, select the Particle Generator.
3. Add another particle generator by double-clicking the Particle Generator node in the Node bin.
4. In the Particle Generator menu, set the second particle generator's Spread and other properties as appropriate. See Setting Particle Properties (page 953).

   NOTE In the Channel Editor's folder for the second particle generator, you can set the Spread_V channel to add variance to the value in the Spread field.

   The following figure shows the schematic and the result of a particle stream that generates a second particle stream. The first particle stream generates spheres and the second particle stream generates quads.





Particles Bouncing into Another Particle Stream

When bouncing particle off a bouncer, you can generate a second stream of particles. For example, use this technique to give the effect of water or sparks bouncing off an image.

There are two different effects that you can create depending on your bouncer's settings. The following example illustrates the Generator bounce type.

To make bouncing particles generate another particle stream:

1. Create a schematic that bounces particles off an image or a bouncer. See Bouncing Particles (page 982).

2 In the schematic, select the Bouncer object.

3 In the Particle Bouncer menu, select Generator from the Bounce Type box.

4 Add another particle generator by double-clicking the Particle Generator node in the Node bin.

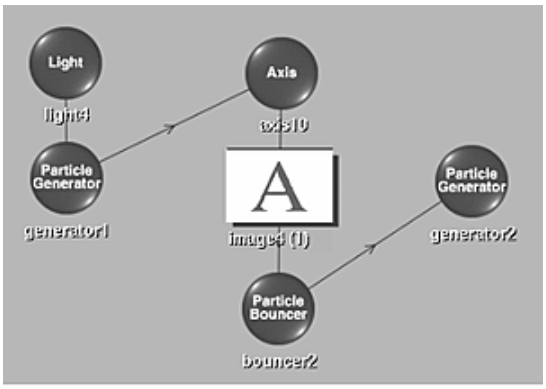
5 In the Particle Generator menu, set the particle generator's other properties as appropriate. See Generating Particles (page 948).

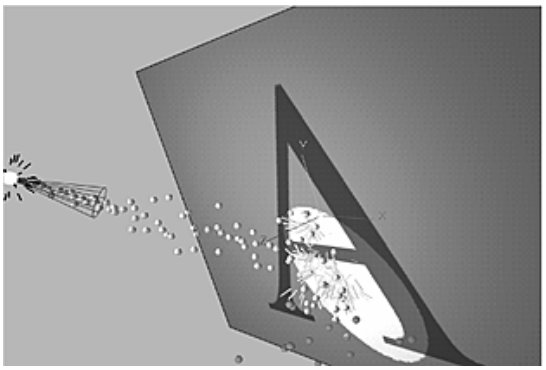
6 In the Channel Editor, open the second particle generator folder to reveal the properties folder.

7 Set the Spread and Spread_V channels to control the spread of the second generator.

If you set the Spread to less than 180, you can also control the rotation of the second particle stream using the Friction field in the Particle Bouncer menu.

The following figure shows the schematic and the result of bouncing particles into another particle stream where the first particle stream continues after bouncing off an image surface. The first particle stream generates spheres and the second particle stream generates quads.





Sample Particle Setups

Ten particle setups are provided in the directory */opt/Autodesk/<product_home>/examples*. To load one of the setup files, open Action using a black frame as the front, back, and matte clips.

A brief description of each example particle setup is provided in the following table.

Setup File Name	Description
Bounce Example	Uses a gravity manipulator and a bouncer attached to an axis. The bouncer appears as a sphere in Camera view. The particles fall down and bounce off the bouncer.
Bounce Example 2	Uses a gravity manipulator and two image surface bouncers.
Bounce Example 3	Uses an Accpoint manipulator and a bouncer linked to an axis. The bouncer follows the Accpoint manipulator.
Explode Example	An example of exploding 3D text. The Geom node is hidden in Schematic view. The setup uses an Explode generator and Polygon particles. To add 3D text (Geom node), go to the Text menu, click the Text field and type the text string, then click Create.

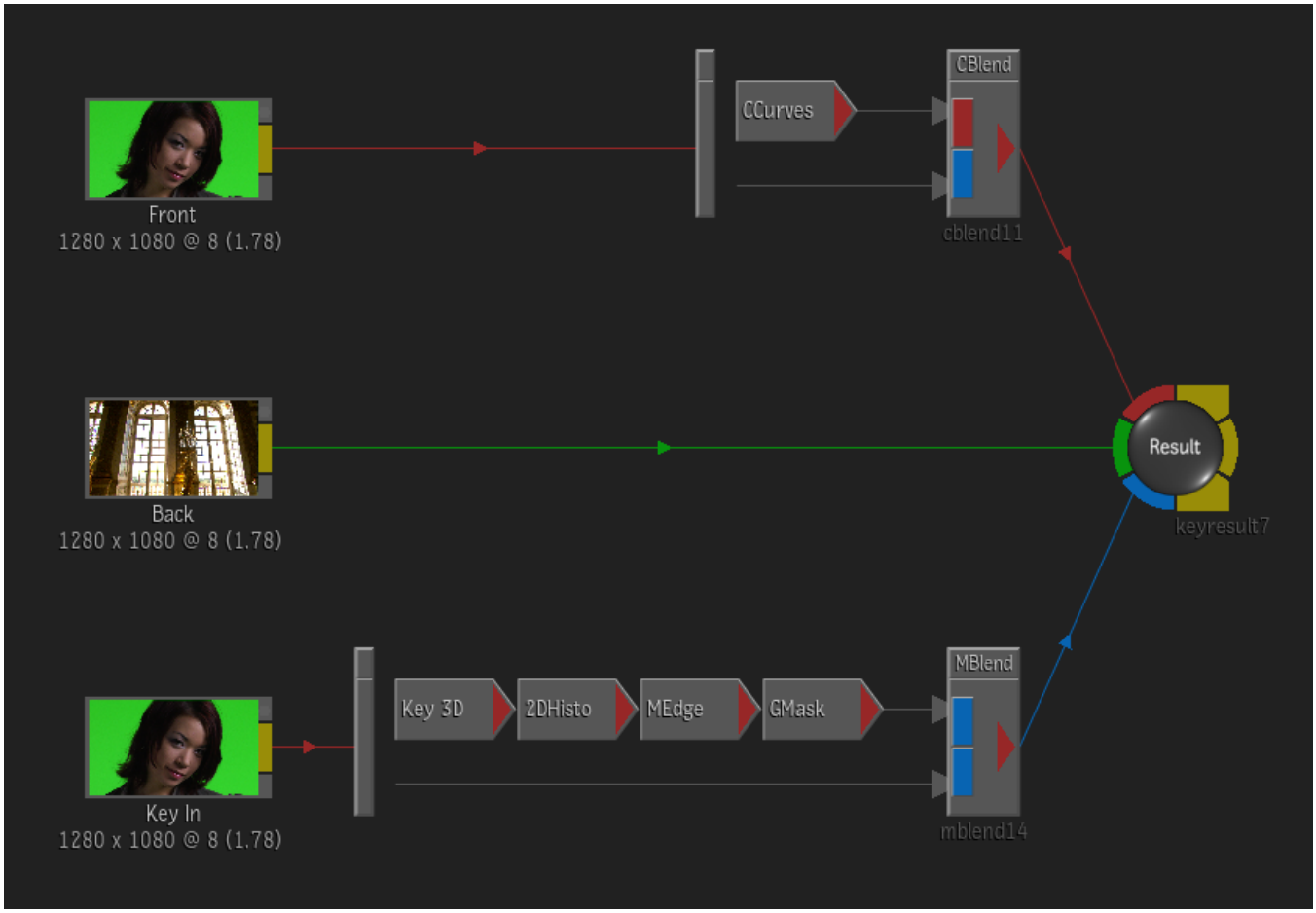
Setup File Name	Description
Explode Example 2	An example of exploding an image surface. The image surface is hidden. Media1 is applied to the generator as a texture and uses Wrap mapping mode. Replace Media 1 with an image of your choice.
FireWorks Example	Uses two linked generators. The first generator generates few particles at each frame. When these reach the end of their lifetime, the second generator is activated.
Function Example	Uses a function manipulator that increases the particles' transparency as they reach the end of their lifetime.
Function Example 2	Uses a function manipulator with sine and cosine functions to modify the speed of the particles.
Node Example	Uses 3D text for the particles. The Geom node (the 3D text) must be named “ParticleDraw”. The generator is parented to the axis of the Geom node, and the selected particle type is Node.
Transform Example	Uses a transform manipulator.

Chroma Keying and Working with Green Screens

18

The Modular Keyer is a schematic environment, like Batch or Batch FX, that is designed specifically for keying. Within the Modular Keyer, you can choose different Start Modes or presets, which generate node schematics adapted to the different keys you want to pull, such as chroma keys.

When you start the Modular Keyer for the first time, Flame builds a default schematic that includes the 3D Keyer node, in addition to a number of other nodes, that all work well together to create and refine a basic key. This is the 3D Keyer Start Mode.



NOTE If you want to reload a keyer setup or BFX from Flame Premium 2013 or prior, consider the following:

- If you enabled the Flame Reactor rendering engine when you created your project, make sure the media in the Setup/BFX is 8 bit or 16 bit floating point, as the Flame Reactor engine only supports 8 bit and 16 bit floating point media.
- If you enabled the Legacy Rendering engine when you created your project, the media in the Setup/BFX can be 8, 10 or 12 bit but not 16 bit floating point as the legacy keyer does not support 16 bit floating point media.

Within the Modular Keyer, the keyers are represented as processing pipelines, which are collections of interconnected nodes. As you perform keying tasks in the Modular Keyer, you navigate from node to node to accomplish the final key, displayed in the Result node.

As a general example, you could proceed as follows:

- Key out the selected colour in the Keyer node;
- Adjust your blacks and your whites in the 2D Histogram node;
- Refine your matte in the Matte Edge node;
- Mask out troublesome areas of your matte in the GMask node;
- Perform colour correction of the front clip in the Colour Curves node;
- View the result in the Result node.

Because it is node-based, you have a lot of flexibility with the Modular Keyer. You can:

- Choose a different Start Mode to start with a different default keying pipeline. Options are:
  - Channel Keyer.
  - Default MK (3D Keyer)
  - GMask. See GMask (page 1102).
  - HLS Keyer.
  - Luminance Keyer.
  - GMask Tracer. See GMask Tracer (page 1099).
  - Master Keyer.
  - RGB Keyer.
  - RGBCMYL Keyer.
  - YUV Keyer.

  IMPORTANT When you select another Start Mode, any work in progress will be lost and the selected keyer is loaded with the default settings.

- Add other effects nodes from the node bin to further refine your key.

NOTE All of the legacy keyers had a colour section available in the settings. This colour section of the legacy keyers is only carried over to the Modular Keyer if the legacy keyer setup used the Master Keyer. If any other legacy keyer was used (RGB, Luminance, etc.), the colour section is discarded when the setup is loaded. To retain the colour section of these legacy keyer setups, enable Classic Engine in the Projects and Users dialog box. Also, motion blur data, from the GMask section of the legacy keyers is not carried over to the Modular Keyer.

Setting Up the Nodes and Media to Pull a Key

To pull a key, you must first set up your composite in the Modular Keyer.

There three ways to set up the composite in the Modular Keyer:

- From the Timeline, through the Action Timeline FX.

- From the Timeline through Batch FX.
- From the Tools menu.

To set up the composite in the Modular Keyer from the Timeline through the Action Timeline FX:

1. Select the track you want to key, and add an Action Timeline FX.

   The track below the selected track is taken as the Background.
2. Double-click the Action entry in the FX ribbon, or click the Editor button to enter Action.
3. In Action, click the Media button to display the Media menu.
4. Double-click the K field in the front/matte row to enter the Modular Keyer.

   The 3D Keyer processing pipeline appears. The 3D Keyer is the default keyer.

   NOTE Once inside the Modular Keyer, the Modular Keyer node bin is displayed at the bottom of the screen. Though very similar to the Batch FX Node bin, the Modular Keyer node bin contains only nodes that are relevant for keying. To return to Batch FX view, click Return.
5. Use the desired Start Mode, and create your key.

To set up the composite in the Modular Keyer from the Timeline through Batch FX :

1. From the timeline, select the segment you want to use as your front.
2. Click FX and select Create Batch FX

   You are taken to the Batch FX view. The schematic is displayed and your front clip is connected to the output node.
3. From the Batch FX node bin, drag the Modular Keyer node between the front clip and the output node and release it when the connection goes from red to yellow.

   The front clip is connected to the front (red) input of the Modular Keyer node, which is connected to the output node.
4. Double-click the back (green) input of the Modular Keyer node.

   You are taken to the Viewing panel and the cursor becomes green and prompts you to select a back clip.
5. In the Viewing panel or the Media panel, click on the clip you want as your back.

   The cursor becomes blue and prompts you to select a matte clip.
6. In the Viewing panel or the Media panel, click on the clip you want as your matte.

   You are taken back to the Batch FX schematic and your front, back and matte clips are connected to the front, back and matte inputs of the Modular Keyer node. You are ready to start creating your key.
7. Double-click the Modular Keyer node.
8. Click the Edit button that appears at the bottom of the screen.

   The 3D Keyer processing pipeline appears with your front, back and matte clips connected to the front, back and matte inputs of the 3D Keyer.. The 3D Keyer is the default keyer.

   NOTE Once inside the Modular Keyer, the Modular Keyer node bin is displayed at the bottom of the screen. Though very similar to the Batch FX Node bin, the Modular Keyer node bin contains only nodes that are relevant for keying. To return to Batch FX view, click Return.
9. Double-click the 3D Keyer node to display the 3D Keyer menu at the bottom of the screen.
10. Select the desired Start Mode, and create your key.

To set up the composite in the Modular Keyer from the Tools menu:

1. Select the Tools tab, on the bottom of the window.

The Tools menu is displayed.

2 Under the Composite tab, click Modular Keyer.

The cursor becomes red and prompts you to select a front clip.

3 From the Media panel or the Viewing panel, click the clip you want as your front.

The cursor becomes green and prompts you to select a back clip.

4 From the Media panel or the Viewing panel, click the clip you want as your back.

The cursor becomes blue and prompts you to select a matte clip.

5 From the Media panel or the Viewing panel, click the clip you want as your back.

The cursor becomes white and prompts you to select a render location in your workspace for your resulting key.

6 Click on a grey area of the Viewing panel or in a folder in the Media panel to select the render location.

You are taken inside the Modular Keyer and the 3D Keyer processing pipeline is displayed, with your front, back and matte clips connected to the front, back and matte inputs of the 3D Keyer. The 3D Keyer is the default keyer.

NOTE Once inside the Modular Keyer, the Modular Keyer node bin is displayed at the bottom of the screen. Though very similar to the Batch FX Node bin, the Modular Keyer node bin contains only nodes that are relevant for keying. To return to Batch FX view, click Return.

7 Double-click the 3D Keyer node to display the 3D Keyer menu at the bottom of the screen.

8 Select the desired Start Mode, and create your key.

NOTE You can also access the Modular Keyer, as well as all the other Keyers in the application through Batch, where each Keyer is represented as an effects node. See Using Batch and ConnectFX (page 495).

After you have set up the nodes and media, set up the viewports for keying (page 992).

Setting Up Viewports for Keying

After you have set up your processing pipeline and your media (page 990), you want to set up the viewports for keying.

When keying, you want to set the viewport to 2-up view, to display the schematic and the result views simultaneously, enabling you to view the result as you work.

To set up the viewports for keying:

1 From the Layout box, select 2-up.

The view area is split in two. The schematic view is displayed on the left, by default.

2 Click in the right-side viewer to select it.

A white frame appears around the viewer to show that it is selected.

3 From the View box, select Result.

The result of your key, in its current state, is displayed in the right-side viewer.

About the 3D Keyer

Like traditional keying, the 3D Keyer in the Modular Keyer and Batch is used to select a range of colours to key out, such as the blue or green background in a clip. In the 3D Keyer, however, this range of colours is

represented visually in an RGB viewer. You can use the RGB viewer to analyse the colour areas in a clip, and then refine the key with increased accuracy.

The 3D Keyer can be used for many purposes other than pulling a key from the key-in clip. For example, in the Modular Keyer, you could use the Keyer-3D node in the Matte pipe of the CBlend node to isolate a region of the front clip to which you want to apply a particular colour correction.

NOTE It is recommended that you be familiar with the use and terminology associated with the Modular Keyer before you use the 3D Keyer.

3D Keying Workflow

The following table shows the recommended workflow for the 3D Keyer.

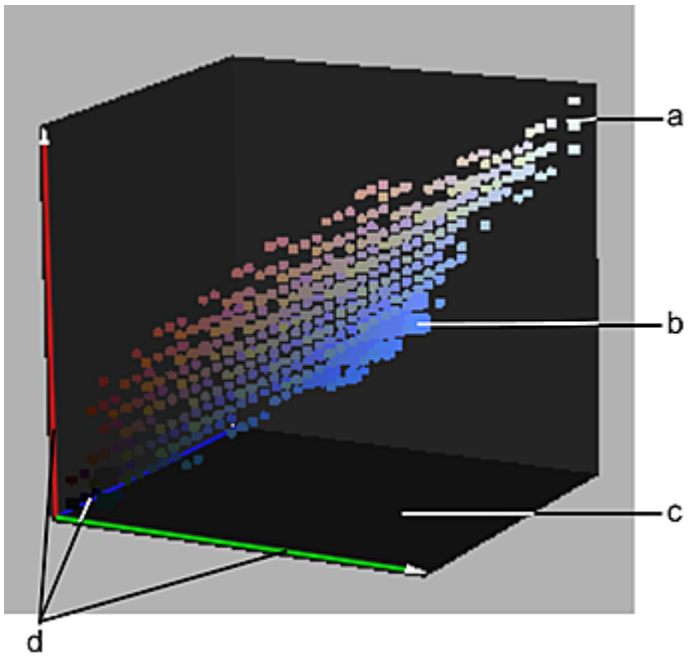
Step:	Refer to:
1. Open the Modular Keyer, select the clips, and then open the 3D Keyer.	Setting Up the Nodes and Media to Pull a Key (page 990).
2. Sample for tolerance.	Setting the Tolerance (page 994).
3. Sample for softness.	Setting the Softness (page 997).
4. Minimize the noise in softened areas.	Removing Noise from Softened Areas (page 999).
5. Remove grey areas from the key using negative sampling and patches, garbage masks, or RGB colour and softness adjustment.	Removing Grey Areas in the Key (page 1002), About Garbage Masks (page 1102), Techniques for Adjusting Softness (page 1030).
6. Adjust the luminance of the key.	Adjusting the Luminance of the Key (page 1047).
7. Adjust the edges of the key.	Modifying the Edges of the Key (page 1053).
8. Remove colour spill.	Adjusting Spill Controls with the Colour Curves Node (page 1056).
9. Adjust front and matte curves.	Adjusting the Front and Matte Luminance Curves (page 1058).
10. Render the clip.	Rendering Your Key (page 1061).

Creating a Precise Key in the 3D Keyer

This section describes the tools you use to create a precise key with the 3D Keyer. You can learn other techniques for perfecting your key in More Keying Techniques (page 1030).

Using the RGB Viewer

The RGB viewer is a 3D colour model for the key-in clip. Its X, Y, and Z axes represent the red, green, and blue components of the key-in clip's colour space. The cubes of colour show the distribution of all the colours of the key-in clip in RGB colour space. This 3D colour map is referred to as the 3D histogram.



(a) The colours in the key-in clip are represented by the colour cubes of the 3D histogram (b) More prevalent colours are represented by larger cubes (c) Black canvas background (d) Red, green and blue axes

When you scroll through a clip, the 3D histogram cubes update to display the colour distribution of each frame.

Use the 3D histogram to analyse the colour mapping of the key-in clip and to locate the colours to be keyed out. When you sample for tolerance and softness, the sampled areas are represented as ellipsoids in the RGB viewer. You can manipulate sampled objects in the viewer to fine-tune the key. When you plot a colour to key in the image, its location is plotted in the RGB viewer.

You can turn the display of the histogram and the black canvas background on and off and set options to control their appearance. Change the display any time as you create the key to help you best visualize the contents of your image and work with the RGB viewer. To learn more about controlling the display of the RGB viewer, see Setting the 3D Histogram Display (page 1014).

Setting the Tolerance

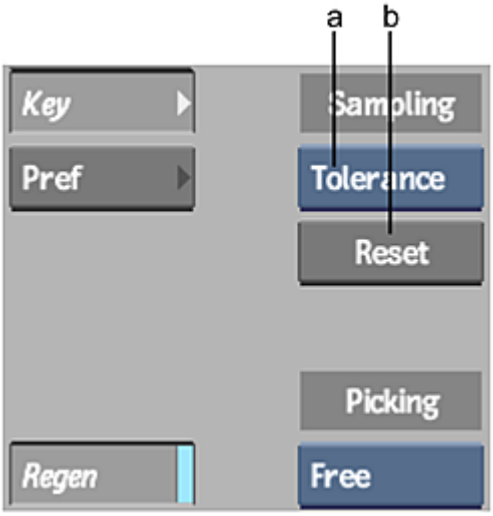
The first step in pulling a key with the 3D Keyer is to set a range of colours to be keyed out from the front clip. This is called the tolerance range. All pixels in the key-in clip in the tolerance range are black in the matte.

To specify the tolerance range, sample the area that you want to key out in the image window. Take one or more samples until you are satisfied with the result. The tolerance range must not be too large or the edge of the composite will be too hard and the subject in the front clip will appear to be pasted into the back clip.

To set the tolerance range:

1 Click Key.

The 3D Key menu appears.



(a) Sampling box (b) Sampling Reset button

2 Make sure you are at frame 1. If you move to a different frame while sampling, you will set a second tolerance keyframe (at the new frame) and animate the tolerance.

NOTE To avoid setting keyframes, disable Auto Key.

3 In the Sampling box, set the sample type to Tolerance.

4 Press `Ctrl` and drag the cursor in the area to be keyed out.

A red rectangle appears. The pixel values in the rectangle are used to define the tolerance range. Notice that the rectangle has a maximum size. This ensures that the result is computed quickly.

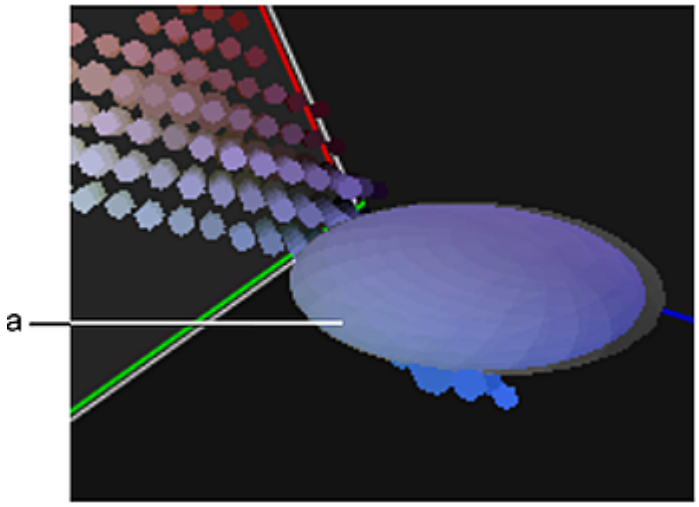
NOTE You can also take a sample by clicking the image and dragging the colour picker in the area to be keyed out. All pixel values that the colour picker samples are used to set the tolerance range.

5 If needed, add to the tolerance range by sampling several more rectangles. The ellipsoid gets bigger as you add to the tolerance.

NOTE If you are not satisfied with the result of the last sample, click Undo. The last sample you performed is cancelled. To reset the entire tolerance sample, click the Sampling Reset button. To remove the entire sample range at the current frame and start over with a new sample, press `Ctrl+Alt` and drag the cursor in the area to be keyed out.

Viewing the Tolerance Sample

The tolerance sample in the RGB viewer is represented as an ellipsoid by default. The ellipsoid shows you the location, in RGB colour space, of the range of colours included in the sample. All colours in the tolerance ellipsoid are black on the matte.



(a) Tolerance ellipsoid

When the histogram is displayed, you can see the location of the sample in relation to the other colours in the image. When it is hidden, you can see the sample more clearly. As you work, show or hide the histogram using the following techniques:

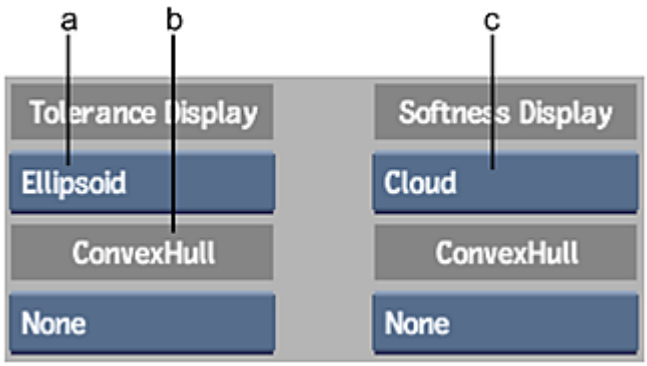
- To display or hide the histogram, press `H`. See Setting the 3D Histogram Display (page 1014).
- To view the sample more closely, rotate the RGB viewer by pressing `Ctrl` and dragging it. To zoom in on the RGB viewer, press `Shift` and drag inside it from left to right. To move the RGB viewer, press `Alt` and drag it.

Viewing the Convex Hull

The tolerance ellipsoid surrounds the exact colours sampled, which are connected in RGB space to form a 3D convex hull. You can see the convex hull in the ellipsoid by changing the tolerance display.

To view the convex hull:

1. Press `H` to remove the histogram.
2. In the Tolerance Display box, set the tolerance display to Cloud, which is semi-transparent.



(a) Tolerance Convex Hull box (b) Tolerance Display box (c) Softness Display box

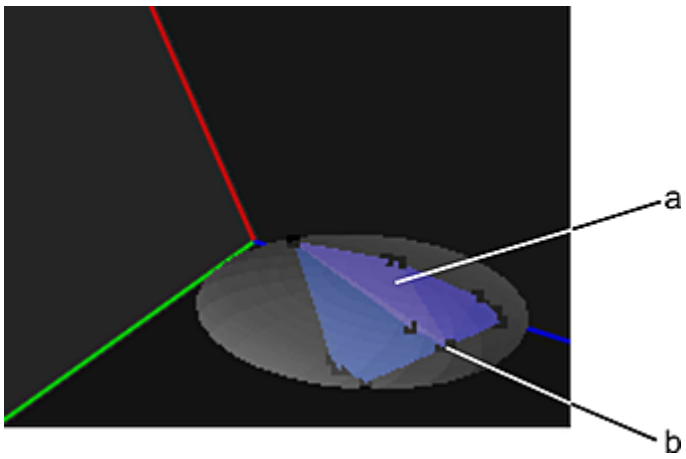
NOTE To learn more about controlling the display of the tolerance sample, see Changing the Display of Key Elements (page 1017).

3. In the Softness Display box, set the softness display to None.

NOTE When you set the tolerance range, the 3D Keyer automatically creates a minimum softness range that is equivalent to the tolerance range. This creates a softness ellipsoid of equal size to the tolerance ellipsoid in the RGB viewer. Sometimes the softness ellipsoid is visible at this point, depending on the way the ellipsoids are displayed.

4 Set the Tolerance Convex Hull box to Vert&Surf.

You can now see the convex hull, with vertices indicating the exact colour areas sampled.



(a) Convex hull (b) Convex hull vertices

The convex hull contains the exact colour values of the tolerance sample, while the ellipsoid represents the actual area in which corresponding pixels will be black in the matte. The 3D Keyer uses the ellipsoid rather than the convex hull to determine the keyed out area because it is much faster to calculate.

You can change the shape of the convex hull directly in the RGB viewer to adjust the tolerance. The ellipsoid re-orients itself according to the new shape. See Reshaping the Convex Hull (page 1028).

5 For the next step, return the softness display to Cloud.

Setting the Softness

The next step is to set a range of colours in the key-in clip to be partially transparent so as to soften the transition between the front and back clips in the composite. This is called the softness range. To specify the softness range, sample the areas to be softened in the image window. The pixels in the key-in clip in the softness range are grey in the matte. The key should have the greatest possible softness value.

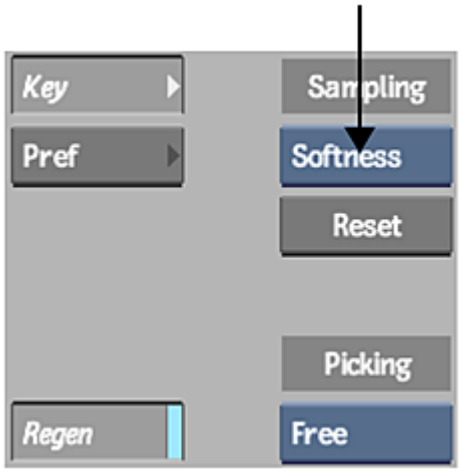
As shown in the following steps, the softness range can be set using the RGB or YUV colour models. RGB Softness is the default setting.

To sample for softness:

1 Make sure you are at frame 1. If you move to a different frame while sampling, you will set a second softness keyframe (at the new frame) and animate the softness.

NOTE To avoid setting keyframes, disable Auto Key.

2 From the Sampling box, select Softness.



3 From the View box, select Front to view the input image for the 3D Keyer.

4 Identify an area where softness is needed. For example, if the clip consists of a talent in front of a blue or green screen, softness is needed at the edge of the talent to soften the transition between the front and back clips. If the clip contains transparency, softness is needed in the semi-transparent area.

5 From the View box, select the current result view (displayed as <Currently Selected Node> Result, in the View box) to monitor the result as softness is applied to the matte while you sample.

TIP You can switch views as you sample the image. The current result view (displayed as <Currently Selected Node> Result, in the View box), shows the result of the processing pipeline up to the selected node. To see the Front Source, select Front from the View box.

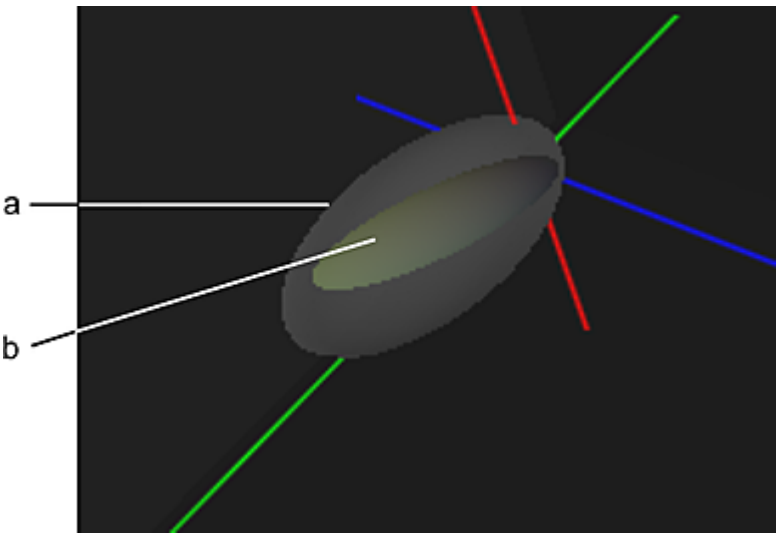
6 Zoom in to see the image more closely.

7 Click in the area you identified.

The cursor changes to a colour picker.

8 Drag the colour picker over the area in the image to be softened. Avoid sampling any area that should not be softened, such as the key itself; however, there is no need to avoid areas that have been keyed out. For transparencies, drag the colour picker in the area that you want to be semi-transparent.

A softness ellipsoid appears in the RGB viewer. Notice the tolerance ellipsoid in the softness ellipsoid.



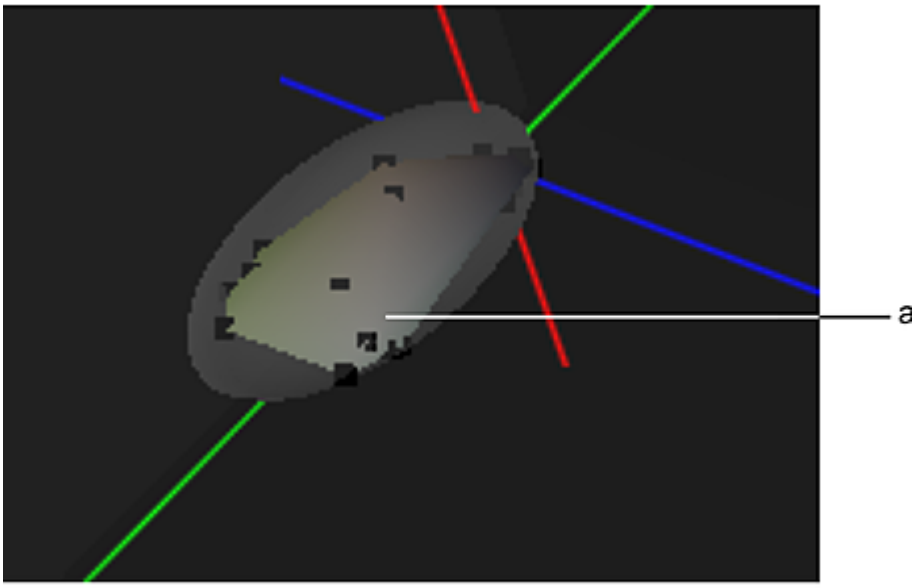
(a) Softness ellipsoid (b) Tolerance ellipsoid

All colours in the softness ellipsoid (except those inside the tolerance ellipsoid) will be grey on the matte. In the softness ellipsoid, the closer a point is to the tolerance ellipsoid, the darker the grey will be. For example, the mid-point between the edge of the tolerance ellipsoid and the edge of the softness ellipsoid is always the same mid-grey.

Notice how the softness ellipsoid increases in size as you sample the edges.

NOTE The softness is rendered interactively on the matte - you see the results immediately. For information on rendering the softness only after releasing the cursor, see Regen (page 1032).

9 Like the tolerance ellipsoid, the softness ellipsoid has a convex hull. To see it clearly, set the Tolerance Display and Tolerance ConvexHull display to None, and set the Softness ConvexHull display to Vert&Surf.



(a) Softness convex hull

10 If needed, take another sample elsewhere on the edges or transparency. Each time you sample, the pixels touched by the colour picker contribute to the previously calculated softness range.

NOTE If you are not satisfied with the result of a sample, click Undo. To reset the entire softness sample, click the R button. To remove the entire sample range and start over with a new sample, press `Ctrl+Alt` and drag the cursor in the area to be softened.

11 From the View box, select MK Schematic and, in the schematic, click the Result node.

12 From the View box, select the current result view (displayed as <Currently Selected Node> Result, in the View box) to view the result.

The edges are softened but may be grainy.

Removing Noise from Softened Areas

Once you sample for softness, use the Minimize Noise option to remove grain from the softened areas. Graininess, also known as noise, can occur at the edges of the key, or in the semi-transparent areas such as water or glass.

First, sample an area containing noise. The 3D Keyer analyses the sampled area. Using Minimize Noise, you can then scale the softness in such a way as to minimize the noise in the softened areas.

Minimize Noise is especially useful for semi-transparent areas and edges requiring a lot of softness, such as smoke, reflection, and shadows.

TIP If the entire clip is grainy, or if you are not satisfied with the results after scaling the softness, consider using another approach. For example, degrain the clip with Degrain before keying with the 3D Keyer. You can add a Degrain node to the beginning of the Matte branch anytime. Additionally, use Minimize Noise on a degrained clip if needed.

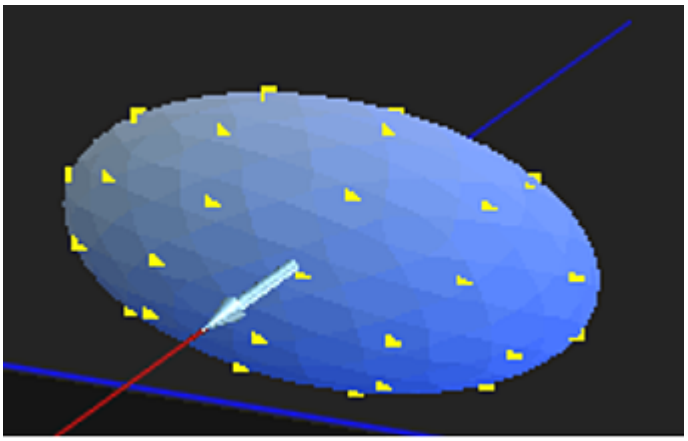
To remove noise:

1. Sample for softness. See Setting the Softness (page 997).
2. Zoom in and choose the area you want to analyse. Look for graininess in the softened areas. For transparencies, look for graininess in the semi-transparent areas.
3. Press N (Noise) and drag a rectangle in the selected area.

   The 3D Keyer analyses the pixels in the rectangle.
4. Select Min Noise from the Softness Scaling box.

(a) Softness Scaling box (b) Scaling field

An arrow appears in the softness ellipsoid, showing the direction in which the softness needs to be increased to reduce graininess in the sampled area.



5 Increase the softness using one of the following methods:

- Drag the cursor over the Scaling field to the right. The softness is increased based on the results of the analysis from the grainy region. Observe that the softness ellipsoid is scaled in the direction of the arrow.

  NOTE Each time you change the Scaling value, it is returned to 1.00.

- Use a colour value plotted in the image window as the basis for virtual point scaling of softness. To do this, use the `O` keyboard shortcut to plot a point in the image window, then press `Ctrl+V`, and click and drag in the Player.

6 If some edges or areas are still not softened, analyse again in that area and repeat the procedure.

7 Return the Softness Scaling to Prop (proportional scaling).

Alternatively, press `Alt` + `N` to scale softness based on the Minimize Noise analysis. This keyboard shortcut is the equivalent of selecting Minimize Noise in the Softness Scaling box, then scaling the softness using the Scaling field.

NOTE When Minimize Noise scaling is selected, it controls the arrow in the RGB viewer. To manually control the arrow, you must return to proportional scaling.

To scale softness using `Alt+N`:

1 Perform a noise analysis.

NOTE `Alt` + `N` only works if an analysis for the noise has been done.

2 Press `Alt` + `N` and click and hold the cursor anywhere in the image window. Drag the cursor to the left to decrease the noise or to the right to increase it.

The softness is scaled according to the noise analysis. Notice that Minimize Noise appears in the Softness Scaling box as you use the keyboard shortcut. When you release the cursor, the Softness Scaling option returns to Prop.

TIP For transparencies, use the `V` keyboard shortcut to remove unwanted grey areas in the matte, then use `Alt+N` to reduce noise in areas you chose to soften (the transparency). Perform the two procedures alternately until you achieve the best result.

Using YUV Softness

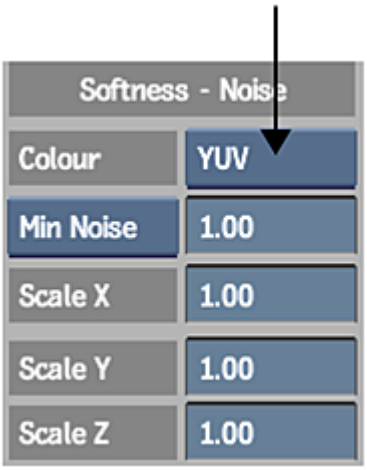
If you are not satisfied with the softness result—for example, if graininess still remains in the softened areas, try using YUV Softness. This option calculates the softness according to the YUV colour space.

YUV Softness often creates softened areas that are smoother and is typically useful for shadows and transparencies. However, since it removes more noise, the resulting softness range is less precise.

NOTE You can switch between RGB and YUV Softness and retain their settings.

To use YUV Softness:

1 Select YUV from the Softness box.

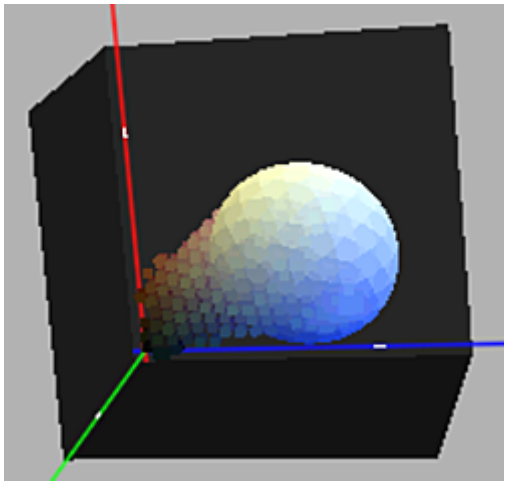


2 Sample for softness as described in Setting the Softness (page 997).

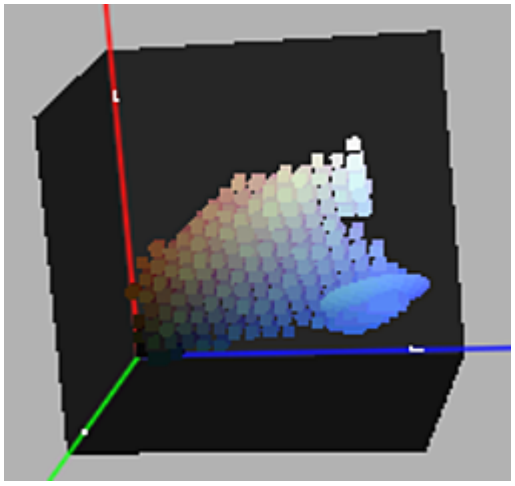
3 Minimize the noise as described in Removing Noise from Softened Areas (page 999).

Compare YUV with RGB Softness

Select RGB Softness from the Softness box. The previous RGB Softness range is still in effect. Compare the results with the YUV Softness. Generally, the softness ellipsoid is larger with YUV Softness, as shown in the following example.



With RGB Softness, the softness ellipsoid is bound by the convex hull



With YUV Softness, the softness ellipsoid is scaled according to the Y component of YUV colour space

If you are satisfied with the result, use YUV Softness. If not, return to RGB softness.

Removing Grey Areas in the Key

Once you have added softness, check to see if there are any grey areas in the key resulting from the setting. If so, there are several techniques to remove the grey areas:

- Perform negative sampling on the areas.
- Use patches.

- Use garbage masks.
- Manipulate the softness element in the RGB viewer.
- Use the Softness fields.

In this section, two methods are shown: using Negative Sampling to refine the softness range, and using the Patch tool to create “selective keys” in the white areas of the matte. To learn other ways to remove unwanted grey areas, see Techniques for Adjusting Softness (page 1030).

Negative Sampling

This is probably the simplest method of removing greys from the matte. You sample in the unwanted grey area to subtract those colour values from the softness range.

Try this method first to see if it solves the problem. However, if the colour values in the foreground subject are too similar to those at the edges, you may remove too much of the softness.

To perform negative sampling:

1. From the View box, select the current result view (displayed as <Currently Selected Node> Result, in the View box). Identify the unwanted grey areas in the matte.
2. Select Sample Softness from the Sampling box.
3. Press `Alt` and click a grey area. Gently drag the cursor over the area.

   If the negative sampling removed too much of the desired softness or brought back graininess, click Undo and try another method.

Adding a Patch

Patches are another type of sample you can take in the image window. Unlike tolerance and softness, you can specify the colour that a patch sample renders on the matte. Patch samples can be rendered as white, black, or any shade of grey on the matte. You can also set the opacity of a patch.

One use for patches is to remove unwanted grey areas from the matte. Use a patch to isolate a range of colours from those that have been included in the softness range, then set the colour you want the pixels to have on the matte. Sampling a patch to remove grey areas in the key is similar to creating a garbage mask by keying instead of drawing.

You can use up to three different patches in a key.

To sample a patch:

1. From the View box, select the current result view (displayed as <Currently Selected Node> Result, in the View box).
2. Identify the unwanted grey areas in the matte.
3. In the Sampling box, set the sample type to Patch1.

4 In the Patches box, select Patch 1.

(a) Patches box (b) Active button (c) Patches Softness field

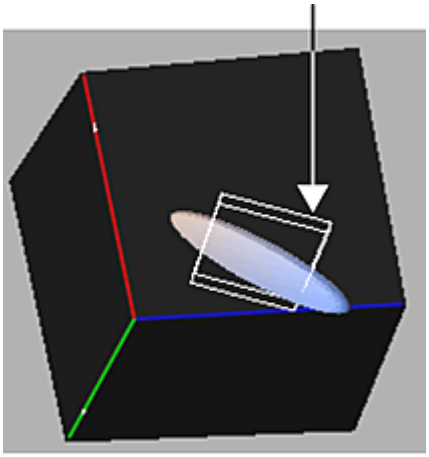
5 Enable Active.

6 Specify the colour in which you want to render the patch. In this example, make the patch white by setting the P1(Patch 1) Colour to 100 (this is the default value). To render the sampled values as black, enter a value of 0. Use a value between 0 and 100 for any shade of grey.

7 Sample the unwanted grey area by dragging the cursor over it.

The grey areas of the matte corresponding to the colour values sampled are changed to white.

All the colour values sampled are included in the patch. Notice the patch in the RGB viewer, represented by a white wireframe box. All colour values in the box will be rendered white on the matte.



8 If necessary, sample again to add to the patch. Watch the edges to make sure they are not affected by the sample.

TIP To remove colour values from the sample, press `Alt` as you sample. To undo a sample, click Undo.

9 If needed, soften the edges of the patch by dragging the cursor to the right or left in the Patches Softness field.

A negative softness value softens the edges of the patch inwards from the edge. A positive value softens the edges outwards beyond the edge, adding softened pixel values to the patch.

TIP You can view softness as a red wireframe box when the patch is displayed in Box mode. See Changing the Patches Display (page 1018).

10 If needed, adjust the opacity of the patch using the P1 Opacity field A value of 100% renders the patch as fully opaque. A value of 0% renders the patch as fully transparent.

The RGB viewer now contains three key elements:

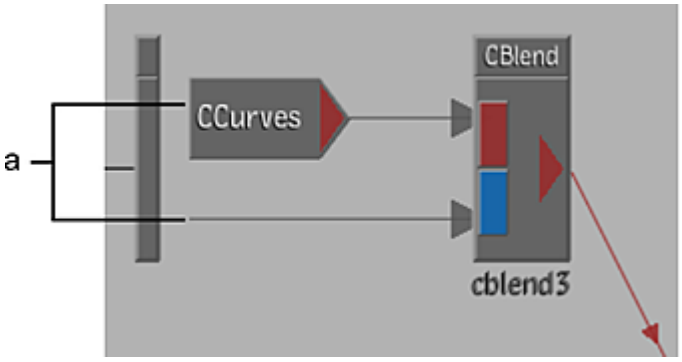
- The tolerance sample (as an ellipsoid)
- The softness sample (as an ellipsoid)
- The patch (as a white wireframe box)

For instructions on completing the key using the basic keying technique, see the sections referred to in 3D Keying Workflow (page 993).

Using Blend Nodes

Blend nodes include the Colour Blend (CBlend) and Matte Blend (MBlend) nodes.

A blend node is essentially a container that you can use to create a “mini-composite” at different parts of the pipeline. CBlend and MBlend nodes each comprise one or more layers that include the components needed for a composite: a front, matte, and back component. These components are referred to as pipes, and they may contain as many nodes as needed to create the desired image.



NOTE The back component is not represented visually as it is applied automatically to the composite.

(a) The Front and Matte pipes of the blend node constitute one layer.

The back component is not represented visually as it is applied automatically to the composite.

Nodes on the pipes are processed sequentially, with a single input and single output to and from each node. Therefore, you cannot add multiple input nodes that require several clips to pipes. However, some multiple input nodes can operate with only one input source, for example, the Gmask, Colour Correct, and Regrain nodes. If you add one of these node types to a blend node pipe, it is "converted" to a single-input node.

You can animate the curves in the MatteCurves, CBlend, MBlend, Colour Curves, or Result nodes.

Colour Blend Node

Use Colour Blend (CBlend) nodes to set up the colour operations you want to perform on a clip.

With a CBlend node, you can:

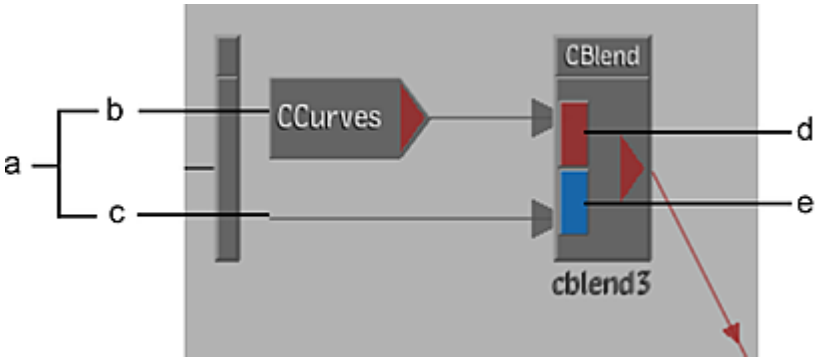
- Use the Front pipe of a layer to adjust the colour of a clip.
- Use the Matte pipe of a layer to isolate a region of the image to which the colour adjustments will be applied. With no matte, the colour correction is applied to the entire image.
- Blend the front and back images for each layer.
- Create layers of colour corrections for your front clip.

How the CBlend Node Works

The Front pipe of the CBlend node is designed to result in a colour image. This is represented visually by a red box at the right of the Front pipe. You can place any node on the Front pipe that results in a colour image. These include the Colour Curves, Colour Correct, RGB Blur, Degrain, and Regrain nodes.

The Matte pipe is designed to result in a matte. This is represented by a blue box at the right of the pipe. You can add any node that results in a matte to the Matte pipe, including the 3D Keyer, MasterK, 2D Histogram, GMask, Edge and Degrain nodes.

The Front and Matte pipes of the CBlend node constitute one layer.



(a) The two pipes constitute one layer (b) Front pipe (c) Matte pipe (d) Red box of Front pipe (e) Blue box of Matte pipe

NOTE The Degrain node results in a colour image; however, it is often useful to place it on the Matte pipe before the Keyer-3D node (which uses a colour image for its input). For this purpose, it can be placed on the Matte pipe.

The two pipes together, along with a back component, constitute one layer. For the first (bottom) layer, the CBlend node uses the source input image for the back component. For example, in the CBlend node on the Front branch of the default pipeline, the Front clip is used for the back component.

Matte Blend Node

Use the MBlend node to set up the matte operations you want to perform on a clip.

Use the Front pipe of a layer to define the matte, and use the Matte pipe to isolate an area where you want the matte to be applied. For example, use the Matte pipe to isolate an area of the key-in clip and apply a different matte to that area.

NOTE When using the Matte pipe for this purpose, you would create a second layer and place the overall matte in the bottom layer and the second, localized matte in the upper layer. This is because layers are processed from the bottom up.

When there are no nodes on the Matte pipe, the matte is applied to the entire image.

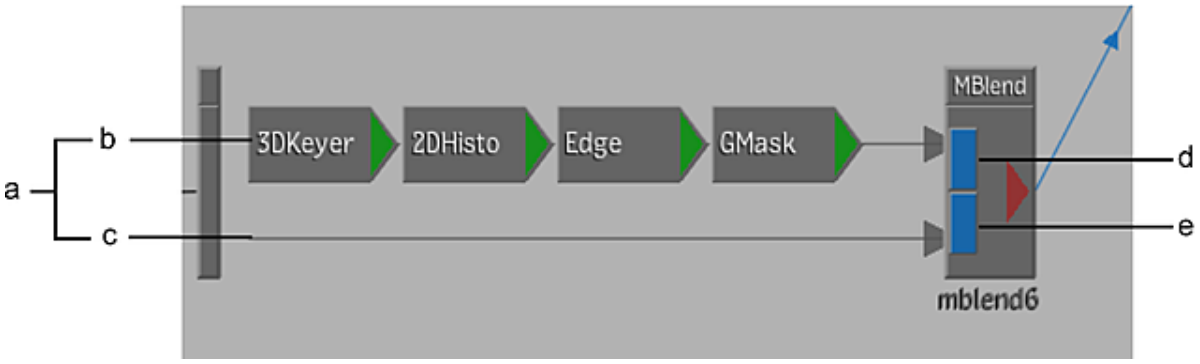
As with the CBlend node, you can blend the front and back images for each layer and create several layers of mattes.

How the MBlend Node Works

The Front and Matte pipes of the MBlend node are both designed to result in a matte. This is represented visually by a blue box at the right of the Front and Matte pipes. You can place any node on the Front and Matte pipes that results in a matte, including the 3D Keyer, MasterK, 2D Histogram, GMask, Edge and Degrain nodes.

NOTE The Degrain node results in a colour image; however, it is often useful to place it on the Matte pipe before the Keyer-3D node (which uses a colour image for its input). For this reason, it can be placed on the Matte pipe.

The Front and Matte pipes of the MBlend node constitute one layer.



(a) One layer (b) Front pipe (c) Matte pipe (d) Blue box of Front pipe (e) Blue box of Matte pipe

The two pipes together, along with a back component, constitute one layer. For the first (bottom) layer, the MBlend node uses a pure black image for the back component.

Working with Blend Nodes

The two pipes of blend nodes provide extra functionality compared to branches in the pipeline:

- You can apply the processes to a selected region of the image. You use the lower of the two pipes to define the area on the image where you want to apply the processes used in the upper pipe. For example, you could add a garbage mask node to the lower branch and create a matte. The processes in the upper pipe are only applied to the foreground subject of the matte. If there are no nodes in the lower pipe, the processes in the upper pipe are applied to the entire image.
- You can blend the two pipes in various ways. Use the menu settings and Blend Curve editor to adjust the blending of the front and back images.
- You can use multiple layers. The two pipes together constitute a layer. You can add one to three additional layers to blend nodes.

Adding a Matte to a Blend Node

Add a matte to the Matte pipe of the CBlend and MBlend nodes to limit the area on the image where the colour correction or matte will be applied.

You can use any matte operation in the Matte pipe—add any combination of nodes that you would want to use to isolate an area of the front image. For example, you could add a garbage mask node to the Matte pipe of the CBlend node.

To add a matte to a blend node:

1 Drag the node to the Matte pipe. Release the cursor.
The node is added to the Matte pipe.

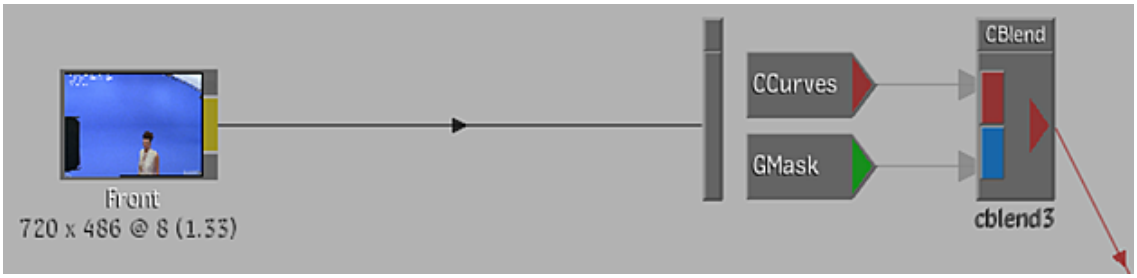


Image courtesy Behavior Communications Inc.

2 Use the node menu to create the matte.

3 Add other nodes you want to use to refine the matte.

Blending the Front and Back Images of a Layer

There are two different methods you can use to blend the front and back images of a layer: Basic Blend and Curve Blend.

Basic Blend is appropriate for layers that do not have a matte component. It allows you to add the front and back images, and adjust the colour and opacity of the result image. In effect, since you are using the entire front image, the back image does not influence the result.

NOTE Basic Blend is only available for MBlend nodes. It would not be useful to blend colour images in a composite in this way.

Curve Blend is suitable for layers that have a matte component. It allows you to use front and back matte curves to adjust the luminance of the front and back mattes, and also perform logical operations on the front and back images.

The two methods are mutually exclusive—you can set the blending in both the Basic Blend and Curve Blend menus, but it is the menu that is displayed that takes effect.

Basic Blend

Use Basic Blend to adjust the opacity and colour of an MBlend layer that has no matte component. This results in adding a second defined layer to the first one. You can control the opacity of the second layer.

NOTE You can also use Basic Blend when you have defined a matte in the Matte pipe. In this case, the blending is applied to the defined matte area only, rather than the entire front image.

To use Basic Blend:

1 Select the MBlend layer by clicking one of the blue boxes to the right of the layer.

The MBlend menu appears and the name of the layer you selected appears in the Layer Name field.

(a) Layer Name field (b) Blend box

By default, if the layer does not have a matte component (that is, there are no nodes in the Matte pipe), the Basic Blend option is selected in the Blend box, and the Basic Blend controls are displayed at the right of the menu.

NOTE If Curves Blend was previously selected, then it is displayed. To display Basic Blend, select it from the Blend box.

2 Display the Result view. This displays the result for the entire blend node.

3 Set the colour of the layer in the Colour field. A value of 100 sets the layer as white, a value of 0 sets it as black, and values in between give it a shade of grey.

4 Set the opacity of the layer in the Opacity field. A value of 100 makes the layer completely opaque, and a value of 0 makes it completely transparent.

Curve Blend

Use Curve Blend to adjust the blending of the front and back images of a CBlend or MBlend node layer that has a matte component.

To use Curves Blend:

1 Select the CBlend or MBlend layer by clicking the red or blue box to the right of the layer.

The CBlend or MBlend menu appears and the name of the layer you selected appears in the Layer Name field.

(a) Layer Name field (b) Blend box (c) Logic Ops box

For MBlend nodes, if the layer has a matte component (that is, there are nodes in the Matte pipe), the Curves Blend option is selected in the Blend box and the Matte Curves graph is displayed at the right of the menu.

NOTE If Basic Blend was previously selected, then it is displayed. To display Curve Blend, select it from the Blend box. The Blend box does not appear for CBlend nodes, since Curve Blend is the only option.

2 Display the Result view. This displays the result for the entire blend node.

3 To perform a logical operation on the front and back images, select one from the Logic Ops box.

4 Use the Blend Curve editor to adjust the front and back matte curves as needed.

Using Several Layers in a Blend Node

You can use up to four layers in blend nodes to create the composite. Using layers, you can avoid doing multiple passes on the same composite. Layers are processed upwards from the bottom layer. In other words, the uppermost layer will appear on top of the next layer, and so on.

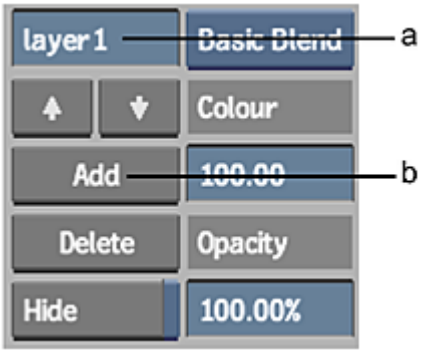
The back component of the lowest layer is a pure black image (for MBlend nodes) and the node input image (for CBlend nodes). For additional layers, however, the back component is different. On both CBlend and MBlend nodes, the result image of the lowest layer is fed into the next highest layer to serve as its back component. Likewise, the result image of this layer is fed into the next highest layer where it serves as that layer's back component, and so on.

NOTE The Current Layer proxy does not display the current layer using the result of the previous layer, as described above. Instead, it displays the layer as if the back clip were a black image (for MBlend nodes) or the front image (for CBlend nodes).

You can change the order of the layers using the Priority Editor.

To create a new layer:

1 Select the blend node by clicking anywhere in the rectangular box at its right end.
The node is highlighted and the Blend Node menu appears.



(a) Layer Name field (b) Add button

2 Click Add to create the new layer.
A new layer is added to the blend node, above the previous layer. The default name for the new layer is layer2.

3 Add nodes to the layer as needed.

4 To change the order of the layers, click the

button to move the selected layer up a layer, or the

button to move it down a layer.

To rename a layer:

1 Select the layer by clicking on the red or blue box to the right of the layer.

2 Click the Layer Name field in the blend node menu.

The on-screen keyboard appears.

3 Press `Esc` to remove the previous name and type a new name.

4 Press `Enter` to save the name.

Bypassing Pipes

On blend nodes, you have the option of inputting a source image from another node on the pipeline, effectively bypassing the pipes (for example, using the key-in clip matte in a colour correction in the Matte pipe of the CBlend node). Instead of copying all the nodes in the Front pipe of the MBlend node, you could just add a branch from the end of the MBlend Front pipe to the blue box on the CBlend node.

When attaching an alternate input source to a Matte pipe, you should use a matte image. If you use a colour image, it will be converted to greyscale.

NOTE To do this, you cannot have any nodes on the pipe receiving an alternate input source.

Animating Key Elements

In a live-action clip, you can animate the tolerance, softness, and patch ranges as the subject in the clip moves. You can also animate the colour, opacity, and softness of patches.

Animating the Range of Key Elements

You animate the tolerance, softness, and patch ranges by changing the range at different frames. In the Channel Editor, range changes appear in the Shape channel. You can set a Shape keyframe by:

- Adding or subtracting pixel values from the range by sampling again in the image.
- Adjusting the key elements in the RGB viewer. All transformations made to a key element are applied to its Shape keyframe, including translation, rotation, scaling, and noise analysis.
- Changing the values in the Softness Scaling X, Y, and Z fields.

If you display the convex hull while animating a key element, the convex hull fills the ellipsoid in between keyframes. The sample is recreated to fill the ellipsoid on interpolated frames. This ensures the smoothest possible transition between keyframes.

To display the Channel Editor, click the Animation button.

NOTE To animate key elements, enable Auto Key.

Shape Keyframes

The shape of ellipsoids and patches is defined by their translation, rotation, and scaling values, so the Shape keyframe is actually composed of multiple parameters. The Y value in the Channel Editor does not have the same significance as it does for a single parameter (for example, opacity). Instead, the Y value is composed of sequential numbers that represent keyframes that have been set. Each consecutive keyframe is assigned a sequential Y value: the first keyframe has a Y value of 1, the second has a Y value of 2, and so on. Although the Y value does not represent a single value, you can still adjust the curve to tweak the shape.

To animate the range of a key element:

1 Perfect the key at frame 1.

2 Enable Auto Key.

3 Scroll to other frames and, where necessary, adjust the range using any of the techniques provided in this chapter.

Keyframes are added at each frame where you change the tolerance, softness, or patch range.

NOTE When you set a tolerance keyframe, a softness keyframe is also set. This is because the softness range is always at least as large as the tolerance range. When you set the tolerance range, the 3D Keyer automatically creates a minimum softness range that is equivalent to the tolerance range. When you adjust the tolerance, the softness range is also adjusted to accommodate the change.

4 Optionally, tweak the animation by adjusting the shape curve in the Channel Editor. The image updates as you make changes.

Changing the Interpolation

Parameter values in between keyframes are interpolated. Change the type of interpolation using the Channel Editor.

The default interpolation for the shape curve is Bézier, which creates a smooth transition between keyframes. If your clip has a very sudden change in the colour values (for example, if a light was switched on at a particular frame), use Constant interpolation between the two keyframes where the change occurs.

Animating the Patch Parameters

Animate the colour, softness, and opacity of patches by changing the values in the Patches menu. Channels for each of these parameters are located in the Patch folders of the Channel Editor.

To animate the patch parameters:

1 Go to the appropriate frame.
2 Enable Auto Key.
3 Change the values in the Colour, Opacity or Softness fields as needed.

Resetting a Channel

There are two ways to reset a channel:
- Using the Reset button under he Sampling box.
- Using the Rst Chn button in the Animation controls.

Mastering the RGB Viewer

The RGB viewer gives you a graphic representation of what you are doing in the key. In this section, learn how to:
- Move, rotate, and zoom in on the RGB viewer.
- Change the display of the 3D histogram and canvas.
- Change the appearance of elements in the RGB viewer.
- Select key elements in the RGB viewer.
- Plot pixels in the image and in the RGB viewer.
- Translate, rotate, and scale key elements in the RGB viewer.

■ Reshape the convex hull by manipulating its vertices.

Moving, Rotating, and Zooming in on the RGB Viewer

The RGB viewer represents the 3D colour space of a key-in clip. As you work on the precision of a key, you will want to change the position, orientation, and size of the RGB viewer. Use `Alt`, `Shift`, and `Ctrl` to move, rotate, and zoom in on the RGB viewer.

To:	Do this:
Move the viewer	`Alt`-click the viewer and drag it.
Zoom in on the viewer	`Shift`-click the viewer and move the cursor to the left or right.
Rotate the viewer	`Ctrl`-click the viewer and move the cursor in any direction. By default this rotates the viewer around its centre. Using Rotation Axis options in the Preferences menu, you can also rotate the viewer around a plotted point, or around the centre of the tolerance or softness ellipsoid. This makes it easier to zoom in on specific elements in the viewer.

Resetting the RGB Viewer

To reset the viewer's position, orientation, and size, click Home in the lower-left corner of the Key, Patches, or Preferences menu.

NOTE Other settings, such as resolution or canvas light, are not reset.

Setting the Display of the RGB Viewer

As you work with sampled elements in the RGB viewer, set the display of the 3D histogram, canvas background, and border lines to suit the operation you are performing. For example, when modifying the softness range, consider hiding the 3D histogram to see the softness ellipsoid more clearly.

Setting the 3D Histogram Display

The 3D histogram provides a visual representation of the colours in the key-in clip. It shows how the colours in your clip are distributed in RGB colour space, with exposure and contrast settings taken into account.

The most prevalent colours are represented by larger cubes; colours that are less prevalent are represented by smaller cubes.

NOTE The size of the cubes approximates the colour usage in the image, but it is not intended as an exact representation. You can increase the resolution of the cubes using the histogram grid.

When a floating-point image is used as a key-in clip, the RGB Viewer will support this input and also activate options to change the display of colour space in the 3D histogram. These options are available in the Histogram panel of the Key menu. When the range in the histogram exceeds 1, a cube outline indicating the colourspace from 0 to 1 is displayed in the RGB viewer.

Frame option box Select an option to display the entire histogram, or a selected range of values.

Select:	**To display:**
All Objects `(Shift+A)`	All objects in the 3D histogram. This is the default setting.
(0,1) `(Shift+0)`	Objects with colour values between 0 and 1.
Free `(Shift+F)`	Objects within a user-defined range. Use the Range field to navigate between views including the full range view and a percentage of the range.
Plot `(Shift+O)`	Objects with colour values between a range that comprises all values between 0 and 1, and the plot value.

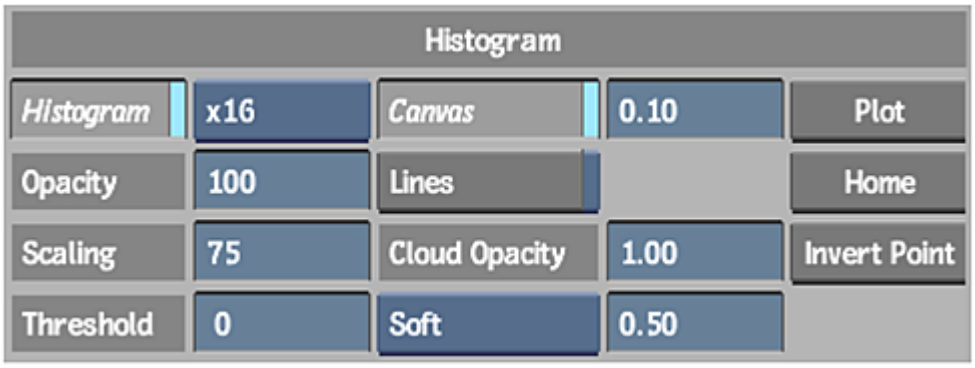
Range field Displays the percentage of the full range that is displayed in the histogram. Editable when the Frame option box is set to Free.

Store button Click to store the current value in the Range field in memory for later use. Active when the Frame option box is set to Free.

Recall button Click to retrieve the last Range value that was stored in the buffer.

Change other display elements of the histogram using the Preferences menu. To access the Preferences menu, click Pref.

Histogram Enable to show or hide the histogram. Alternatively, use the H keyboard shortcut.



Histogram grid Enable to displays the resolution for the cubes:

- x16 (the default)
- x8 displays fewer and larger cubes
- x32 displays smaller cubes

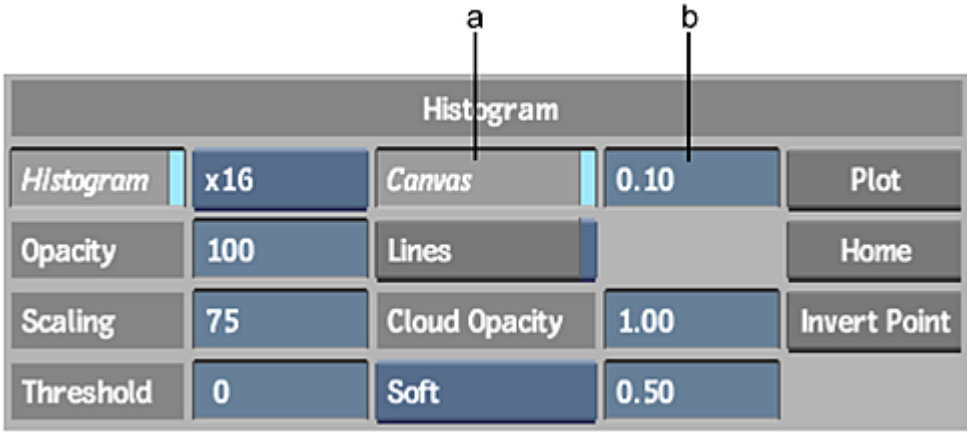
Opacity field Displays the opacity of the cubes. As you adjust the opacity, smaller cubes become transparent at a faster rate than larger ones. The default setting is 100% opacity.

Scaling field Resizes the cubes. Cubes representing less prevalent colours (the smaller cubes) are scaled down more than those representing more prevalent colours. The cube of the most prevalent colour remains the same size (as set in the Histogram Grid box). By increasing the size difference between the colours of greater and lesser prevalence, you can see the main components of the colour space more clearly.

Threshold field Removes the display of colours that are less prevalent in the image. As you increase the threshold, less prevalent colours are increasingly removed from the histogram.

Setting the Canvas Display

The canvas is the black background of the RGB viewer. Control the display of the canvas using the Preferences menu.



(a) Canvas button (b) Canvas Light field

Canvas button Enable to show or hide the canvas. Alternatively, use the C keyboard shortcut.

Canvas Light field Displays the amount of lighting in the RGB viewer. The light source emanates from behind the viewer.

Setting the Line Display

You can show or hide a white outline of the borders of the RGB viewer, as well as green wireframe boxes around selected ellipsoids. Show the boxes to quickly identify the selected element, or hide them to reduce "clutter" in the viewer.

To show or hide lines in the RGB viewer:

1 Enable or disable the Lines button in the Setup menu.

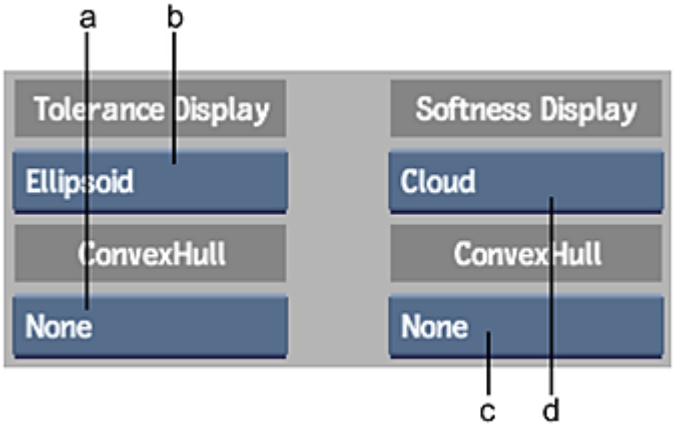
Changing the Display of Key Elements

You can change the display of the tolerance ellipsoid, softness ellipsoid, and patches as you work on the key to suit the operation you are performing. For example, to adjust the tolerance range after setting the softness, you could set the softness display to wireframe, or remove its display altogether, for a better view of the tolerance. See Setting the Tolerance (page 994) and Setting the Softness (page 997).

Setting the Tolerance and Softness Ellipsoids

As you sample the tolerance and softness in a key, corresponding key elements appear in the RGB viewer. Set the appearance of the tolerance and softness ellipsoids to make it easier to work with them while refining the precision of the key.

Use the Key menu to set the display of the tolerance and softness in the RGB viewer.



(a) Tolerance Convex Hull box (b) Tolerance Display box (c) Softness Convex Hull box (d) Softness Display box

Tolerance and Softness Display boxes Select an option to set the appearance of the tolerance and softness ellipsoids.

Select:	To display:
Ellipsoid	An opaque ellipsoid of the exact colours sampled.
Cloud	A semi-transparent grey ellipsoid. You can set the transparency of the ellipsoid using the Cloud Opacity field in the Preferences menu. See Setting the Cloud Opacity (page 1018).
Wireframe	The ellipsoid as a wireframe outline

Select:	To display:
None	No ellipsoid. Note that when an ellipsoid is not displayed, it still has an effect on the image

Tolerance and Softness Convex Hull boxes Select an option to set the appearance of the tolerance and softness convex hulls.

Select:	To display:
Vertices	Only the vertices of the convex hull.
Surface	Only the surface of the convex hull.
Vert&Surf	Both the surface and vertices of the convex hull.
None	No convex hull.

NOTE Vertices do not appear on keyed elements when Tolerance Display or Softness Display is set to Ellipsoid (because an ellipsoid is opaque).

Changing the Patches Display

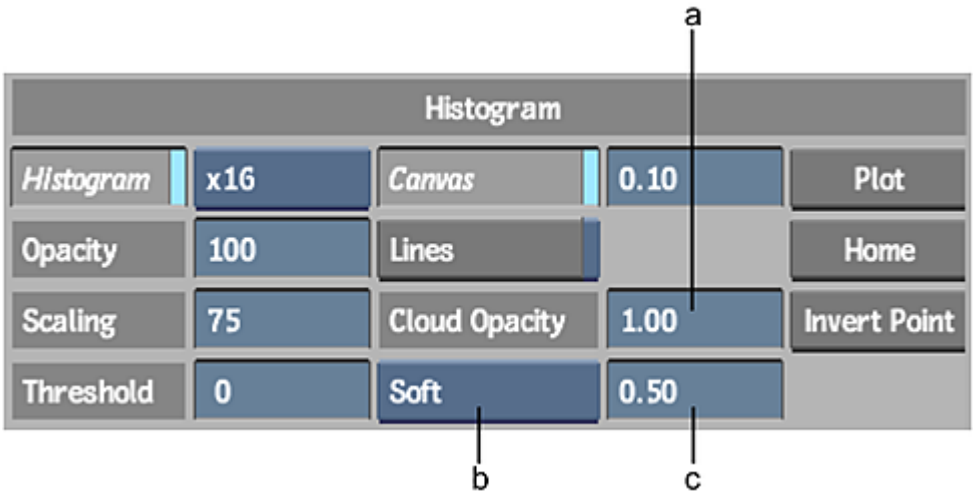
You set the display of each of the three patches individually. In the Patches menu, use the Display boxes to set the display of patches 1, 2 and 3.

Display box Select an option to set the appearance of the current patch.

Select:	To display:
Box	The patch as a wireframe box.
Surface	The convex hull of the patch.
Box&Surf	Both the wireframe box and convex hull of the patch.
Cloud	The patch as a semi-transparent grey box. You can set the transparency of the box using the Cloud Opacity field in the Preferences menu. See Setting the Cloud Opacity (page 1018).
None	No patch. When a patch is not displayed, it still has an effect on the image.

Setting the Cloud Opacity

You can control the opacity of the Cloud setting of ellipsoids and patches as they appear in the RGB viewer using the Cloud Opacity fields in the Preferences menu.



(a) Overall Opacity field (b) Key Element box (c) Individual Opacity field

You can adjust the opacity for all key elements or for individual key elements:

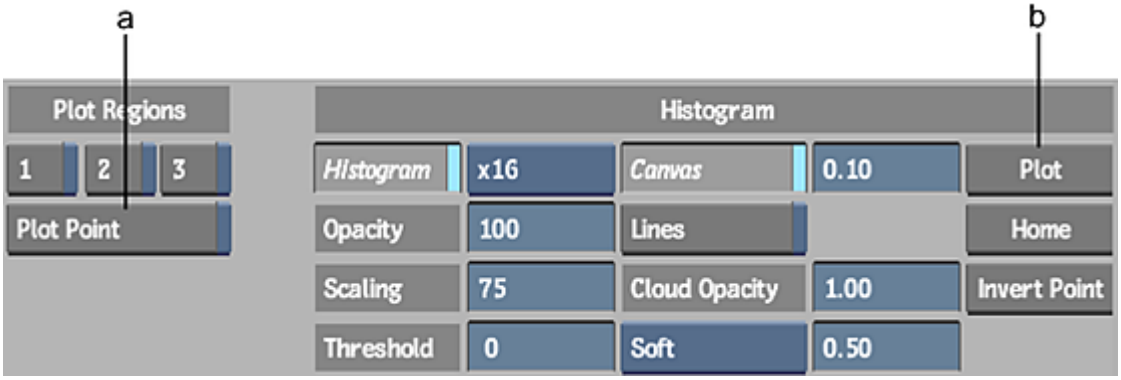
- To set the cloud opacity for all ellipses and patches, use the Overall Opacity field.
- To set the cloud opacity for an individual key element, select the element from the Key Element box, then set the opacity in the Individual Opacity field.

Displaying Plotted Points and Regions

When you plot colour values and ranges of colour values in the key image, you can display or hide the plotted points and regions in the RGB viewer.

To show or hide plotted points:

1 Enable or disable the Plot Point button in the Preferences menu.



(a) Plot Point button (b) Plot button

NOTE When you plot a colour value, the Plot Point button is automatically enabled to display the colour value in the viewer.

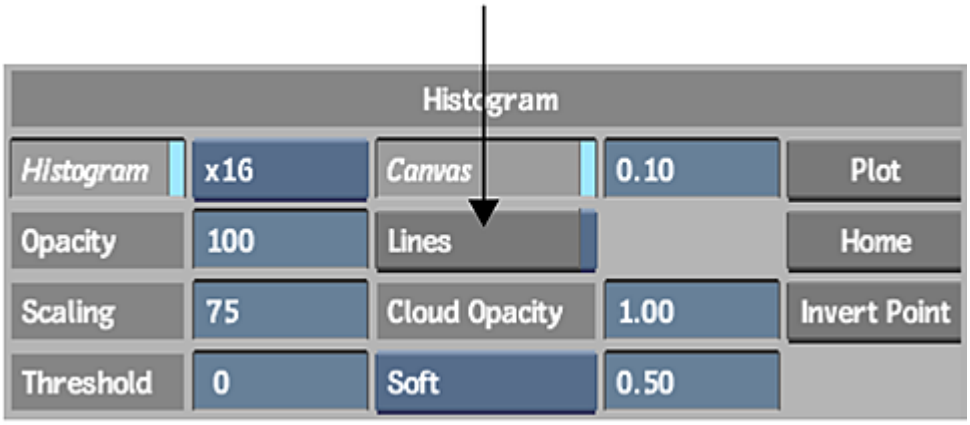
To show or hide Plot Regions 1, 2, or 3:

1 Enable or disable the corresponding buttons in the Preferences menu.

Selecting Key Elements in the RGB Viewer

As you translate, scale, and rotate key elements in the RGB viewer, you must select them individually. The 3D Keyer provides two methods for selecting key elements—clicking on them in the viewer or using the Picking box. The Picking box enables precise selection especially when tolerance, softness, and patch elements overlap each other in the RGB viewer.

The Preferences menu contains the Lines button. Use this button to display lines around key elements when they are selected in the viewer. This can make it easier to identify the selected key element.



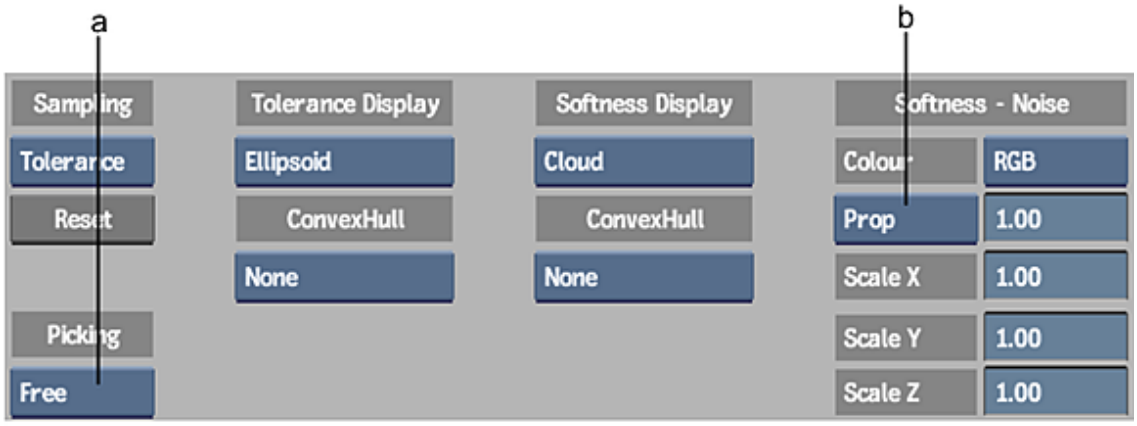
Clicking Key Elements

When you click key elements in the RGB viewer, a directional arrow appears through the selected element. The arrow points in a particular direction according to where you click on the element. See Controlling Elements with the Directional Arrow (page 1022).

The Picking box must be set to Free to select an element by clicking it. Also, the Softness Scaling must be set to Prop not Minimize Noise (Minimize Noise selects softness and controls the arrow direction).

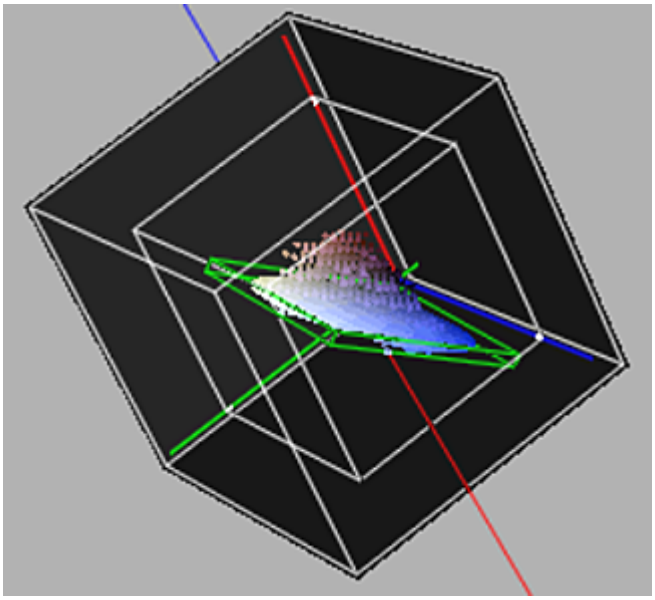
To select key elements by clicking:

1. In the Key menu, select Free from the Picking box and make sure that Prop is selected in the Softness Scaling box.



(a) Softness Scaling box (b) Picking box

2 Select a key element in the RGB viewer by clicking it.



A directional arrow appears through the selected key element and, if the Lines button is enabled, a green wireframe box appears around the selected key element.

3 At this point, you are ready to proceed with modifying the selected key element. See Translating, Scaling, and Rotating Key Elements (page 1027) and Reshaping the Convex Hull (page 1028).

If you are unable to select the element, the element may be behind or inside another element. You can hide the element that is in the way by setting its display to None. See Changing the Display of Key Elements (page 1017). Alternatively, use the Picking box to select the element, as described in the next procedure.

Using the Picking Box

When an element is selected with the Picking box, you cannot select other elements in the viewer by clicking on them. This helps prevent other key elements from being selected by accident while you work with a particular key element.

The Picking box is also useful because it allows you to select a key element without changing its current arrow position. For example, if you accidentally click elsewhere in the viewer and lose the selection, and you want to re-select the element without changing the arrow position, use the Picking box.

NOTE When you use the Picking box to select a key element that has not been previously selected, the arrow is not displayed. You must actually click the element to set the arrow direction.

To select key elements using the Picking box:

1. In the Key menu, verify that the Softness Scaling is set to Prop (Minimize Noise forces the softness to be selected).
2. Select the key element from the Picking box.

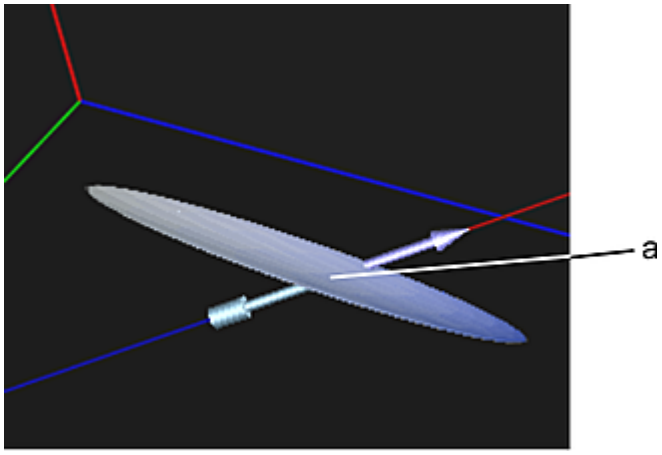
   If lines are displayed in the viewer, a selected ellipsoid has a green wireframe box around it and patch boxes change from white to green. If the element's arrow direction has been set, the arrow is displayed.
3. To display the element's arrow and set its direction, click the element.

   At this point, you are ready to proceed with adjusting the range of the selected key element.

Controlling Elements with the Directional Arrow

Each key element (tolerance, softness, and patches) has a directional arrow that passes through its centre point. Use this arrow to indicate the direction in which you want to perform a particular action, such as scaling or translating the element.

- To display an element's arrow, select the element by clicking on it.
- To set the arrow direction manually, click inside the key element anywhere between its centre and the area towards which you want the arrow to point.



(a) Click in the element to set the arrow direction.

NOTE You can only click on a part of the element's surface that is visible. If necessary, rotate the RGB viewer so that the correct area of the surface is visible.

- To point the arrow towards a plotted point, simply select the element (either by clicking on it or using the Picking box) before plotting. The arrow will automatically point towards the plotted point.
- To reverse the direction of the arrow, click the Reverse Point button in the Key menu.

NOTE When Minimize Noise is enabled, this tool controls the arrow direction. See Removing Noise from Softened Areas (page 999).

Plotting Colour Values

You can plot pixels in the key image and view the location of the pixel colour value in the RGB viewer. You can plot a single colour value or a range of colour values. These plotted areas do not affect the key at all—they simply identify colour values in the viewer.

You can plot a colour value to view its location in relation to key elements in the viewer. Additionally, you can employ a number of techniques to modify a key element in relation to a plotted colour value or range of values. For example, you could plot a colour value corresponding to an unwanted grey area in the foreground subject, and then translate or scale the softness ellipsoid such that the colour value is outside the ellipsoid.

Plotting Single Colour Values

Plotting a pixel in the key image sets the exact location of the colour value in the RGB viewer. The colour value is identified by a small, white wireframe cube in the viewer. As an alternative, you can use the Plot tool to select a colour value directly in the viewer by clicking on a 3D histogram cube.

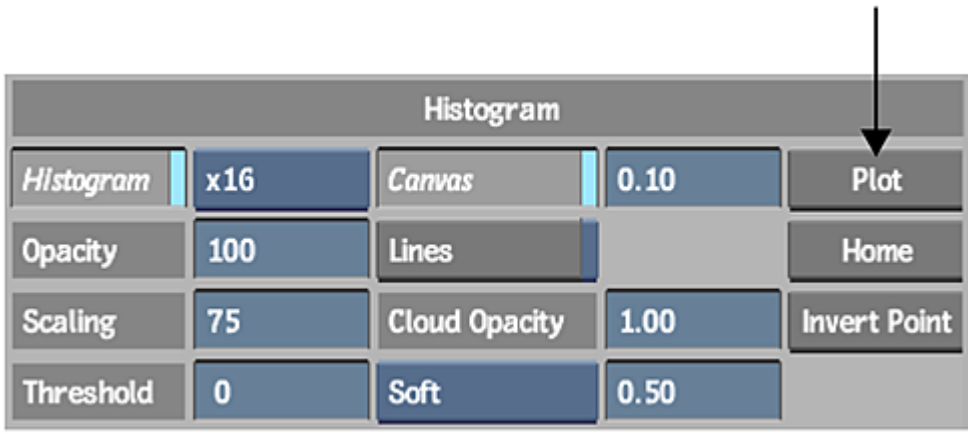
If a key element in the viewer is selected before you plot the colour value, the element's directional arrow moves to point towards that colour. Plotting pixels in this way allows you to quickly refine the matte by translating or scaling a key element with respect to the plotted colour.

To plot a pixel on the key image:

1. In the RGB viewer, select the key element that you want to modify in relation to a plotted colour value. For example, click a patch key element.

   If you have problems selecting the element, see Selecting Key Elements in the RGB Viewer (page 1020).
2. If needed, zoom in on the key image so that you can select a pixel more easily.

3 Click Plot.



Plot is enabled.

4 Click a pixel in the key image. You can also drag the cursor in the image, then release the cursor when the colour picker is over the desired pixel.

In the RGB viewer, the colour value of the selected pixel is plotted and the arrow of the selected key element moves to point towards the plotted colour value.

If needed, rotate the viewer by pressing `Ctrl` and dragging it to get a better view of the key element with respect to the plotted value.

5 Proceed with modifying the selected key element. Using the keyboard shortcuts, you can move the selected key element to refine the key.

Press:	To:
`V`-click	Sample pixels where you click in the image. The arrow is repositioned in the RGB viewer. Drag left or right to scale along this axis.
`6` and drag key element	Move the selected key element in the direction of the arrow.
`7` and drag key element	Rotate the selected key element in any direction around its centre point.
`8` and drag key element	Scale the selected key element proportionally from its centre. Drag left to scale down and right to scale up.
`9` and drag key element	Scale the selected key element in the direction of the arrow (non-proportionally) from its centre. Drag left to scale down and right to scale up.
`0` and drag key element	Scale the selected key element in the direction of the arrow (non-proportionally), starting from the point on the key element's surface where the back of the arrow remains anchored in place.

As you move the key element away from the plotted pixel, you can see the effect on the key image or the matte.

To select a colour value in the RGB viewer:

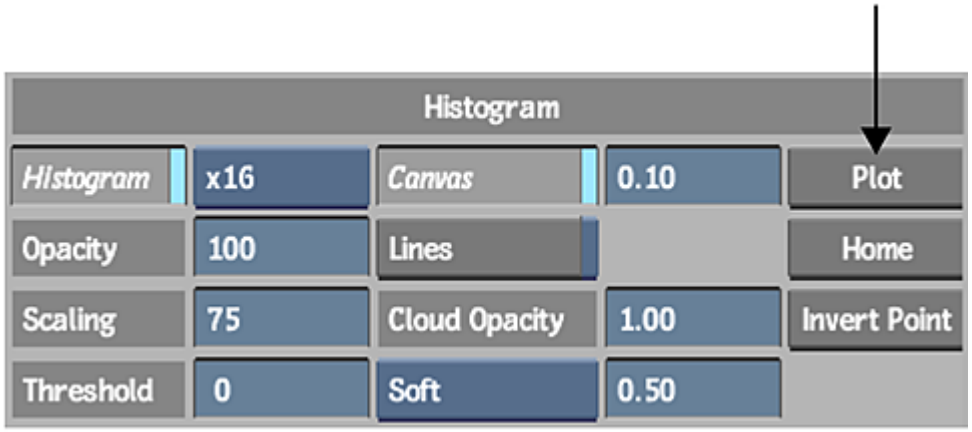
1 In the RGB viewer, select the key element that you want to modify in relation to a plotted colour value. For example, click a patch key element.

If you have problems selecting the element, see Selecting Key Elements in the RGB Viewer (page 1020).

2 Press H to display the 3D histogram.

3 If needed, zoom in on the viewer or rotate it so that you can select a histogram cube more easily.

4 Click Plot.



Plot mode is activated and the cursor changes to a colour picker.

5 Select a cube in the histogram of the RGB viewer. If needed, drag the colour picker around the viewer until it is pointing to the correct colour value.

The selected cube is surrounded by a white wireframe box and the direction of the key element arrow moves toward the selected cube.

Plotting a Range of Colour Values

When you plot a range of colour values in the image, the resulting plotted region in the RGB viewer appears as a 3D convex hull. You can plot up to three colour ranges in the image. The plotted regions do not in themselves affect the key image in any way.

A plotted region allows you to compare the intersection of a key element with a particular colour range in the image, providing you with yet another alternative for adding precision to your key.

For example, use this tool to plot a colour range that you do not want to be softened. Using the plotting region as the reference, you translate and scale the softness ellipsoid away from the plotted region to ensure the softness precision of the key. Plotted regions also help you to determine whether or not to use a garbage mask. If a colour that you want to soften is the exact colour that you want key out, then you should apply a garbage mask.

To plot a region:

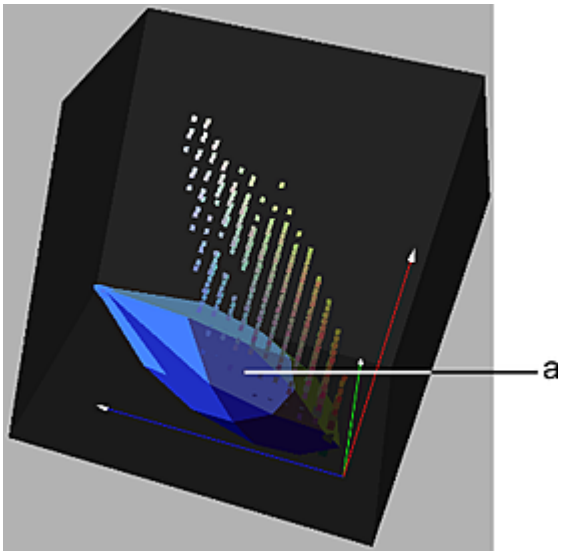
1 In the Preferences menu, enable the Plot Regions 1 button.

The display of Plot region 1 is enabled.

2 In the Sampling box, select Plot1.

3 In the key image, sample one or more pixels using the same methods that you use for sampling tolerance or softness by dragging in the image.

A 3D convex hull appears in the RGB viewer.



(a) Each vertex contributes to the convex hull that represents the plotted region.

4 Continue sampling pixels in the key image until you are finished plotting the region.

Plotting Several Regions You can plot a second and third region, and compare the positions of the convex hulls. To plot a second region, use the procedure described above, selecting Plot 2 from the Sampling box and enabling the Plot Regions 2 button.

For example, sample one plot region in an area of the matte containing unwanted grey areas, and another plot region where the grey area is intentional. You can then observe where the colours of the two regions are the same (where the plotted regions intersect), and adjust the softness accordingly.

To:	Do this:
Remove the grey areas from the foreground subject.	Move the softness key element away from the plotted region intersection.

To:	Do this:
Prevent the matte from becoming over softened or transparent.	Use a patch.
Keep part of the key that has the exact colour as the area being softened.	Use a garbage mask.

Translating, Scaling, and Rotating Key Elements

You can fine-tune the key by translating, scaling, or rotating the tolerance ellipsoid, softness ellipsoid, and patches in the RGB viewer. For example, scale the softness ellipsoid in a particular direction to increase the softness. You use the same methods for all three key element types.

To translate or scale an element in a particular direction, use its arrow to specify the direction. When scaling an element along the axis of the arrow, you can scale it equally in both directions from its centre, or only in the forward direction of the arrow. You can also scale an element proportionally in all directions (X, Y and Z directions).

First set up the RGB viewer to best display the elements you are working with. Zoom in on the element, decide if you want to show the histogram, and hide elements that are in the way. See Changing the Display of Key Elements (page 1017).

TIP Display the 3D histogram as you make modifications to see the areas of colour that you are working in. Try reducing the size and opacity of histogram cubes to see more clearly. See Setting the 3D Histogram Display (page 1014)

To modify the tolerance or softness using these techniques, the ellipsoid must be displayed. The Tolerance Display or Softness Display must be set to Ellipsoid, Wireframe, or Cloud. Likewise, when reshaping patches, the patch box must be displayed. The box is displayed when the Display is set to Cloud, Box, or Box&Surf.

See More Keying Techniques (page 1030) to learn ways to apply these techniques in specific situations.

To translate a key element:

1. Select the element. See Selecting Key Elements in the RGB Viewer (page 1020).
2. Identify the direction in which you want to translate the element and set the arrow in this direction. You can do this manually. See Controlling Elements with the Directional Arrow (page 1022).
3. Alternatively, you can also plot a pixel on the image or a histogram cube (see Plotting Single Colour Values (page 1023)). When you plot a pixel on the image or a histogram cube, the arrow of the selected key element automatically points towards the plotted pixel.
4. Press 6 and drag the cursor in the RGB viewer. Drag to the right to translate the element forward along the arrow axis, or to the left to translate it backward.

To rotate a key element:

1. Select the element. See Selecting Key Elements in the RGB Viewer (page 1020).
2. Press 7 and drag the cursor in the key element.

   The key element rotates in any direction around its centre point.

To scale a key element proportionally:

1. Select the element. See Selecting Key Elements in the RGB Viewer (page 1020).

2 Press 8 on the keyboard and drag in the RGB viewer. Drag to the right to increase its size, or to the left to make it smaller.

To scale a key element from its centre:

1 Select the element. See Selecting Key Elements in the RGB Viewer (page 1020).

2 Set the arrow in the desired direction. You can do this manually. See Controlling Elements with the Directional Arrow (page 1022).

You can also set the arrow by plotting a point on either the image or a histogram cube. When you plot a point on the image or a histogram cube, the arrow of the selected key element automatically points towards the plotted pixel. See More Keying Techniques (page 1030).

3 Press 9 and drag the cursor in the RGB viewer. Drag to the right to scale it larger along the axis of the arrow, and to the left to scale it smaller.

The element is scaled equally in the forward and backward directions of the arrow.

To scale a key element in one arrow direction:

1 Select the element. See Selecting Key Elements in the RGB Viewer (page 1020).

2 Set the arrow in the desired direction. You can do this manually. See Controlling Elements with the Directional Arrow (page 1022).

3 You can also set the arrow by plotting a point on either the image or a histogram cube. When you plot a point on the image or a histogram cube, the arrow of the selected key element automatically points towards the plotted pixel. See More Keying Techniques (page 1030).

4 Press 0 and drag the cursor in the RGB viewer. Drag to the right to scale it larger in the forward direction of the arrow, or to the left to scale it smaller towards the centre of the element.

The element is scaled in the forward direction of the arrow only.

Reshaping the Convex Hull

You can exclude or add specific ranges of colour to a sample by modifying the shape of the convex hull of the softness ellipsoid, tolerance ellipsoid, and patches. You do this by manipulating the vertices on the hull. The ellipsoid or patch box changes shape according to the new shape of the convex hull.

This can be an accurate way of modifying the sampled ranges since you can see the area of RGB colour space in which you are working and because you can plot specific parts of the image and add them to the hull. Also, scaling the ellipsoid using one vertex of the convex hull provides a more precise (localized) result than scaling using the arrow alone (using 0).

To change the shape of the convex hull, you can add and move vertices, or delete vertices from the convex hull entirely.

Before performing the following operations, set up the RGB viewer to best display the elements you are working with. Zoom in on the element, decide if you want to show the histogram, and hide elements that are in the way.

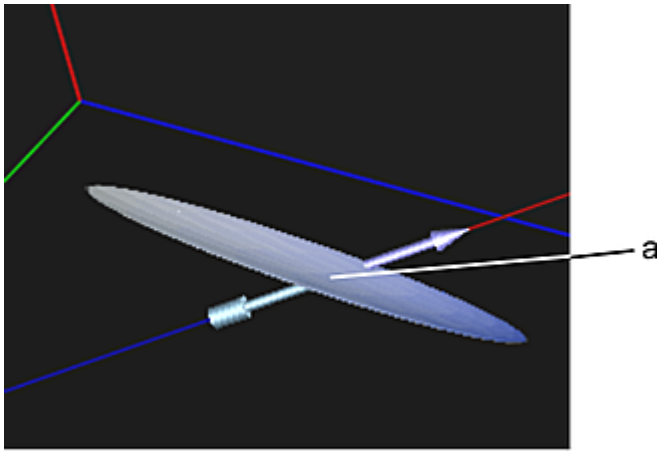
To reshape the convex hull of the tolerance or softness, the ellipsoid must be displayed. It must be set to Ellipsoid, Wireframe, or Cloud. Likewise, when reshaping patches, the patch box must be displayed. The box is displayed when the Display is set to Cloud, Box, or Box&Surf.

It is best to show the convex hull. Although it is not necessary to perform the operation—unless you are deleting vertices, you can see what you are doing more clearly. See previous sections in this chapter, such as Changing the Display of Key Elements (page 1017).

To add vertices to the hull and move them:

1. Select the key element by clicking it.
2. Position the arrow of the selected element in the direction in which you want the ellipsoid or patch to grow or shrink. For example, point the arrow towards colours you want to include in your sample. Alternatively, plot a point to position the arrow. See Controlling Elements with the Directional Arrow (page 1022).
3. Press and hold V on the keyboard and click anywhere in the RGB viewer.

   A vertex is added to the hull at the intersection of the arrow and the edge of the ellipsoid or patch.



(a) Vertex is added to the hull

NOTE When using V in the RGB viewer, you can scale any convex hull (tolerance, softness or patches). If you use V in the image window, it will apply to softness only.

4. To add colours to the sample, drag the cursor to the right. This moves the vertex in the forward direction of the arrow.

   To remove colours from the sample, drag the cursor to the left. This moves the vertex backward towards the centre of the element.

   When you are satisfied with the result in the image window, release the cursor.

   The hull is reshaped as you move the vertex. If you did not display the convex hull, it is temporarily displayed while you are moving the vertex. The direction of the arrow may change slightly as you move the vertex, but the direction it moves in is always the direction in which the arrow was initially pointing.

   NOTE You can release the V key and the cursor, then press them again to scale the same vertex. You retain control over the new vertex until you change the position of the arrow.

To remove a vertex from the convex hull:

1. Display the convex hull of the key element, including the vertices. See Changing the Display of Key Elements (page 1017).

   NOTE You do not have to select a key element to delete a vertex from its convex hull.

2. If necessary, zoom in on the element to distinguish the vertices more clearly.
3. Press and hold the - sign on the keypad (not the one on the keyboard) and click on the vertex that you want to remove.

   See Techniques for Adjusting Softness (page 1030).

More Keying Techniques

In this section, learn how to adjust the key using various methods to manipulate the key elements (tolerance, softness, and patches) in the RGB viewer, as well as the Softness Scaling controls in the Key menu.

It is recommended to read Creating a Precise Key in the 3D Keyer (page 993) and Mastering the RGB Viewer (page 1013) before proceeding.

Techniques for Adjusting Softness

This section includes three additional methods for adjusting the softness of the key.

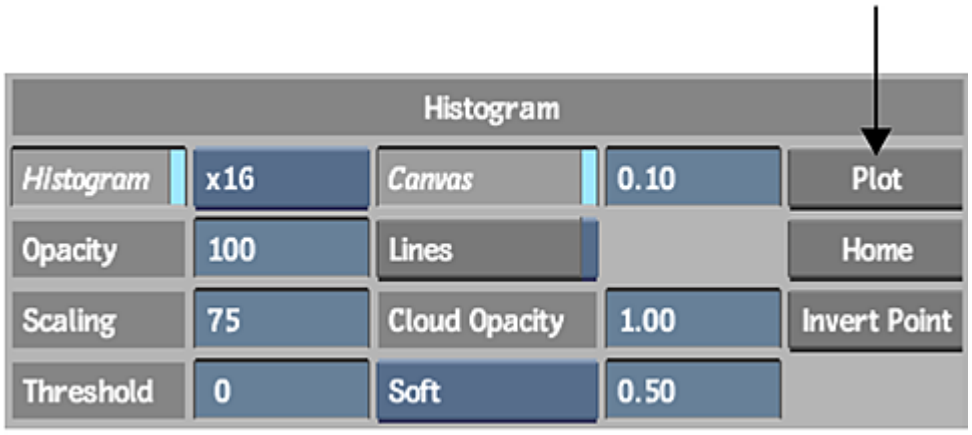
Scaling the Softness Ellipsoid to Remove Grey Areas

Increasing the softness may create unwanted grey areas in the foreground subject. There are several ways to remove these grey areas. One method is to plot a pixel in a grey area, then translate, scale, or rotate the softness ellipsoid such that the plotted region is no longer in the ellipsoid. In effect, you will no longer be softening the plotted region.

If you translate, rotate, or proportionally scale the softness ellipsoid, all areas of the ellipsoid are affected to a degree. By contrast, when you use vertex scaling, you can affect a more localized area of the ellipsoid, leaving the rest of the ellipsoid unchanged. Vertex scaling is therefore often the most accurate method to use.

To scale the softness ellipsoid:

1. Show the canvas (press C) and hide the histogram (press H).
2. Use the Soft Display box to change the display of the softness ellipsoid to Cloud.
3. Select the softness ellipsoid in the RGB viewer by clicking on it or selecting Softness from the Picking box.
4. From the View box, select Keyer 3D Result to see the matte.
5. Zoom in to see the grey pixels up close.
6. Click Pref.
7. In the Preferences menu, click Plot.



8. Plot a grey pixel in the foreground subject by clicking it.

   The colour value of the pixel you clicked is plotted in the RGB viewer. The softness ellipsoid arrow passes through the plotted point.

9 To view the exact location of the plotted colour value in relation to the softness ellipsoid, rotate and scale the viewer.

The plotted point inside the softened region is represented by the softness ellipsoid.

10 Press V, click in the viewer and drag the cursor to the left to scale the ellipsoid down along the axis of the arrow. Watch the image update interactively. As soon as the grey area in the subject disappears, release the cursor to stop scaling the ellipsoid.

NOTE If you are not satisfied with the result, click Undo to return the ellipsoid to its previous shape, then scale again.

11 If some grey areas remain, plot those areas and repeat the procedure.

Scaling the Softness Ellipsoid to Add to Softness

You can also use the technique described in the previous section to increase the softness at the edges of the key. Start by identifying the region where you want to introduce more softness, and then plot a pixel in that area. You see the colour value in the viewer. Press the V keyboard shortcut, then either enlarge or decrease the size of the softness ellipsoid minimally by dragging the cursor to the right or left. Include the colours you want and exclude the rest.

TIP Display the 3D histogram to help you figure out the direction in which you want to scale the tolerance or softness.

Using the Softness X, Y, and Z Fields

You can also adjust the softness in the key by changing the values in the Softness Scaling X, Y and Z fields. Drag the cursor over the fields, watching the result in the image window. You can often get a good result just by experimenting.

NOTE The Sampling box does not have to be set to Softness to use these fields.

Adjusting Transparencies with Patches

When keying transparencies, the transparency is sometimes either not visible enough or too prominent in the final composite. You can use patches to lighten or darken transparencies.

To adjust a transparency using a patch:

1. Use the basic keying technique to get the transparency as soft as possible.
2. Sample a patch in the transparency. See Adding a Patch (page 1003).
3. View the final composite by displaying the CurResult view.
4. Do one of the following:
   - To lighten the transparency, set the Colour of the patch to 0 and the Opacity to 100, then lower the Opacity.
   - To darken the transparency, set the Colour of the patch to 100 and the Opacity to 0, then raise the Opacity.

Setting 3D Keyer Preferences

The following section explains the available preferences for working in the 3D Keyer.

Auto Key

Enable this button to set keyframes to animate the key. See Animating Key Elements (page 1012) for details.

Regen

When you sample (softness, tolerance, and so on) using the colour picker or the numeric fields, you have the option of having the image update interactively as you move the cursor, or having the image update only when you release the cursor.

It is useful to update the image only after sampling if interaction during sampling is slow. This can happen if you are working with high resolution images, or if you are sampling while viewing a context from further along the pipeline. This can also occur in the current result view (displayed as <Currently Selected Node> Result, in the View box), where a large amount of processing is required to display the result.

To update the image interactively:

1. Enable Regen.

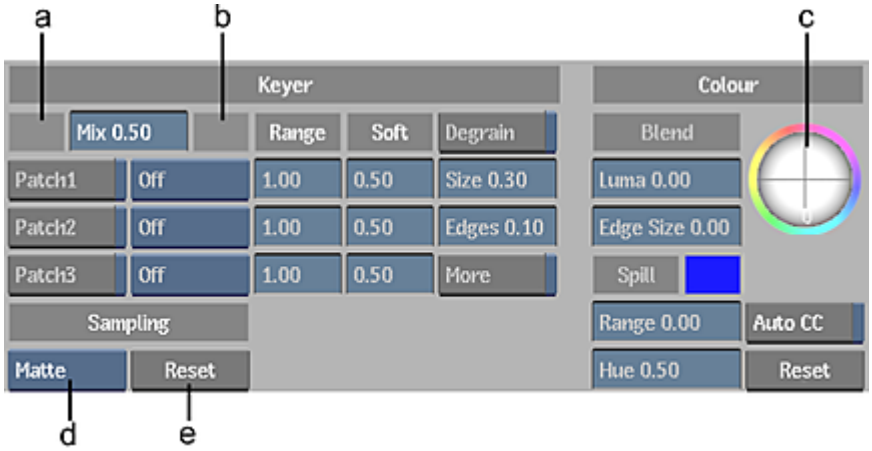
To update the image after completing the sample:

1. Disable Regen.

Creating and Refining a Key in the Master Keyer

Use the Master Keyer method to create a complete key—extract colours to generate a matte and then refine the result—using only the Master Keyer menu. The Master Keyer uses an algorithm that quickly isolates a colour and is very good for chroma keying.

With the Master Keyer, you can refine the key by gesturally modifying the matte, removing colour spill, blending edges, applying patches, and removing grain.



(a) Primary Sample colour pot (b) Secondary Sample colour pot (c) Edge Balance trackball (d) Sampling box (e) Reset button

The following procedure is a recommended workflow for keying a clip with the Master Keyer. You may not need to complete all the procedures. You may also revisit procedures as you develop the key.

To create a complete key:

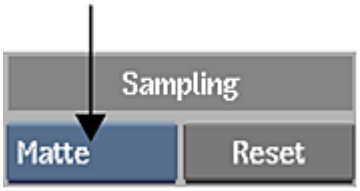
1. Generate and refine a matte using the Matte controls.
2. Remove any colour spill.
3. Improve the blend between the front and back clips.
4. If the image is grainy, apply the Degrain algorithm.
5. Create any necessary patches to remove unwanted grey areas from the matte.

Generating Mattes

Use the Matte controls in the Master Keyer to generate and refine a matte for your chroma key.

To generate and refine a matte:

1. In the Master Keyer menu, select Matte from the Sampling box.



2. Set the view to Result or Matte view.
3. Click the Primary Sample colour pot and then sample the image. Click the image to sample a single pixel.

   The Auto CC button is enabled and the initial pure colour value for the key is set.
4. Click the Secondary Sample colour pot and then sample the image. Sample an area where you do not want any softness in the matte.
5. Drag in the Mix field to adjust the mix between the primary and secondary sample. Drag right to include more of the secondary sample or left to include less.

TIP You can adjust the mix at any time as you develop the key.

6 Gesturally refine the matte. In the image window, click an area of the matte that you want to refine—only those parameters that pertain to the area you click appear. Parameters are displayed in order of importance, from top to bottom (those that are brightest and at the top have the greatest effect on the image). You can then modify a parameter by dragging its highlighted slider.

- To add softness, drag a slider to the right.
- To remove softness, drag a slider to the left.

The red indicator shows the original value and the yellow indicator shows the current value.

7 To modify more than one parameter, move the mouse between the parameters to highlight a parameter, or drag the pen vertically. When you highlight the parameter you want to adjust, drag the slider.

TIP If you do not like the result, you can click Undo to reset parameters directly after you complete the operation. There is only one level of undo in the Keyer menu.

8 When you are finished modifying the displayed parameters, click another area of the image without highlighting a parameter to hide them. Alternatively, you can press any key, such as `spacebar` or `Esc`.

The parameters are no longer displayed.

9 Repeat steps 6 to 8 in other parts of the matte to further refine it. Only the parameters that apply to the problem area will appear.

10 To scroll through the image and display the pertinent parameters, Alt-drag the image without clicking it. The parameters update as you drag. You can then modify the displayed parameters by clicking the image and dragging the highlighted slider.

NOTE To reset matte parameters, click the Reset button, next to the sampling box. All matte parameters are reset, except the Mix field and the key colour.

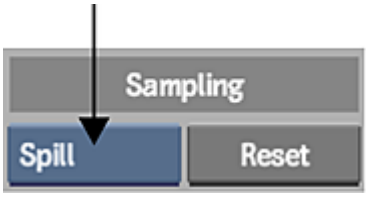
After you have generated your matte, perform basic colour spill removal (page 1034).

Basic Colour Spill Removal

After you create and refine your matte (page 1033) for the key, you can perform basic colour spill removal by gesturally modifying the Spill parameters.

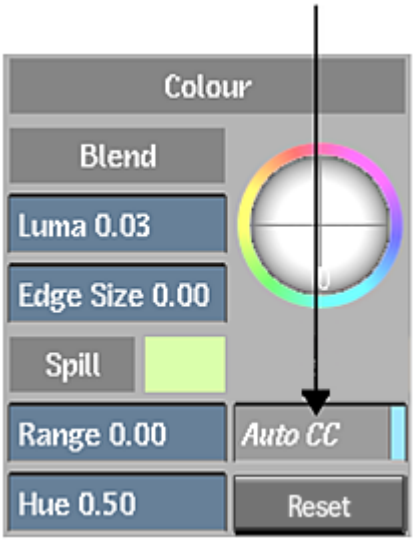
To gesturally remove colour spill:

1 In the Master Keyer menu, select Spill from the Sampling box.



2 Set the view to Result view.

3 Enable Auto CC to apply the Spill settings to the front clip.



4 Remove any colour spill. Click anywhere in the image window, and then modify the parameters that appear by dragging the sliders—you do not need to click a specific area.

Drag:	To:
Range	Set the range for the removal of colour spill along the edges of the key. Drag right to soften the edge and remove colour spill further into the key. Drag left to harden, or create a thinner, edge.
Hue	Modify and suppress colours that are adjacent to the primary sample.

The red indicator shows the original value and the yellow indicator shows the current value. The values also appear in the Spill fields. The original values should be good, so make small adjustments only.

5 To modify both parameters, move the mouse between them to highlight a parameter, or drag the pen vertically. When you highlight the parameter you want to adjust, drag the slider.

If you do not like the result, you can click Undo to reset parameters directly after you complete an operation. There is only one level of undo in the Modular Keyer.

6 When you are finished modifying the displayed parameters, click another area of the image without highlighting a parameter to hide them. Alternatively, you can press any key, such as `spacebar` or `Esc`.

The parameters are no longer displayed.

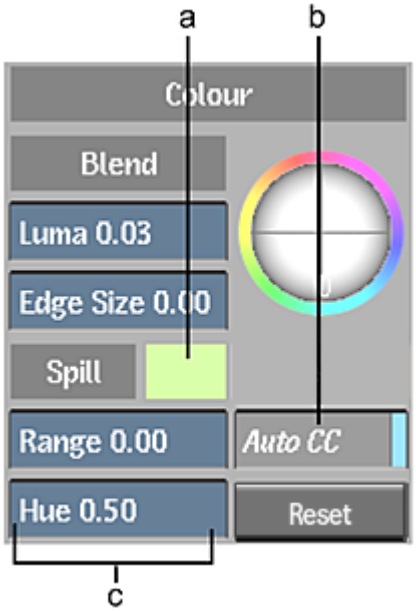
NOTE To reset Spill parameters, click the Reset button, next to the sampling box.

If you are satisfied with your colour spill removal, move on to blending your front and back clips (page 1037).

If you want to perform additional colour spill suppressions, see Advanced Colour Spill Removal (page 1035).

Advanced Colour Spill Removal

If you want to suppress colour spill beyond what was shown in Basic Colour Spill Removal (page 1034), you can use the Spill controls in the Colour menu to eliminate and disguise the colour spill in the Master Keyer. You can sample the colour you want to remove, and then adjust the Range and Hue fields to suppress the selected colour and shift its adjacent colours.



(a) Spill Colour pot (b) Auto CC button in Range menu (c) Spill fields

When you sample an image to generate a key from a Range menu, the sampled colour in the Spill colour pot is also updated. However, the inverse is not true—if you sample a colour using the Spill colour pot, sampled colours used to generate keys are not updated. You can therefore sample colour spill without affecting the original sample.

NOTE To apply these settings to the key, the Auto CC button must be enabled.

To remove colour spill from a key:

1. Click the Result node so that you can see the changes in the right side viewer. If you do not see the result, see Setting Up Viewports for Keying (page 992).
2. Enable Auto CC in the Colour menu to apply the Spill settings to the front clip.
3. The Spill colour pot displays the colour that will be suppressed in the clip. By default, the original colour extracted from the key-in clip is displayed. In many cases, you can use this colour because the colour spill is the same colour as the original colour. However, if the colour spill is not the same, change the colour sample. Click the colour pot, use the colour picker to sample the colour spill in the image window, and then click the colour pot again.

   NOTE When you select the colour you want to key from the Master Keyer menu, the colour that appears in the Spill colour pot is automatically updated to match the key colour. However, the inverse is not true—when you sample a colour from the Spill colour pot, the sampled colour used to generate a key is not updated.

4. Drag in the Spill fields to remove the colour spill.

Drag:	To:
Range	Suppress the primary sample colour where there is colour spill (along the edges of the key).
Hue	Modify colours that are adjacent to the primary sample and further remove colour spill.

NOTE You can reset the Spill parameters, Blend parameters, and the Edge Balance trackball by clicking Reset, located to the right of the Hue field.

When you are satisfied with your colour spill removal, move on to blending your front and back clips (page 1037).

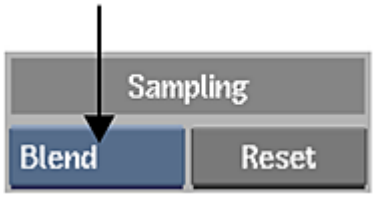
Blending Front and Back Clips

After you remove colour spill (page 1034), you can use the Blend parameters in the Master Keyer to gesturally modify the luminance at the edge of the key so that it blends with the luminance in the background clip. For example, when the front clip is darker than the back clip, you can use the Master Keyer's Blend parameters to lighten the edge of the key.

When you blend front and back clips, you can add a cast to the edge of the key and further merge the clips.

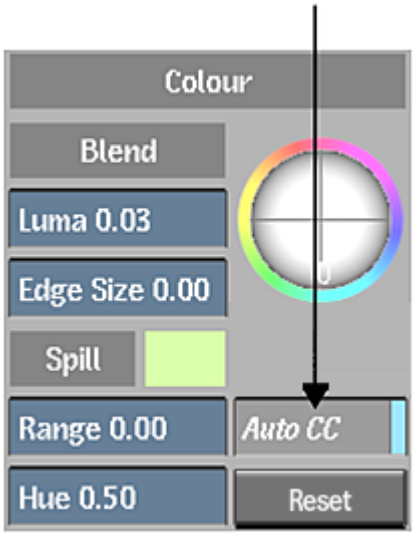
To blend the front and back clips:

1 In the Master Keyer menu, select Blend from the Sampling box.



2 Set the view to Result view.

3 Enable Auto CC to apply the Spill settings to the front clip.



4 Blend the front and back clip. Click anywhere in the image window, and then modify the parameters that appear by dragging the sliders—you do not need to click a specific area.

Drag:	To:
Luma	Darken or lighten the edge of the key. Luma only affects the luma of the edge.

Drag:	To:
Edge Size	Set the range for the blend. Drag right to soften the edge and blend further into the key. Drag left to harden, or create a thinner, edge. Edge Size affects both the Luma field and the Edge Balance trackball.

TIP To drag faster, apply more pressure with the pen, or hold Alt+Spacebar while you drag.

The red indicator shows the original value and the yellow indicator shows the current value. The values also appear in the Blend fields.

5 To modify both parameters, move the mouse between them to highlight a parameter, or drag the pen vertically. When you highlight the parameter you want to adjust, drag the slider.

TIP If you do not like the result, you can click Undo to reset parameters.

6 When you are finished modifying the displayed parameters, click another area of the image without highlighting a parameter to hide them. Alternatively, you can press any key, such as `spacebar` or `Esc`.

The parameters are no longer displayed.

7 To add a cast to the edge of the key and improve the overall look by matching the edge with a colour cast in the back clip, drag the Edge Balance trackball toward the colour you want to add. The trackball only affects the chroma of the edge.

NOTE To reset Blend parameters, click the Reset button, next to the sampling box. All blend parameters are reset, except the Edge Balance trackball. Ctrl-click the Edge Balance trackball to reset it.

After you have set your blend parameters for the front and back clip, remove grain from your clips (page 1039).

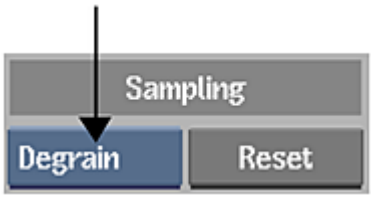
Removing Grain

After you have set your blend parameters for the front and back clips (page 1037), you may want to remove grain from your clips. Graininess can make it difficult to pull a clean and effective key. Use the Degrain tools in the Master Keyer to remove grain from clips.

NOTE Degrain parameters are animatable. Degrain samples are also animatable and appear in the Channel Editor as the Degrain_Matrix channel, located in the Key_Degrain folder. See Animating Keyframes (page 1687).

To remove grain:

1 In the Master Keyer menu, select Degrain from the Sampling box.



2 Set the view to Result view.

3 Sample a grainy area of the image. To sample a single pixel, click the image. To sample an area of the image, Ctrl-drag a selection box.

The Degrain button is enabled and the algorithm is applied to the image—grain is removed from the image.

NOTE The Degrain button is either enabled or disabled for the entire clip—you cannot turn it on and off for different frames in the clip.

4 Drag in the Degrain fields to modify the grain size and restore edge sharpness.

(a) Size field (b) Edges field

Drag:	To:
Size	Estimate the size of the grain in the image.
Edges	De-sharpen the edge of the image. By default, Degrain sharpens the edges. Use the Edges field to restore the natural look of the edges in the image.

5 If you are not satisfied with the result, you can start over with a new sample, and then adjust the Degrain fields. To resample an area of the image, Ctrl+Alt-drag the image.

The sample is outlined in green, indicating that you are resampling the image.

NOTE You can also reset Degrain parameters. To reset Degrain parameters, click the Reset button, next to the sampling box.

6 If resampling the image and adjusting the Degrain fields does not sufficiently remove grain, enable More to increase the overall effect of Degrain.

NOTE The More button is either enabled or disabled for the entire clip—you cannot turn it on and off for different frames in the clip. When you enable More, processing speed slows down.

After you have removed grain from your clips, you may still have to remove unwanted greys (page 1040).

Removing Unwanted Greys

If you have unwanted grey areas in the matte, you can use up to three patches in the Master Keyer to isolate a range of colours to be included in, or excluded from, the key.

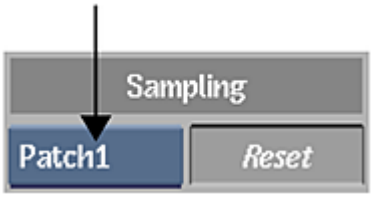
NOTE Patch parameters are animatable. Patch samples are also animatable and appear in the Channel Editor as a Matrix channel, located in the Patch folder.

There are three types of patches in the Master Keyer, that are applied to different areas of the image.

Selected patch:	Is applied to:
Black	Areas of the image to be included in the black part of the matte.
White	Areas of the image to be included in the white part of the matte.
Edge Analysis	Areas of the image that are along the edge of the key. Edge Analysis is useful when there is a specific edge you want to erode but cannot do so with the Matte parameters. You can then increase or decrease the softness of this patch using the Soft field.

To remove unwanted greys:

1 In the Master Keyer menu, select Patch1, Patch2, or Patch3 from the Sampling box.

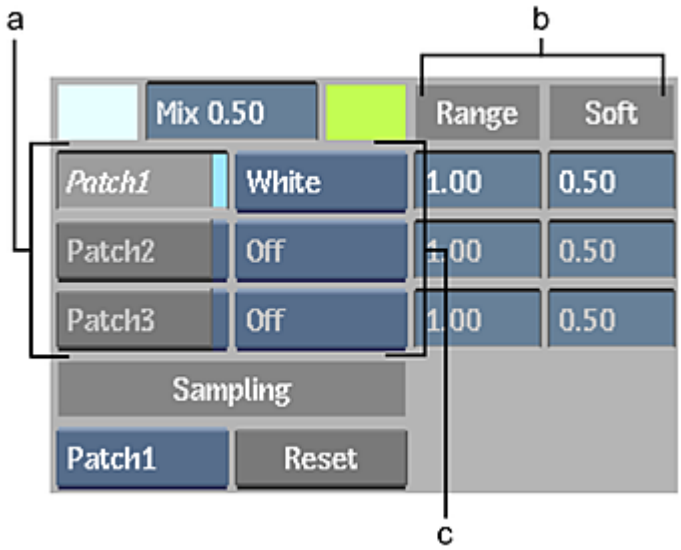


2 Set the view to Matte view or Result view.

3 Sample the image where you want to apply the patch. To sample a single pixel, click the image. To sample an area of the image, Ctrl-drag a selection box.

When you Ctrl-drag to sample, the sample is outlined in red. In the Master Keyer menu, the Patch button is enabled and the appropriate patch appears in the Patch box and is applied to the image.

NOTE A Patch button is either enabled or disabled for the entire clip—you cannot turn it on and off for different frames in the clip.



(a) Patch buttons (b) Patch controls (c) Patch boxes

4 To add more colour to the patch, resample the image.

5 To use the same patch but start with a new sample, Ctrl+Alt-drag the image.

The sample is outlined in green, indicating that you are resampling the current patch.

6 To manually select a patch type, select it from the Patch box.

NOTE If you want to reset the patch so that you can automatically select the patch type, you must disable the patch and set the patch type to Off. You can then resample an area in the image and generate a patch type.

7 To improve the patch, use the Patch controls.

Drag:	To:
Range	Increase or decrease the colour range that is included in the patch.
Soft	Soften the edge.

NOTE To reset Patch parameters, click the Reset button, next to the sampling box.

Creating a Key by Extracting a Single Colour with the Channel Keyer

You can extract one of the three primary colours (red, green, or blue), or a custom colour from the key-in clip. This method is useful for clips containing transparencies such as glass or smoke.

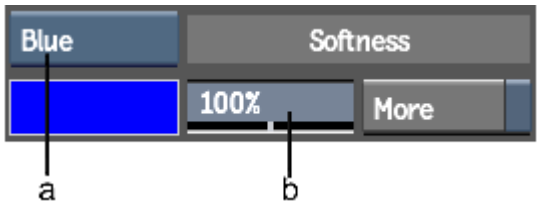
Extracting a Primary Colour

Extract one of the three primary colours from the key-in clip when you have a front clip shot in front of a blue, red, or green screen.

To create a key by extracting a primary colour:

1 From the Start Mode dropdown list, select Reset to Channel and click Confirm.
The Channel keyer processing pipeline is displayed in the schematic view.

2 Double-click the Channel Keyer node.
The Channel controls appear.



(a) Channel Mode box (b) Softness field

3 Select one of the primary colour options (Red, Green, Blue) from the Channel Mode box.

4 Enable More to enhance the keying effect.
The More option extends the range of the colour to be extracted from the key-in clip.

5 Increase or decrease the softness for the key using the Softness field.
The softness value determines how much of the key-in clip is partially transparent in the matte. Softness creates a smoother transition between the front and back clips in the composite clip.

6 Click Result to see the result of your key.

Extracting a Custom Colour

Use the Custom colour channel option in the Channel Keyer to select the colour to be extracted from the key-in clip. This option can give good results when keying transparencies, particularly by experimenting with the Softness and Colour Spread values.

To create a key by extracting a custom colour:

1. From the Start Mode dropdown list, select Reset to Channel and click Confirm.
   The Channel keyer processing pipeline is displayed in the schematic view.
2. Double-click the Channel Keyer node.
   The Channel controls appear.

(a) Channel Mode box (b) Average Colour pot (c) Softness field (d) Colour Spread field

3. Select Custom from the Channel Mode box.
4. Click the Average Colour pot.
   The cursor changes to a colour picker.
5. Click the image to select a single colour, or drag the colour picker across a region to obtain the average of the colours encountered by the colour picker. Try selecting different colours in the background area to get the best result.

   TIP To keep shadows, click just outside them. To remove shadows, click inside them.
6. Adjust the softness for the key using the Softness field.
   Increasing softness raises the level of grey in the matte. Lowering it makes the matte sharper.
7. Adjust the colour spread for the key using the Spread field.
   Increasing the colour spread value extends the range of colours extracted from the key-in clip.
8. Click the Result node so that you can see the changes in the right side viewer. If you do not see the result, see Setting Up Viewports for Keying (page 992).

   NOTE To improve the key at this point, try adjusting it with the histogram in the 2D Histogram node. See Adjusting the Luminance of the Key (page 1047).

Creating a Key by Extracting a Range of Colours with the HLS, YUV, RGB and RGBCMYL Keyers

Another technique for creating a key is to extract a range of colours from the key-in clip. Use this technique for clips where the colour you are extracting contains impurities.

When you build a key by defining a colour range, you can use one of four colour models: RGB, YUV, HLS, or RGBCMYL. Each model interprets the key-in clip differently and gives a slightly different result.

After you choose a colour model, you set a range of colours to become partially transparent in the key-in clip to soften the transition between the front and back clips. This is called the *softness range*. All pixels in

the front clip within the softness range become grey in the matte. The key should have the greatest possible softness value.

After setting the softness range, you set a range of colours to be keyed out in the key-in clip. This is called the *tolerance range*. All pixels in the front clip within the tolerance range become black in the matte. The tolerance range must not be too large or the edge of the composite will be too hard and the subject in the front clip will appear to be pasted into the back clip.

About the Different Colour Models

There are four colour models you can use.

HLS

In the HLS Keyer menu, you set the softness and tolerance ranges using the hue, luminance, and saturation channels.

YUV

In the YUV Keyer menu, you set the softness and tolerance ranges using the luma and chroma signals of YUV component video.

RGB

In the RGB Keyer menu, you set the softness and tolerance ranges using the red, green, and blue channels.

RGBCMYL

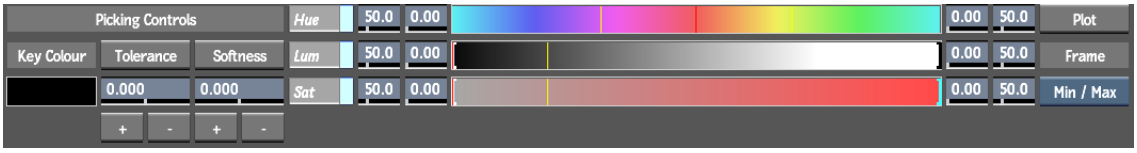
In the RGBCMYL Keyer menu, you set the softness and tolerance ranges using the red, green, blue, cyan, magenta, yellow, and luminance channels. This colour model provides subpixel resolution.

Selecting a Colour Model Based Component Keyer

To determine which component keyer will give you the best key, try creating a sample matte for each keyer.

To create a sample matte for each keyer:

1. From the Start Mode dropdown list, select one of the following: Reset to RGB, Reset to YUV, Reset to HLS, Reset to RGBCMYL.
2. Select Key In in the View box to view the key-in clip.
3. Double-click the Keyer node in the schematic.



   The softness for each model is set automatically to 50, which helps you to quickly gauge the model that will create the best key for your clip.
4. At frame 1, click the Average Colour pot.
5. Drag the colour picker around the area you want to key out.

The average colour sampled by the colour picker appears in the Average Colour pot. In each channel of the colour model, the channel value of the average colour appears as a white line. All pixels in the key-in clip with the selected colour value are keyed out.

The yellow lines in the colour model channels indicate the limits of the softness range. The pixels in the front clip with colour values at the centre of the range are black (transparent) in the matte. As you move away from the centre, the pixels become more opaque.

6 From the View box, select Matte.

7 Repeat this procedure to plot the average colour using the other colour models, and then view each resulting matte to determine which model gives the best result. After you decide on a colour model, use the tools described in the next sections to refine your matte.

After you have selected a colour model and set the average colour, set the softness range (page 1045).

Setting the Softness Range

After you choose a colour model based component keyer and set the average colour (page 1044), adjust the softness range. The softness at the edges of the matte increases as you increase the softness range. To set the softness, you can use the colour picker, the numeric fields, or the Master Softness field.

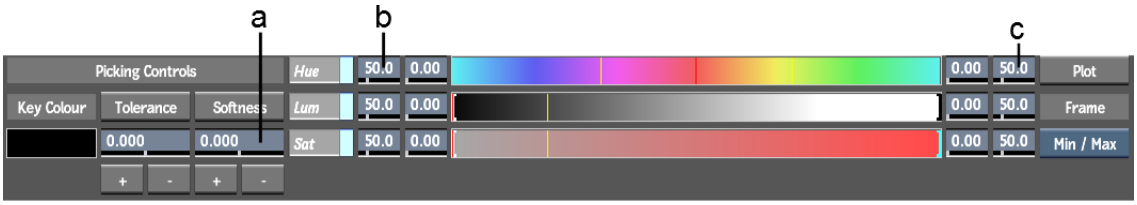
To set the softness using the colour picker:

1 Zoom in to see the edges of the matte more clearly.

2 Click Softness in the selected colour model menu.

The cursor changes to a colour picker.

3 Position the colour picker at the edge of the matte. To increase the softness, click and slowly drag the colour picker toward the centre of the matte. The values within the area you selected are used to adjust the maximum and minimum values for the softness range. The positions of the yellow lines change as you drag the cursor on the image.



(a) Master Softness field (b) Minimum Softness field (c) Maximum Softness field

4 To increase or decrease the softness range, click the + or - button beside the Softness button and then click an area of the image.

To set the softness range using the numeric fields:

1 Set the minimum value for the softness range using the Minimum Softness field on the left side of the colour bar.

2 Set the maximum value for the softness range using the Maximum Softness field on the right side of the colour bar.

NOTE You can also set the softness range by entering a numeric value in the Master Softness field.

TIP As a reference, when setting softness and tolerance ranges, you can display the colour value for any pixel in the key-in clip using the Plot tool. You can then adjust the tolerance or softness range so the pixel falls within one of the ranges. For example, plot pixels at the edges of the matte to check for softness, or plot pixels in the background to check for tolerance.

To plot a pixel's colour values:

1. Click Plot.

   The cursor changes to a colour picker.
2. Select a pixel in the image area.

   A red bar appears in each colour gradient showing the colour value of the pixel.

After you have adjusted the softness range, set the tolerance range (page 1046).

Setting the Tolerance Range

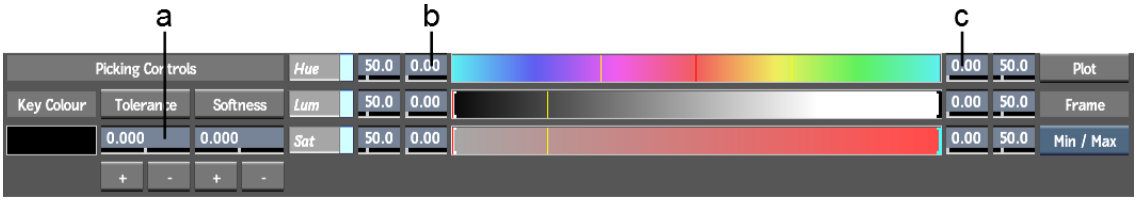
After you define the softness range in a colour model based component keyer, you can set the tolerance range to remove the greys outside the key shape. The maximum and minimum tolerance values define the range of colours to be keyed out in the key-in clip.

To set the tolerance range using the colour picker:

1. Click Tolerance.

   The cursor changes to a colour picker.
2. Drag the colour picker around the area of the image that you want to key out.

   The colour values that the colour picker samples are used to set the maximum and minimum values for the tolerance range. The white lines define the limits of the tolerance range. All colour values between the white lines are extracted from the key-in clip.



(a) Master Tolerance field (b) Minimum Tolerance field (c) Maximum Tolerance field

3. To increase or decrease the tolerance range, click the + or - button beside the Tolerance button.

   NOTE An alternative method of setting the tolerance range is to press Ctrl and draw a rectangle in the area of the image you want to key out.

To set the tolerance range using the numeric fields:

1. Set the minimum value for the tolerance range using the Minimum Tolerance field on the left side of the colour bar.
2. Set the maximum value for the tolerance range using the Maximum Tolerance field on the right side of the colour bar.

NOTE You can also set the tolerance range by entering a numeric value in the Master Tolerance field.

TIP As a reference, when setting softness and tolerance ranges, you can display the colour value for any pixel in the key-in clip using the Plot tool. You can then adjust the tolerance or softness range so the pixel falls within one of the ranges. For example, plot pixels at the edges of the matte to check for softness, or plot pixels in the background to check for tolerance.

To plot a pixel's colour values:

1. Click Plot.
   The cursor changes to a colour picker.
2. Select a pixel in the image area.
   A red bar appears in each colour gradient showing the colour value of the pixel.

Creating a Key by Setting the Luminance

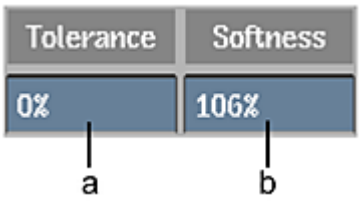
You can create a key using only the Luminance channel. The softness and tolerance values are expressed as percentages.

This technique is useful for clips with high contrast or filmed against a black background. You can also use the Luminance channel to adjust a matte that has already been rendered. Load the matte as the key-in clip, and then adjust it in the Luminance menu.

NOTE When you load a matte as the key-in clip and open the Luminance menu with default menu values, the resulting Keyer matte is identical to the original matte.

To create a key by setting the luminance:

1. From the Start Mode dropdown list, select Reset to Luminance and click Confirm.
   The Luminance Keyer processing pipeline is displayed in the schematic view.
2. Double-click the Luminance Keyer node.
   The Luminance controls appear.



(a) Master Tolerance field (b) Master Softness field

3. Set the softness in the Master Softness field.
4. Set the tolerance in the Master Tolerance field.
   A value of 100 for the tolerance creates an entirely opaque matte.

Refining Your Key

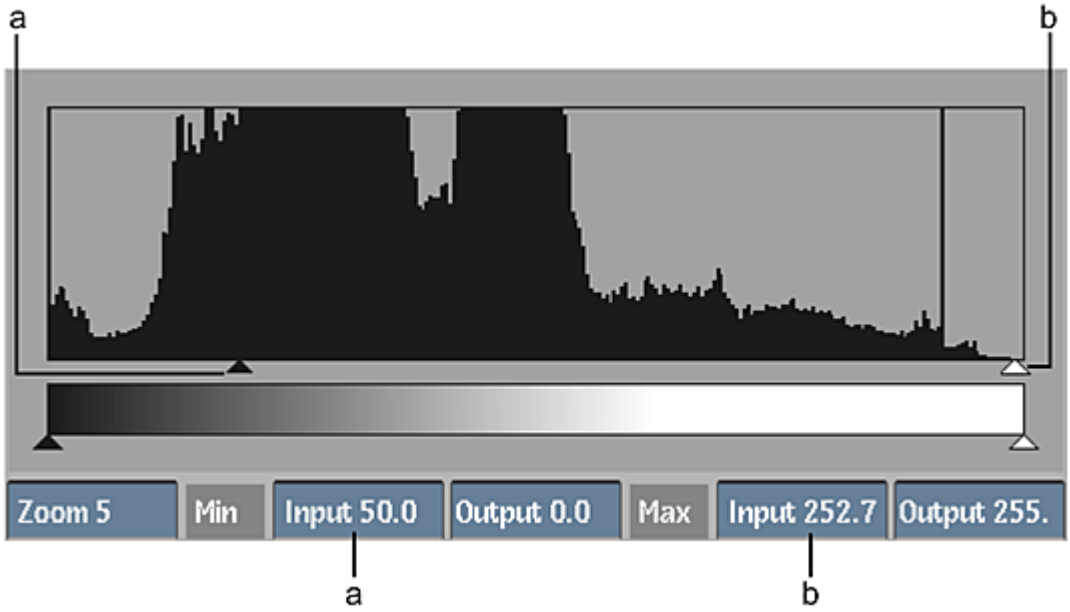
Adjusting the Luminance of the Key

After creating a key, you can adjust luminance values to fine-tune the result. The 2D Histogram node in the schematic displays the distribution of luminance values within the matte. The horizontal axis of the histogram

represents the range of luminance values in the matte and spans from 0 (black) to 255 (white). The vertical axis shows the number of pixels at each luminance value.

Setting the Range of the Luminance Values

Use the List and Gain fields with the Input Level controls, in the 2D Histogram node in the schematic, to set the range of luminance values in the matte. You can darken black areas of the matte or remove grey from white areas of the matte.



(a) Minimum Input level (b) Maximum Input level

Removing Grey from the Black Areas of the Matte

The Minimum Input level sets the start of the range of luminance values. Pixels with luminance values below the Minimum Input level are mapped to black (0).

You can set the Minimum Input level by dragging the black triangle, or by setting a value in the Minimum Input Level field.

The matte before adjusting the Input levels

The matte after lowering the Minimum Input level

Removing Grey from the White Areas of the Matte

The Maximum Input level sets the end of the range of luminance values. Pixels with luminance values greater than the Maximum Input level are mapped to white (255).

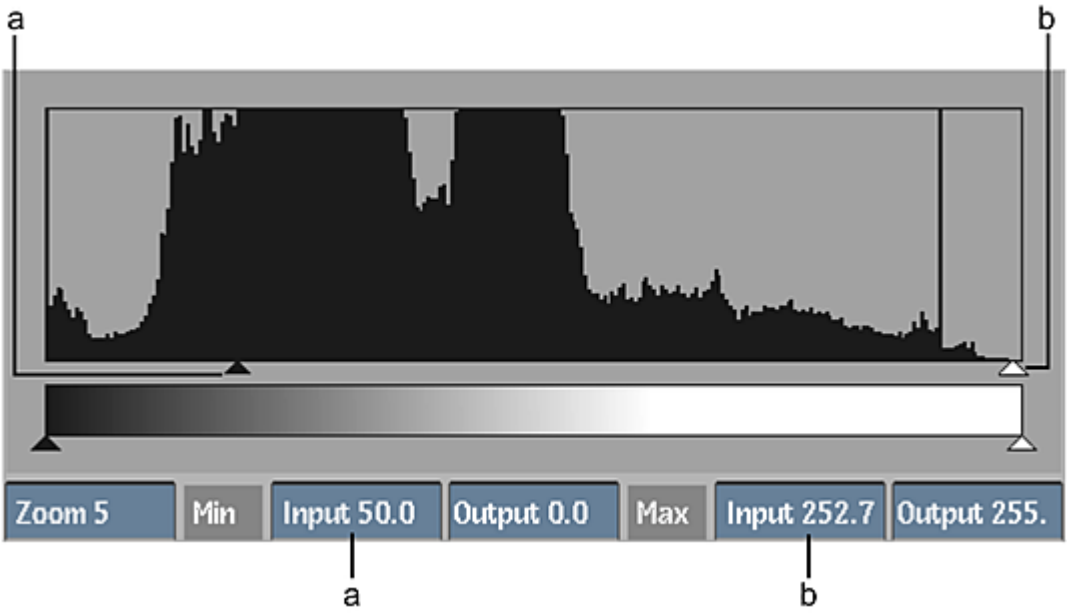
You can set the Maximum Input level by dragging the white triangle, or by setting a value in the Maximum Input Level field.

The matte before adjusting the Input levels

The matte after lowering the Maximum Input level

Remapping Black and White

Use the Output levels in the 2D Histogram node in the schematic to remap the luminance values for black (0) and white (255). You can brighten the dark areas of the matte or darken the white areas.



(a) Minimum Output level (b) Maximum Output level

Brightening the Matte

The Minimum Output level sets the luminance value of all black (0) pixels in the matte. Black pixels in the matte are mapped to the luminance value set by the Minimum Output level.

You can set the Minimum Output level by dragging the black triangle, or by setting the value in the Minimum Output Level field.

The matte before adjusting the Output levels

The matte after lowering the Minimum Output level

Darkening the Matte

The Maximum Output level sets the luminance value of all white (255) pixels in the matte. White pixels in the matte are mapped to the luminance value set by the Maximum Output level.

You can set the Maximum Output level by dragging the white triangle or by setting the value in the Maximum Output Level field.

The matte before adjusting the Output levels

The matte after lowering the Maximum Output level

Boosting the Luminance of the Key Using Gain and Lift

You can remove grey from the key by increasing the gain and lowering the lift values in the 2D Histogram node in the schematic. Increase the gain to eliminate the light greys that may be in the white area of the matte, and decrease the lift to eliminate dark greys in the black area of the matte.

NOTE Adjusting these values increases the contrast and may harden the edges of the matte.

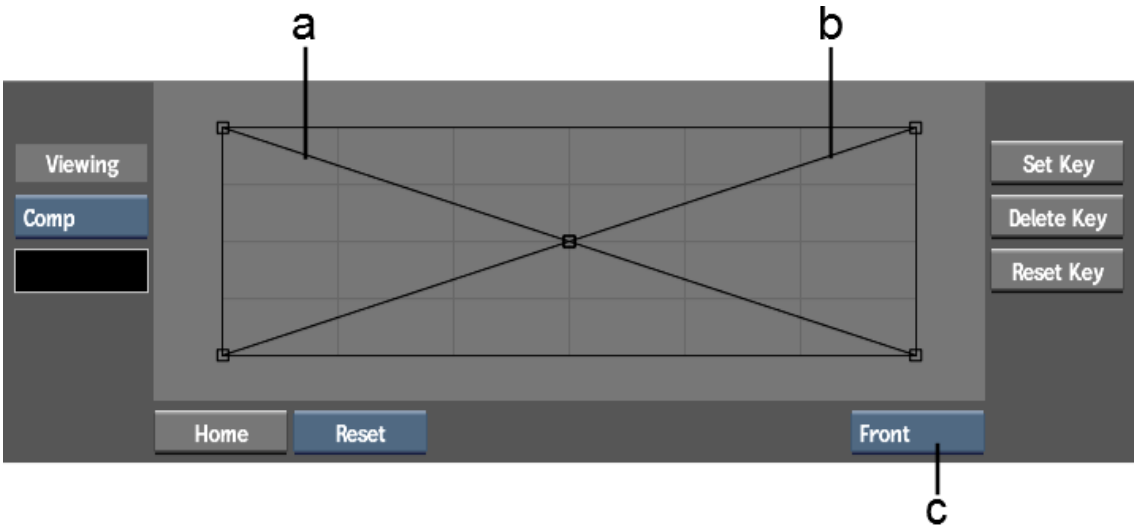
Using the Luminance Curves

When you create a matte for the front clip, the Keyer automatically creates a matte for the back clip to specify which part of the back clip is used for the composite. By default, the back matte is the inverse of the front matte.

You can adjust the luminance of the front matte and back matte separately in the Blending menu in the Result node in the schematic. For example, increase the luminance of the back matte so that more of the back clip shows through at the edges of the key. This creates a better blend at the edges of the key.

To adjust the luminance curve:

1. Double-click the Result node in the schematic.
   The Luminance curve appears.



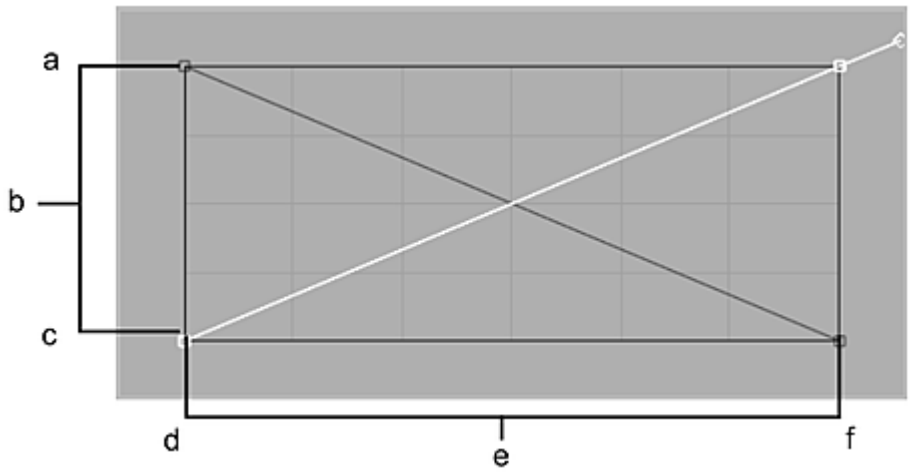
(a) Front matte curve (b) Back matte curve (c) Front/Back box

2 Use the Viewing box to select the image you want to view as you adjust the curves.

Select:	To view:
Result	The composite clip.
Matte	The front matte. You cannot see changes made to the back matte when this view is selected.
Bmatte	The back matte. You cannot see changes made to the front matte when this view is selected.
Comp	The composite with a coloured background. The default colour is black. To select a different colour, click the colour pot to the right of the Comp button. The colour picker appears.
Key In	The image that was used to pull the key.
Back	The background image that is revealed when the front is keyed.
Front	The image to which you applied the key.

3 To adjust the luminance curve for the front matte, select Front from the Front/Back box. To adjust the back matte curve, select Back. Alternatively, click a curve to select it.

The selected curve changes to white.



(a) 255 (White) (b) Output (remapping of luminance values) (c) 0 (Black) (d) 0 (Black) (e) Input (current luminance values) (f) 255 (White)

4 Click a point to display its tangent handle and drag the handle to adjust the curve. Use other modes in the Tools box to further adjust the curve, adding or deleting points or breaking tangent handles, as needed.

Modifying the Edges of the Key

Use the Shrink, Erode, and Blur controls in the Matte Edge node in the schematic, to enhance the edge of the keyed image.

Use:	To:
Edges	Detect the edges of your matte and fine-tune the edges with tolerance controls. This is useful for cleaning up difficult mattes.
Shrink	Remove pixels from the edge of the matte. This control should not be used when the object in the front clip has soft edges, such as hair.
Erode	Blend the light and dark edges of the matte.
Blur	Apply a softening filter to the edge of the matte. You can select either a Gaussian filter or Box filter.

To access the Edge Matte controls:

1 Double-click the Matte Edge node in the schematic.

The Edge Matte controls appear.

To detect the edges of your matte:

1 In the Matte Edge menu, enable Edges.

The edges af your matte are detected and displayed in the viewer.

2 Fine-tune the edges by modifying the Min and Max fields (tolerance) and the Width field, which determines the width, in number of pixels, of the detected edges.

To shrink the edge of the matte:

1. In the Matte Edge menu, enable Shrink.
2. Set a value in the Shrink Width field.

This value specifies the width of the border, in number of pixels, that is removed from the edge of the matte.

The matte before enabling the Shrink filter

The matte after setting the shrink width value to 1.00

To erode the edge of the matte:

1. In the Edge Matte menu, enable Erode.
2. Set a value in the Erode Width field.

This value specifies the width of the matte border, in number of pixels, that will be softened.

The matte before enabling the Erode filter

The matte after setting the erode width value to 1.00

To blur the edge of the matte:

1. In the Edge Matte menu, enable Blur.
2. Set values in the Blur Width and Height fields.

These values specify the width and height of the Blur filter applied to the edge of the matte.

The matte before enabling the Blur filter

The matte after setting the blur width and height values to 1.00

Adjusting Spill Controls with the Colour Curves Node

After you create a key and key out any trouble areas, some of the background colour may have spilled over at the edge of the key. Colour spill suppression in the component keyers is done through the Colour Curves node. The Colour Curves menu lets you sample the colour you want to suppress and then suppress that colour where necessary, using the suppression curve.

- Adjust the Suppression curve in the Colour Curves node to suppress a selected colour.
- Adjust the Hue Shift curve in the Colour Curves node to perform a hue shift on a selected colour.

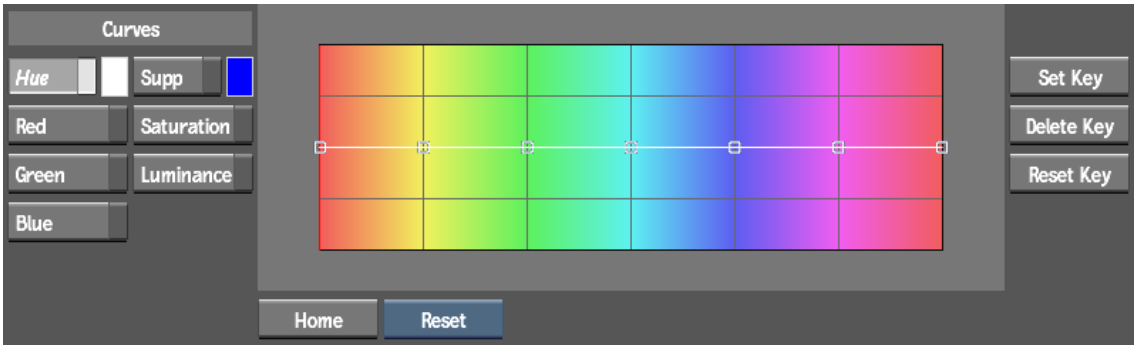
NOTE To remove colour spill in the Master Keyer, see Basic Colour Spill Removal (page 1034) and Advanced Colour Spill Removal (page 1035). Alternatively, you can connect a Colour Curves node between the front clip and the front input of the Master Keyer and follow the procedures below.

Adjusting the Suppression Curve

To suppress colour spill using the suppression curve:

1. Set the view to Result, so that you can see the changes as you make them.
2. Double-click the Colour Curves node in the schematic.

   The Colour menu appears.



The Colour menu displays colour curves over a hue spectrum. When you modify the shape of a curve over a region of the spectrum, only those colours are affected.

3 Click Supp to modify the Suppression curve.

4 The colour pot next to the Suppress button displays the colour that will be suppressed in the clip when you modify the curve. By default, the blue colour is displayed. To change the colour sample, click the colour pot, use the colour picker to sample the colour spill in the image window, and then click the colour pot again.

5 Click the colour pot next to the Supp button.

The cursor turns into a colour picker.

6 Select a pixel within the spill.

A red vertical bar appears in the hue spectrum identifying the colour to be suppressed.

7 Use the cursor to move the points along the Suppression curve.

8 On the Suppression curve, drag the point closest to the plotted colour down to a value of 25, intersecting the plotted colour.

The colour spill is suppressed.

9 Continue modifying the shape of the curve until you are satisfied with the result.

10 Click Saturation to adjust the saturation of the spill.

NOTE Once you remove the saturation from a spill, you may want to increase the values for the other curves (for example, red and green if you removed a blue spill) to reconstruct some of the natural colours at the edge of the keyed image.

11 Click Red, Green, or Blue to edit individual colour curves.

12 Click Luminance to adjust the luminance of the spill.

Adjusting the Hue Shift Curve

You can disguise colour spill by shifting its hue so the colour blends better with the background. Sample the colour to which you want to shift the colour spill and then adjust the Hue Shift curve to shift the colour spill accordingly.

To Create a Hue Shift:

1 Set the view to Result, so that you can see the changes as you make them.

2 Double-click the Colour Curves node in the schematic.

The Colour menu appears.

The Colour menu displays colour curves over a hue spectrum. When you modify the shape of a curve over a region of the spectrum, only those colours are affected.

3 Enable Hue to modify the Hue Shift curve.

4 Select a pixel within the spill in the Result image.

   A red vertical bar appears in the hue spectrum identifying the colour to be shifted.

5 Click the colour pot next to the Hue button.

   The colour picker appears.

6 Select or pick the colour you want to shift the spill to.

7 Use the cursor to move the points along the Suppression curve.

8 On the Hue Shift curve, drag the point closest to the plotted colour down to a value of 75, intersecting the plotted colour.

   The colour spill is shifted toward the Hue colour.

9 Continue modifying the shape of the curve until you are satisfied with the result.

Inverting a Matte

Use the Negative node to invert a matte. By inserting the Negative node between the Matte Edge and the GMask nodes in the processing pipeline, the matte is automatically inverted. The Negative node has no settings to configure.

To invert a matte:

1 From the Modular Keyer Node bin, drag the Negative node to the schematic.

2 Hold Shift and drag the Negative node to the Matte Edge node so their tabs touch and repeat for the GMask node.

   The Negative node is inserted to the schematic and your matte is inverted.

Adjusting the Front and Matte Luminance Curves

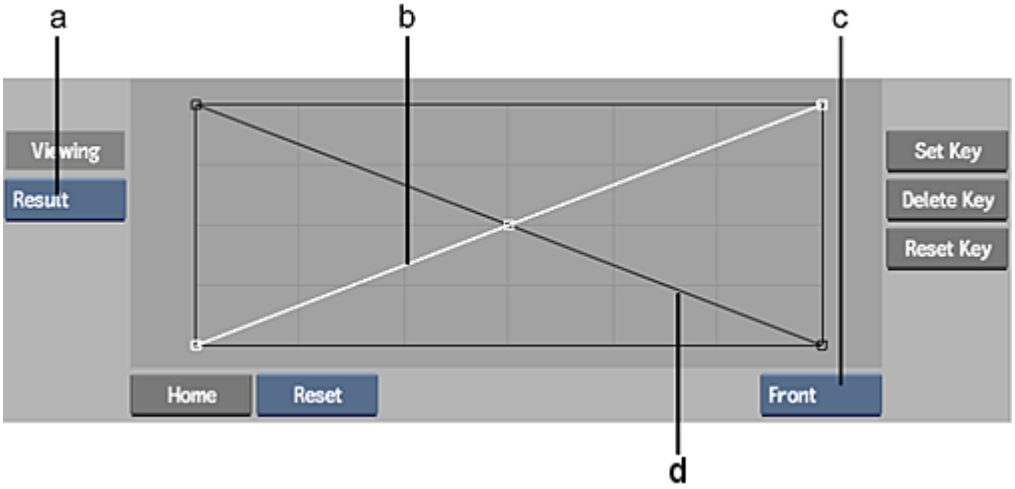
When you create a matte for the front clip, the Modular Keyer automatically creates a matte for the back clip to specify which part of the back clip is used for the composite. By default, the back matte is the inverse of the front matte.

You can adjust the luminance of the front matte and back matte separately in the Matte Curves menu. For example, increase the luminance of the back matte so that more of the back clip shows through at the edges of the key. This creates a better blend at the edges.

To adjust the luminance curves:

1 Click the Result node in the pipeline.

   The Matte Curves menu appears.



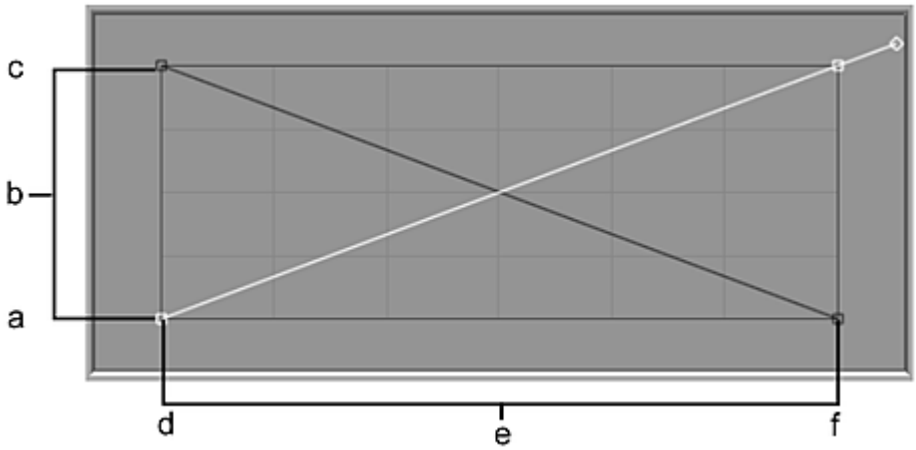
(a) Result box (b) Front matte curve (c) Matte box (d) View box (e) Back matte curve

2 Select Result view from the Result box. This allows you to select a particular image to view as you adjust the curve.

3 From the Result box, select the image you want to view as you adjust the curves.

Select:	To view:
Result	The composite clip.
Matte	The front matte. You cannot see changes made to the back matte when this view is selected.
Bmatte	The back matte. You cannot see changes made to the front matte when this view is selected.
Comp	The composite with a coloured background. The default colour is white. To select a different colour, click the colour swatch below the Tools box. The colour picker appears.

4 To adjust the luminance curve for the front matte, select Front from the Matte box. To adjust the back matte curve, select Back. Alternatively, click a curve to select it.



(a) 255 (White) (b) Output (remapping of luminance values) (c) 0 (Black) (d) 0 (Black) (e) Input (current luminance values) (f) 255 (White)

In Move edit mode, click a point to display its tangent handle and drag the handle to adjust the curve. Use other modes in the Tools box (Add, Delete, or Break, for example) to further adjust the curve, adding or deleting points, or breaking tangent handles as needed.

Viewing a Key with a Solid Colour Background

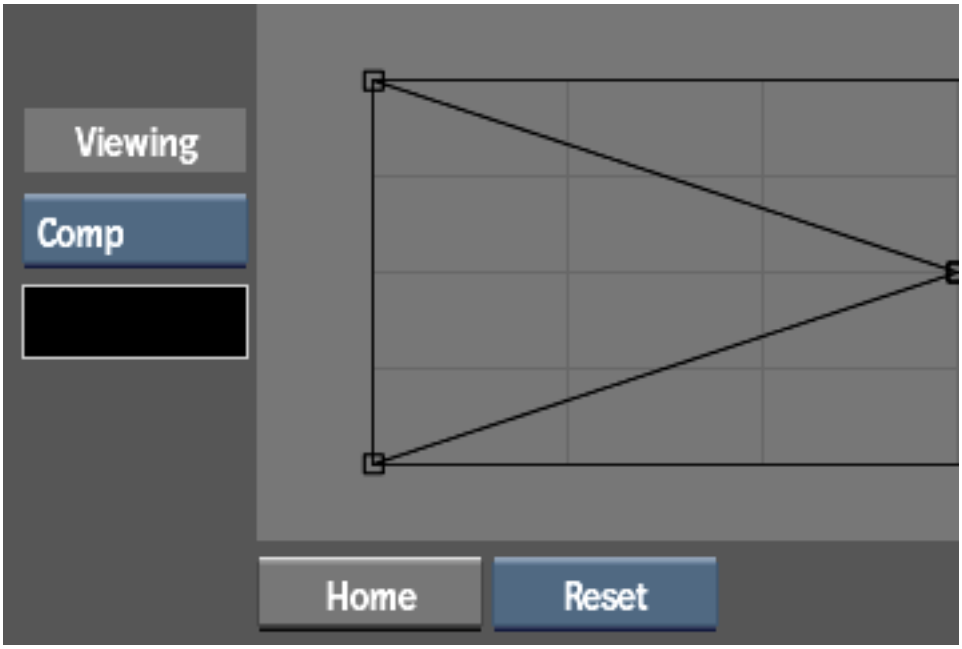
The following optional technique can help you create cleaner keys more quickly.

While creating the key, you can view the composite using a solid colour for the background in place of the back clip. This can help you see details in the image, such as colour spill, that you may otherwise miss. You can use the default colour (black) or select a colour using the colour picker.

NOTE This view does not affect the final render.

To use a solid colour background:

1. Double-click the Result node.
   The Matte Luminance Curves menu appears.
2. In the Result box, select Comp.
   A colour pot appears.



3. Click the colour pot under the Comp view output option.
   The colour picker appears.
4. Select a colour for the background using the colour picker.
   Your key is displayed on a solid colour background in the Result view.

Animating Your Key

The following parameters of a key can be animated:

- The average colour selected using the Average Colour pot in the Keyer menu

- The minimum and maximum Tolerance and Softness values
- The Lift, Gain, Shrink, and Erode values
- The blur factors for the matte and key-in clip
- The Maximum and Minimum Input and Output level values for the histogram

To display the Animation controls in the Keyers, click Animation to the left of the keyer menus. Using the Channel Editor, animate the parameters.

Resetting Your Key

You can reset the individual node within the Modular Keyer processing pipeline. You can also reset the entire Modular Keyer processing pipeline.

To reset individual node within the Modular Keyer processing pipeline:

1. Select the node you want to reset.
2. Right-click the node and select Reset.
   The selected node is reset.

   NOTE You can reset multiple nodes by holding the Ctrl key, selecting the nodes and right-clicking to reset.

To reset the entire Modular Keyer processing pipeline:

1. Shift+click the Result node.
2. Right-click the Result node and select Reset.
   The entire Modular Keyer processing pipeline is reset.

Rendering Your Key

To view your key in the timeline, you must render it. You can render the key inside the Batch FX view or in the timeline.

To render your key in the Batch FX view:

1. When you are satisfied with your key, exit the Modular Keyer by clicking the Return button.
   You are taken to the Batch FX view.
2. In the Batch FX view, click the Render Combo box and select Render. If the Render option is already displayed, simply click Render.
   The rendering starts. A progress bar appears at the bottom of the screen.
3. After the rendering is complete, click EXIT BFX to return to the timeline.
   - If you accessed the Modular Keyer from the timeline, your original source clip is updated in the timeline and in the Viewing panel and displays the result of your key.
   - If you accessed the Modular Keyer from the Tools tab, the result of your key is rendered as new media and is displayed in the Viewing panel.

   A small BFX icon appears on the clip and the timeline segment.

To render your key in the timeline:

1. Once you are happy with your key, exit the Modular Keyer by clicking the Return.

You are taken to the Batch FX view.

2 In the Batch FX view, click EXIT BFX to return to the timeline.

Your key is displayed in the timeline. A dotted line is displayed on the segment and you cannot view the result, as the segment needs to be rendered.

3 From the Editing panel, click the Render Combo box and select Render. If the Render option is already displayed, simply click Render.

The rendering starts. A progress bar appears at the bottom of the screen.

4 Once the rendering is complete, the dotted line becomes a full line and you can view the result of your key. Your original source clip is updated in the Viewing panel and displays the result of your key. A small BFX icon appears on the clip and the timeline segment.

Stabilizing and Tracking 19

Use the Stabilizer to remove camera instability and motion jitter, and to track reference points in your clips. You can also use the Stabilizer to produce 2D or, in the context of Action, 3D motion, anchor a surface to the clip's background or anchor the UV points of the surface to features of a clip. With tracking, a point or points on the clip are tracked as they move through the scene. You can then apply the resulting motion path to an object on another layer so that it follows the same path as the object you tracked.

Stabilizing is the inverse of tracking. With stabilizing, the motion path is used to shift the scene so that the point that is tracked remains fixed at one position.

Tracking and stabilizing are often processes of trial and error. It is recommended that you track or stabilize using the default settings. If the tracker box strays from its original point, you can fine-tune the analysis.

The Stabilizer viewport uses the Viewing Rules for colour management. Enter the View settings panel to bypass the colour management or select a viewing rule other than the primary one for that media.

Accessing the Stabilizer

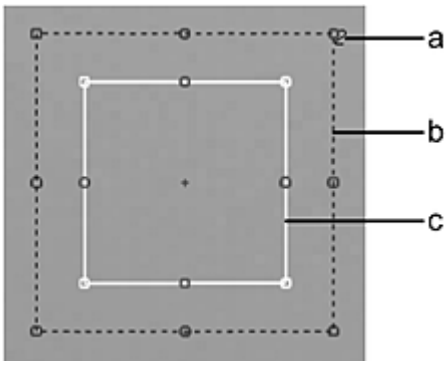
Accessing the Stabilizer

Access the Stabilizer from:	To:
The Tools tab	Stabilize.
Action Axis	Track or stabilize.
Action Perspective Grid	Perform 4-corner tracking.
Action GMask node	Perform Cluster or Vertex tracking.
Action Surface	Perform Bilinear, Extended Bicubic, or Perspective tracking.
Action Analyzer	Provide a 2D tracking path for 3D manual tracking.
GMask	Track a garbage mask or the vertices of a GMask.
Distort	Track vertices or the axis of a spline when warping or morphing.

Access the Stabilizer from:	To:
Warper	Track points or an axis of a mesh when warping or morphing.
Paint Tool	Track an AutoPaint stroke.
Blur and Glow	Track the center point of the radial blur.
2D Transform	Provide correction for 2D motion, rotation and scaling.

How the Stabilizer Works

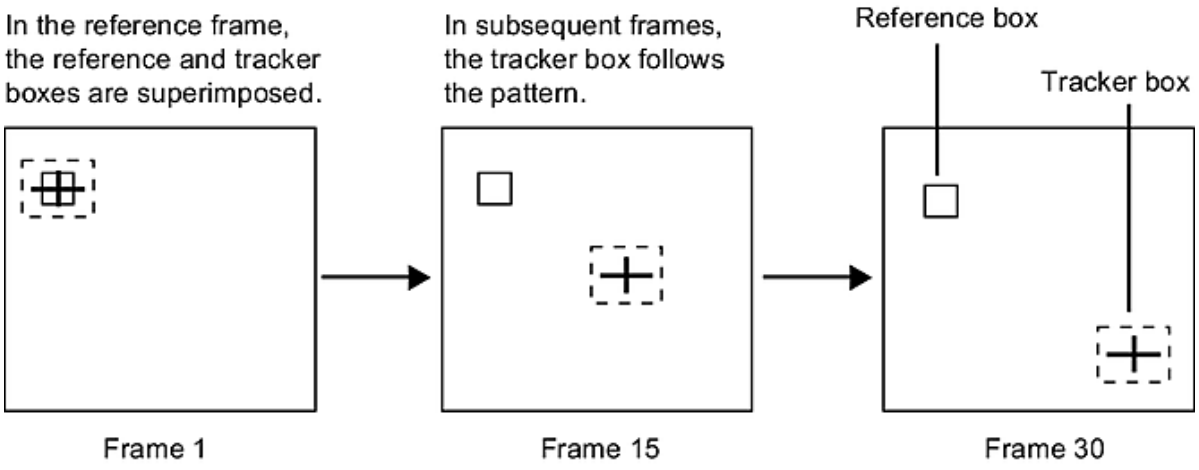
The Stabilizer uses trackers to generate tracking data. Each tracker consists of a solid box, called the *reference box*, and a dashed box, called the *tracker box*. The reference box establishes the reference point (the feature to track or stabilize) in any frame of the sequence. The tracker box indicates to the Stabilizer where to locate the reference point. The tracker box follows the frame-to-frame movement of the reference point.



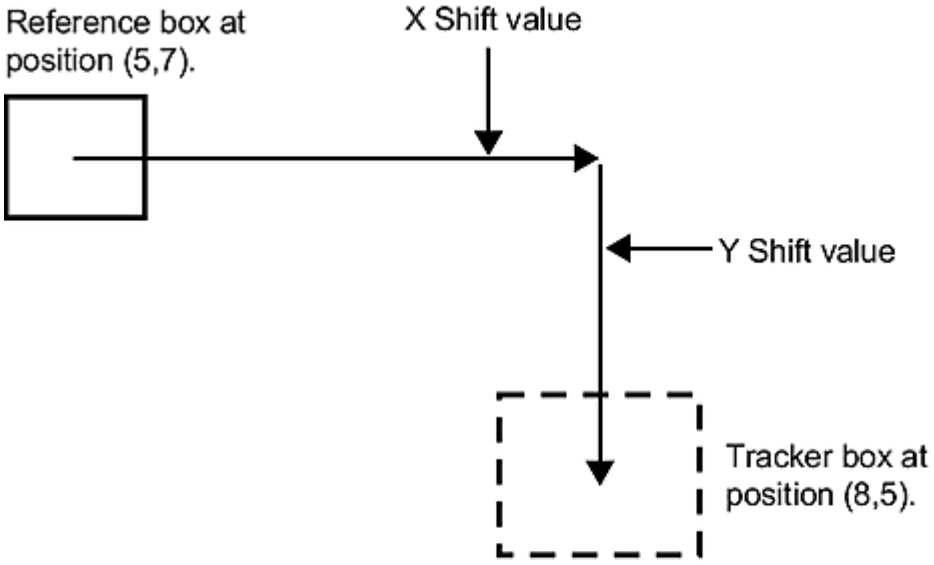
(a) Tracker number (b) Tracker box (c) Reference box

You start by selecting one or more reference points on your clip. Locate the first frame containing the movement to be tracked (the reference frame). In general, the reference frame is the first frame of the sequence. The choice of the reference point depends on whether you are tracking or stabilizing. When tracking, the reference point is a feature you want to track; when stabilizing, the reference point represents the point around which the image is stabilized. See Selecting a Reference Point (page 1067) for details. Place the reference box(es) around the selected feature(s).

Once you have set the tracker positions, start the tracking process, also referred to as analyzing the clip. During the analysis, the tracker box associated with each tracker moves as the Stabilizer looks for a pattern that matches the reference in each frame of the clip.



The Stabilizer calculates the difference between the position of the tracker box and the position of the reference box to produce X and Y Shift values. Shift values represent a measurement in pixels and subpixels of how much the reference point has moved.

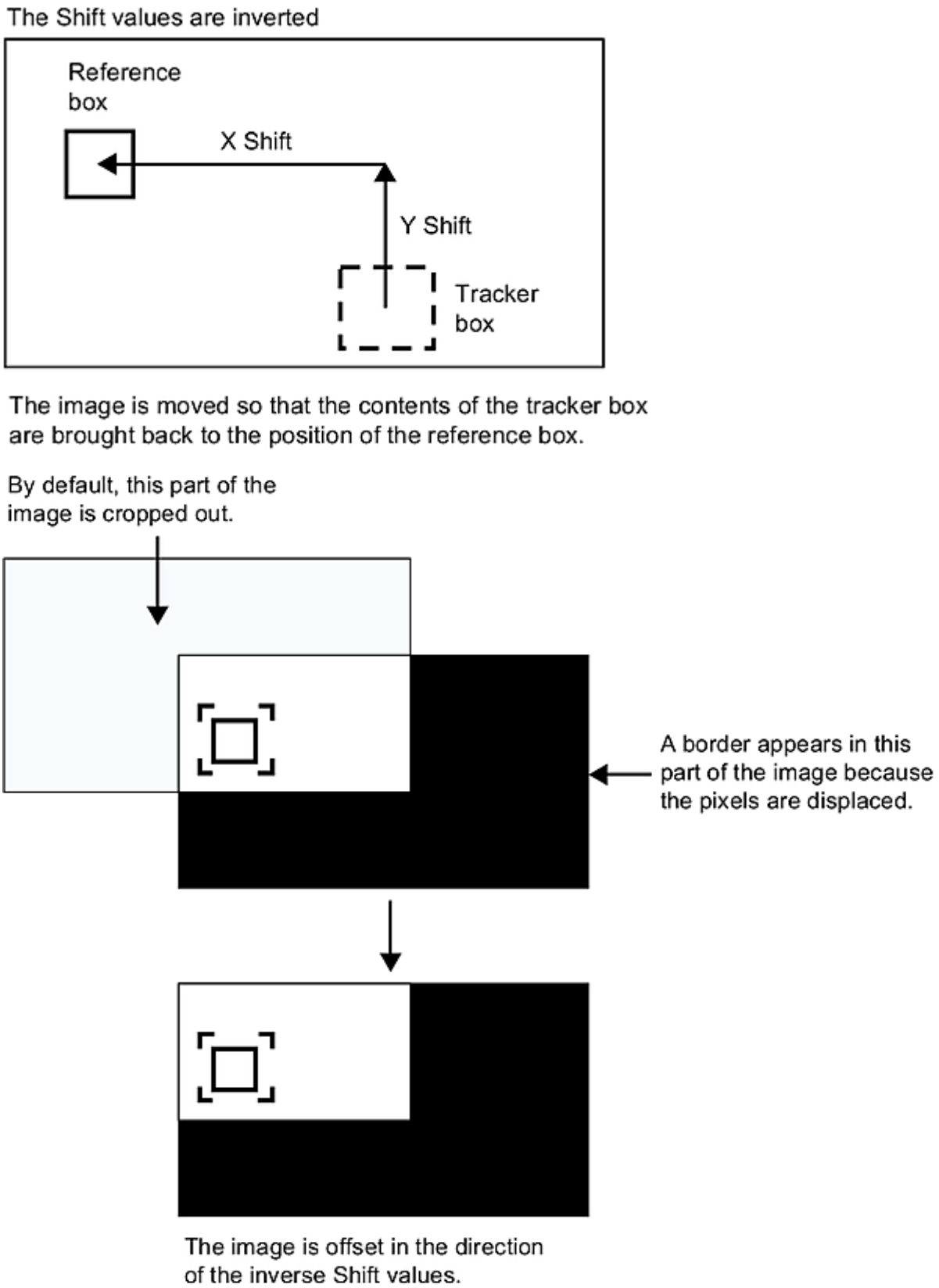


Reference position in X -	8
Tracker position in X	5
Shift value	3

Reference position in Y -	5
Tracker position in Y	7-
YShift value	2

When the analysis is complete, you fine-tune it if a tracker box has strayed from the reference it was supposed to follow. Once you are satisfied with the results, you can apply the data to the clip.

To track, the Stabilizer applies the Shift values "as is." To stabilize, the Stabilizer inverts the X and Y Shift values in each frame of the sequence, and moves the image according to these values. This gives the impression that the reference point stays in the same position throughout the sequence. Because the image is moved during stabilization, a border appears on one or more edges, which means that you lose some pixels. The following illustrations summarize the process.


The Shift values are inverted
Reference box
X Shift
Y Shift
Tracker box
The image is moved so that the contents of the tracker box are brought back to the position of the reference box.
By default, this part of the image is cropped out.
A border appears in this part of the image because the pixels are displaced.
The image is offset in the direction of the inverse Shift values.

Working with Trackers

This section provides information that is common to many procedures. It is recommended that you first read the procedure you want to perform in Selecting a Stabilizer Method (page 1071), and then consult this section when needed.

Selecting a Reference Point

A good reference point is a high-contrast pattern that has good definition both vertically and horizontally.

Selecting a good reference point is a process of trial and error. Play the clip several times to become familiar with the material. Ideally, you should try to find a pattern that is present in every frame. In some cases, this is not always possible and you may have to track two different patterns, track an object that disappears behind another one, or track an object that moves out of the frame. For more information, see Tracking Difficult Shots and Correcting Errors (page 1091).

The frame you use for the reference point should be the frame where the reference image is most representative in terms of shape, size, and rotation. You can select the reference point from a frame in the middle of the sequence, if the pattern you want to use as a reference is subject to a lot of rotation or scaling. In most cases, you will position the reference box over the reference point in the first frame of the sequence.

Selecting a Tracker

You can use any of the following methods to select a tracker:

- Select any tracker Select Tool, from the Stabilizer Tools menu and click the tracker box, the tracking path, or the reference box of the tracker in the viewer.
- Select a tracker by clicking its corresponding button (Tracker1, Tracker2, and so on).
- Select a tracker by pressing the corresponding number key on your keyboard. You must use the number keys on the alphanumeric keyboard, not the numeric keypad.
- Select any tracker by pressing the `up` and `down arrow` keys on your keyboard.
- Select any tracker by selecting its channel in the Channel Editor.

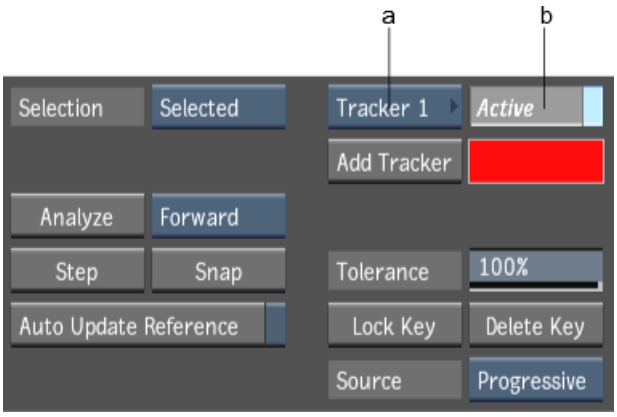
When you select a reference box, a tracker box, or a keyframe in the image window, the sequence automatically updates to the frame where the keyframe was set.

Positioning the Reference and Tracker Boxes

When you position the reference box, the tracker box automatically follows. You can leave the boxes together if you place the reference box in the first or last frame of the clip. However, if you start the analysis on a different frame from where you position the reference box, you should position the tracker box in the first frame of the clip on the pattern you want to track. This way, the Stabilizer knows where to look for the pattern to track when you analyse the clip.

To position the reference and tracker boxes:

1. Go to the frame where you want to position the reference and tracker boxes.
2. If the tracker and reference boxes do not appear on the image, select the appropriate Tracker, from the Tracker box and enable Active.



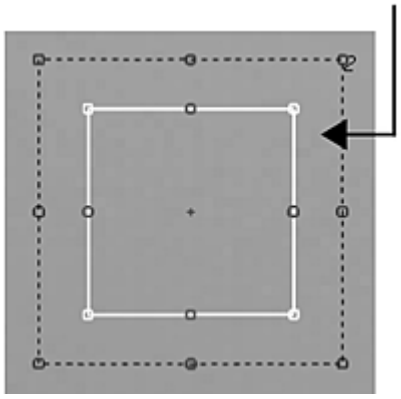
(a) Tracker box (b) Active button

NOTE Each tracker is automatically assigned a different colour. However, you can customize the tracker colour at any time.

3 Click inside the reference box to select it, and drag it over the pattern you want to track.

The reference box changes into a magnifying glass. By default, the tracker box follows the reference box.

4 To position the tracker box on a different frame from the reference box, go to that frame and move the tracker box over the pattern to track. To move only the tracker box, click anywhere outside the reference box.



Resetting the Reference and Tracker Boxes

You can reset the Reference, Tracking or Shift values independently for the selected tracker or all these values simultaneously, by selecting the appropriate option in the Reset box.

1 Do one of the following:

- To reset the Reference values of the selected tracker, select Reset Ref from the Tracker box.
- To reset the Tracking values of the selected tracker, select Reset Track from the Reset box.
- To reset the Shift values of the selected tracker, select Reset Shift from the Reset box.
- To reset the Reference, Tracking and Shift values of the selected tracker, select Reset from the Reset box.

2 Click Confirm.

Resizing the Reference and Tracker Boxes

Resize the reference box to make it surround the feature you chose to track more closely. By only including the recognizable feature in the box with no other details that may change throughout the clip, you minimize the possibility of the tracker losing the reference point.

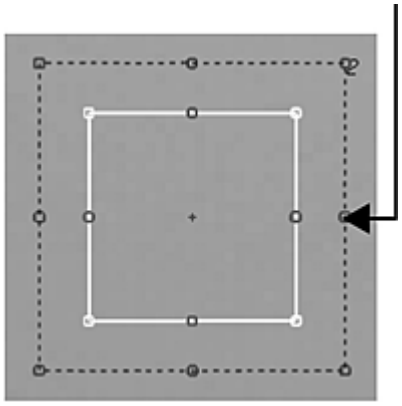
TIP Sometimes a small reference point does not give good results. Try enlarging the box to include more of the selected feature. Note that the larger the box, the slower the analysing speed.

Resize the tracker box so that it is large enough to accommodate the most frame-to-frame movement of the reference point. For example, if the movement of the reference point is mostly horizontal, you can increase the width and reduce the height of the tracker box. If there is a large amount of movement both horizontally and vertically, increase both the width and the height of the tracker box. Note that the smaller the tracker box, the faster the analysis.

To resize the reference and tracker boxes:

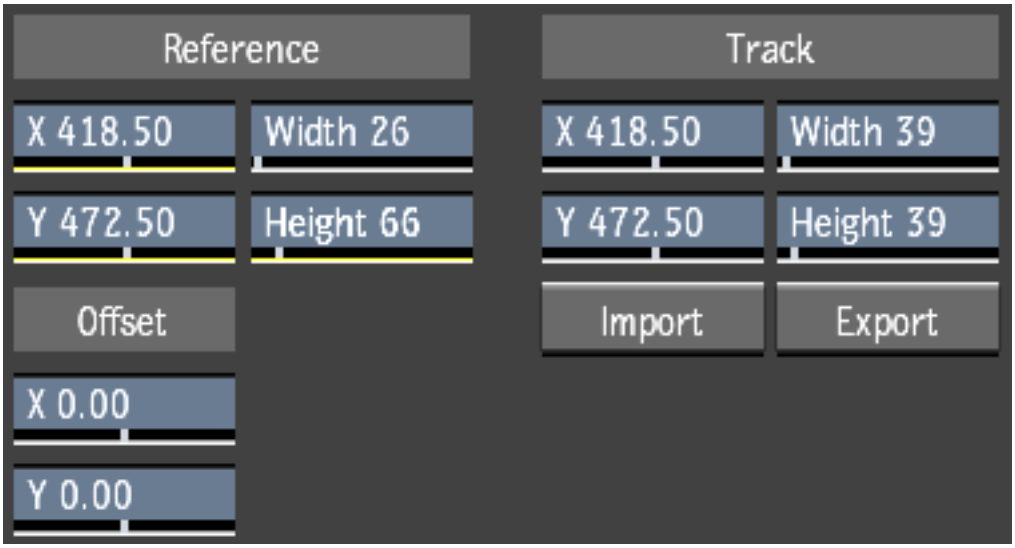
1 Do one of the following:

- To resize the reference box and tracker box interactively on the frame, press a resize handle on the box and drag.



- To use the menu to move or resize the reference or tracker box for the current tracker, enter values in the Reference or Track fields.

  You can also set the position of the offset the position of the reference, via the Offset X and Y fields. See Offsetting the Tracking Motion of the Reference (page 1088).

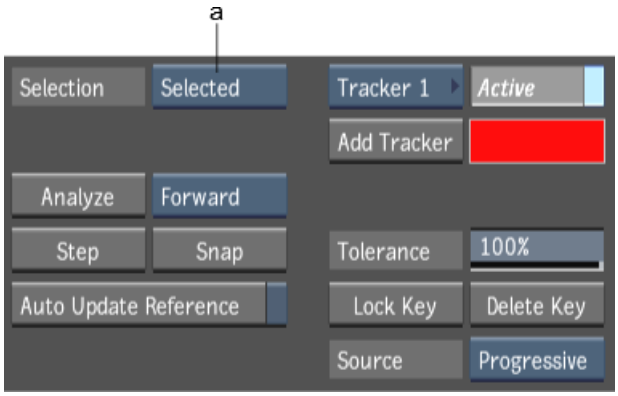


TIP Use the image window Zoom command to get a closer view of the tracker and reference boxes.

NOTE To reset the reference or tracker box to its default position and size, select either Reset Ref, Reset Track or Reset Shift from the Reset box.

Working with Multiple Trackers

You can change a parameter for all active trackers at once. For example, you can change the dimensions of the tracker box for all trackers or set Fixed to off for all trackers, by selecting Gang, from the Tracker Selection box. Inversely, it can be useful to hide all but a selected tracker, to edit its tracking path, for example. This is achieved by selecting Solo, from the Tracker Selection box. By choosing Selected, only the selected tracker is affected, but the other trackers are still displayed.



(a) Tracker Selection box

Choose:	To affect:
Selected	Only the selected tracker while still showing all trackers in the image window.
Solo	Only the selected tracker and hide all other trackers.
Gang	All the active trackers, except when changing the colour of the trackers.

Selecting a Stabilizer Method

Use the stabilizer to remove unwanted shaky motion and to smooth out the camera motion. The Stabilizer can smooth out motion relative to translation, rotation and scaling in an image sequence to produce a stabilized image.

The first step to stabilizing a clip is to select an appropriate stabilization method.

Auto Stabilize

- Can stabilize simple 2D motion, scaling and rotation, and perspective transformation.
- Can smooth out or pin the detected motion.
- Can remove jitter but keep camera motion.
- Can adapt resolution and advanced padding options.

Stabilizer

- Supports manual trackers.
- Enable simple padding and texture repeat options.
- Can stabilize simple 2D motion.

2D Transform

- Can stabilize simple 2D motion, scaling and rotation.
- Can adapt resolution and advanced padding options.

Simple Stabilization

Use simple stabilization to stabilize a clip where there is no pan or tilt.

To stabilize when there is no pan or tilt with the Stabilizer:

1. Position Tracker1 over a pattern that you want to stabilize.
2. Click Analyze to generate the stabilization data.
3. Fine-tune the data if necessary. See Tracking Difficult Shots and Correcting Errors (page 1091).
4. Adjust the framing options as you need (Roll, Fill, Crop Edges, Letterbox, Shift).
5. Once you are satisfied with the stabilization, go to frame 1 and click Render.

Auto Stabilize is accessed from either the Tools tab or from Batch .

To stabilize using Auto Stabilize:

1. Select 2D Stabilization from the Stabilization Method box.
   You are performing a two-dimensional analysis.
2. Switch to Front View.
3. **Optional:** Enable the Region Of Interest button.
   Indicate the region you want to analyze on the front clip.
4. Click the Analyze button.

Analysis takes place. This may take some time however you will see the percentage of progress in the analysis in the Progress field.

5 Switch to Result view to see the stabilized shot.

6 Enable the Components (Position X, Position Y) that you want to stabilize.

7 For these components select either Fixed or Smooth.

- Fixed pins the component during the analysis.
- Smooth removes the jittering with 0% being the original motion and 100% removing the highest frequency jitter.

The analysis is complete.

NOTE The default reference frame is the first frame of the clip. You can change the reference frame of the analysis by defining a new reference point.

To define a new reference point:

1 Place the positioner at the desired frame.

2 Click the Set Frame button.

The analysis will readapt the timing of the transformation; stabilization will occur before and after the set frame.

2D Transform is accessed from either the Tools tab or from Batch .

To stabilize using 2D Transform:

1 Select Pan & Scan from the Transform Type box.

You are now limited to modifying only the position and scale.

2 In the Stabilization tab, select the Enter Stabilizer button.

You are in Stabilizer.

3 Position Tracker 1 over a pattern that you want to stabilize.

4 Click Analyze to generate the stabilization data.

5 Fine-tune the data if necessary. See Tracking Difficult Shots and Correcting Errors (page 1091).

6 Once you are satisfied with the stabilization, exit back to 2D Transform.

7 Enable the Components (Position X, Position Y) that you want to stabilize.

8 Adjust the framing options as you need (Roll, Fill, Crop Edges, Letterbox, Shift) .

NOTE The default reference frame is the first frame of the clip. You can change the reference frame of the analysis by defining a new reference point.

To define a new reference point:

1 Place the positioner at the desired frame.

2 Click the Set Frame button.

The analysis will readapt the timing of the transformation; stabilization will occur before and after the set frame.

Stabilizing Motion, Scaling, and Rotation

Auto Stabilize is accessed either from the Tools tab or from Batch .

To stabilize Motion, Scaling and Rotation using Auto Stabilize:

1. Select 2D Stabilization from the Stabilization Method box.

   You are performing a two-dimensional analysis.
2. Switch to Front View.
3. **Optional:** Enable the Region Of Interest button.

   Indicate the region you want to analyze on the front clip.
4. Click the Analyze button.

   Analysis takes place. This may take some time however you will see the percentage of progress in the analysis in the Progress field.
5. Switch to Result view to see the stabilized shot.
6. Enable the Components (Position X, Position Y, Scaling, Rotation) that you want to stabilize.
7. For these components select either Fixed or Smooth.
   - Fixed pins the component during the analysis.
   - Smooth removes the jittering with 0% being the original motion and 100% removing the highest frequency jitter.

   The analysis is complete.

NOTE The default reference frame is the first frame of the clip. You can change the reference frame of the analysis by defining a new reference point.

To define a new reference point:

1. Position the positioner to the desired frame.
2. Click the Set Frame button.

   The analysis will readapt the timing of the transformation; stabilization will occur before and after the set frame.

2D Transform is accessed either from the Tools tab or from Batch .

To stabilize Motion, Scaling and Rotation using 2D Transform:

1. Select Pan & Scan from the Transform Type box.
2. In the Stabilization tab, select the Enter Stabilizer button.

   You are in Stabilizer.
3. Position Tracker 1 over a pattern that you want to stabilize.
4. For stabilizing scaling and rotation you must position Tracker 2 over a pattern.
5. Click Analyze to generate the stabilization data.
6. Fine-tune the data if necessary. See Tracking Difficult Shots and Correcting Errors (page 1091).
7. Once you are satisfied with the stabilization, exit back to 2D Transform.
8. Enable the Components (Position X, Position Y, Scaling, Rotation)that you want to stabilize.
9. Adjust the framing options as you need (Roll, Fill, Crop Edges, Letterbox, Shift).

NOTE The default reference frame is the first frame of the clip. You can change the reference frame of the analysis by defining a new reference point.

To define a new reference point:

1. Position the positioner to the desired frame.
2. Click the Set Frame button.

   The analysis will readapt the timing of the transformation; stabilization will occur before and after the set frame.

Stabilizing a Clip from Action Axis

You can import stabilizing data to lock the position of an axis in relation to a reference point on attached media. This means that any movement in the media is matched by the axis.

To stabilize a clip from Action Axis:

1. From the Action Axis menu's Tracking tab, select the media to be stabilized from the Use Media field (to stabilize the Back media, select 0).
2. In the Axis menu for the selected media, make sure that the motion path is disabled (Path button).

   NOTE Do not move the axis from its default position at the centre of the image window before entering the Stabilizer, or else the tracking data will be overwritten. Use the offset axis to add an offset to the tracker movement instead.

3. Enable Rotation, Scale or both, as needed.
4. In the Tracking Mode box, select Stabilize and click the Enter Tracker button.

   The media is automatically loaded into the Stabilizer and the Stabilizer menu appears.
5. In the Stabilizer menu, enable Context to use the Action scene as the reference.
6. In the Stabilizer, position Tracker1 over the reference point that you want to track.
7. If you enable Rotation or Scale (or both), position Tracker 2 over a point that represents the rotation or change in size of the pattern.
8. Click Analyze to generate the translation data.

   NOTE You can fine-tune the tracking data if necessary. See Tracking Difficult Shots and Correcting Errors (page 1091).

9. Once the analysis is complete, click Return.

   When you exit the Stabilizer, the X and Y Shift values for the reference position are automatically copied into the X and Y translation channels for the selected axis in Action.

   TIP After stabilizing, you can set a reference frame that has no transformations by enabling the Adjust Offset button (in the Axis tab) at the selected frame.

Removing Jitter While Keeping Overall Motion

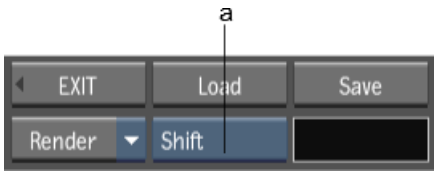
Use the Jitter option to remove the jitter from a clip while retaining the overall motion. The Stabilizer calculates the difference between an average applied to the channel by using the current Over value and the original tracking data. Applying this curve leaves only the jitter values as keyframes. Copying these curves can also be useful if you want to extract the jitter values to apply to another clip.

The Jitter option removes jitter in a clip on both the X and Y axes simultaneously, and averages the pan over time so that it appears even. Jitter control offers more flexibility than the Remove Vertical Jitter and Remove Horizontal Jitter buttons, which remove motion in one direction only. Use Remove Vertical Jitter and Remove Horizontal Jitter in simple situations, or to produce a result quickly.

Removing jitter is a process of trial and error. Try different Over values until you find one that yields good results. As a general rule, start with a large Over value over *n* frames to remove slow jitter, and start with a small Over value to remove fast jitter.

To remove jitter and keep overall motion:

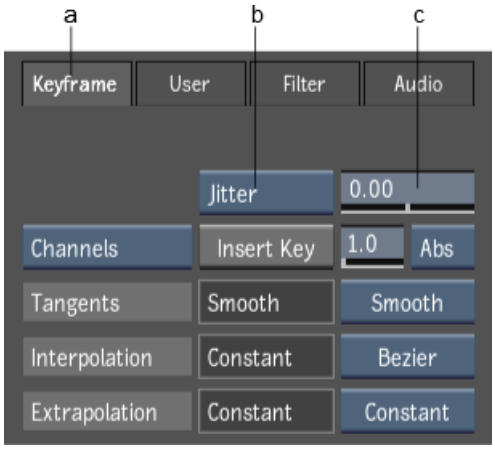
1. Position Tracker1 over a pattern you want to stabilize.
2. To automatically remove the border at the edges of the clip, select Crop Edges from the Scale and Shift Option box.



(a) Scale and Shift Option box

3. Go to the reference frame, and click Analyze to generate the Shift data.
4. Click Animation to display the Channel Editor. Select the tracker number and expand the Shift folder.

   The Shift curves turn white when the Shift channel is selected.
5. Select the Keyframe tab from the Animation Controls tabs.



(a) Keyframe tab (b) Curve option box (c) Curve Value field

6. Select Jitter from the Curve option box.
7. Enter a value in the Curve Value field.

   The transformation is applied to the curves. The curves show the amount of motion that the Stabilizer will remove. They should wrap around the zero point and should not be entirely flat; otherwise, the Stabilizer will not remove any jitter.

Quickly Removing Jitter in X and Y

The Remove Vertical Jitter and Remove Horizontal Jitter buttons remove motion in one direction only. Enable the Remove Vertical Jitter button to remove vertical jitter only. Movement on the horizontal (X)

axis, such as a camera pan, is not affected. Enable the Remove Horiztonal Jitter button to remove horizontal jitter. Movement on the vertical (Y) axis is not affected.

Smoothing Out Camera Motion

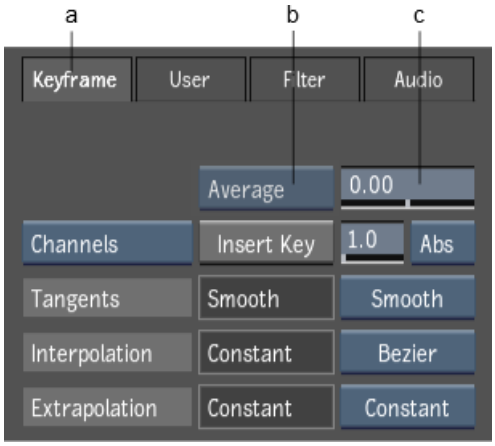
Use the Average curve options to smooth out uneven camera motion. For example, if the camera moves at a certain rate and suddenly drops or increases in speed, the Average option can stabilize the camera motion.

The Average curve smooths out camera motion over a group of keyframes, by affecting the Stabilizer's Shift values. The Shift values represent the amount of movement in a clip. Use the Curve Value field to determine how much stabilizing is applied to the clip. A larger value averages camera motion over more keyframes and increases camera smoothness. This is largely a process of trial and error.

Analyze your clip to generate the initial channel data, then select the channel to apply the average to.

To smooth out camera motion:

1. Position the tracker(s) on your image and click Analyse.
2. Click Animation to open the Channel Editor and expand the Shift folder for the track(s).
3. Do one of the following:
   - To affect the the X and Y Shift values simultaneously, select the Shift folder.
   - To affect the X Shift value, select the X parameter.
   - To affect the Y Shift value, select the Y parameter.
4. Select the Keyframe tab from the Animation Controls tabs
5. Select Average from the Curve option box.



(a) Keyframe tab(b) Curve option box (c) Curve Value field

6. Enter an average value in the Curve Value field.

The transformation is applied to the curves. The Curve Value field defaults back to 0, where 0 always represents the average of the current curve.

One-Point and Two-Point Axis Tracking

In both one-point and two-point tracking, you select the object that follows the pattern on the Action media by assigning the tracking data to its axis in Action.

To perform one-point or two-point tracking in Action Axis:

1. From the Action Axis menu's Tracking tab, select the media to be stabilized from the Use Media field (to stabilize the Back media, select 0).
2. In the Axis menu for the selected media, make sure that the motion path is disabled (Path button).
3. In the Tracking tab, enable Rotation, Scale or both, as needed. The choice depends on the movement of the object to track and on the camera movement in the clip.
4. In the Tracking Mode box, select Track and click the Enter Tracker button.

   The media is automatically loaded into the Stabilizer and the Stabilizer menu appears.
5. In the Stabilizer menu, enable Context to use the Action scene as the reference.
6. Position the tracker(s) over the pattern(s) that you want to track, and click Analyze to generate the tracking data. For one-point tracking, you use only one tracker to generate position information. For two-point tracking, you use a second tracker to generate rotation and/or scaling information. The Stabilizer obtains this information by comparing the position of Tracker2 to that of Tracker1.
   - Tracker1 follows the horizontal and vertical translation of the reference point. You should position Tracker1 over a point on the pattern that you want to track.

     NOTE The position of the axis is synced with the position of Tracker1. Therefore, any existing animation on the Axis is overwritten by the tracking data.

   - Tracker2 tracks the rotation and/or the change in size of the pattern. In the first frame, the rotation is always 0 and the scaling factor is always 100%. In subsequent frames, a rotation and/or scaling factor is added if the relative position of the two trackers changes. You should position Tracker2 over a point that represents the rotation or change in size of the pattern.

   NOTE You can fine-tune the tracking data if necessary. See Tracking Difficult Shots and Correcting Errors (page 1091).
7. Once you are satisfied with the tracking, click Return.

   The Axis menu in Action reappears. The tracking data is applied to the Axis media.

4 Corner Perspective Grid Tracking

You can track the four corners of an Action Perspective Grid in the Stabilizer. Since the Perspective Grid acts like an axis in Action, any tracking data can be applied to other Action objects.

To perform 4 corner Perspective Grid tracking:

1. From the Action Perspective Grid menu's Tracking tab, select the media to be tracked from the Use Media field (to stabilize the Back media, select 0).
2. Set your perspective grid corners.
3. Select 4 Corners from the Tracking Mode box.

4 Click the Enter Tracker button.

The media is automatically loaded into the Stabilizer and the Stabilizer menu appears.

5 In the Stabilizer menu, enable Context to use the Action scene as the reference.

6 Optional: Refine the position the trackers over the corners that you want to track, and click Analyze to generate the tracking data.

NOTE You can fine-tune the tracking data if necessary. See Tracking Difficult Shots and Correcting Errors (page 1091).

7 Once you are satisfied with the tracking, click Return.

The Perspective Grid menu in Action reappears. The tracking data is applied to the Perspective Grid media.

Vertex and Cluster Action GMask Tracking

You can track mask vertices in your Action scene from the GMask object's Tracking tab.

To track mask vertices:

1 From the Action GMask menu's Tracking tab, select the media to be tracked from the Use Media field (to track the Back media, select 0).

2 In the image window, select the vertices (page 906) you want to track.

3 Select a tracking mode:

Two types of vertex tracking are available:

- Each Vertex: Trackers are available for each selected vertex of the GMask.
- Cluster: Tracker1 represents the selected vertices, and Tracker2 represents rotation and scale (if selected).

If you select Cluster, enable Rotation, Scale or both, as needed.

4 Adjust any other Tracking Settings (page 910), as needed.

5 Click Enter Tracker.

The Tracker menu is displayed.

6 In the Tracker menu, enable Context to use the Action scene as the reference.

7 Gang the trackers.

8 Enable Auto Update Reference.

9 Click Analyze to generate the tracking data.

NOTE You can fine-tune the tracking data if necessary. See Tracking Difficult Shots and Correcting Errors (page 1091).

10 After the analysis is complete, click Return.

The GMask menu in Action reappears. The tracking data is automatically applied to the GMask media.

11 Enable Edit Track in the GMask Vertices menu to edit tracking data directly from the image window without having to switch to the Animation channel. The vertices are displayed in blue when Edit Track is enabled.

TIP If you want to add new vertices to your tracked GMask, make sure to enable Edit Track before selecting Add Points, so that the new points also follow the tracking data.

Bilinear Surface Tracking

With bilinear tracking, or four-point tracking, you use four trackers in the Stabilizer to generate tracking data.

- For anchoring the four corners of a bilinear surface to the background clip.
- For anchoring the four UV points of the surface to features of the clip.

The bilinear surface then tracked to the horizontal and vertical translation of the reference points.

See Stabilizer Menu Settings (page 1642) for an overview of the settings and options in the Stabilizer.

To track a bilinear surface:

1. Load front and back clips in Action.
2. Double-click the image node in the schematic to display the Surface menu.
3. Select Bilinear from the Surface Type box.
4. Do one of the following:
   - To track the Vertices relative to the background, click the Vertices tab to display the Stabilizer button and tracking options.
   - To track the UV points of the surface to features of the clip, click the UV Points tab to display the Stabilizer button and tracking options.
5. Select a tracking mode. Options are:
   - **Each Point**: Track the surface using a tracker for each selected vertex or UV point.
   - **Pos/Rot**: Track the surface using two proxy trackers that extrapolate the position of the selected vertices or UV points, based on changes in the position and rotation.
   - **Pos/Scale**: Track the surface using two proxy trackers that extrapolate the movement of the selected vertices or UV points, based on changes in the position and scaling.
   - **Pos/Rot/Scale**: Track the surface using two proxy trackers that extrapolate the movement of the selected vertices or UV points, based on changes in the position, scaling and rotation.
6. Click Stabilizer.

   The Stabilizer menu is displayed.

   NOTE The Position, Rotation and Scale settings of the vertices or UV points in Z are reset when entering the Stabilizer.
7. Do one of the following:
   - Position the trackers on the background clip (Vertice tracking).
   - Position the trackers on the surface texture (UV points tracking).
8. Gang the trackers.
9. Enable Auto Update Reference.
10. Click Analyze to generate the tracking data.
11. After the analysis is complete, click Return.

    The Surface menu in Action reappears. The tracking data is automatically applied to the four corners of the bilinear surface or texture.
12. Enable Edit Track in the Vertices menu to edit tracking data directly from the image window without having to switch to the Animation channel. The vertices are displayed in blue when Edit Track is enabled.

Extended Bicubic Surface Tracking

When tracking an extended bicubic surface, trackers are placed on each vertex or UV point, enabling you to track complex surfaces that move in 3D space. This is useful for tracking non-rigid objects like faces, bodies, cloth, etc. Extended bicubic tracking is particularly well suited for these types of deformations, as the vertices and the UV points are tracked individually.

With extended bicubic tracking, you use multiple trackers in the Stabilizer, based on the number of subdivisions applied to the surface. This generates tracking data for anchoring the vertices of an extended bicubic surface to the background clip or for anchoring the UV points to the extended bicubic surface.

See Stabilizer Menu Settings (page 1642) for an overview of the settings and options in the Stabilizer.

To perform extended bicubic surface:

1. Load front and back clips in Action.
2. Double-click the image node in the schematic to display the Surface menu.
3. Select Extended Bicubic from the Surface Type box.
4. Subdivide the surface, and position the vertices or UV points as needed.
5. Do one of the following:
   - To track the Vertices relative to the background, click the Vertices tab to display the Stabilizer button and tracking options.
   - To track the UV points of the surface to features of the clip, click the UV Points tab to display the Stabilizer button and tracking options.
6. Select a tracking mode. Options are:
   - **Each Point**: Track the surface using a tracker for each selected vertex or UV point.
   - **Pos/Rot**: Track the surface using two proxy trackers that extrapolate the position of the selected vertices or UV points, based on changes in the position and rotation.
   - **Pos/Scale**: Track the surface using two proxy trackers that extrapolate the movement of the selected vertices or UV points, based on changes in the position and scaling.
   - **Pos/Rot/Scale**: Track the surface using two proxy trackers that extrapolate the movement of the selected vertices or UV points, based on changes in the position, scaling and rotation.
7. Click Stabilizer.

   The Stabilizer menu is displayed.

   NOTE The Position, Rotation and Scale settings of the vertices or UV points in Z are reset when entering the Stabilizer. A notification appears on the screen when this is the case.
8. Do one of the following:
   - Position the trackers on the background clip (Vertice tracking).
   - Position the trackers on the surface texture (UV points tracking).
9. Gang the trackers.
10. Enable Auto Update Reference.
11. Click Analyze to generate the tracking data.
12. After the analysis is complete, click Return.

    The Surface menu in Action reappears. The tracking data is automatically applied to the surface or texture.
13. Enable Edit Track in the Vertices menu to edit tracking data directly from the image window without having to switch to the Animation channel. The vertices are displayed in blue when Edit Track is enabled.

Extended Bicubic SurfaceTracking Example

In this example, we want to match the motion of the cloth deformation in this t-shirt clip with the fire clip and superimpose it on top of the t-shirt.

Image courtesy of P. Fua, CVLab, EPFL, Switzerland

Proceed as follows:

1. Load front and back clips in Action.
2. Select the Fire clip in the Media list and double-click the image proxy to add it to the scene.
3. Access the Surface menu and select Extended Bicubic.
4. Access the Vertices menu, select all of the vertices and scale the image to match the t-shirt.
5. Adjust the four corners to match the t-shirt.

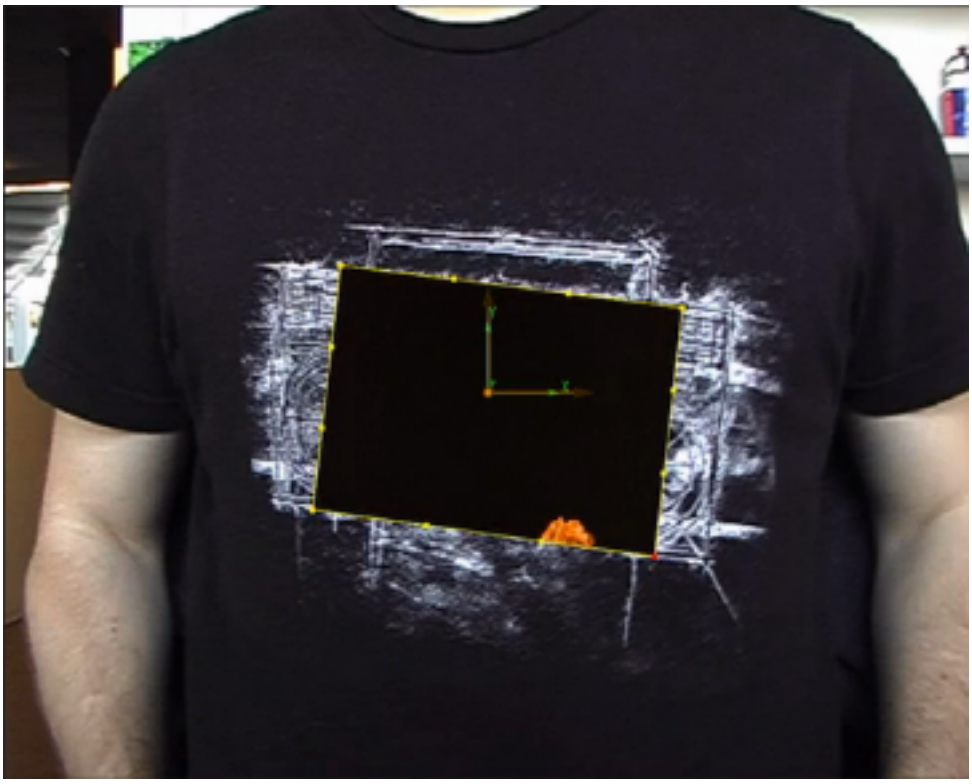


Image courtesy of P. Fua, CVLab, EPFL, Switzerland

6. Because the t-shirt deforms much more than the four corners, subdivide the surface, by clicking the Subdivide button twice.

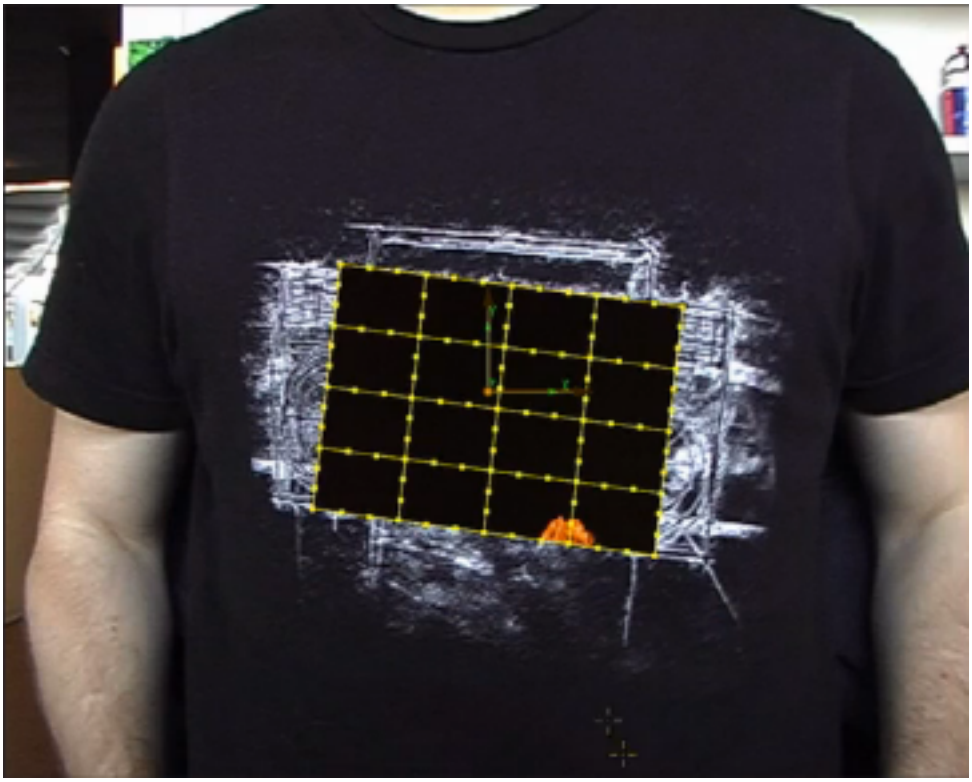


Image courtesy of P. Fua, CVLab, EPFL, Switzerland

7 Select Each Point from the Tracking Mode box.

8 Select all of the vertices.

9 Enter the Stabilizer.

We have one tracker for each vertex.

10 Gang the trackers.

11 Enable Auto Update Reference.

12 Adjust the tracker size, to adjust the sampling area. In this case, we want to make them a little bit bigger.

13 Press Analyze.

The motion is tracked.

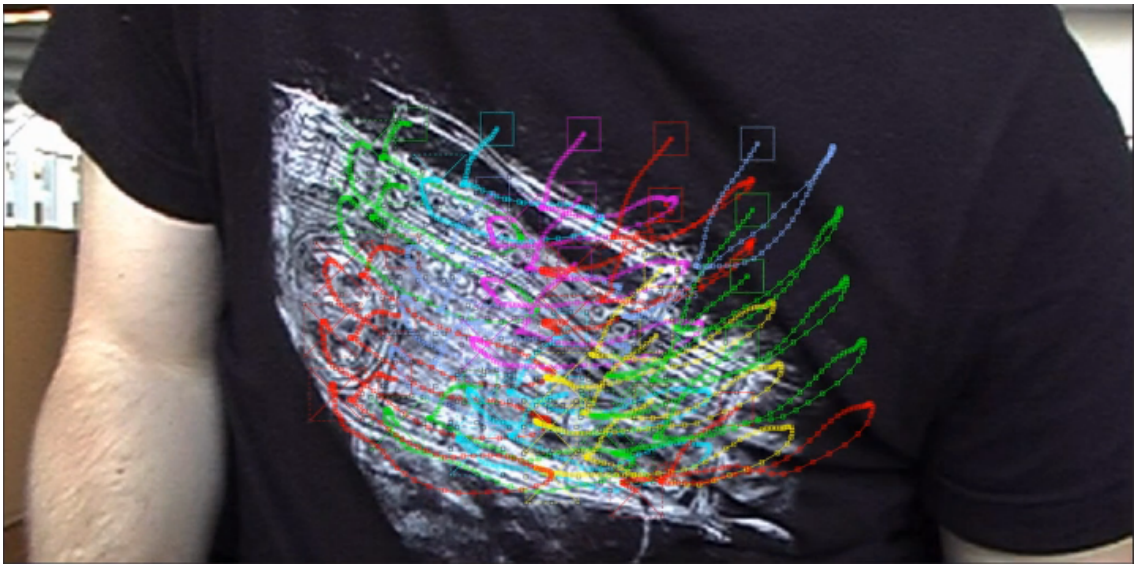


Image courtesy of P. Fua, CVLab, EPFL, Switzerland

14 Exit the Stabilizer.

The fire surface organically follows the t-shirt displacement.

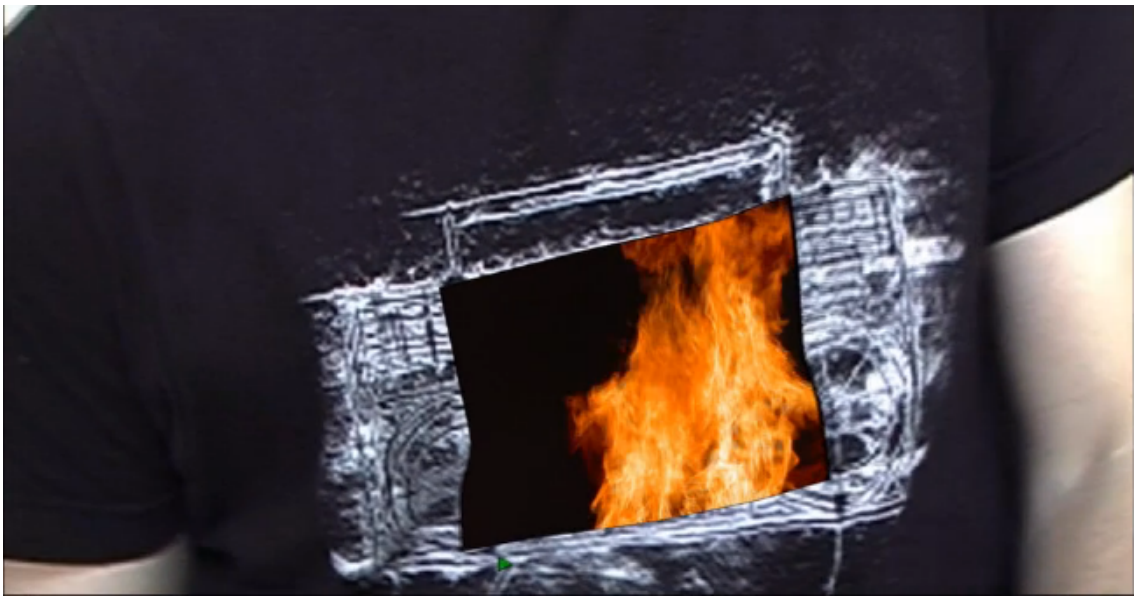


Image courtesy of P. Fua, CVLab, EPFL, Switzerland

Perspective Surface Tracking

Track a perspective surface to straighten out elements captured at an angle in the scene, to insert surface elements within a scene with camera motion and for logo replacement.

With perspective tracking, you have the ability to either track the perspective using the Vertices or to correct and stabilize the perspective using the UV Points.

See Stabilizer Menu Settings (page 1642) for an overview of the settings and options in the Stabilizer.

To track the perspective motion:

1. Load front and back clips in Action.
2. Double-click the image node in the schematic to display the Surface menu.
3. Select Perspective from the Surface Type box.
4. Click the Vertices tab to display the Stabilizer button and tracking options, to track the Vertices relative to the background (track the perspective).
5. Select Perspective from the Tracking Mode box.

   This tracking mode allows you to add as many trackers as you want and position them in the same plane as your surface, to cover as much of the perspective transformation as possible.
6. Click Stabilizer.

   The Stabilizer menu is displayed.

   NOTE The Position, Rotation and Scale settings of the vertices or UV points in Z are reset when entering the Stabilizer.
7. Position and add trackers on the background clip, as needed.
8. Gang the trackers.
9. Enable Auto Update Reference.
10. Click Analyze to generate the tracking data.

The motion is tracked

11. Exit the Stabilizer.

    The Surface menu in Action reappears. The tracking data is automatically applied to the perspective surface.
12. From the Surface menu, extrapolate the surface in X and Y, as needed.

13 Enable Edit Track in the Vertices menu to edit tracking data directly from the image window without having to switch to the Animation channel. The vertices are displayed in blue when Edit Track is enabled.

To correct and stabilize the perspective:

1 Access the Surface menu.

2 Select Perspective from the Surface Type box.

3 Position the UV points to isolate and correct the perspective on the desired portion of the shot.

4 Select the four UV points.

5 Click the UV Points tab to display the Stabilizer button and the tracking options.

6 Select Perspective from the Tracking Mode box.

This tracking mode allows you to add as many trackers as you want and position them in the same plane as your surface, to cover as much of the perspective transformation as possible.

7 In the Stabilizer Option box, make sure Track is selected.

8 Click Stabilizer.

The Stabilizer menu is displayed.

NOTE The Position, Rotation and Scale settings of the vertices or UV points in Z are reset when entering the Stabilizer.

9 Add trackers and position them as needed.

10 Gang the trackers.

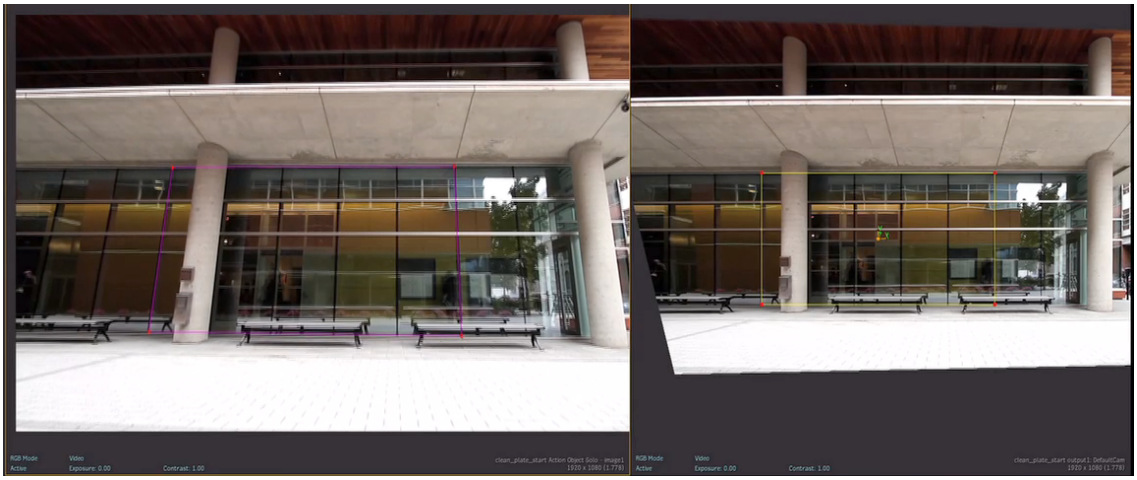
11 Enable Auto Update Reference.

12 Click Analyze to generate the tracking data.

The motion is tracked.

13 Exit the Stabilizer.

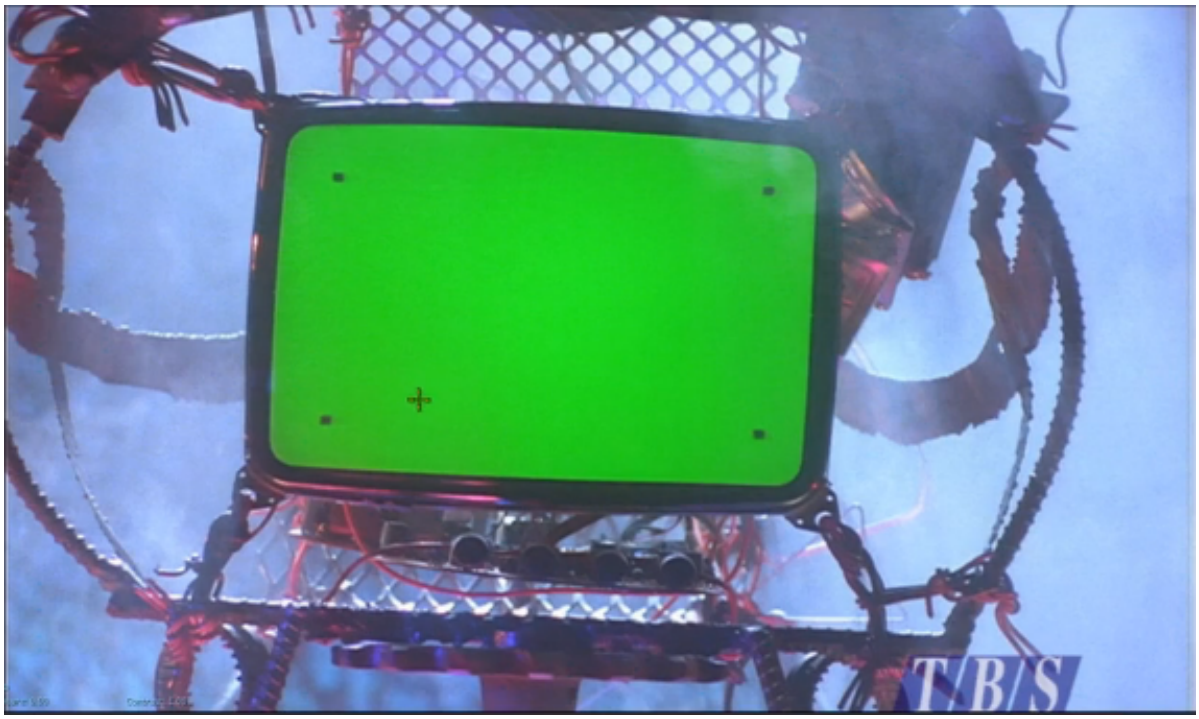
The perspective is corrected and stabilized.



Offsetting the Tracking Motion of the Reference

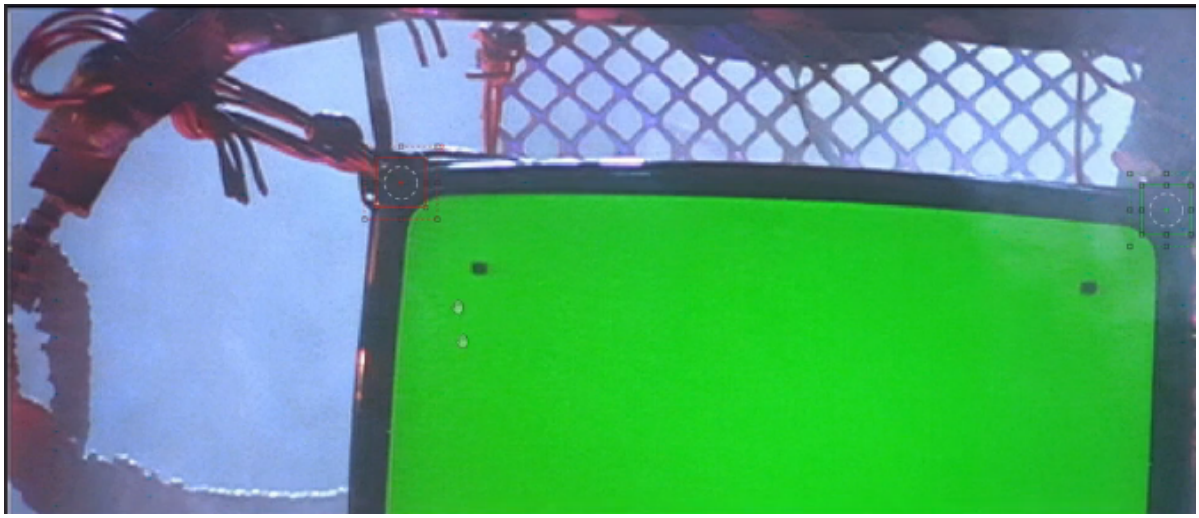
You can apply the tracking information from one point to another point in the image. A dotted line is displayed between the tracker and the offset target, using the Offset Reference tool. This is useful when you want to track a portion of an image that does not have an adequate tracking anchor point.

In this example, we want to replace the green screen in the TV clip with another shot.



Images courtesy of Turner Broadcasting Systems

However, when entering the stabilizer, we can see that we don't have proper anchor points on the TV clip, where the four corners of our replacement shot are positioned.



To resolve this, we can offset the reference. To do this, proceed as follows:

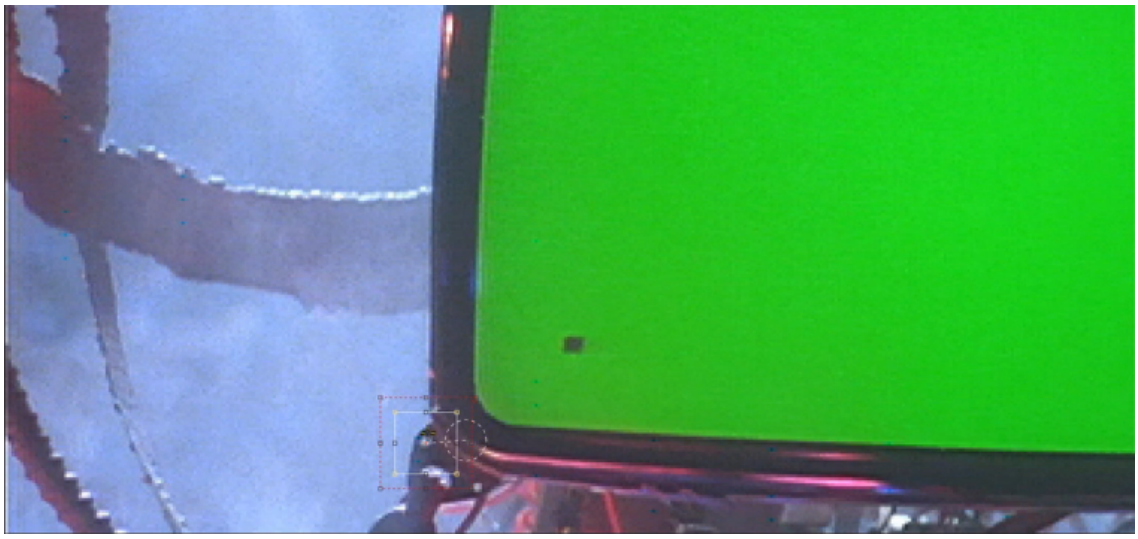
1 From the Stabilizer Tools menu, select Offset Reference.

This enables you to place the tracker on a proper anchor point, but apply the tracking data to the original portion of the image, which does not have an adequate anchor point.

2 Place the trackers on adequate anchor points.

NOTE The trackers should be placed relatively close to the offset target, or on an anchor point that follows the same movement as the offset target, otherwise the detected motion will be off and will drift from the motion of the offset target.

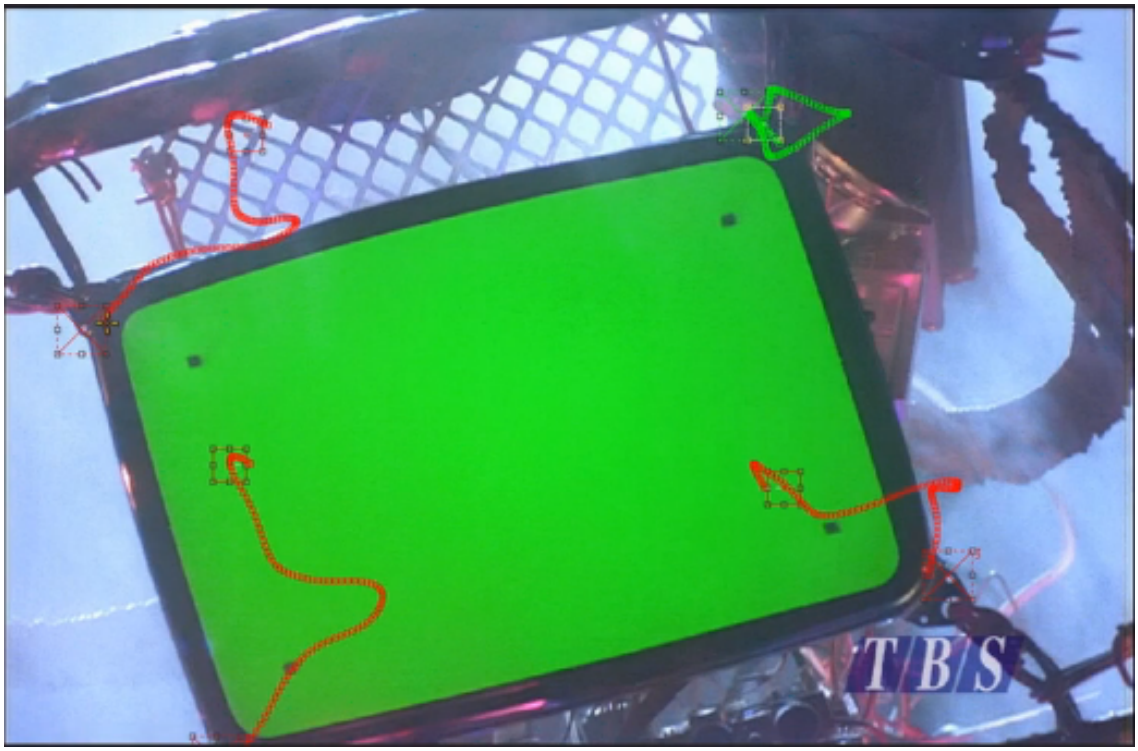
A dotted line is displayed between the tracker and the offset target.



3 Gang the trackers.

4 Enable Auto Update Reference.

5 Press Analyze.



The motion is tracked. The reference is the anchor point on which the tracker was placed, but the motion is applied to the offset target.

6 Exit the Stabilizer.

The tracking data is applied to the offset targets. The corners of the replacement image are still at their original position, where there were no adequate anchor points, but the motion and the positioning is correct.

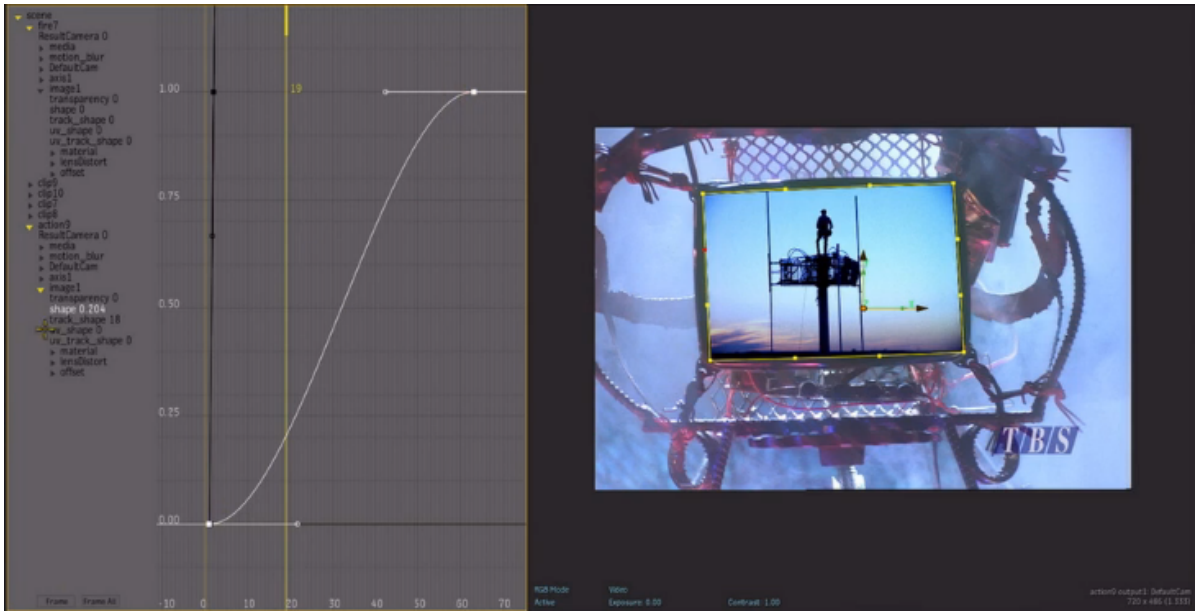
Adjusting the Surface With the User Shape Channel

When tracking surfaces in Action, you have access to a user shape channel, that you can use to fine tune or to add additional deformation to the existing tracking animation, without affecting the tracking animation.

Once the surface has been tracked, you can adjust the vertices, the UV points or the tangents of the surface, to add deformation or to adjust the position of the surface. This creates a keyframe at the timecode you specified, which is stored in the user shape channel and does not interfere with the tracking animation.

In this example, we adjust the tangents of the tracked extended bicubic surface, at frame 63, curving the surface for it to fill the TV screen.

Looking at the user shape channel, in the Animation Editor, we see that only one keyframe is added, at frame 63. The adjustment or deformation persists after the last keyframe of the user shape channel and does not interfere with the existing tracking animation.



Tracking Difficult Shots and Correcting Errors

If the tracker box strays from the reference point that it is supposed to be tracking, incorrect shift keyframes result. If such an error occurs, you can stop the analysis, correct it, and restart it at any frame.

This section covers the different strategies you can use to track difficult shots. After trying any of the strategies, you will need to redo the analysis. Click Analyze to generate new keyframes based on the updated information you provide.

Adjusting the Size of the Tracker Box

If the Stabilizer cannot find the reference point within the boundaries of the tracker box during analysis, the tracker box will stray from the reference point and produce incorrect keyframes. Although you can manually correct these keyframes, it is easier to make the tracker box large enough to accommodate the movement of the reference point. Note that processing time increases as the size of the tracker box increases. See Resizing the Reference and Tracker Boxes (page 1069).

To adjust the size of the tracker box:

1. Press on the pen or hold down the left mouse button to stop the analysis.

   NOTE Make sure to press below the viewer to avoid moving the tracker box.

2. Go back to the last good frame before the tracker box strayed from the reference point.
3. Adjust the size of the tracker box so that it is large enough to accommodate the largest frame-to-frame movement of the reference point.
4. Click Snap, and then click Analyze.

Positioning the Tracker Box Manually During Analysis

When the reference point is temporarily covered by another object, position the tracker box manually.

When you manually position the tracker box, the new position is recorded in the Track X and Track Y channels of the Channel Editor. These values have precedence over the Shift X and Shift Y values.

To manually position the tracker box:

1. Press on the pen or hold down the left mouse button to stop the analysis.

   NOTE Make sure to press below the viewer to avoid moving the tracker box.

2. Advance the clip to the frame where the reference point becomes visible again, and reposition the tracker box over the reference point. Click Analyse to restart the analysis at this frame.

   Since the X and Y shifts are recorded as keyframes in channels, the Stabilizer calculates the translation values for the frames in which the reference point was covered. The final result will be a smooth motion.

Locking Keyframes in Place

You can lock Shift keyframes so that they stay in place even if you try to move them manually or perform an analysis. This is useful, for example, when you set keyframes manually and want to prevent Flame from overwriting them when you redo the analysis.

In the following procedure, you select the keyframes directly in the image window. However, you can also select the Shift keyframes in the Channel Editor.

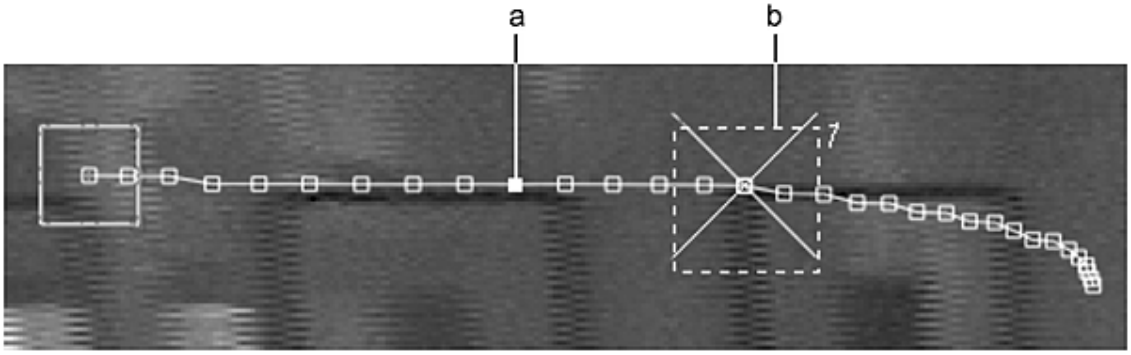
NOTE You cannot lock several keyframes at once. You must lock each keyframe individually.

To lock keyframes in place:

1. Zoom in on the image until you can see the points clearly and select the point you want to lock on the tracking path.

   The tracking path becomes white when you click it, and the frame that corresponds to the keyframe appears in the image window.

   NOTE If the tracking path does not appear in the image window, enable Path in the Setup menu.



(a) The inside of locked keyframes are filled (b) The tracker box appears when you select a keyframe

2. Click Lock Key.

3 To unlock a keyframe, select it and click Unlock Key. The Lock Key button becomes Unlock Key when a locked frame is selected.

Changing the Reference Point

Change the reference point to another feature if the feature you have been tracking moves out of the frame, or alters such that it is no longer trackable. You should do this before the original reference point moves out of the frame. This allows the Stabilizer to calculate the offset between the two reference points and predict the position of the original reference point. If the Path button in the Setup menu is enabled, you will see the extrapolated path of the original reference point.

Change the reference point before analyzing the clip.

To change the reference point:

1 Click Setup and then enable Path.
2 Select the tracker for which you want to change the reference point.
3 Display the desired frame in the image window.
4 Using the Select tool from the Stabilizer Tools menu, drag the reference box and position it on the new reference point.
5 Go to the first frame of the sequence and click Analyze.

Finding the Best Tolerance Value

The Tolerance value determines how much discrimination the Stabilizer uses in matching the reference point from frame to frame and in setting a keyframe for that frame. At 100% tolerance (no discrimination), almost anything is considered as a match and a keyframe is set. At 0% tolerance (complete discrimination), only a perfect match is accepted.

If the reference point becomes hidden by another object, you can use the Tolerance value to make the tracker ignore the reference point in parts of the clip where it is hidden, then continue tracking it normally when it reappears. Choosing a good Tolerance value reduces the need to manually reposition the tracker box during analysis.

TIP You can also adjust the tolerance to make a valid keyframe that has been filtered out reappear.

To find the best Tolerance value:

1 Click Setup and then enable Path to view the tracking path.
2 Analyze the clip with full (100%) tolerance.

3 Go to the first frame where the reference point is hidden by another object and adjust the tolerance value until the crosshair reappears in the tracker box.

   The crosshair reappears when there is a keyframe.

4 Go to the first frame in the sequence and click Analyze.

Once you determine the optimal tolerance value, further analysis automatically discards the unwanted keyframes, and the position of the reference point is extrapolated until the reference point reappears.

Analyzing Backward

Analyze the clip backwards when the pattern you want to track grows larger or when it is off screen at the beginning of the clip.

To analyze backward:

1 Select Backward from the Direction box.
2 Go to the last frame of the sequence.
3 Position the reference and tracker boxes.
4 Click Analyze.

Manually Tracking a Pattern that Disappears

Track manually when the pattern you are tracking disappears behind an object for several frames, moves out of the frame, or is difficult to track.

To track a pattern that disappears:

1 Go to the last keyframe before the reference point disappears.
2 Click Snap to redefine the reference point on this frame.

3 Using the Select tool from the Stabilizer Tools menu, drag the tracker box to the required position in the current frame.

Use the reference image you see in transparency as a guide to find the new position. A keyframe is set and the timeline advances one frame forward or backward when you release the pen. The direction of the timeline depends on the setting in the Direction box.

NOTE If you do not see the reference image in transparency, set the Opacity option in the Setup menu to approximately 50%.

4 Repeat step 4 as necessary.

TIP Lock the keyframes as you add them.

Analyzing One Frame at a Time

Analyzing one frame at a time is useful for difficult shots because you can adjust the position of the tracker after each frame. Click the Step button to analyse a single frame and advance to the next frame.

Editing the Stabilizer Channels on the Image

You can edit the tracker channels directly on the image instead of editing them in the Channel Editor. Note that the curve on the image shows the X and Y components together and not separately like in the Channel Editor. The tracking path on the image shows the actual displacement of the tracker box.

To edit the curve directly in the image window, Path must be enabled in the Setup menu. When you select a keyframe on the curve, the positioner goes to the frame that corresponds to the keyframe. When you select the reference box, the positioner goes to the frame where you set the reference.

Tracking a Degraded Copy of the Clip

If other options do not work, you can try degrading a copy of the image until the contrast of the pattern to track is high enough. You then generate tracking data from this copy, save the setup, and apply it to the original clip. Try the following methods for degrading the clip:

- Saturate the colours in the Colour Corrector.
- Increase the contrast in the Colour Corrector.
- Apply filters (for example, Emboss).
- Remove film grain using Degrain.

Changing the Colour of Trackers

When you add a new tracker, the system automatically assigns it a unique colour so that you can easily distinguish between multiple trackers. You can change the colour of an individual tracker or of all the trackers at once.

To change the colour of the tracker:

1. From the Tracker box, select the tracker you want to change.

   TIP To change all trackers, select Gang in the Tracker Selection box.

2. Click the Tracker colour pot.
3. Select a colour with the colour picker.
4. Click in the Tracker colour pot to apply the new colour to the tracker.

Pretracking a Clip

Before analyzing, you can use the Pretracking option in the Setup menu to preview the motion path for a specified number of frames. You can then adjust the tracker position, if necessary, to find the best reference point. The Pretracking option applies only to the selected tracker, regardless of whether you selected Solo, Selected, or Gang in the Tracker Selection box. If you move or resize a tracker with Pretracking enabled, the next frames are analyzed.

To pretrack a clip:

1. Position and select a tracker.
2. In the Setup menu, make sure that Path is enabled. If it is disabled, Pretracking is not available.
3. Enter the number of frames to pretrack.

   NOTE If you enter a number larger than the length of the clip, only existing frames are pre-tracked.

4. Enable Pretracking.
   The motion path appears.

   NOTE Frames, not fields, are pretracked. If you are in Fields mode, frames will be pretracked.

Copying Shift Channels

Using the Channel Editor, you can copy a tracker's Shift channel values and paste them into any other channel in any module. For example, you can copy jitter values to the X and Y position channels of an axis to add realism to a static scene, or to a channel to add noise.

To copy and paste a shift channel:

1. In the Channel Editor, open the folder of the tracker whose shift values you want to copy.
2. Select the X or Y Shift channel, or the entire Shift folder to select both X and Y Shift channels.
3. Click Copy.
4. If necessary, load the clip to which you want to apply the Shift values into the appropriate tool.
5. Open the folder of the channel to receive the copied shift values and select the appropriate channel(s).
6. Click Paste.

Importing and Exporting Data

Import and export tracking and shift data values to an ASCII file using the Import and Export buttons located in the Track and Shift areas of the Stabilizer menu.

The tracking data specifies the X and Y position of the tracker box in relation to the origin point of the image; these are absolute coordinate values. The origin point has coordinates (0,0), and is located at the lower-left corner of the image.

The shift data specifies the difference between the position of the reference box (0,0) and the position of the tracker box in the current frame. These are relative coordinate values.

Both the Tracking values and the Shift values are formatted as follows:

frame#: X position, Y position

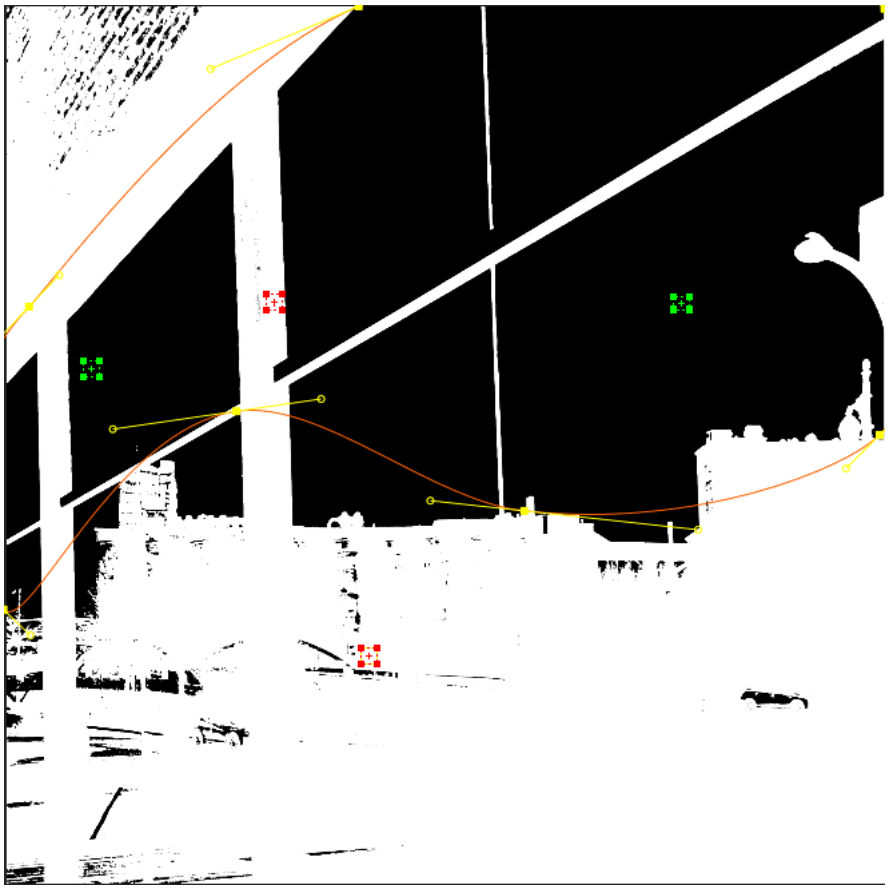
NOTE If the position of the reference box is changed during the analysis, the offset is compensated for and the exported tracking data still shows a continuous tracker path.

Masking and Rotoscoping

20

GMask Tracer

The GMask Tracer is a tool dedicated to matte creation, and is based on the Action ecosystem; thus offering a number of advantages over traditional Garbage Masks.



Accessing the GMask Tracer

You can access the GMask Tracer using the following methods:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.
  Use the GMask Tracer Start Mode in the Modular Keyer as the default keying pipeline.

  TIP You can also use the GMask Tracer node in the Colour Blend (CBlend) and Matte Blend (MBlend) pipes in Batch, Batch FX, and the Modular Keyer.

- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
  When you access the GMask Tracer as a Timeline FX, you can enable Use Matte to use the matte input as part of the effect processing.

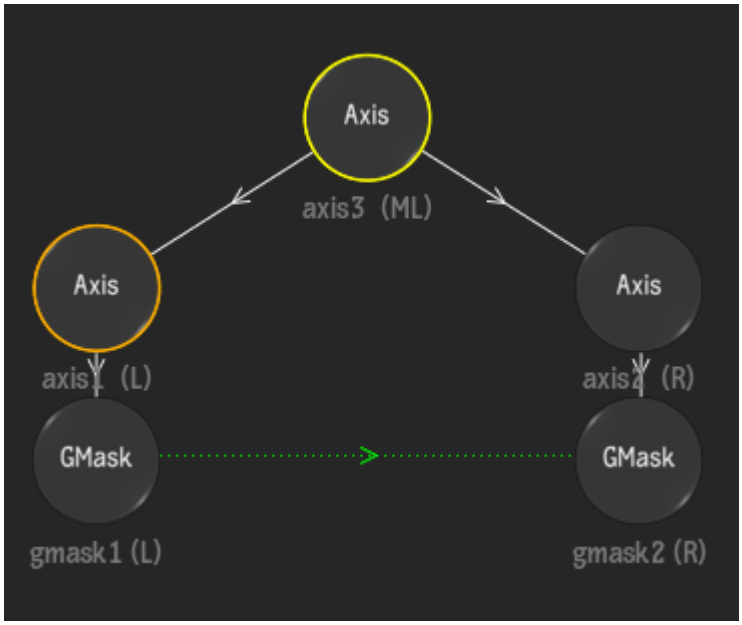
NOTE By default, the GMask Tracer accepts a front and matte clip, and outputs an outmatte.

Using the GMask Tracer in Stereo Mode

If you want to work in stereo mode with the GMask Tracer:

- From the Tools menu, make sure that you select a stereo input (Left Front/Matte / Right Front/Matte or Left Front / Right Front).
- From Batch: make sure that you enable Stereo Mode (below the node bin) first before adding the GMask Tracer node, so that the proper stereo inputs are present.

In this case, once you enter the GMask Tracer and add a GMask node, a stereo rig and stereo camera are added to the schematic. In the menu for the parent Axis node, you can set the stereo convergence (Convergence is displayed as Offset in the Channel Editor under the parent Axis folder).



The GMask Tracer User Interface

Once you enter the GMask Tracer user interface, you should be familiar with the look and layout, as it resembles the Action interface (page 559), with a schematic, node bin, priority editor, and output menu.

The GMask node is usually the primary node that you use within the GMask Tracer, but many other nodes are available in the GMask Tracer node bin to help you create your mattes. You can create a new GMask quickly without using the node bin by using the GMask: Create Node shortcut. Press **Alt+N** to automatically switch to Draw Shape mode, then once you add your first point, a GMask node is automatically added to the schematic.

Follow the links to the comparable Action sections:

- 3D Shape (page 752)
- 3D Text (page 723)
- Analyzer (Mono or Stereo) (page 871)
- Axis (page 618)
- Camera (page 840)
- Camera 3D (page 856)
- GMask (including Ellipse, Plane, and Rectangle presets) (page 900)
- Image (Surface) (page 623)
- Import (3D Geometry) (page 699)
- Matchbox (attached to textures or cameras) (page 821)
- Perspective Grid (page 653)
- Replica (page 832)
- Stereo Object (page 647)

Using the GMask Tracer

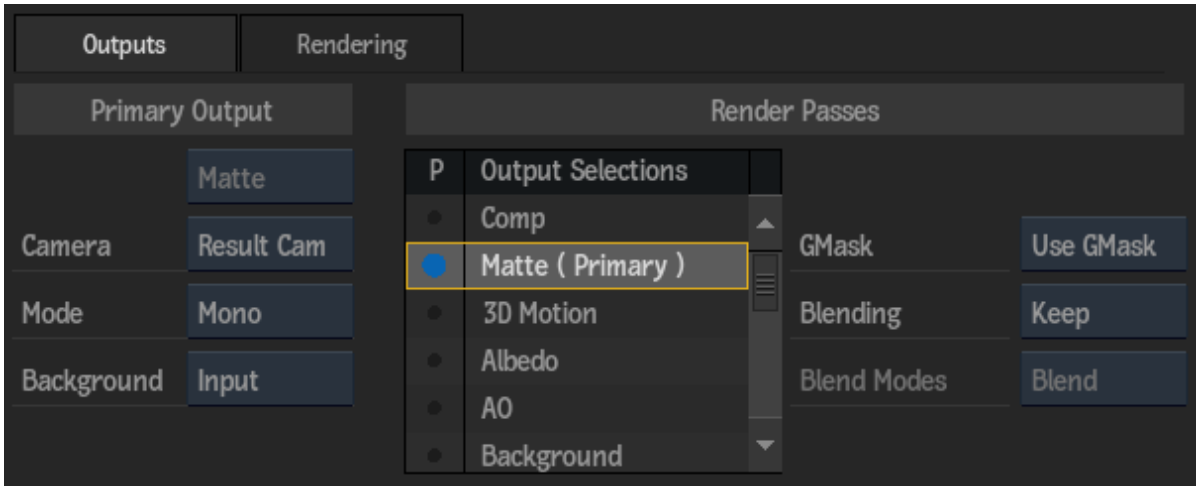
The same workflow (with a few minor differences) is possible whether you work in the GMask Tracer node or Action to perform matte creation. If you only need to create a matte for a particular image, then the GMask Tracer may be less taxing than Action to accomplish the task.

Here are a few tips for working in the GMask Tracer:

- You can save and load setups between GMask Tracer nodes, and also between GMask Tracer and Action nodes. You can also copy and paste compatible nodes between GMask Tracer and Action nodes.
- When accessed from the GMask Tracer, the GMask menu' s Mask tab has one extra setting: the GMask Tracer Colour field is available for you to set the blend percentage between the outgoing and incoming images inside the mask.
- You may notice a difference when using GMask blend modes within the GMask Tracer node versus Action, since the GMask Tracer works with images, while Action GMasks work with geometries.
- You can perform location-based keying operations on a GMask within the GMask Tracer or Action, using the Tracer settings. The tracer allows each GMask to host a key based on its associated media. See Creating a Location-Based Key With the GMask Tracer (page 912).

Rendering Outputs from the GMask Tracer

Use the multi-pass rendering capabilities of the GMask Tracer's Output menu to manage your outputs. You are limited to Matte primary outputs, but you can have multiple render passes. The overall functionality of the Output menu is otherwise similar to the Action Output menu (page 598).



The Comp output of a GMask Tracer outputs the input, so it can be used as a pass-through, or can be used in the Camera FX pipeline.

One difference from Action, is that you have an extra option in the Matte Background box for GMask Tracer outputs. The Input option allows you to set the incoming Media Front and Matte as the Background for both the Comp and Matte outputs respectively.

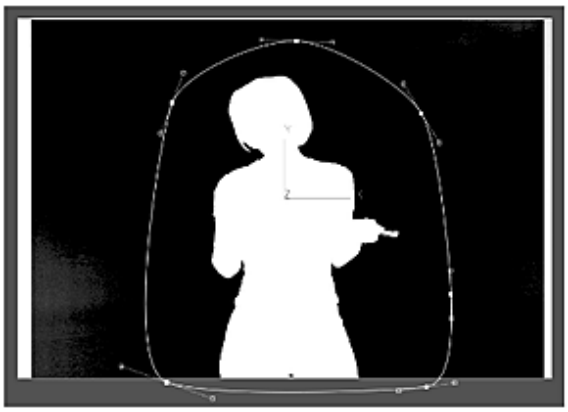
NOTE If you accessed the GMask Tracer as a Timeline FX, you are limited one Matte output.

GMask

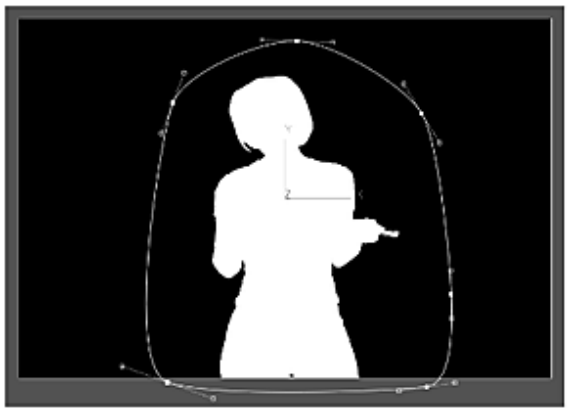
When pulling a key, use garbage masks (GMasks) to isolate particular areas of an image to include with, or exclude from, the opaque area of the matte. Garbage masks are spline-based objects you draw directly on a clip or image. Depending on what you want to accomplish, you can set the area inside each garbage mask to be either opaque (white), transparent (black), or semi-transparent (any shade of grey).

Garbage masks are useful in many situations, such as when removing unwanted elements like equipment or people from a blue-screen shot, when removing unwanted borders from an image, or when creating complex keys by blending portions of the front and back clips. You can animate the shape of a garbage mask, apply motion blur to it, and even use the Stabilizer to make a garbage mask follow a moving element in a clip.

The following example illustrates keying out an unwanted white border using a garbage mask.



Matte before garbage mask is applied



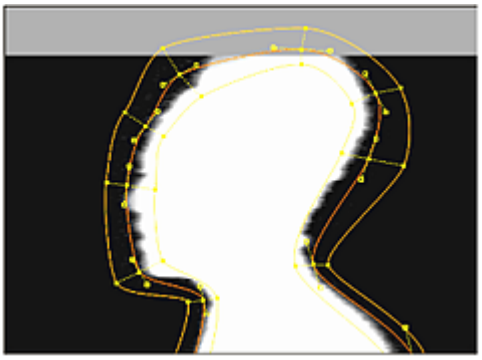
Matte after garbage mask is applied

You can also apply softness to the edge of a mask using a uniform or advanced gradient. Instead of having an abrupt transition from white to black in the matte, you can control the gradations of grey between the key and the background. For example, softness can compensate for uneven edges of a key by blending the front and back clip. You can apply uniform softness around the edge of a mask or vary the softness range for different parts of the mask.

An advanced gradient adds two splines around the garbage mask. By moving points on these splines, you can control the gradient according to how far the surrounding spline is offset from the garbage mask.

Softness applied around edge of garbage mask spline

Image courtesy of Behavior Communications Inc.



Advanced gradient varies softness for different parts of the garbage mask

Image courtesy of Behavior Communications Inc.

Accessing the Garbage Mask Menu

Access the Garbage Mask menu from the Modular Keyer or the GMask node when you want to use multiple garbage masks or use the Tracer for detailed masks.

To access the garbage mask, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).

  When you access the GMask as a Timeline FX, you can enable Use Matte to use the matte input as part of the effect processing.
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

NOTE A different GMask node is available from within Action (page 900) and the Mask Tracer (page 1099).

About Garbage Mask Options

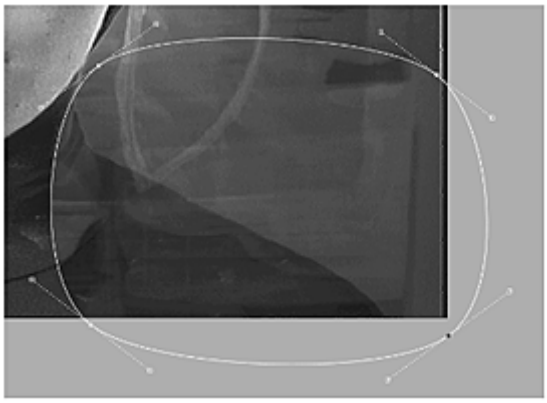
You can customize a mask's properties in the Garbage Mask menu.

Render Mask button Turns masks on or off as you work. A mask that is turned off can be seen in the image window, but it has no effect on the image. Masks are turned on by default.garbage masks:turning on and offConstant shape, for garbage masks

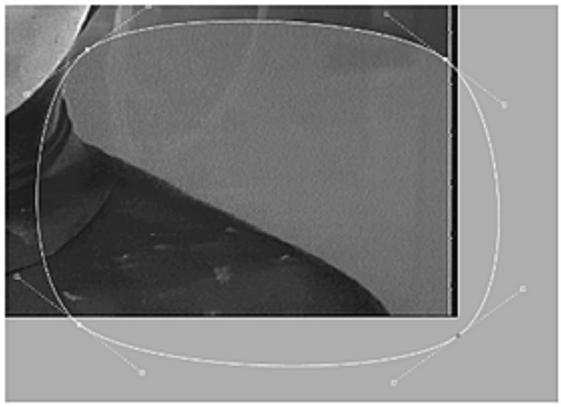
NOTE This parameter cannot be animated.

Outside button Applies the effect to the part of the image that is outside the mask.

Colour field Sets the blend value between the front and back image in the area that the mask is affecting (that is, either outside or inside the mask). A value of 50% is a 50/50 blend between the front and back clip. A value of 100% is the front clip. A value of 0% is the back clip.

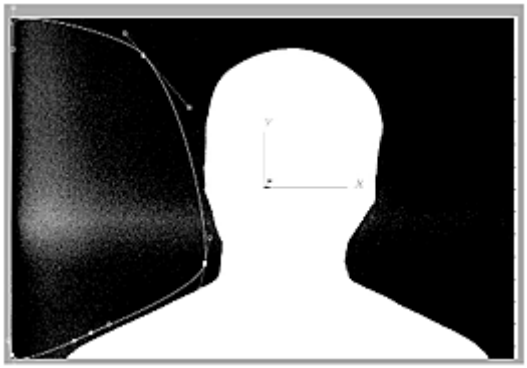


Mask with 25% colour

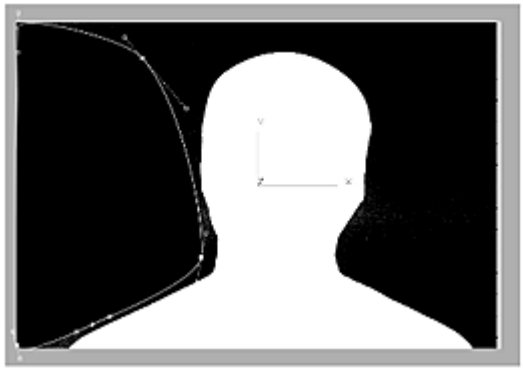


Mask with 75% colour

Opacity field Sets the opacity of the matte in the area affected by the mask (that is, either inside or outside the mask). A value of 100% means the matte is completely opaque; 50% means the matte is 50% transparent; and 0% has no effect on the image.



Mask with 0% opacity



Mask with 100% opacity

Lasso Fit field Increase or decrease the number of points in the segments of the mask that are drawn freehand. See Adjusting the Number of Points in Freehand Segments (page 1109).

Edge Softness box Apply a uniform gradient or use an advanced gradient to set different levels of softness for different parts of the mask edge. In the GMask node in Batch or Batch FX or the Modular Keyer, use the Tracer to set variable softness around the mask edge using pickers. See Refining the Mask (page 1124).

Alpha field Defines the transparency of the gradient from the edge of the mask.

Offset field Defines the border position of the gradient from the edge of the mask.

Inner Edge, **Outer Edge**, **Distance fields** Applies the distance and degree of change between the opaque and transparent part of the mask edge.

Linear Interp Enable to use linear interpolation of the mask border between keyframes. Disable to use rounded interpolation.

Constant Shape button Enable to modify the mask's shape without setting keyframes. This forces all animatable parameters (except the Tracer parameter Sample On) to be set for the whole clip rather than for only the current frame. It also removes any existing keyframes and applies the shape of the current frame to the rest of the clip.

Splines button Enable the display of softness, borders and pickers.

Borders button Enable the display of borders. The Border button is active when the Advanced Gradient and Tracer menus are displayed.

Pickers button Enable the display of pickers. This element is only available when accessing the Tracer tool in the Garbage Mask menu from the GMask node in Batch or Batch FX or the Modular Keyer.

X/Y fields Defines the mask's X and Y offset from its axis.

Setting Mask Options

You can control how a mask affects the matte. For example:

- Set the opacity, colour, and edge softness of the mask.
- Specify whether the effect is applied to the inside or outside of the mask.
- Adjust the offset of the mask from its axis.
- Turn a mask on or off.

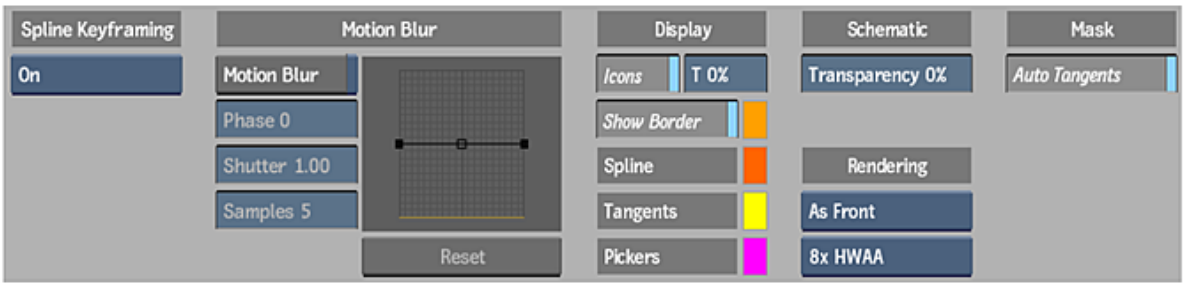
To set mask options:

1. Display the Matte or Result view.

2. Select the mask in the image window:
   - Click anywhere on the garbage mask spline.
   - Click the Geom node in the schematic.
3. Access the Garbage Mask menu.
4. Set the mask options. See About Garbage Mask Options (page 1103).

About Drawing Options

Use the Mask Setup menu to access mask drawing options.



Spline Keyframing box Determines under what conditions keys are set, and for which parameters, when you are animating a garbage mask spline. See Animating Masks Using Spline Keyframing (page 1115).

Motion Blur settings Controls motion blur applied to garbage masks that can be used to match the movement of objects in a clip. See Applying Motion Blur to Garbage Masks (page 1120).

NOTE There are additional setup options to do with animating garbage masks. These only appear when you disable Shape Animation in the Garbage Mask menu. For more information, see Animating Masks Using Spline Keyframing (page 1115).

Icons button Globally displays or hides the garbage mask splines and axes.

NOTE If you hide garbage maks splines and axes with the Icons button, the Widget All / Widget Sel button in the View menu has no effect.

Transparency field Sets the transparency for the garbage mask splines and axes.

Show Border button Displays the Softness Offset wireframe border, defined in the Shape menu.wireframe:in garbage masks

Show Border colour pot Sets the colour for the Softness Offset wireframe.

Spline colour pot Sets the colour for the wireframe of the garbage mask.

Tangents colour pot Sets the colour of the garbage mask tangents and vertices.

Pickers Display colour pot Sets the colour of the Tracer's pickers. This element is only available when accessing the Garbage Mask Setup menu from the GMask node.

Schematic Display Transparency field Sets the transparency of the nodes in the garbage mask schematic.

Rendering box Select the rendering method.

Hardware Anti-aliasing Sample box Select a hardware anti-aliasing sampling level to accelerate edge anti-aliasing with no performance penalty. The graphics hardware automatically renders the image at full speed with approximately the equivalent of up to 32 samples of anti-aliasing (depending on your graphics card and project graphic bit depth). Hardware anti-aliasing also gives anti-aliasing during normal interaction instead of only while rendering.

Auto Tangents button Positions a tangent for each vertex set to create a smooth curve between the vertices. When Auto Tangents is disabled, the tangents are positioned under the vertex, resulting in straight lines between vertices. When you draw freehand segments in a mask with Auto Tangents off, vertices are added with broken tangents, allowing the mask to follow your cursor movement.

Drawing a Mask

After setting the drawing options, you can draw the mask.

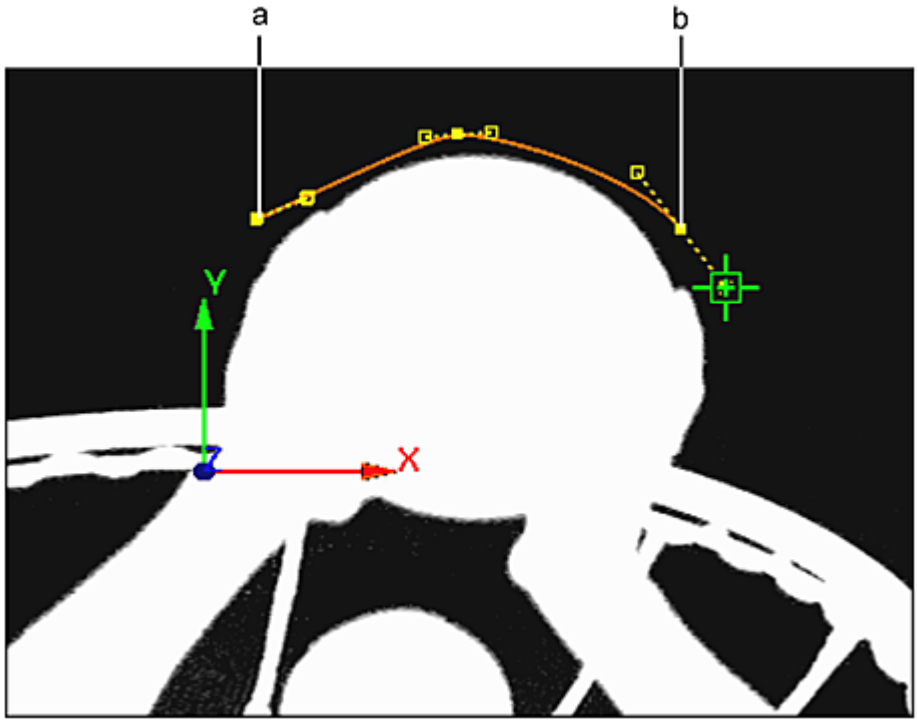
To draw a garbage mask:

1. Access the Garbage Mask menu.
2. Select the view where you want to draw the mask from the View box.

   Which view you use depends on the context of your key and why you are drawing the mask. For example, if you want to crop the edges of a clip by drawing an opaque mask around the subject and then turning on Outside, try drawing the mask using the Matte view. Or, if you are rotoscoping a subject that has not been shot on a blue or green screen, try either the Front or Result view.

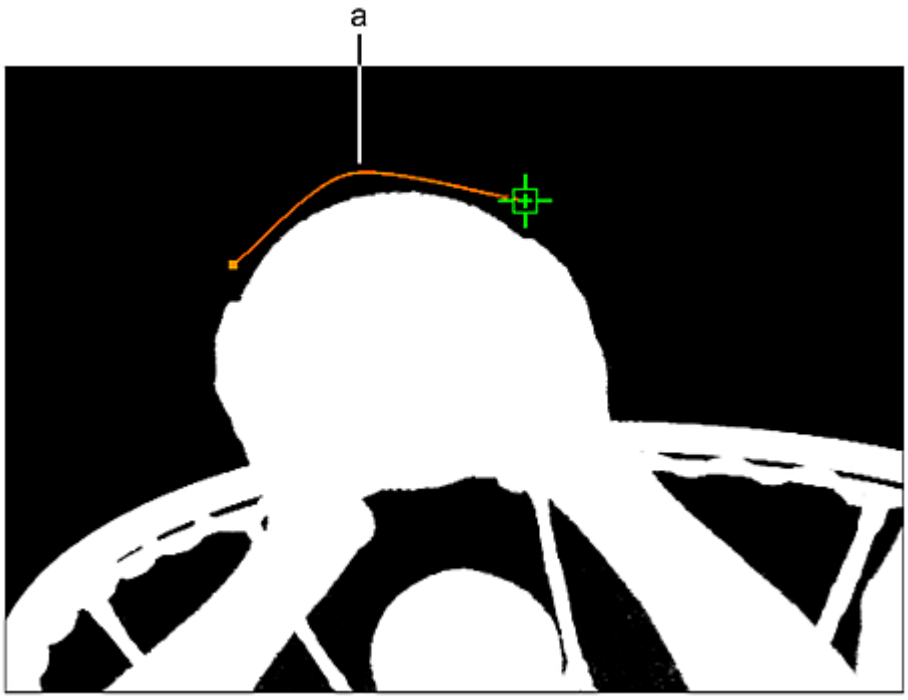
   TIP In Batch or Batch FX or the Modular Keyer, you can use multiple viewports as you draw the garbage mask. For example, you can draw the mask in Matte view, while also displaying Result view. Using multiple viewports, you can immediately see how your mask affects the clip.

3. Do one of the following:
   - Click Add.
   - Select Create from the Tools box.
   - Press `N`.
4. Do any of the following to draw the garbage mask:
   - Click to add vertices.



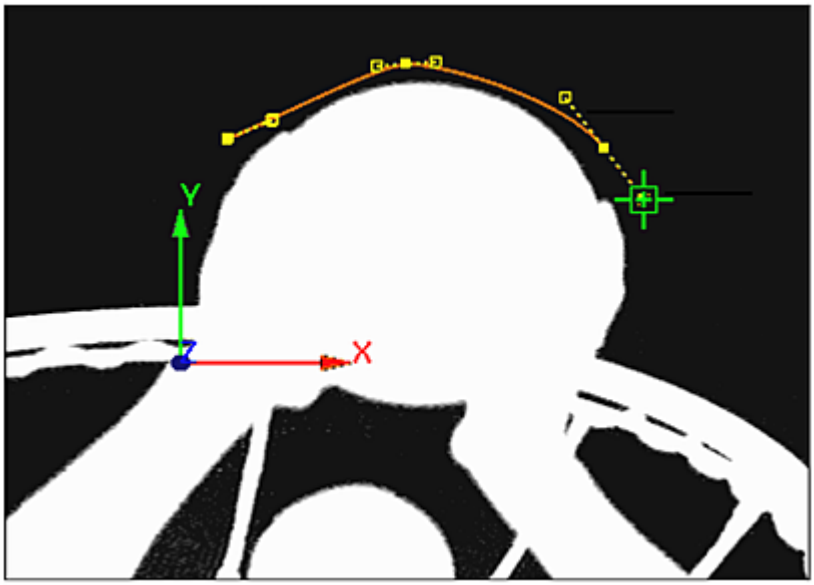
(a) Vertex added by first click (b) Vertex added by third click

- Press `Shift` to temporarily hide the vertices and tangents as you are drawing the mask.



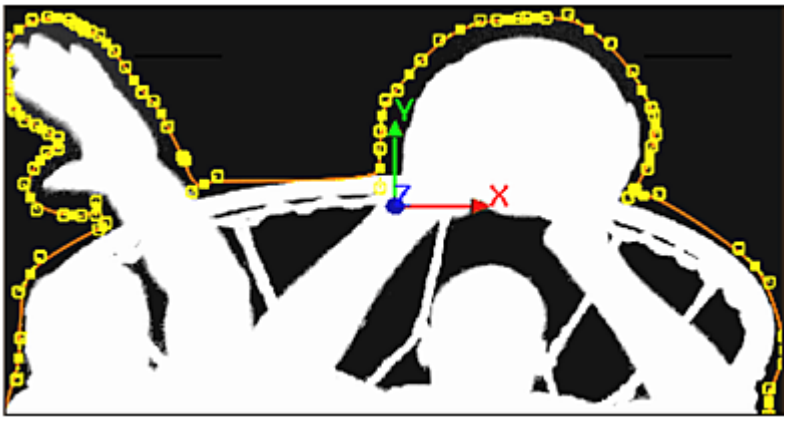
(a) Vertices are temporarily hidden while holding Shift

- Click-drag to control the tangency of any vertex as you are drawing the mask. When you release, you can continue to add vertices.



Tangency handles extend as you drag from a vertex

- `Shift`-drag to add freehand segments to the mask. The vertices and tangents of the mask you are drawing are temporarily hidden. Vertices are added where you drag, and appear when you release `Shift`. After closing the mask, you can use the Lasso Fit parameter to increase or decrease the number of vertices that define the freehand segments of the mask. See Adjusting the Number of Points in Freehand Segments (page 1109).



Segments of garbage mask drawn freehand

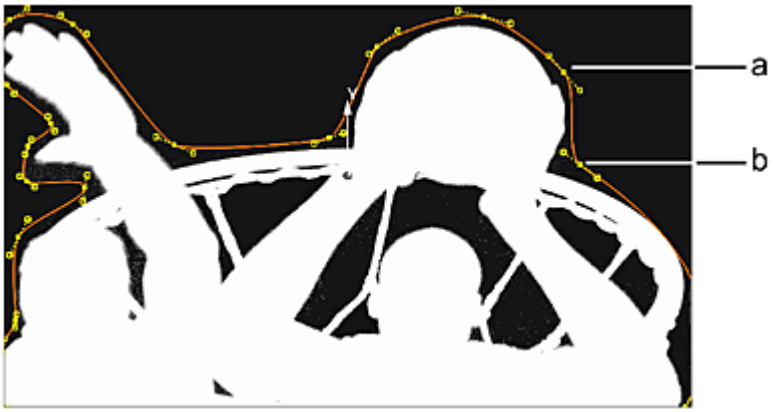
NOTE If you are drawing the mask with Auto Tangents turned off, freehand segments are drawn with broken tangents. See Breaking Tangents (page 1112).

5 To close the mask, do one of the following:

- Click Close.
- Click the first vertex.
- Drag to the first vertex.

TIP You can also close the mask by pressing the keyboard shortcut for the edit mode you want to use next: `M` for Move, `S` for Select, or `B` for Break. The mask closes and you are ready to work in that mode.

When the garbage mask is closed, its vertices and tangents can be edited and animated.



(a) Vertex (b) Tangent

If you want to view the nodes that are added with each new garbage mask, select Schematic from the World View box. You can use the Schematic view to access a menu, create parent-child relationships between masks and axes, delete masks, as well as perform other organizational tasks.

Adjusting the Number of Points in Freehand Segments

You can increase or decrease the number of points in freehand segments of a mask by adjusting the Lasso Fit value. An entire garbage mask can consist of freehand segments, in which case the Lasso Fit value affects the whole garbage mask. Otherwise, the Lasso Fit value only affects the part of the mask drawn freehand.

The Lasso Fit parameter loses its influence over freehand segments of a mask if you have edited points as follows:

- Added or deleted points using the Add or Delete Edit modes
- Saved and loaded a garbage mask
- Started a new session with the application

To adjust the number of vertices using Lasso Fit:

1. If necessary, select the garbage mask you want to affect.
2. Drag the Lasso Fit field. Increase the value to decrease the number of points in the freehand segments of the mask. Decrease the value to increase the number of points in the freehand segments.

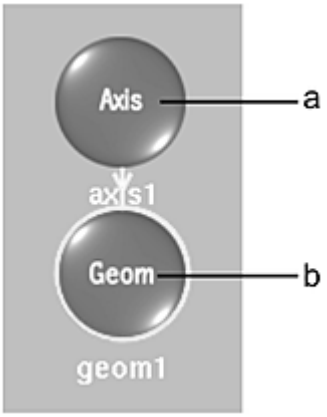
   Segments of the mask that have been drawn freehand are affected, while segments created by simple clicks remain unaffected.

   NOTE If you move vertices of a garbage mask and then change the Lasso Fit value, all the vertices snap back to the original position where they were first drawn, and all animation is lost. This occurs only if the Lasso Fit value retains its influence on the garbage mask.

Garbage Mask Nodes

When you create a garbage mask, an Axis node and a Geom node are added to the schematic. The Axis node contains rotation, position and scaling data. The Geom node contains information about how the mask will affect the image. There is also a Camera node, which is always part of the scene. The Camera node can parent the Axis node for greater precision over the garbage mask, particularly if you want to rotate or orbit the camera with respect to the garbage mask.

To view the schematic, select Schematic from the World View box to toggle between the Schematic view and the previous view.



(a) Axis node (b) Geom node

Changing the Priority Order of Masks

When you create several garbage masks on an image, you can change the order in which they are drawn, or layered in the scene. This affects the resulting image because a mask with a higher priority has precedence over one with a lower priority. You can use the priority order to create complex effects involving multiple garbage masks.

To set the priority of a mask:

1. Select the mask.
2. Click Push to move the mask down a layer; click Pop to move the mask up a layer.

NOTE You cannot animate the priority by setting Push and Pop at different frames. When you set a mask's priority with Push or Pop, it is set for the entire clip.

Transforming the Mask

You can apply transformations such as translation, rotation, scaling, shearing, and centre changes to a garbage mask. The transformation data for the mask is stored in its Axis node. You apply transformations to the mask using the Axis controls.

All transformation parameters in the Axis menu can be animated. You can use a motion path to animate the position of a mask.

You can also apply motion blur to a garbage mask. See Applying Motion Blur to Garbage Masks (page 1120).

To transform a mask:

1. Access the Garbage Mask menu.
   The Axis controls appear.

   NOTE The Axis controls are similar to the Axis menu in Action.

2. Use the Transformation fields to modify the position, shape, and size of the mask.

Manipulating Vertices and Tangents

Use the options in the Tools box to manipulate the vertices on the mask. You can edit the mask most easily in Front, Key-In, or Matte view.

Selecting Vertices and Tangents

You can select vertices and tangent handles when Select, Move, or Scale is selected in the Tools box:

- To select an individual vertex or tangent handle, click the vertex or tangent handle.

- To select multiple vertices, `Ctrl`-drag a selection box over a series of vertices.
- To select multiple vertices using the pen, press the pen button and drag a selection box over a series of vertices.
- To add another vertex to a multiple selection, `Shift`-click the vertex.
- To add several more vertices, press `Shift+Ctrl` and drag a selection box over the additional vertices.
- To deselect all vertices, click anywhere outside the mask.

Selecting Pickers and Softness Vertices

You can select pickers or softness vertices using the following techniques. Pickers are an element of the Tracer tool, which is only available in the GMask node in Batch or Batch FX and the Modular Keyer. See Applying Softness Using Pickers (page 1126).

To select a single picker or softness vertex:

1. Click the picker or vertex that you want to select.

To select multiple pickers or softness vertices:

1. Select the mask vertices that the pickers or softness vertices are associated with (see previous section).
2. Click a picker or softness vertex corresponding to one of the selected mask vertices.
   The pickers or softness vertices are selected for all the selected mask vertices.

Moving Vertices and Tangents

You can move vertices and tangents using the Move and Scale options of the Tools box:

- To move a vertex or tangent in any direction, select Move. Click the vertex or tangent and drag.

  TIP If you have trouble selecting a tangent handle that is very close to the vertex, hold `Q` then select it.

- To move selected vertices in any direction, select Move. Click one of the selected vertices and drag.
- To move one or more selected vertices in the direction perpendicular to their tangents, select Scale. Click one of the selected vertices and drag.
- To ensure better continuity on a garbage mask, automatically adjust tangents as you move vertices or scale the mask. Press and hold `G` and then drag a vertex. See Automatically Adjusting Tangent Handles of Adjacent Vertices (page 1113).

Breaking Tangents

You can separate two tangent handles (“break” the tangent) and move them separately using the Break option from the Tools box:

- To break and move a tangent handle, select Break and click the tangent handle. The tangent is displayed as a solid line, indicating it is “broken.”
- To reconnect two broken tangent handles, select Auto and click either of the two tangent handles. The tangent is displayed as a dotted line.
- To change the position of an individual tangent handle after releasing the cursor, select Move.

Removing and Adding Tangents

You can also use the Break option to remove tangents from vertices by clicking the vertex while in Break mode. When you remove a tangent, the curve defined by the tangent is removed.

The shape of the border line differs depending on whether adjacent vertices are broken:

- If adjacent vertices are unbroken, the border line curves as it approaches the vertices.
- If adjacent vertices are broken, the border line is straight as it approaches the vertices.

You can create a garbage mask composed entirely of straight edges by removing the tangents from all the vertices:

- To remove the tangent of a vertex, select Break and click the vertex.
- To add a tangent back to a vertex, select Auto and click the vertex. Alternatively, using the Break option, click the vertex and "drag" the tangent out again.

Automatically Adjusting Tangent Handles of Adjacent Vertices

When you move vertices, the tangent handles of the two adjacent vertices remain fixed in their current position. You can make them automatically adjust to create smooth curves between the vertices by pressing the `G` keyboard shortcut as you move vertices.

You can reverse the behaviour so that dragging vertices automatically adjusts tangents without using the keyboard shortcut. You reverse the behaviour by setting the following environment variable:

```
DL_GMASK_AUTO_MOVE_MODE
```

To reverse functionality of the `G` keyboard shortcut:

1. From the home directory of the application, open the *.cshrc* file in a text editor.
2. Add the following line to the end of the *.cshrc* file:

   ```
   setenv DL_GMASK_AUTO_MOVE_MODE
   ```
3. Save and close the *.cshrc* file.
4. In the home directory, type:

   source .cshrc
5. Launch or restart the application.

   When you click or move a vertex on a garbage mask, the tangents are automatically repositioned. When you press **G** and click or move a vertex, the tangents are not repositioned.

   TIP Alternatively, you can set the environment variable in the shell pointing to the home directory of the application. When you set an environment variable in the shell, the environment variable is enabled for the current session only.

Creating Right-Angled Vertices

You can create right-angled vertices using the following technique.

Press:	To:
`Ctrl+A`	Select all vertices.

Press:	To:
B	Break apart tangents.
G	Create right-angled vertices of selected vertices.

Animating Garbage Masks

You can animate a garbage mask using the Shape channel in the Channel Editor or using spline keyframing, which animates individual Vertex channels. Vertex channels correspond to the vertices—or handles—that appear along the perimeter of the garbage mask. The Vertex channel names are contained in the Spline folder in the Channel Editor.

You can apply tracking data to a mask's axis to animate a garbage mask. Existing keyframes are disregarded, after applying tracking data, you can animate individual vertices and make adjustments.

Animating Masks Using the Shape Channel

The Shape channel shows when the shape of the garbage mask changes during the animation.

To animate a garbage mask using the Shape channel:

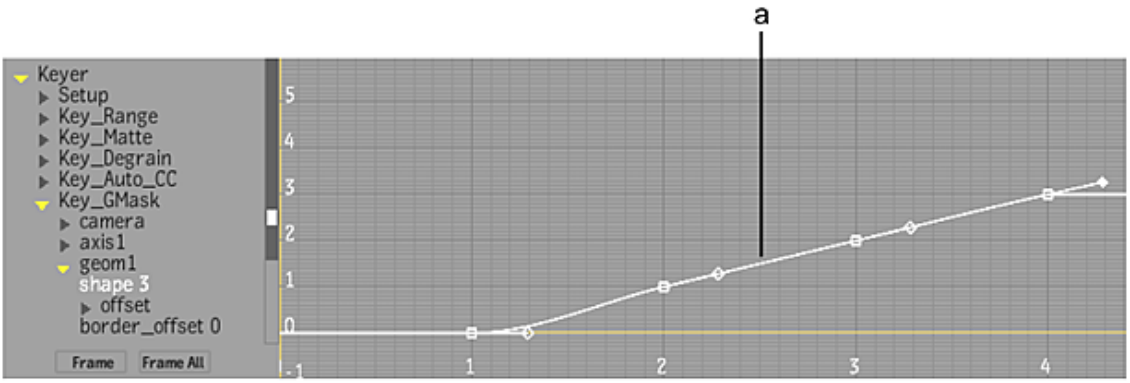
1. In the Garbage Mask menu, click Mask Setup.
2. In the Garbage Mask Setup menu, turn on Spline Keyframing.
   Each time you move a vertex, a shape key is added at the current frame.

   NOTE Disable Spline Keyframing if you do not want a keyframe added to the Shape channel.

3. Click Mask Setup again.
   The Garbage Mask menu appears.

(a) Shape Animation button (b) Linear Interpolation button

4. Enable Shape Animation.
   If keyframes are already set for specific vertex channels, the following message appears: “Convert explicit channels to a single shape channel?”
5. Click Confirm.
   The Shape channel appears in the Channel Editor and any Vertex channel keyframes are converted.



(a) Shape channel with 4 keyframes

NOTE Enabling Shape Animation disables the Vertex Keyframing and Channel Selecting boxes in the Garbage Mask Setup menu. These contain options for animating individual vertex channels. See Animating Masks Using Spline Keyframing (page 1115).

6 Depending on how you want to animate the garbage mask, use the Linear Interpolation and Constant Shape buttons as follows.

Enable:	To:
Linear Interpolation	Make the garbage mask trajectory linear. Enable this button when rotoscoping. For smooth garbage mask animation, disable this button.
Constant Shape	Keep the garbage mask shape the same for all frames. When you enable this button, the following message appears: "Remove all keyframes except current one?" Click Confirm.

Animating Masks Using Spline Keyframing

You can animate the shape of a mask by moving the vertices of a garbage mask spline at different frames throughout the clip. Each vertex has its own X, Y and Z position channels in the Channel Editor, as well as several other parameters, as shown in the following table.

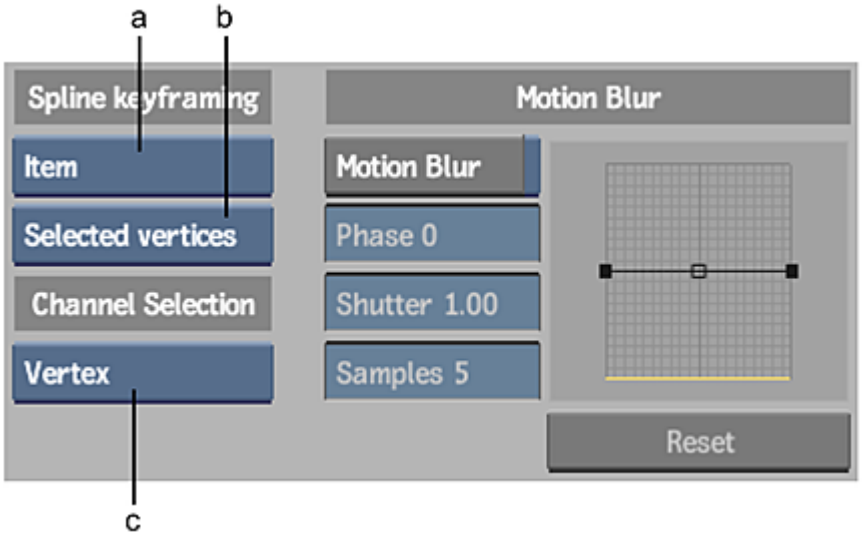
Parameter	Channel Editor Folder & Name(s)	Channel Editor Values
The position of the mask vertices.	*position: X, Y, Z*	
The position of the left and right tangent handles relative to the mask vertex.	*tangents: left, right*	
The tangent continuity (handles broken or unbroken).	*tangents: continuity*	0 = broken handles 1 = unbroken handles
The active/inactive status of mask vertices. See Adding and Removing Vertices (page 1117).	*Vertex_x: active*	0= inactive 1 = active

Before you start to animate the mask, set the Spline Keyframing options in the Garbage Mask Setup menu. In the GMask node in Batch or Batch FX, these options are located in the Node Setup menu. Use these options to specify which elements will get a keyframe as you animate the mask.

As you adjust the mask, you can opt to apply keyframes to selected vertices or to all vertices. To quickly animate a mask, keyframe all vertices every time you adjust a single vertex. This way, what you see at a given frame will be exactly what you get no matter how you animate the "surrounding" frames.

To have more control over the shape of the mask frame by frame, animate one vertex at a time. This method is useful for tracking garbage masks and keeping control over tangent animation. See Animating Masks Using Tracking (page 1119).

By combining the Spline and Vertex Keyframing options in different ways, you can set an animation mode to suit the task at hand. For example, by selecting Item and All Vertices, you could animate just the tangent handles of all vertices on the mask.



(a) Spline Keyframing box (b) Vertex Keyframing box (c) Channel Selecting box

Spline Keyframing box Sets the conditions under which keyframes are set, and for which parameters.

NOTE The Spline & Tracer, Spline only, and Item options are only available if you disable Shape Animation in the Garbage Mask menu.

Select:	To:
Spline & Tracer	Set a keyframe for all the parameters of the vertex or vertices and related elements, with the exception of the Sample On parameter. This option is only available when accessing the Garbage Mask Setup menu from the GMask node.
Spline only	Set a keyframe for one or more vertices, or related elements, for the specified parameter, as well as for the mask vertex position and the tangent position channels.
Item	Set a keyframe for a parameter of a vertex, or of a related element. Only the particular parameter gets a keyframe.
On	Set a keyframe for the shape channel when a change to a vertex is made (such as moving a vertex or tangent handle, breaking tangent handles, or changing the Active/Inactive status of a vertex). This option is only available when Shape Animation is enabled in the Garbage Mask menu.

Select:	To:
Off	No keyframes are set.

Vertex Keyframing box Sets the conditions under which parameters are applied to specified vertices in the mask.

NOTE The Vertex Keyframing box is only available if you disable Shape Animation from the Garbage Mask menu.

Select:	To:
All Vertices	Set a keyframe for the parameter you are animating for all the vertices of the mask. Related elements, such as tangents, also all get a keyframe when you animate a parameter on the mask.
Selected Vertices	Set a keyframe for the parameter you are animating for all selected vertices on the mask. Related elements, such as tangents, that are selected, also all get a keyframe when you animate one on the mask.

Channel Selecting box Determines which channels are selected in the Channel Editor when you select one or more vertices.

NOTE The Channel Selecting box is only available if you disable Shape Animation from the Garbage Mask menu.

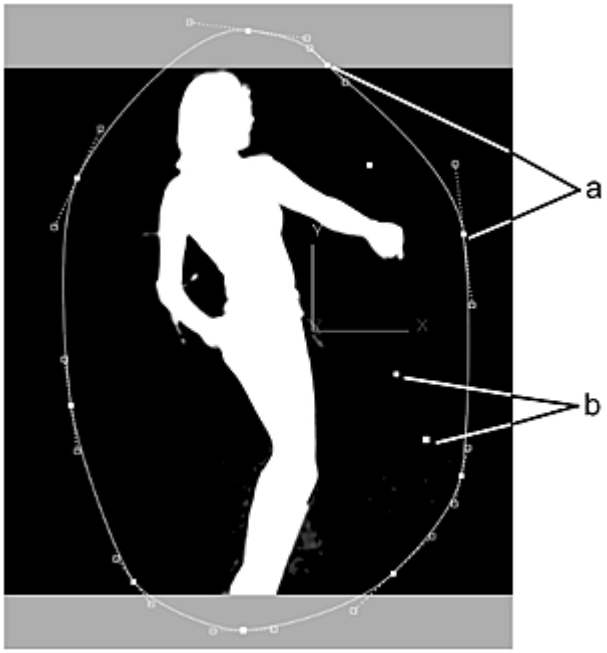
Select:	To:
Vertex	Select all the channels related to the vertices you select on the garbage mask. For example, whether you select a vertex or one of its tangents, its active, position, and tangent channels are also selected in the Channel Editor.
Item	Select only the particular channel corresponding to the selected vertices. For example, if you select the right tangent handle of a vertex, only the corresponding right channel is selected in the Channel Editor.

Adding and Removing Vertices

To further control the shape of a mask throughout the clip, you can add and remove vertices from your mask at any frame of the clip. For example, if the shape changes radically or gets bigger part way through the clip, you can add vertices to reshape the mask at that frame. When you add a vertex part way through the clip, it is added to the whole clip but respects the existing shape of the mask in all previous and subsequent frames. Therefore, any reshaping that you have done to the mask is not affected by the new vertices.

NOTE Adding or deleting vertices on masks with freehand segments causes the Lasso Fit value to lose influence. See Adjusting the Number of Points in Freehand Segments (page 1109).

If you delete a vertex, it becomes “inactive” from that frame to the next active/inactive keyframe set for that vertex, or, if there are no keyframes, to the end of the clip. An inactive vertex has no effect on the shape of the mask. On previous frames, the vertex is still active, so the mask shape is unchanged by the deletion. To delete the vertex completely (that is, throughout the whole clip), click it a second time.



(a) Active vertices (b) Inactive vertices

Use the options in the Tools box to add and delete vertices from the mask, and to control which vertices are active and inactive.

To add a vertex to the entire mask:

1. Select Add from the Tools box.
2. On any frame in the clip, click the mask where you want to add the vertex.

   The new vertex is added and is active throughout the clip. It does not affect the existing mask animation, regardless of where in the clip you add the vertex.

To make an active vertex inactive:

1. Switch to Delete mode.
2. Click the vertex.

   The vertex is inactive from the current frame to the next frame where an Active/Inactive keyframe has been set, or, if there are no keyframes, to the end of the clip. The status of the vertex on previous frames is unchanged.

   NOTE Alternatively, use the Active button in the Shape menu to make vertices inactive. Select the vertex you want to make inactive (you must be in Select, Move, or Scale mode), and then click Active.

To make an inactive vertex active:

1. Switch to Add mode.
2. Click the inactive vertex.

   The vertex is active from the current frame to the next frame where an Active/Inactive keyframe has been set, or, if there are no keyframes, to the end of the clip. The status of the vertex on previous frames is unchanged.

   NOTE Alternatively, use the Active button in the Shape menu to make vertices active. Select the vertex you want to make active (you must be in Select, Move, or Scale mode), and then click Active.

To delete a vertex from the mask:

1. At any frame, switch to Delete mode.
2. Click the vertex.

   It becomes inactive.
3. Click the vertex a second time.

   It is removed from the mask for the duration of the clip.

   NOTE You cannot delete the last three vertices of a mask, as they are the minimum necessary to define it. If you want to delete the mask, you delete its Geom node in the Garbage Mask Schematic.

Animating Masks Using Tracking

You can animate a mask by applying tracking data to it. This is useful when you want a garbage mask to follow a moving element in a clip. To do this, you can either:

- Animate the entire mask by applying the data to the mask's axis.

  You can track an object on the front or back clip and apply the tracking data to the axis of a mask, or to a hierarchy of masks.
- Animate selected vertices according to reference points that you set in the Stabilizer.

  Each vertex you select is assigned a tracker in the Stabilizer. The vertices are repositioned according to the reference points that you set in the Stabilizer.

When you apply tracking data to a mask, only the mask axis, or position of selected vertices, is animated. Any other keyframes set for the mask are disregarded. You can, however, animate individual vertices after applying tracking data to your mask's axis or to individual vertices. It is especially useful to manually adjust the tangent handles of the vertices at different frames where needed.

To animate an entire mask with tracking:

1. Access the Garbage Mask menu.

   The Garbage Mask menu appears.
2. Select the mask.
3. From the Clip box, select the clip you want to track.
4. If necessary, adjust the mask using the Position, Rotation, Scale, Shear, and Centre fields.
5. Set the Rotation and Scaling options. To track rotation, select Rotation On; to track scaling, select Scaling On.
6. Select an option from the Adjust box.

Select:	If the selected mask is:
Adj Offset	Parented to one axis.
Adj Axis	Parented to a hierarchy of objects.

7. Go to the frame where you want to start tracking.
8. Click S.

   The Stabilizer appears.
9. Position of the trackers.

NOTE The first tracker is for position data and the second tracker is for rotation and scaling data. If rotation and scaling were not selected, the second tracker has no effect.

10 Click Analyse.

Fine-tune your analysis if necessary. For more information, see Stabilizing and Tracking (page 1063).

11 Click Return.

The Garbage Mask menu reappears. The tracking data is applied to the mask.

12 Fine-tune your mask if necessary.

To animate individual vertices on a mask with tracking:

1 Select a vertex or a group of vertices on a mask.

2 Access the Garbage Mask menu.

The Garbage Mask menu appears.

(a) Clip box (b) Adjust box

3 From the Clip box, select the clip you want to track.

NOTE When tracking vertices on a mask using the Garbage Mask menu in the GMask node in Batch or Batch FX or the Modular Keyer, you can only track the front clip.

4 From the Adjust box, select Adj Tangents.

The tangents for the selected vertices are adjusted while the points are being tracked.

5 Go to the frame where you want to start tracking.

6 Click S.

The Stabilizer appears. The trackers are automatically placed in the position of the selected vertices. Reposition if needed.

7 Click Analyse.

Fine-tune your analysis if necessary.

8 Click Return.

The Garbage Mask menu reappears. The tracking data is applied to the selected vertices on the mask.

9 Fine-tune your mask if necessary.

Applying Motion Blur to Garbage Masks

Use motion blur on a garbage mask to compensate for the movement of an object in a clip. Motion blur is created according to the animated movement of a mask's vertices. The amount of blurring is affected by the speed of the mask's movement.

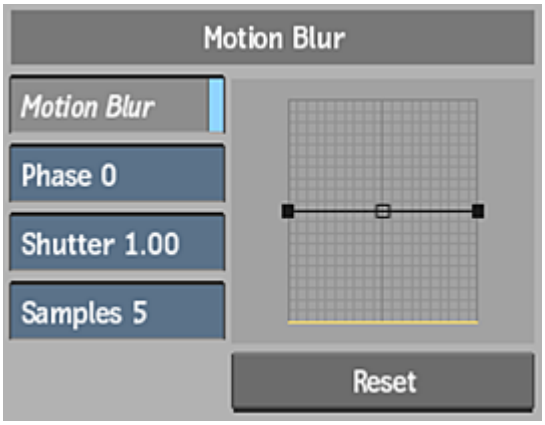
Motion blur can account for the natural blurring of an object as it moves or rotates in space. For example, assume you are drawing a garbage mask around a car as it turns a corner. Because you are using an overhead shot, the edges of the car where motion blur occurs change as the car accelerates and rounds the corner. By

applying motion blur to the mask, which has animated position and rotation values to match the car's movement, appropriate blurring of the car's edges into the background clip occurs.

TIP Besides the practical use of blurring a garbage mask, that is, to match the movement of an object you are keying, you can try blurring masks for artistic effects in multi-layered composites.

To apply motion blur to a garbage mask:

1 Access the Garbage Mask Setup menu.



2 Set the motion blur as follows.

Motion Blur button Enable to apply motion blur to garbage masks.

Phase field Specifies whether the motion blur is based on the movement before or after the current frame. A value of -100 places the motion blur before the frame, while a value of 100 places the motion blur after the frame. A value of 0 is centred, which evenly distributes the motion blur. The default value is 0.

Shutter field Controls the duration of the motion blur at each frame, which affects the size of the motion blur. Increasing Shutter value does not increase the processing time.

Samples field Determines the quality of the motion blur produced by the number of samples taken at each frame. Increasing the Samples value causes the processing time to increase linearly.

NOTE You can animate the Motion blur button, as well as the Phase, Shutter, and Samples fields. They can be found in the Channel Editor under the *motion_blur* folder.

Motion Blur curve Determines the transparency of the samples that create the blurring effect. The left side of the curve controls the samples of the incoming movement of the subject in the frame. The right side of the curve controls the samples of the outgoing movement of the subject in the frame.

For example, to add blur to the left edge of the mask as it travels from left to right in the clip, adjust the motion blur curve so it slopes down. If you want to add a bit of blur on each side of the mask, adjust the motion blur curve so it peaks in the middle.

Select one of the three vertices on the curve to move it. When you select a vertex, its tangent handle is displayed. You use the tangent handle to adjust the slope of the curve.

NOTE The garbage mask motion blur settings are similar to those in the Axis menu in Action.

Motion blur occurs on the mask according to the settings you select and the speed its vertices traverse the clip.

3 Enable Motion Blur. This makes the motion blur available for application on any garbage mask or axis in the setup.

NOTE Motion blur settings are applied uniformly to all garbage masks in the setup.

4 To reset all motion blur settings back to their default settings, click Reset and then click Confirm.

5 To apply motion blur to a garbage mask, access the Garbage Mask Shape controls, select the garbage mask, and then enable MBlur.

The settings specified in the Garbage Mask Setup menu will be used to create the motion blur effect on the garbage mask. You can select one garbage mask at a time.

6 To apply motion blur to an axis, select the axis, and then enable MBlur.

The settings specified in the Garbage Mask Setup menu will be used to create the motion blur effect on the axis. You can select one axis at a time.

Blurring a Stationary Garbage Mask

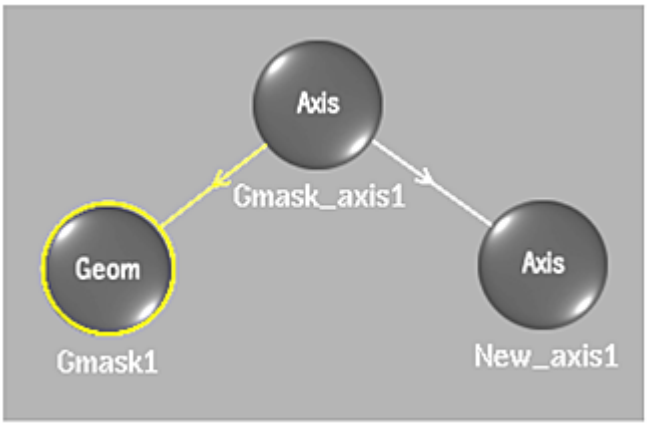
You can create a motion blur effect for a garbage mask without having the mask actually change position. This could be useful, for example, if you would like to simulate movement in a stationary image.

To apply axis motion blur:

1 Toggle to schematic view and add a new axis.

2 Parent the new axis to the garbage mask axis.

The garbage mask and the new axis should now both be parented to the garbage mask axis.



3 Apply motion blur to the new axis.

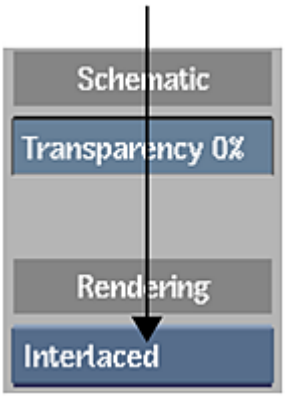
Motion blur is applied to the stationary garbage mask.

Rendering Garbage Masks in Interlaced Mode

Using the GMask node, you can render garbage masks in Interlaced mode to accommodate video material.

To render garbage masks in Interlaced mode:

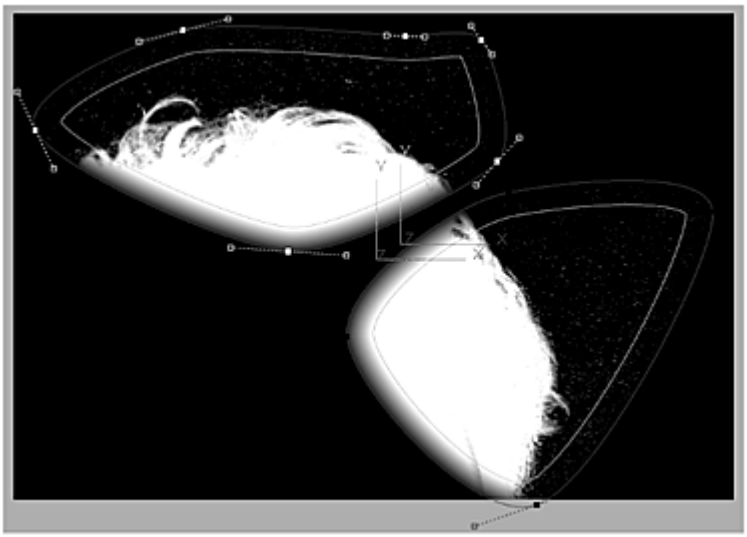
1 From the GMask node's menu, select Interlaced from the Rendering box.



Using Multiple Garbage Masks

Use the Region of Interest feature on clips containing multiple garbage masks that each hide a portion of the matte. Region of Interest ensures that multiple overlapping or non-overlapping garbage masks all respect the areas defined by one another. Furthermore, you can individually modify the opacity of each mask.

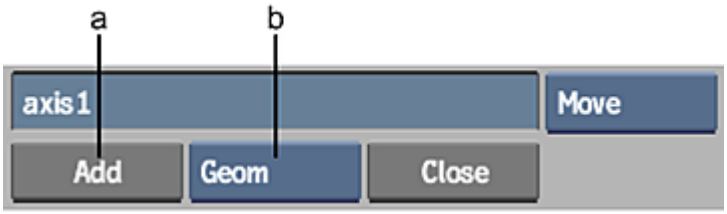
Region of Interest is only available when accessing the Garbage Mask menu from the GMask node in Batch or Batch FX.



Two garbage masks with Region of Interest enabled

To draw more than one garbage mask revealing a matte:

1 In the Garbage Mask menu, select Geom from the Object box and click Add.



(a) Add button (b) Object box

2 Draw the first garbage mask.

3 Switch to Result view.

4 Enable Region of Interest.

Outside is enabled and the Colour is set to 0.0 for the mask.

NOTE Disabling Region of Interest toggles the Colour setting back to what it was before.

5 Adjust offsets and softness as required.

6 To adjust the individual opacity for each mask, select the mask and then adjust the value in the Opacity field.

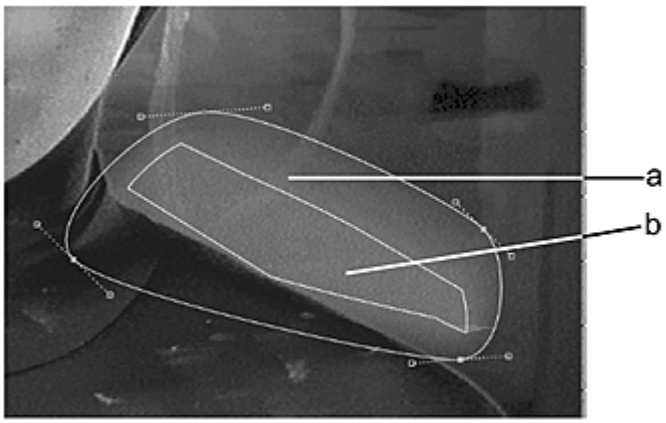
7 Draw any other garbage masks as needed.

NOTE Each time you add a garbage mask with ROI enabled, Result view is temporarily cleared of all masks so that you see the entire matte. This way you have a better view of what you want to reveal and mask.

Refining the Mask

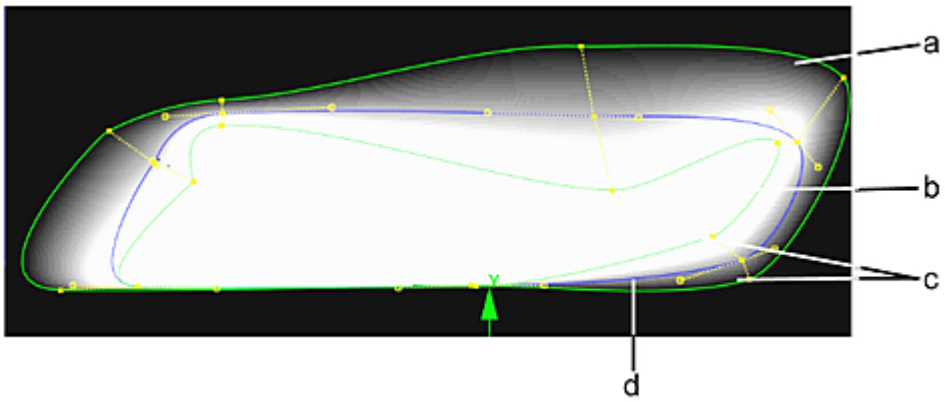
You can adjust the softness gradient of a mask to smoothen its edges. You can smoothen the gradient towards the inside edge, the outside edge, as well as the area where the inside and outside adjustments have an effect. You can create a uniform gradient around the edge of the mask, or use an advanced gradient to control the shape of the gradient at different parts of the mask.

To create a uniform gradient, you define how far you want the gradient to be offset from the edge of the mask and then set its transparency. To vary the shape of the gradient, you move vertices on inner and outer softness splines. The gradient will be based on how far each vertex point is offset from the mask.



(a) Alpha region (b) Inner offset

Advanced gradient provides a versatile method for setting the softness of the mask edge. It applies a gradient according to the distance of the softness borders from the garbage mask spline. It allows you to customize the softness gradient at different parts of the mask. The advanced gradient has two softness borders, one inside and one outside the mask border. It also includes inner and outer softness vertices for each regular mask vertex.



(a) Outer Softness border (b) Inner Softness border (c) Softness vertices (d) Mask border

Some potential uses for the advanced gradient are:

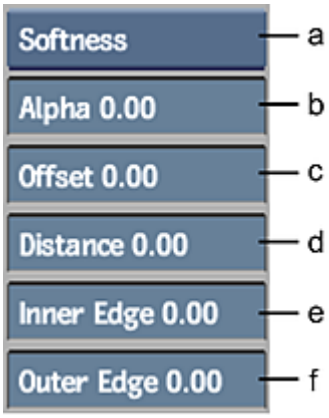
- Creating compositing effects when using Action
- Removing wires
- Pulling a key on areas of the mask where pickers are not needed

When using the Tracer in the GMask node in Batch or Batch FX or the Modular Keyer, you can work in two modes: advanced gradient and pickers. Individual vertices can be set to either of these modes.

You can change the default mode of the Tracer from advanced gradient to pickers, a system of localized keys. Tracer analyses the colour information both inside and outside the mask, then compares the values and uses a keying algorithm (the same one used in the 3D Keyer) to derive a key for the mask edge. This system allows you to key images with a lot of detail at the edges.

NOTE For good results with the Tracer, the object must have sufficient (and relatively consistent) chroma and luma differences between its foreground and background.

To control softness and advanced gradients, use the Garbage Mask Shape controls.



(a) Edge Softness box (b) Alpha field (c) Offset field (d) Distance field (e) Inner Edge field (f) Outer Edge field

Applying Softness Using a Uniform Gradient

Use the Softness fields to apply a constant range of softness to the edge of the mask. The Alpha and Offset fields define the fade-out of the softness gradient from the edge of the mask.

Once gradient transparency and offset have been determined, the degree of smoothness can be set using the Inner Edge and Outer Edge fields. You can then use the Distance field to determine the area where Inner Edge and Outer Edge settings have an effect.

To apply uniform softness:

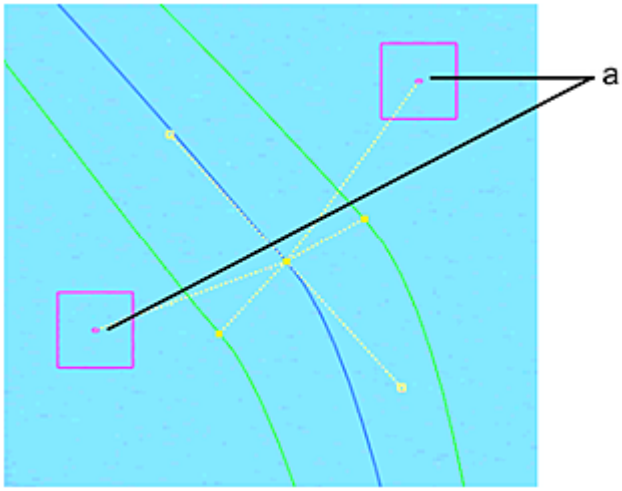
1. Access the Garbage Mask menu.
2. Select a mask in the image window.
3. If necessary, toggle the Edge Softness box to Softness.

4. Set the border of the softness gradient using the Offset field. The gradient will be affected by how far the softness border is offset from the edge of the mask.
5. Set the transparency of the gradient using the Alpha field.
6. Use the Inner Edge field to smoothen the softness gradient towards the inside.
7. Use the Outer Edge field to smoothen the softness gradient towards the outside. Adjusting this value is especially noticeable if you are smoothening a transition from a black inside edge to a white outside edge.
8. Use the Distance field to modify the area of which the Inner and Outer Edge adjustments have an effect.

Applying Softness Using Pickers

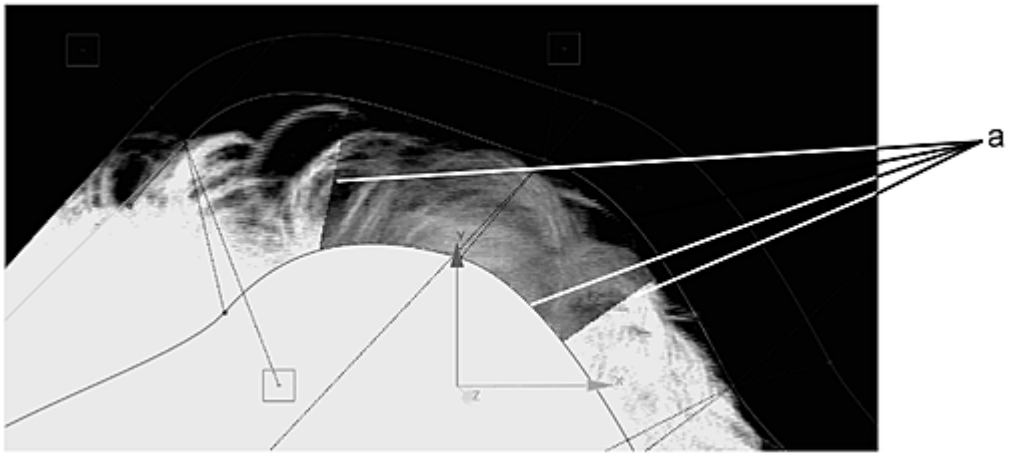
You can change the default mode of the Tracer from advanced gradient to pickers, allowing you to sample colours inside and outside the garbage mask spline, to apply softness according to the colour transition in the clip. Individual vertices can be set to either of these modes.

When great control over the mask edge is needed, for example, for fine edge detail, use pickers to effectively key out the background. The Tracer uses pairs of pickers to do luma and chroma analysis of the area inside and outside of the mask and derives localized edge keys from this information.



(a) Pickers—Colour analysis is done in the area inside the pickers.

The area that each picker affects extends halfway towards the two adjacent mask vertices, and up to the two softness borders, as shown in the following example.



(a) Borders of area of pickers' influence.

Using pickers, you can key objects that otherwise would be extremely difficult to key. Imagine a golden horse with a very fine mane on an unfocused background of various colours. As long as you have some chroma/luma differences in the background (green vegetation, blue sky, black earth or rocks), you can 'force' the outside pickers to sample these colour values. The inside pickers can sample the golden average values, and the Tracer can generate a soft-edged matte based on the difference between the two sets of values.

Pickers can be selected and manipulated independently of the vertices they are associated with. See Selecting Pickers and Softness Vertices (page 1112).

Mixing Advanced Gradient and Pickers

When the subject has a mixture of fine, wispy edges and hard, clean edges, use a mix of localized keys and advanced gradients. Pickers are better for the fine edge areas and advanced gradient is sometimes better for the hard edge areas. You can set the state of each vertex to advanced gradient (pickers off) or localized key (pickers on) mode.

- To convert a section of a mask from advanced gradient to pickers, select one or more vertices with no pickers and enable the Picker button. Adjust the pickers as needed.

- To convert a section of a mask from pickers to advanced gradient, select one or more vertices with pickers and disable the Picker button.
- To toggle selected vertices between the two modes, use the 0 (zero) keyboard shortcut.

To completely remove gradient (for hard edges):

1. Select the mask vertices in the area of the mask border where you want to remove the gradient.
2. Click any one softness vertex corresponding to one of the selected mask vertices (either an inner or outer vertex).

   All the corresponding inner or outer softness vertices are selected.
3. From the Tools box, select Scale.
4. Click any one of the selected softness vertices and drag until the softness border is lined up with the mask border.
5. Repeat for the second softness border (inner or outer).

To separate the softness border from the mask border:

1. Click a mask vertex, and then drag to the right to move the softness vertex out.

Applying Softness Using the Tracer

Drawing a Tracer-aided matte entails four basic steps.

To draw a Tracer-aided matte:

1. Analyse the image to determine which areas are appropriate for pickers and which areas are better served by advanced gradients.
2. Draw the garbage mask.
3. Assign pickers to vertices where they are needed.
4. Adjust the softness borders, offsets, tangents, vertices, and pickers (where used).

To analyse the image:

1. Examine the edges of your talent to decide which portions would benefit from advanced gradient treatment and which would respond better to localized key treatment. If your clip has colours that change greatly throughout the clip, using pickers may be difficult.

   For more information, see Applying Softness Using Pickers (page 1126).

To draw the garbage mask:

1. In the Modular Keyer or Batch or Batch FX, add a context point further down the processing pipeline or processing tree. You can then use the Context view to see the effect of the matte on the result image.

   NOTE When using the Tracer to pull a key from the Modular Keyer, remove the nodes before the GMask node in the default pipeline.
2. Display the Garbage Mask menu.
3. From the View box, select Reference.

   This view is a reference image of the clip.

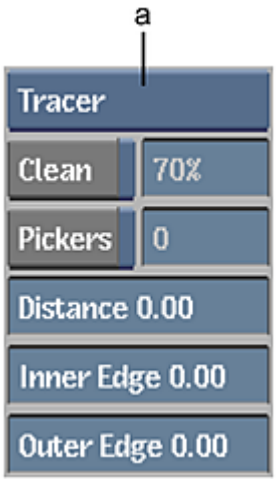
4 Draw a garbage mask around the subject and close it by clicking the first vertex drawn. Use as many vertices as required to adequately enclose the subject, but try to avoid excessive vertices. This will minimize unnecessary tweaking and manipulation later.

TIP Place more vertices in areas with variation in the background.

To assign pickers to selected vertices:

1 From the Edge Softness box, select Tracer.

The Tracer menu appears, and two softness borders (the green lines) are added to the mask.



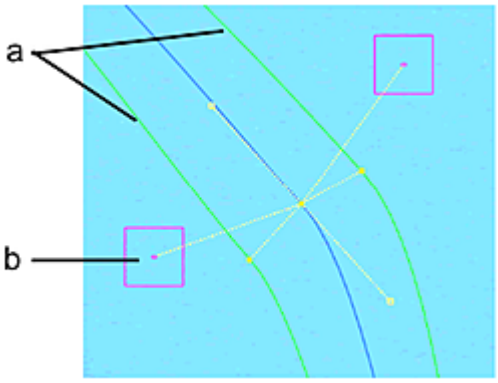
(a) Edge Softness box

2 Select the vertices to which you want to apply pickers. See Selecting Vertices and Tangents (page 1111).

3 Enable Pickers.

NOTE When one or more pickers are enabled, mask characteristics are automatically set to Outside disabled and a Colour value of 0 so that the mask will be white inside and black outside. Do not set the Outside button unless you want to reverse the effect.

A pair of pickers is provided for each selected vertex.



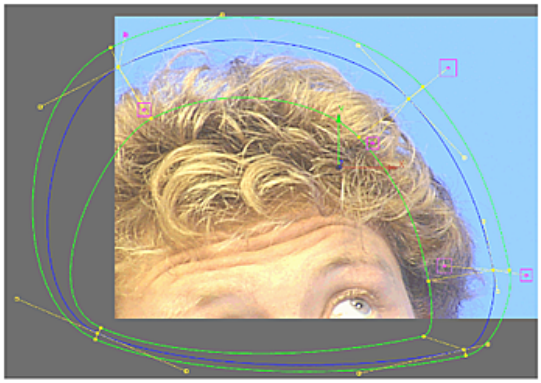
(a) Softness borders (b) Picker

- On vertices without pickers, the softness borders delineate the area where a softness gradient is applied.
- On vertices with pickers, the softness borders delineate the area to which softness is applied according to the picker values.

To fine-tune the matte:

1 Adjust the softness borders and mask border by moving the vertices and tangents:

- The mask border should follow the general contour of the subject as closely as possible.
- The outer softness border should completely surround all details that you want to include in the matte—all wispy details and edges must be within this line.
- The inner softness border should be well within the area where softness control is needed.



When moving tangents, you can opt to have the pickers follow the tangent movement (the default behaviour), or be independent of tangent movement. To make pickers independent of tangents, switch to Break mode and click either picker. You can animate this behaviour. See Animating a Tracer Mask (page 1131).

NOTE If the tangents are broken, the pickers are automatically unlinked from the tangents.

2 For each pair of pickers, one requires a sample of the area outside the mask, while the other requires a sample of the mask interior.

Place one picker outside the subject to sample values you do not want to include in the matte (for example, a blue screen). Place the other picker within the subject for a colour value sample of an area you do want included in the matte. Try to select areas where the colour values do not change too much throughout the clip.

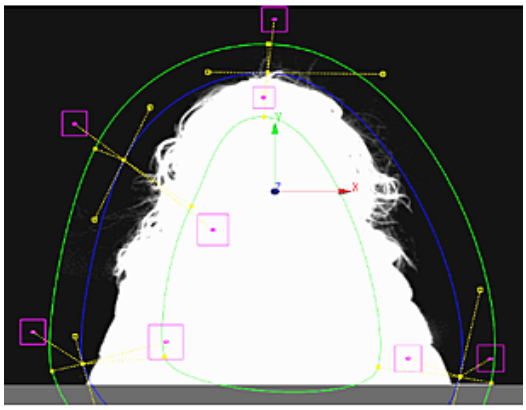
3 Click Matte or Result (Matte will provide a clearer view) to see your progress.

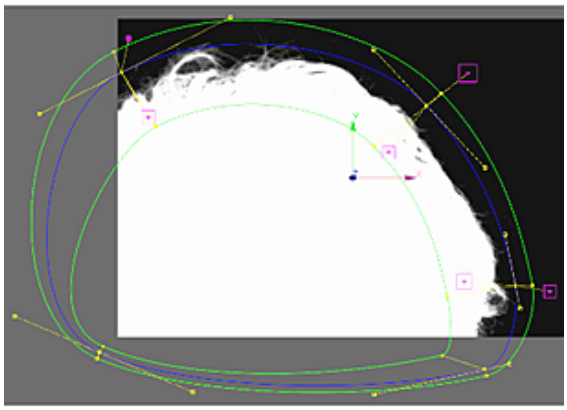
4 Scale the pickers by doing one of the following:

- Select Scale edit mode, click within the picker, and drag to the left to reduce the size or to the right to increase the size.
- Select the picker and then drag the cursor over the Sample Size field.

5 Move the pickers around and enlarge or reduce them to interactively see how you can obtain the best results. This will require toggling between the Front and Matte views.

The following figure illustrates the Matte view result of the pickers placed in the previous figures.





Animating a Tracer Mask

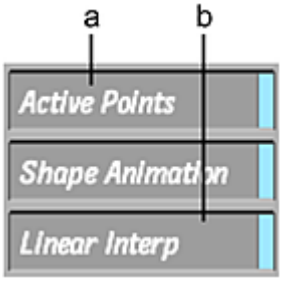
When using the Tracer, you can animate the basic vertex parameters, as well as the following additional parameters.

Parameter	Channel Folder and Name(s)	Channel Values
The position of the two softness vertices relative to the mask vertex.	border: upper = outer vertex* bottom = inner vertex*	
Whether pickers are linked to tangents or not. Linked—Pickers move with tangents. Unlinked—Pickers remain in their current position when tangents are moved.	pickers: fixed	0 = unlinked 1 = linked
The on/off status of pickers.	pickers: active	0 = off, 1 = on
The position of the pickers relative to the mask vertex.	pickers: upper = outer picker* bottom = inner picker*	
The picker size.	pickers: u-size = outer picker size* b-size = inner picker size*	
The Sample on/off status of pickers. See Animating Picker Values (page 1132).	sample: active	0 = Sample off 1 = Sample on
The interpolation mode for picker values when Sample is off. See Animating Picker Values (page 1132).	sample: interp.	0 = Constant 1 = Linear

*Channel Editor designations for “upper” and “bottom” assume the mask was drawn clockwise.

Animating Picker Values

Two options are available to control how picker values (that is, the sampled colour values) are animated: Sample On and Interpolation mode.

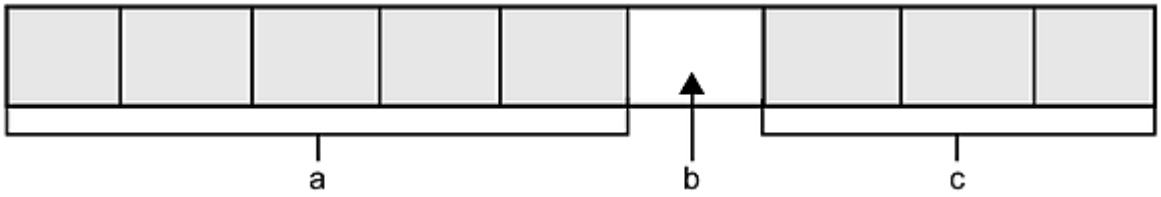


(a) Sample box (b) Interpolation button

Sample box Active resamples picker values at every frame (the default). Passive disables resampling for one or more frames. When Passive at a particular frame, the picker values previously sampled at another frame are used. It is useful to adjust the size and position of pickers at one particular frame until the result is optimal, and then apply these values to other frames. In this way, if movement in the clip in other frames causes the area being sampled to move away from the pickers, you do not need to readjust the pickers to get a good reading.

You can select Active or Passive for one or more selected pickers.

Selecting Passive works best when colour values remain fairly consistent throughout the clip. In this case, one picker sampling will often be enough for the entire clip.



Typical technique for using Sample On: one sample is used for the entire clip

(a) Sample Off (b) Sample On (c) Sample Off

When there is more colour variation in the clip, you may need to take samples at several frames.

To use one picker sampling throughout a clip:

1. Choose a frame in which the colour values are representative of the average colour values found in the clip.
2. Select the pickers you want to sample. See Selecting Pickers and Softness Vertices (page 1112).
3. At that frame, make sure Active is selected from the Sample box (the default).
4. Set the picker values by setting their position and size until you get the optimal result.
5. Go to the next frame and select Passive from the Sample box.

   The sample values read at the frame you chose will be used to compute the softness for all other frames in the clip.

To take samples at several frames:

1. Follow the steps in the previous procedure to set a picker sampling for the clip.

2 Starting at the first frame sampled, examine your results frame by frame (either forward or backward through the clip).

3 When you come to an unsatisfactory result, select the pickers you want to re-sample and select Active.

4 Adjust the picker position and size until you get a good result.

5 Go to the next frame and select Passive from the Sample box.

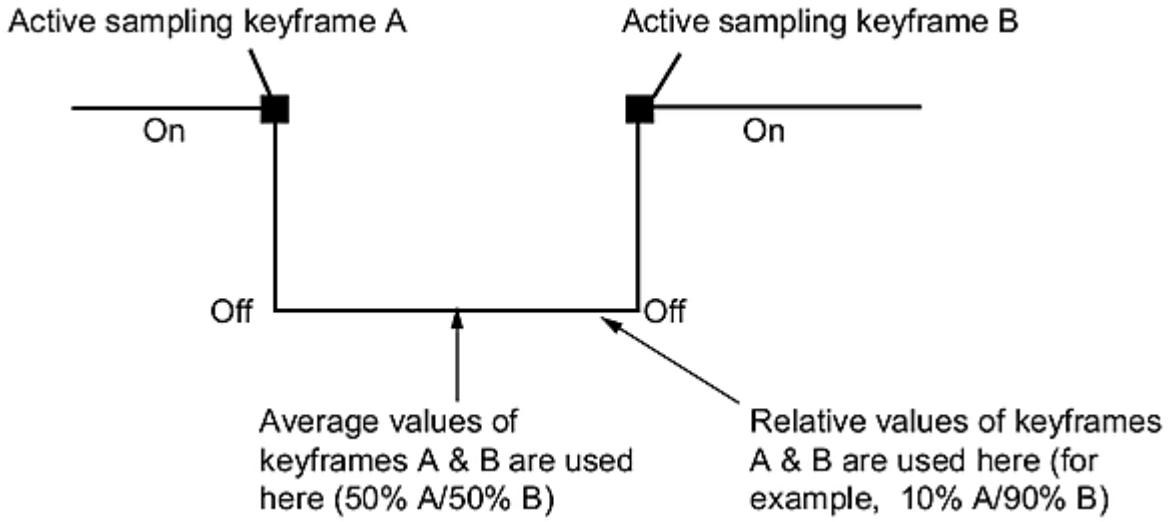
6 Continue verifying frames until you come to one that needs to be adjusted.

7 Repeat steps 3-5.

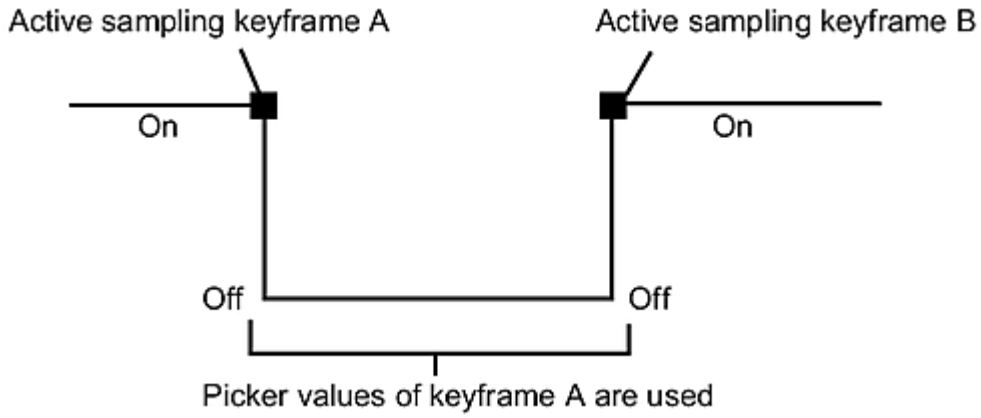
8 Repeat for the remainder of the clip.

Interpolation button Displays the interpolation between keyframes that have Active selected as the Sampling option.

- Linear (enabled): Picker values are interpolated between keyframes with active sampling. The picker values for each frame are computed based on the values of the previous and next active sampling keyframes and the proximity of the frame to those keyframes.



- Constant (disabled): Picker values are fixed between keyframes with active sampling. This means that the picker values at the first active sampling keyframe will be used for all subsequent frames up until the sampling is active again.



Tracking with the Tracer

Tracking with the Tracer is done using the same method as with regular garbage masks. See Animating Masks Using Tracking (page 1119).

NOTE When animating selected vertices with the Stabilizer, the vertices do the tracking, not the pickers. Once you obtain the tracking data, you will need to go back through your clip from frame 1 onwards, making adjustments to the pickers where necessary.

Troubleshooting the Tracer

Every clip has its own challenges and requires a certain amount of tweaking and fine-tuning to achieve the best results.

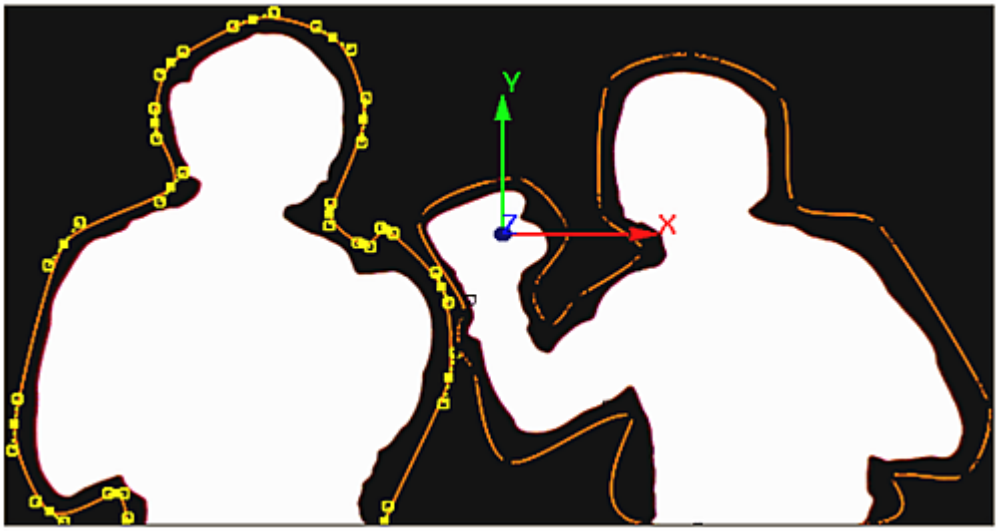
Problem	One of the Tracer segments (localized keys) on my matte shows noisy black and white artefacts and nothing else. What's wrong?
Possible Cause	The two pickers are sampling luma and/or chroma values that are too similar.
Solution	Try moving one of the pickers to a differently coloured area, or enlarge or reduce the size of one or both of the pickers to include more varied colour information. If there is simply no area in the clip that differs enough, you will need to use an advanced gradient for that segment.
Problem	One of the Tracer segments (localized keys) on my matte appears to be showing the black/white matte information reversed, or as a negative. What's wrong?
Possible Cause	You may have reversed the position of the two pickers.
Solution	Try exchanging the inside picker for the outside one and vice-versa.
Problem	I can't get a good colour sample from the area outside the mask edge in a certain portion of the image. What can I do?
Possible Cause	There is simply none (or not enough) of the needed colour in the immediate area of that particular picker.
Solution	You can drag the picker somewhere further away from the vertex to get a better colour sample. The picker can be located anywhere on the image in order to facilitate the best possible colour sample. Do not, however, place a picker outside the image.
Problem	I'm getting really nice edge detail on my matte, but I'm also getting black/white holes in my matte. Is there anything I can do?
Possible Cause	This may be caused by the generation of random noise and/or artefacts.

Solution 1	Try the Clean algorithm by clicking the box labelled Clean. You may want to try several different percentages to find the optimal level of cleaning that doesn't effect your edges (the default percentage is 70%). The Clean algorithm works only with the Tracer and pickers on. If you have advanced gradient segments, they will be unaffected. The Clean command works on all vertices with pickers, whether or not they are selected. The area affected by the Clean command is the same area that the pickers affect (see Applying Softness Using Pickers (page 1126)).
Solution 2	Use the inner border to isolate the problem areas.
Solution 3	Add a second garbage mask.
Problem	I'm repositioning and rescaling the pickers and the matte goes from having too much softness (overly transparent) to having too little (overly opaque). What can I do?
Possible Cause	The sampled colour range is either too broad or too narrow.
Solution	Locate the areas that have a tendency to become too softened and the picker that is 're-sponsible' for this by moving the pickers until the matte results change for the worse. Include a larger portion of these softer areas in a picker box. By sampling a fuller range of chroma/luma in this area, a more uniform, averaged effect is created.

Using Region of Interest and the Tracer

Certain garbage mask features are supported only in GMask Timeline FX, and the GMask node in Batch or Batch FX and the Modular Keyer.

In GMask Timeline FX, and the GMask node in Batch or Batch FX and the Modular Keyer, you can use a parameter called Region of Interest (ROI) when drawing multiple garbage masks on a clip to remove the portion of the matte that falls outside of each mask while respecting the area defined by all masks.



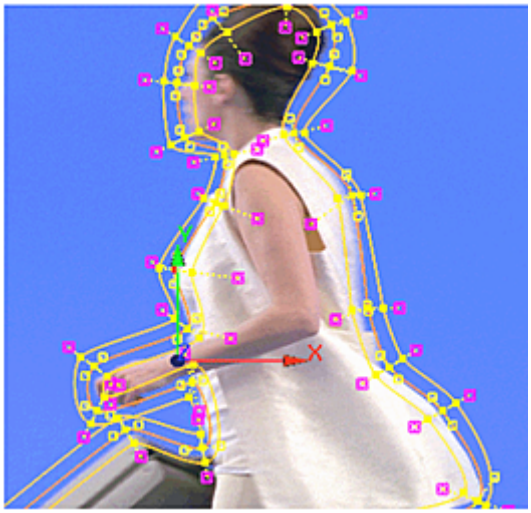
ROI enables you to reveal multiple parts of the matte.

Image courtesy of The Post Group

In the GMask node in Batch or Batch FX and the Modular Keyer,you also have the option of using the Tracer with an advanced gradient. The Tracer is an additional softness tool you can use to pull a key from scratch. This allows you to key images with a lot of detail at the edges. It uses a system of localized keys, called *pickers*, which analyse the colour information both inside and outside the mask to derive a key for the mask edge. You can use both the Tracer and the advanced gradient within the same garbage mask.

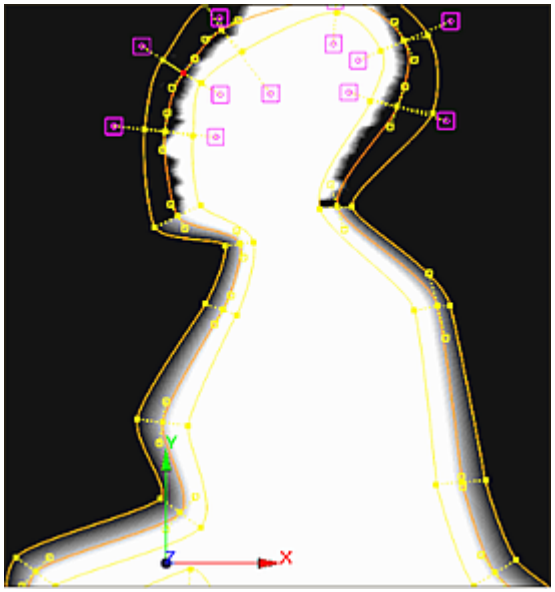
The Tracer is particularly useful for keying challenges such as:

- Very fine, wispy talent edges (hair, fur, or lace) where a blue or green screen may show through.
- An object not shot on a blue or green screen.



Garbage mask with pickers sampling image for more advanced softness

Image courtesy of Behavior Communications Inc.



Garbage mask with pickers applied to only select vertices, the remainder using an advanced gradient

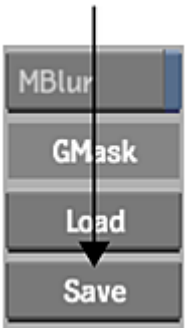
Image courtesy of Behavior Communications Inc.

Saving Garbage Mask Setups

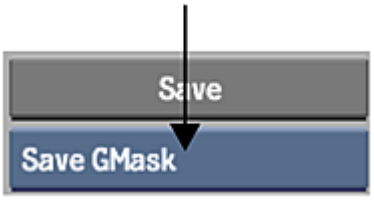
Garbage masks can be saved and loaded as setup files, with all vertex and animation information preserved.

To save garbage masks as a setup:

1 In the Garbage Mask menu, click Save to open the file browser.



2 From the GMask Type box, select Save GMask.



3 Type the name of the setup, and click Save.

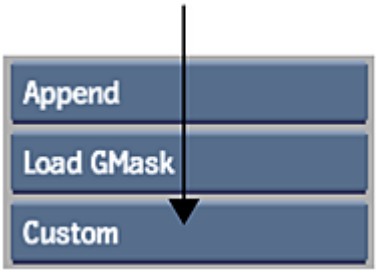
Loading Garbage Mask Setups

In the file browser, when loading garbage mask setups, you can quickly navigate to either the default gmask directory or a custom directory for the current project.

In Action, you can load a garbage mask as a 3D model. See Using Garbage Masks as 3D Geometry (page 722).

To navigate to the default or custom directory when loading garbage mask setups:

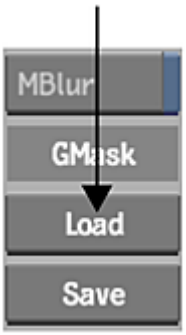
1 Select Default or Custom from the GMask Directory box.



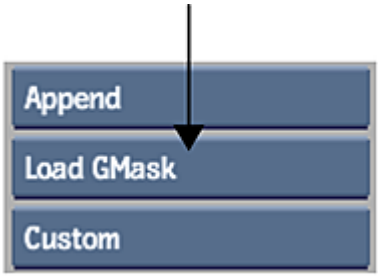
NOTE The custom directory is updated every time you navigate to a new directory.

To load a garbage mask setup:

1 In the Garbage Mask menu, click Load to open the file browser.

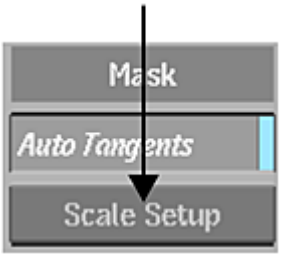


2 From the GMask Type box, select Load GMask.



3 From the file browser and select the name of the setup to load.

If the GMask setup you are loading was created in a different resolution than the current project, click Scale Setup to scale the GMask.



NOTE The Scale Setup button is available on the GMask Setup menu only when accessed from the GMask node in Batch or Batch FX or the Modular Keyer.

Preset Garbage Mask Setups

A library of preset garbage mask setups is available. To load a preset garbage mask, navigate to the */opt/Autodesk/project/<project_name>/gmask/default/* directory and select one of the following:

- 4_point_star
- 5_point_star
- 6_point_star
- arrowhead
- cat_eye
- circle
- diagonal
- diamond
- heart
- hexagon
- keyhole
- oval
- pentagon
- rectangle
- round_rectangle
- s_shape
- square
- triangle
- v_shape

You can also view proxies for each of these setups using the file browser.

Importing and Exporting Raw Setup Files

You can import and export raw setup files for animated garbage masks. A garbage mask exported as a raw setup file includes the following information:

- The number of keyframes in the animation of the spline
- The number of vertices in the shape
- The x, y, and z position of every vertex at each keyframe

To export a raw setup file:

1. In the Garbage Mask menu, click Save to open the file browser.
2. From the GMask Type box, select Export RAW.
3. Type the name of the file to be exported, and click Save.

To import a raw setup file:

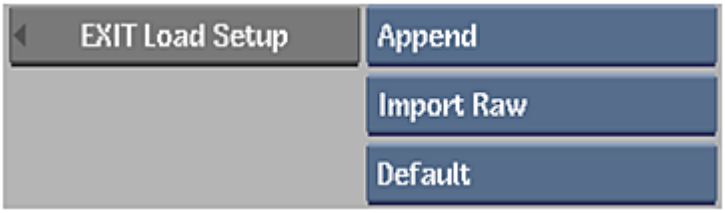
1. In the Garbage Mask menu, click Load to open the file browser.
2. From the GMask Type box, select Import RAW.
3. Navigate to the appropriate directory and select the raw setup file to import.

   The raw setup file is imported into the Keyer.

To import a raw setup file in the Modular Keyer:

1. In the Modular Keyer, click the GMask node in the processing pipeline. See Accessing the Garbage Mask Menu (page 1103).
2. In the GMask menu, click Load.

   The Load menu and file browser appear.
3. In the Load menu, select Import Raw and Default from the Load option boxes.



4. Enable Append or Replace.
5. In the file browser, specify the name and path of the file to import.

To export a raw setup file in the Modular Keyer:

1. In the Modular Keyer, click the GMask node in the processing pipeline. See Accessing the Garbage Mask Menu (page 1103).
2. In the GMask Setup menu, click Save.

   The Save menu and file browser appear.

(a) Save option box

3. In the Save menu, select Export Raw from the Save option box.
4. In the file browser, specify the name and path of the file to export.
5. Click Save.

Creating Customized Wipes with Garbage Masks

Use masks on the incoming clip to create a customized transition that wipes into the outgoing clip.

Setting the Mask Drawing Options

Before you create a mask, set how the points of the mask are drawn.

To set the drawing options:

1. From the Wipe Editor, click Setup.
2. Specify the setup options.

Enable:	To:
Auto Tangents	Create tangents for every new point you set.
Show Border	View the border defined in the Offset field of the Shape menu. The colour pot next to the Show Border button defines the colour of the border.
Invert	Reverse your wipe. For instance, if you originally create an expanding box wipe, enabling Invert creates a shrinking box.
Spline Keyframing	Allow animation of points on the mask.

3. To change the colour of the mask's wireframe, colour points, or offset border, click a colour pot and pick a colour.

Colour Correcting 21

About Colour Correcting

The Colour Corrector includes tools that provide precise control over colour values. You can modify luma ranges in a clip (shadows, midtones, and highlights), sample colours, and adjust the colour balance. You can rewire colour channels and suppress colours, as well as animate a colour correction by manipulating the animation curve in the Channel Editor.

When working with large images, you can free up additional screen space with the Overlay user interface. This feature is exclusive to the Colour Corrector and the Colour Warper. The Colour Warper's Overlay user interface is not available when accessing the Colour Warper as a Batch or Batch FX node.

For details on using the Overlay user interface, see Overlay User Interface (page 408).

Accessing the Colour Corrector

To access the Colour Corrector, you must load clips of the same resolution. If the clips you want to load have different resolutions, resize them so that they have the same resolution.

You can load a front clip, a front and back clip, or a front, back, and matte clip for colour correction. Changes in colour are applied to the front clip.

To access the Colour Corrector menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- Modular Keyer, Action or Paint tool.

Accessing the Colour Corrector from Modular Keyer, Action, or Paint

You can colour correct clips loaded into the Modular Keyer, Action or Paint.

Back clips and key-in clips cannot be colour corrected.

To access the Colour Corrector from the Modular Keyer:

1. Double-click on, or drag, the Colour Correct node from the node bar to the schematic.
2. Double-click the Colour Correct node.
   You are in the Colour Correct editor.

To access the Colour Corrector from Action:

1. From the Media menu, select the media containing the clip you want to colour correct.
2. Double-click the CC field for the media you want to colour correct.
   The clip is loaded into the Colour Corrector.

To access the Colour Corrector from Paint:

1. Do one of the following:
   - To colour correct a clip, click Setup in the Paint menu. In a Colour Correction field, enable CC.
     The clip is loaded into the Colour Corrector.
   - To colour correct a cutout, click Texture in the Paint menu, create a cutout. Click CC.
     The cutout is loaded into the Colour Corrector.

Hiding the Colour Corrector Menu

You can hide the Colour Corrector menu while adjusting an image if you find the menu distracting. With the menu hidden you can modify numeric fields, use the colour wheel, or access buttons via their corresponding keyboard shortcuts.

To hide the Colour Corrector menu while modifying a numeric field or the colour wheel:

1. Click and hold any numeric field or the colour wheel. For example, click and hold the Hue field.
2. Press `Alt+'` (on the Tilde key) while continuing to hold the cursor.
   The Colour Corrector menu is hidden.
3. Drag the cursor to modify the field value or adjust the colour wheel.
   The changes you apply will be visible in the image as you drag if Regen is enabled (see Dynamic Updating (page 1144)). Otherwise, the changes appear when you release the mouse.
4. Press `Alt+`` (on the Tilde key) to display the menu.

To hide the Colour Corrector menu while accessing buttons via their corresponding keyboard shortcuts:

1. Press `Alt+`` (on the Tilde key).
2. Use the Colour Corrector keyboard shortcuts.
3. Press `Alt+`` (on the Tilde key) to display the menu.

Viewing Clips

You can view a front, back, matte, or result clip in the image window. The front, back, and matte clips show the source clips used. The result clip shows your colour corrections as you apply them.

You can also view the following information for the clip:
- Channel: displays the numerical data used by the components in the clip.

- Reference: displays any referenced clips.
- Tracks: displays when an effect is used in the timeline
- Info: lists the channel, components, any related values, and the number of keys used in each effect in the clip.

To display a clip:

1 From the View box, select the clip you want to view. For example, select Result to view the result clip.

TIP Use the controls below the image window to change the frame displayed in the image window. Use the viewing tools in the upper-right corner of the panel to enlarge and pan images.

To change the clip that is displayed using the keyboard shortcuts:

1 Consult the following table.

To display:	Press:
The front clip	F1
The back clip	F2
The matte clip	F3
The result clip	F4
The channel data	F5
The track timeline	F5
The Info table	F5
The Referenced clips	Select Reference from the View box

Colour Corrector Settings

In the Colour Corrector Setup menu, you can change matte clip settings, the way that colour changes are updated, and background brightness. You can also reset colour corrections made from the Colour Corrector.

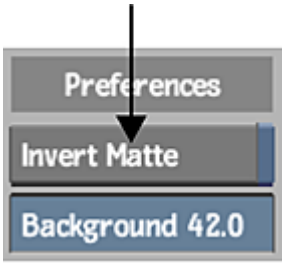
Inverting the Matte

You can invert the matte clip. Inverting the matte swaps the transparent and opaque areas of the matte and redefines the area of the front image that can be colour corrected, as well as the area of the back image that is shown.

To invert the matte clip:

1 Click Setup in the Colour Corrector menu.

2 Enable Invert Matte.

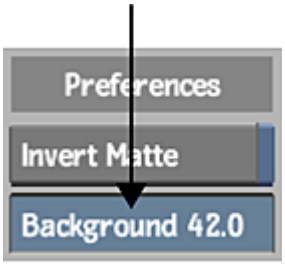


Changing the Background Brightness

You can adjust the brightness of your work area background. Changing the background brightness is a global setting. Changing the brightness through the Colour Corrector menu has the same effect as changing it through the Flame Preferences menu.

To change the background brightness:

1 Click Setup in the Colour Corrector menu.
2 Enter a value in the Background field.



Dynamic Updating

Use the Regen button to enable dynamic updating of changes you make in the Colour Corrector. When Regen is enabled, you see the changes interactively as you adjust the values in the various displays. When Regen is disabled, you do not see changes until values are entered after releasing the cursor, or by clicking the `Enter` key in the calculator.

To enable dynamic updating:

1 Select **Setup ➤ Master**.
2 Enable Regen.

Viewing Reference Clips

You can use the split bar to view and sample a reference clip while you work. Reference clips are particularly useful when you want to match colours or compare colour corrections. Carefully selecting a suitable reference clip will help you make your colour corrections quickly and efficiently. For example, you can use a reference clip from the Batch or Batch FX schematic to match saturation, whites and blacks, or colours. You can also switch between different references to ensure continuity throughout the project.

(a) Focus clip (b) Split bar (c) Reference clip

TIP If the Overlay user interface is enabled, the split bar can extend beneath the Overlay user interface. To access it, grab from an area where there are no menu elements.

Resetting Colour Corrections

You can reset colour corrections created with the Colour Corrector. You can also reset the corrections you make inside the Colour Corrector using the Reset Basics button, or by using the Range, Histogram, or Curve graphs.

To reset a colour correction:

1 Do one of the following:
- In the Colour Corrector Timeline FX or Tools editor, click Reset All. Click Confirm.
- In Batch or Batch FX, right-click the Colour Correct node in the schematic, and select Reset.

To reset a colour correction within the Colour Corrector:

1 Select one of the reset buttons depending on what you want to reset.

Select:	To reset:
Reset Basics	The basic properties of the Colour Corrector. Any changes made using the Curve, Histogram, or Range charts are unaffected.
Reset in the Curve menu	The settings defined by the Curve chart. The Basic properties of the Colour Corrector are unaffected.
Reset in the Histogram menu	The settings defined by the Histogram. The Basic properties of the Colour Corrector are unaffected.
Reset in the Range menu	The settings defined by the Range chart. The Basic properties of the Colour Corrector are unaffected.

Saving Setups and Preferences

In the Colour Corrector or Colour Warper, you can save or load colour correction or colour warper setups using the Save or Load button. Any setup can be loaded from the file browser for use with another set of clips using the same efffect. You can also save or load colour correction or colour warper preferences using the Load or Save button.

- To save setups or preferences, click Save.
- To load setups or preferences, click Load.

TIP Click Revert to revert to the last saved setup. All changes made since the previous Save operation are undone.

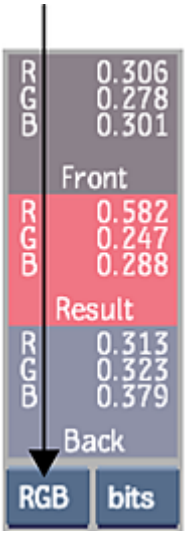
Colour Sampling

You can sample colours from the front and back clips, and instantly see how the front clip will be affected by the colour correction.

Colours are sampled using colour patches. The sampled colours appear in the patches with their colour values. You can display the RGB, HLS, or Y (luma) values for the sampled colours. You can choose to display numerical sample data as either RGB bit values or percentages.

To sample a colour:

1 Below the colour patches, select the sample option you want to use.



Select:	To:
Y	Display the NTSC luma values for the colours sampled. Either NTSC 601 or HD 709 luma values are displayed, depending on the settings in the configuration file.
HLS	Display the hue, lightness, and saturation values for the colours sampled.
RGB	Display the red, green, and blue values for the colours sampled.
Off	Turn off the colour information display.

2 Click the Front colour patch to sample a colour from the front clip. Click the Back colour patch to sample a colour from the back clip. Click both patches to sample front and back clips simultaneously.

The cursor becomes a colour picker.

3 To sample an individual pixel, click a colour in the image.

4 To sample an average colour, `Alt`-drag in the image, or press `Ctrl` and draw a selection box.

TIP Zoom in for more precise colour sampling.

The sampled colour appears in the colour patch. The sampled colour also appears in each menu.

In the:	A sample is displayed:
Colour Wheel	As a reference point on the colour wheel. A black reference point indicates the colour you sampled from the front clip. A white reference point indicates the sample from the back clip.
Histogram menu	As a red vertical line on the gradient bar, indicating the luma value. Only the front sample is displayed.
Curves menu	As red, green, and blue values mapped to the colour curves.
Ranges menu	As a red vertical line on the Luminance gradient bar, indicating the luma value. Only the front sample is displayed.

Colour Sampling and Processing Order

You can use any combination of colour correction commands to modify the front clip. The commands are processed in the following order:

- RGB Rewiring, Monochrome, Negative
- Hue Shifting
- Saturation
- Colour Balancing
- Gamma, Gain, Offset, and Contrast
- Histogram
- Colour Curves
- Chroma Suppression

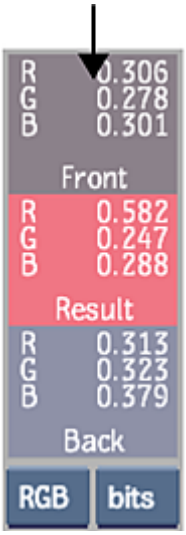
The processing order is important in determining the colour that appears in the middle (result) section of the colour patch. The middle section displays the colour resulting from the application of all colour correction setups; in other words, the colour at the end of the processing order. This colour is updated as you change any parameter value. The original front colour appears in the left section of the patch.

Matching Colours

You can match and colour correct the colours of the front clip using the back clip as a reference. To perform a match operation, sample colours from the front and back clip and then use the Match button in the Curves menu.

To match colours between two clips:

1. Load a front and back clip from the Thumbnail view.
2. Use the split bar to view the front and back clips in the image window.
3. Click the Curves tab to display the Curves menu.
4. Click the Front colour patch.

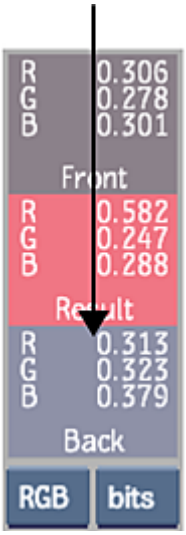


The colour picker appears.

5 Using the colour picker, select a colour in the front clip to be corrected.

The colour is added to the Front and Result colour patches.

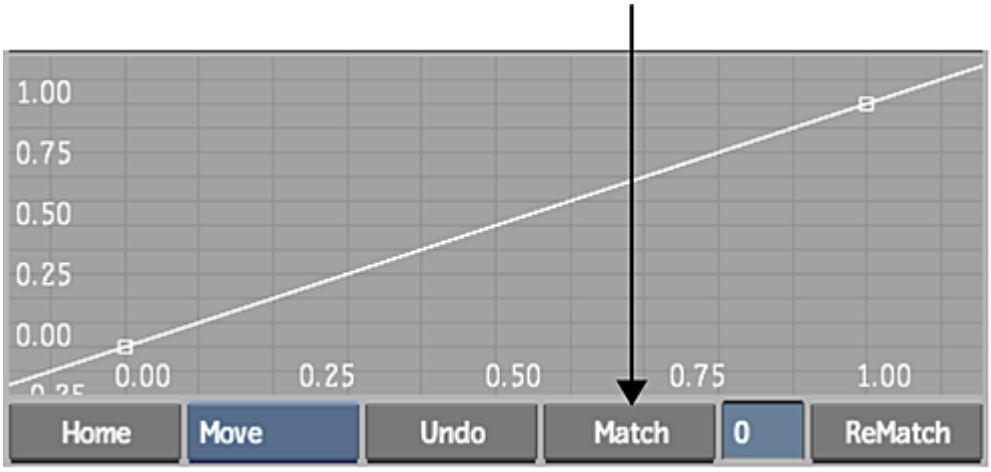
6 Click the Back colour patch.



The colour picker appears.

7 Using the colour picker, select a colour in the back clip to use as a reference colour.

8 Click Match.



The reference colour is applied to the result clip. The reference colour is also added to the Result colour patch and points are added to the colour curves.

When you use Match, a number is displayed in the numerical field beside the Rematch button. The first match you perform is indicated by a 1 in the field, the second by a 2, and so on. To revert to a previous match, click the numerical field and type the number of the match you want to recall. The colour patches and colour curves reset to the values stored in the specified match.

To reset the red, green, blue, or luminance curve, select a curve and click Reset in the Curve menu. The Curves menu also has its own Undo button that is separate from the overall Colour Corrector Undo.

Colour Correcting

Use the colour correction controls to perform many operations. You can:

- Colour correct shadows, midtones, and highlights separately or together.
- Adjust the hue, saturation, or contrast pivot point of an image.
- Rewire the red, green, or blue channels of an image.
- Create a negative or monochrome image.
- Adjust the gamma of the individual red, green, blue, or RGB channels.
- Adjust the red, green, blue, or RGB values in an image by modifying the gain, offset and contrast values for the channel.
- Perform colour suppression.
- Adjust the colour balance.

To reset any of these changes, click Reset Basics. To reset all the changes in the Colour Corrector, click Reset All.

Colour Correcting Shadows, Midtones, and Highlights

When you modify Saturation, Gamma, Gain, Offset, and Contrast, you can colour correct the shadows, midtones, and highlights ranges in the image individually or all together. Select the luma range you want to modify. Monochrome, Negative, Chroma Suppression, and RGB Rewiring always affect the Master tonal range.

Shadows Dark areas in the image.

Midtones Midlevel areas in the image.

Highlights Light areas in the image.

Master The entire image. Modification made in the Master range are applied after individual range modifications.

Rewiring the RGB Channels

Use the Red, Green, and Blue Channel Rewiring fields to rewire the red, green, and blue channels of an image. Use the Rewire option box to create a monochrome or negative image.

When you rewire a channel, the values for the current colour channel are replaced with those of the new channel. For example, if you select the R

G option in the Red Channel Rewiring field, the colour values of the red channel are replaced with those of the green channel.

The RGB Rewiring commands can be applied to the Master range only.

To rewire the red channel:

1 Select an option from the Red Channel Rewiring box.

Select:	To:
R←R	Use colour values for the red channel.
R←G	Replace colour values of the red channel with those of the green channel.
R←B	Replace colour values of the red channel with those of the blue channel.
R←Y	Replace colour values of the red channel with the luma of all channels.

Select:	To:
R←1-R	Replace colour values of the red channel with its inverse. For example, 1 corresponds to the maximum value of the given colour channel. In 8 bits, this value is 255.

NOTE The green and blue channels are rewired in the same way as the red channel (described in the previous table).

Creating a Negative or Monochrome Image

You can convert a colour image into a monochrome or negative image using the Rewire option box.

TIP You can also convert the image manually by clicking the appropriate Rewire selection box (R, G or B) and selecting the Y option for monochrome or 1- R for a negative image.

To convert a colour image into a monochrome or negative image:

1 Select Mono or Negative from the Rewire option box.

The image is automatically converted to a monochrome or negative image.

2 To cancel the changes, click Reset Basics.

Suppressing Colours

You can suppress the RGB or CMYw colours in an image. Chroma suppression is useful for removing blue spill from a blue-screen composite.

To suppress a colour:

1 Click Master.

NOTE Chroma suppression can only be applied to the Master tonal range.

2 Enable the appropriate button for the colour to suppress.

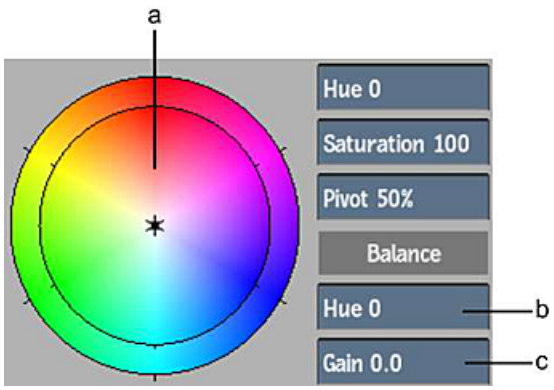
Enable:	To suppress:
R	Red
G	Green
B	Blue
C	Cyan
M	Magenta
Yw	Yellow

The status box on the button is light blue when the button is enabled.

NOTE Suppression of RGB and CMYw always reduces luma.

Adjusting the Colour Balance

You can adjust the colour balance in an image using the colour wheel or the Balance Tools Hue and Gain fields. The value of Hue determines the colour to add and the value of Gain specifies the amount of colour to use.



(a) Colour wheel (b) Hue field (c) Gain field

Pure red is the 0-degree point for hue on the colour wheel. When you increase the Hue value, you move counterclockwise on the colour wheel. When you decrease the Hue, you move clockwise on the colour wheel.

The centre of the colour wheel represents 0 Gain. As you increase the value of Gain, you move towards the edge of the colour wheel and add more of the selected colour to the image.

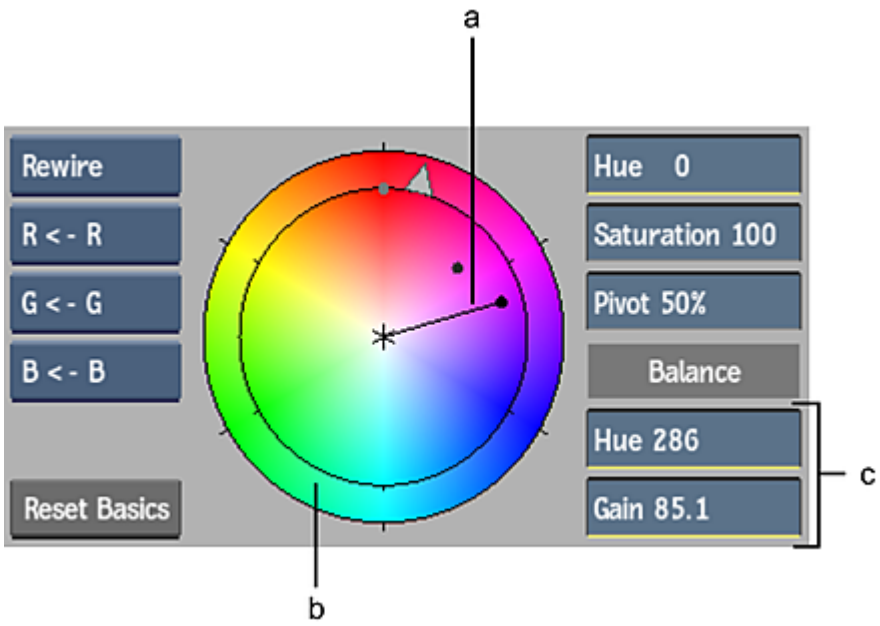
NOTE If Saturation is set to 0, no colour balancing is performed, regardless of the Gain value. In the Channel Editor, Balance channels are labelled hue_balance and gain_balance, respectively.

Creating and Modifying a Colour Balance Point

You can make colour balance points for each range (highlights, midtones, and shadows) or for all ranges simultaneously (master). Click inside the colour wheel to create a balance point for the current tonal range.

The current range's colour balance point is connected to the hub of the colour wheel by a line. You can constrain how the point is modified according to how you select it. If you select the line, the radius of the line Strength is constrained. If you select the point, the direction of the line Tint is constrained. You can make unconstrained modifications by clicking anywhere in the wheel, or by entering values in the Balance fields.

NOTE As you move the colour balance point on the colour wheel, the values in the Balance Tools Hue and Gain fields update automatically. You can also set the values for Tint and Strength directly in these fields.



(a) Current colour balance point (b) Hue shift/saturation comparison (outside ring is Result; inside ring is Source) (c) Balance tools

Colour balancing is performed in RGB colour space. The luma of the image is not changed. For example, if Hue (tint) is set to 0 and Gain (strength) is set to 100%, pure red is added to the image. However, a black pixel remains black since it keeps the same luma value.

To change the colour balance in the image:

1. Enable Regen so that the image will update as you change the colour balance.
2. Select the range you want to modify.
3. On the colour wheel, click and drag toward the colour you want to use.

   As you drag the cursor, a colour balance point follows the cursor. Depending on the range you are modifying, the colour balance points are shaded to make for easier identification (from light grey to black, in this order: Highlights, Shadows, Midtones, Master).

   You can continue dragging outside of the colour wheel. The colour balance point is replaced by a triangle in the outer circle of the colour wheel. The triangles are also shaded depending on the range selected (from white to grey, in this order: Master, Highlights, Shadows, Midtones).

   The image dynamically updates as you drag.

   TIP You can interact directly in the image window to change the colour balance. While holding the range keyboard shortcut (`V` for Master; `Z` for Shadows; `X` for Midtones; or `C` for Highlights), drag directly in the image window. The colour wheel and Balance Tools fields update accordingly.

4. If you are not satisfied with the resulting image, click and drag the colour balance point or triangle again. You can continue to move the colour balance point or triangle until the desired effect is achieved.

Adjusting the Colour Range

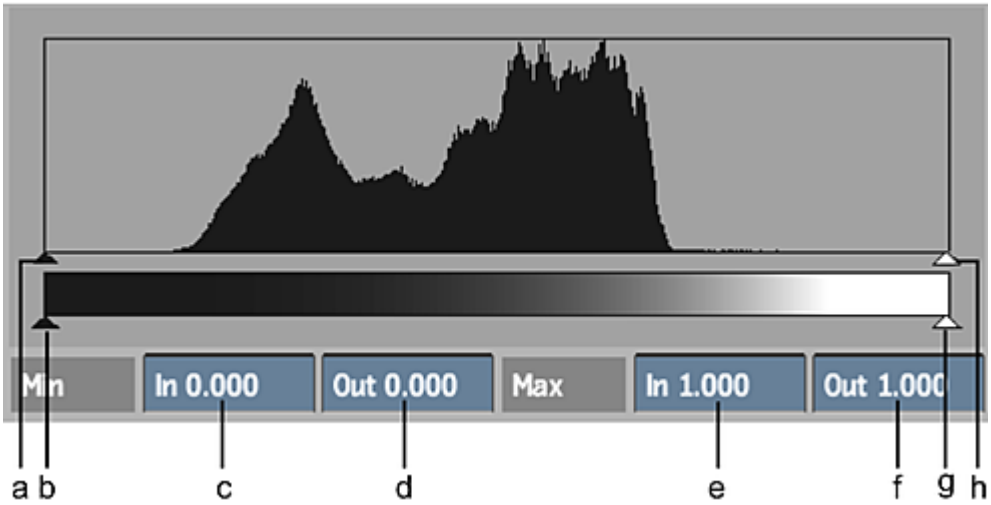
You can use the Histogram menu to adjust the range of values used for the red, green, blue, or luminance channels of an image. The histogram shows the colour distribution of pixels in the front and result image for the selected channel. The horizontal axis represents the values of pixels, from black at the left to white at the right. The vertical axis represents the number of pixels with these colour values.

When working with 16-bit floating point images, the values of the histogram are represented on a logarithmic scale. When working with integer images (8, 10, or12-bit), the values of the histogram are represented on a linear scale.

To access the Histogram menu:

1 Click the Histogram tab.

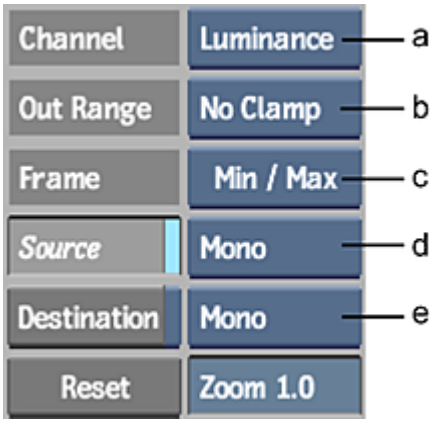
The Histogram menu options are described as follows.



(a) Minimum Input slider (b) Minimum Output slider (c) Minimum Input field box (d) Minimum Output field (e) Maximum Input field (f) Maximum Output field (g) Maximum Output slider (h) Maximum Input slider

Input controls Use the minimum and maximum input sliders or fields to set the input levels of your image.

Output controls Use the minimum and maximum output sliders or fields to set the output levels of your image.



(a) Channel Selection box (b) Out Range Option box (c) Frame Selection box (d) Source View Option box (e) Destination View Option box

Select	To Frame:
Min / Max	Based on the minimum and maximum slider values.
Full Range	The complete range of histogram values.
Plot Colour	The plot and reference colours.
Home	Based on a [0:1] horizontal and vertical range.

Increasing Image Contrast

The slider controls directly below the histogram are the Input Level controls. These controls are used to set the range of colour values used in the image:

- The white triangle on the right sets the maximum value for the range.
- The black triangle on the left sets the minimum value for the range.

You can use the Input Level controls to increase the contrast in the image. For example, if you set the minimum value to 50, pixels with colour values less than 50 are remapped to 0 (black). Pixels with colour values greater than 50 are remapped to the appropriate values. This darkens the image and increases the contrast in the shadow areas.

You can also set the maximum and minimum limits for the colour range by entering the values directly in the Minimum and Maximum Input fields on either side of the histogram.

To increase the contrast in an image:

1. Display the Histogram menu.
2. Click the Master, Shadows, Midtones, or Highlights button to select the parts of the image you want to modify.
3. From the Channel Selection box, select Luminance, Red, Green, or Blue as the channel you want to work with.
4. Position the cursor on the black triangle below the histogram. Drag right to darken the shadow areas in the image.

The value of the lower limit for the colour range appears in the Minimum Input field.

5 Position the cursor on the white triangle below the histogram. Drag left to brighten in the highlight areas in the image.

The value of the upper limit for the colour range appears in the Maximum Input field.

NOTE If you are in Luminance, you can invert the image by reversing the order of the black and white triangles.

6 To cancel the changes, click Reset Basics.

Reducing Image Contrast

The slider controls on the gradient bar below the histogram are the Output Level controls. These controls are used to set the range of colours used in the image:

- The white triangle on the right sets the maximum value for the range.
- The black triangle on the left sets the minimum value for the range.

You can use the Output Level controls to decrease the contrast in the image. For example, suppose that you set the minimum value to 100. A pixel with a colour value of 0 is remapped to 100. Pixels with colour values greater than 0 are remapped to the appropriate values. This has the effect of lightening the image and decreasing the contrast in the shadow areas.

Suppose that you set the maximum value for the range to 200. A pixel with a colour value of 255 is remapped to 200. Pixels with colour values less than 255 are remapped to the corresponding values. This has the effect of darkening the image and decreasing the contrast in the highlight areas.

You can also set the maximum and minimum limits for the colour range by setting the values directly in the Minimum and Maximum Output fields.

To reduce the contrast in an image:

1 Display the Histogram menu.

2 Click the Master, Shadows, Midtones, or Highlights button to select the parts of the image you want to modify.

3 From the Channel Selection box, select Luminance, Red, Green, or Blue as the channel you want to work with.

4 Position the cursor on the black triangle below the Output Level bar. Drag right to lighten the shadow areas in the image.

The value of the lower limit for the colour range appears in the Minimum Output field.

5 Position the cursor on the white triangle below the gradient bar. Drag left to decrease the brightness in the highlight areas.

The value of the upper limit appears in the Maximum Output field.

Defining Luma Ranges

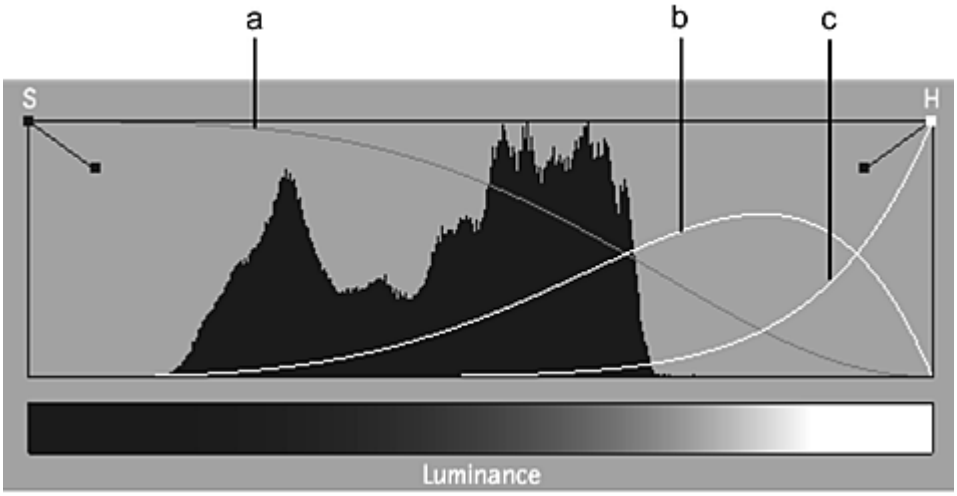
Use the histogram in the Ranges menu to modify the shadow, midtone, or highlight ranges.

When working with 16-bit floating point images, the values of the histogram are represented on a logarithmic scale. When working with integer images (8, 10, or12-bit), the values of the histogram are represented on a linear scale.

To access the Ranges menu:

1 Click the Ranges tab.

The Ranges menu options are described as follows.



(a) Shadows curve (b) Midtones curve (c) Highlights curve

The X-axis is luma and the Y-axis is weight. The Shadow curve is labelled “S”, the Highlight curve is labelled “H”, and the Midtones curve is not labelled.

(a) Frame Selection box (b) Source View option box (c) Destination View option box

Frame Selection box Choose how you want to frame the Ranges Histogram.

Select	To Frame:
Min / Max	Based on the minimum and maximum slider values.
Full Range	The complete range of histogram values.
Plot Colour	The plot and reference colours.
Home	Based on a [0:1] horizontal and vertical range.

To see the effect of adjusting these curves:

1. Adjust the colour balance for each of the Shadows, Midtones, and Highlights channels. Note the effect that this setup has on the image.
2. Display the Ranges menu. Adjust the curves using the tangent handles.

   You should see that the resulting image is different from that in step 1. The difference is the result of the changes you made to the luma curves for the shadows, midtones, and highlights.

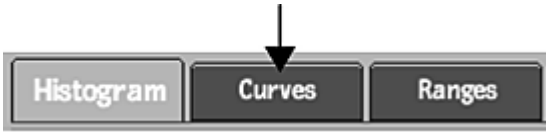
Remapping Colour Values

Like the Histogram menu, the Curves menu allows you to remap the colour values for the individual red, green, blue, and luminance channels of the image. However, instead of adjusting the colour values by resetting just the maximum and minimum values for the colour range, the Curves menu allows you to remap any value in the colour range precisely.

To reset any changes to the Curves chart, click Reset. To reset all the changes in the Colour Corrector, click Reset All. The Curves menu also has its own Undo button that is separate from the overall Colour Corrector Undo.

To access the Curves menu:

1. Click the Curves tab.



There is one colour curve for each of the red, green, blue, and luminance channels of an image. The colour curves are generated by plotting the input values for the source image versus the output values for the resulting image.

For integer images, the input values range from 0 to 255, and are plotted along the horizontal axis. The output values range from 0 to 255, and are plotted along the vertical axis. The range for both the input values and the output values is from 0 to 255 in 8-bit mode, and from 0 to 4095 in 12-bit mode.

For 16-bit floating point images, the input values range from 0 to 1, and are plotted along the horizontal axis. The output values range from 0 to 1, and are plotted along the vertical axis. By default, there are 2 points on the curve, mapping 0 to 0 and 1 to 1. Use the Out Range option box to define whether the curves are constant (select Clamp) or linear (select No Clamp) before the first point of the curve and after the last point of the curve.

The default curve for each channel is a diagonal line that extends from the lower-left corner to the upper-right corner of the graph. The default curve represents the initial state in which the colour values for the pixels of the source image are equal to the values for the corresponding pixels of the resulting image. For example, all pixels that have a value of 100 in the source image also have a value of 100 in the resulting image.

The colour values of an image are remapped when you change the shape of a colour curve. Use the Curves Editor to add, delete, or move vertices on the curve. For example, move the end vertices on the curve to change the maximum and minimum values for the colour range. Add a vertex anywhere on the curve to remap a particular colour value.

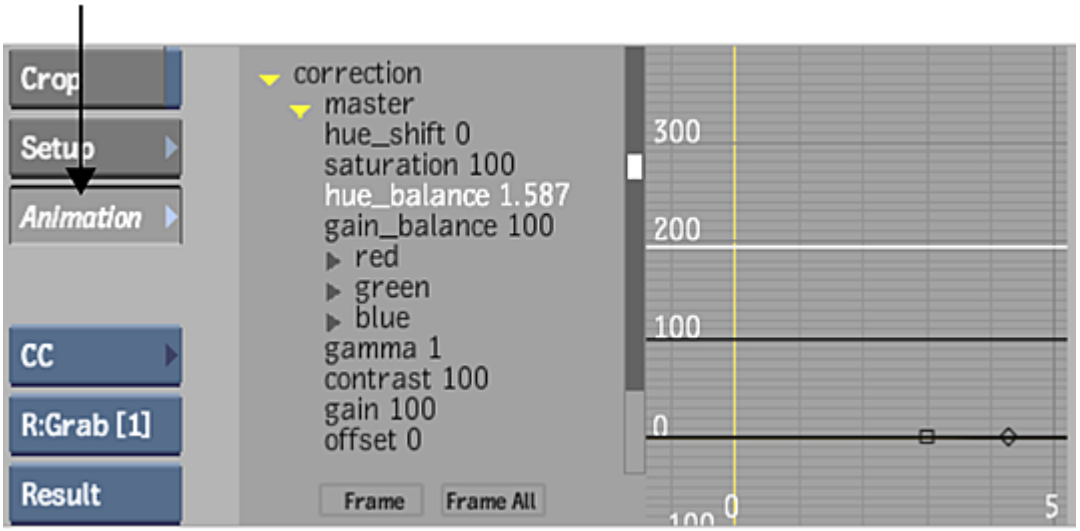
The Curves Editor behaves in much the same way as an animation curve in the Channel Editor. To edit colour curves, select Add, Delete, or Move from the Tools box.

Animating a Colour Correction

Use the Animation controls to animate a colour correction that has been applied to a clip. You can animate any value in the Colour Corrector.

To access the Animation controls:

1 In the Colour Corrector menu, click Animation.



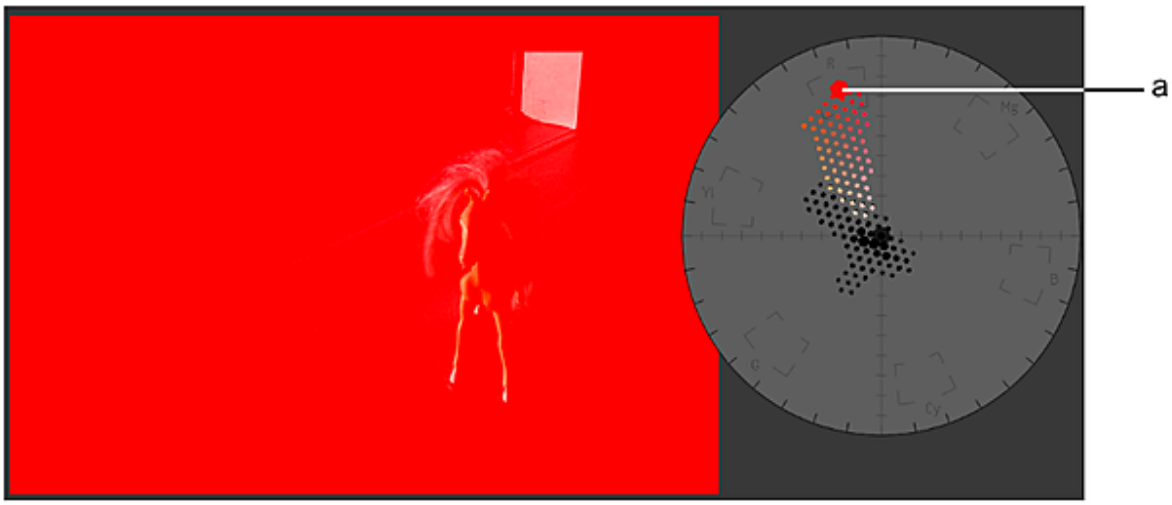
About the Colour Warper

When you colour correct an image or clip, use the Colour Warper to perform advanced colour corrections and create artistic colour effects. The way in which you approach these tasks depends on your goal, the number of clips you are using, and the type of clips being used.

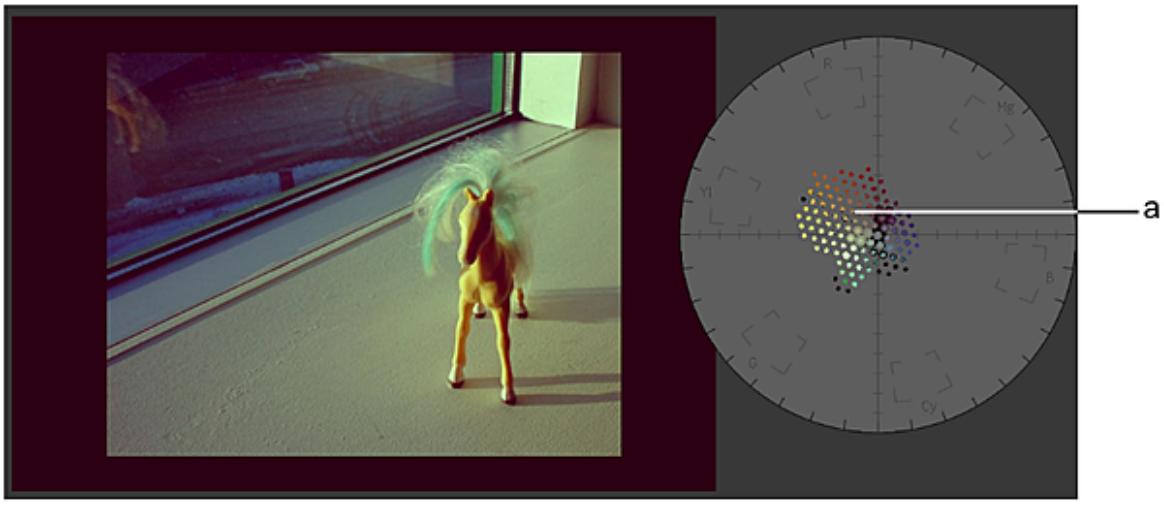
Clips created from source material shot with the same camera equipment under the same lighting conditions may be colour corrected quickly and easily to correct lighting and colour imbalances. Clips created from source material shot at different times of the day, in different seasons, at different locations, or using different equipment require more work. With the Colour Warper, you can manipulate colours with precision and ease, working on the entire clip as you would with traditional tools or working with a matte to adjust a range of colour in the clip.

Use the Colour Warper to gesturally set black and white levels, adjust specific colours and colour ranges, and accurately match colours in one clip to another. You can also perform hue shifts and suppress colour to remove colour spill or create visual effects such as a colour cast. While you manipulate the colour content of a clip, you can monitor reference clips as well as changes in the colour distribution to ensure that you achieve the result you want. Original data is always preserved, so you can adjust colours without the risk of permanently losing colour information.

When working with 16-bit floating point images in the Colour Warper, you can plot colours outside of the 0 to 1 range. Even when working with integer images, clamped colour information (colours that go beyond the RGB range) can be retrieved using the Colour Warper controls.



(a) Clamped colour



(a) Colour information restored

When working with large images, you can free up additional screen space with the Overlay user interface. This feature is exclusive to the Colour Corrector and the Colour Warper. The Colour Warper's Overlay user interface is not available when accessing the Colour Warper as a Batch or Batch FX node.

For details on using the Overlay user interface, see Overlay User Interface (page 408).

Accessing the Colour Warper

Use the Colour Warper to modify the colour content in your clips. You can manipulate colour content with intuitive controls that provide precise colour correction, and view histograms that help you visualize your image's colours. You can also output a matte corresponding to selected colours.

You can access the Colour Warper as a Batch or Batch FX node that you drag and drop into the process tree or the Modular Keyer's processing pipeline. Access the Colour Warper from the Modular Keyer to remove colour spill, or access it from Batch or Batch FX to modify the colour content of your clips.

To access the Colour Warper menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).

- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- Modular Keyer, Action or Paint tool.

Setting Up Your Work Environment

Set up your work environment to streamline the colour correction process. Use the tools provided to examine the clip or image and efficiently perform advanced colour corrections. As your work progresses, you can modify the Colour Warper environment to suit the task at hand.

You can:

- Display multiple viewports.
- View reference clips.
- View the colour and luma content of a source (front) clip and result clip.
- Apply a colour correction to all or part of a clip.
- Dynamically update colour information as you work.
- Reset entire colour corrections.
- Clear or reset individual values.
- Save and load colour correction setups and preferences.
- Use the Undo/Redo list.

Multiple Viewports

When working in the Colour Warper, you can display up to four viewports at a time in the image window, including a view of the Channel Editor. Multiple viewports are convenient when you want to modify colour values, create mattes, and compare your result with other images in different viewports simultaneously.

NOTE The Overlay user interface cannot be used with multiple views in the Colour Warper tool.

Viewing Reference Clips

You can use the split bar to view and sample a reference clip while you work. Reference clips are particularly useful when you want to match colours or compare colour corrections. Carefully selecting a suitable reference clip will help you make your colour corrections quickly and efficiently. For example, you can use a reference clip from the Batch or Batch FX schematic to match saturation, whites and blacks, or colours. You can also switch between different references to ensure continuity throughout the project.

(a) Focus clip (b) Split bar (c) Reference clip

TIP If the Overlay user interface is enabled, the split bar can extend beneath the Overlay user interface. To access it, grab from an area where there are no menu elements.

Viewing Colour Information with a Vectorscope

In the Colour Warper, you can use a 2D or 3D vectorscope to help match colours, adjust shadows and highlights, view colour distribution, and ensure that broadcast standards are met. The vectorscopes show the changes you make to colour content in a clip or image.

Each vectorscope can display:

- A histogram showing the distribution of image pixels across luma and hue ranges. The histogram dynamically updates to reflect your changes as you modify colours in the image. Exposure and contrast settings in the image display viewer are taken into account.
- Source and destination colours. Source colours are the colour values in the front clip and destination colours are the colour values in the result clip.
- Plotted and reference colours that are obtained by sampling images in the image window. See Sampling Clips in the Image Window (page 1171).

NOTE The 2D and 3D vectorscopes only appear in Result view.

To view colour information in either the 2D or 3D vectorscope:

1. In the Colour Warper menu, enable Scope.

   The vectorscope appears in the image window.

   NOTE You can only view one vectorscope at a time.

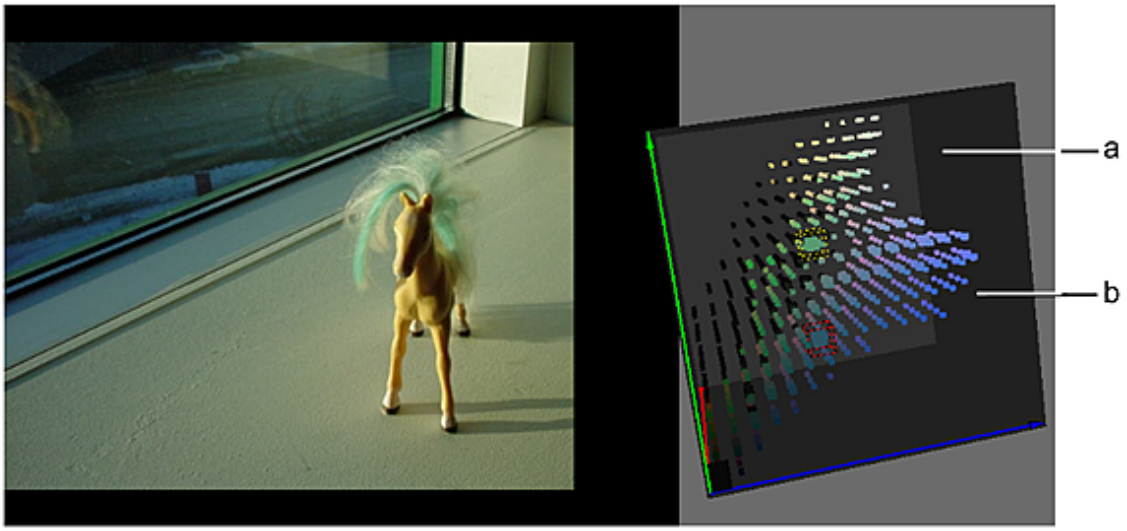
2. Click Setup.
3. From the Scope box, select the vectorscope you want to use to view the clip.

   2DDisplays the 2D vectorscope. Use the 2D vectorscope to analyse the colour content of the clip and locate specific colours in terms of their chroma values. When you view a clip in the 2D vectorscope,

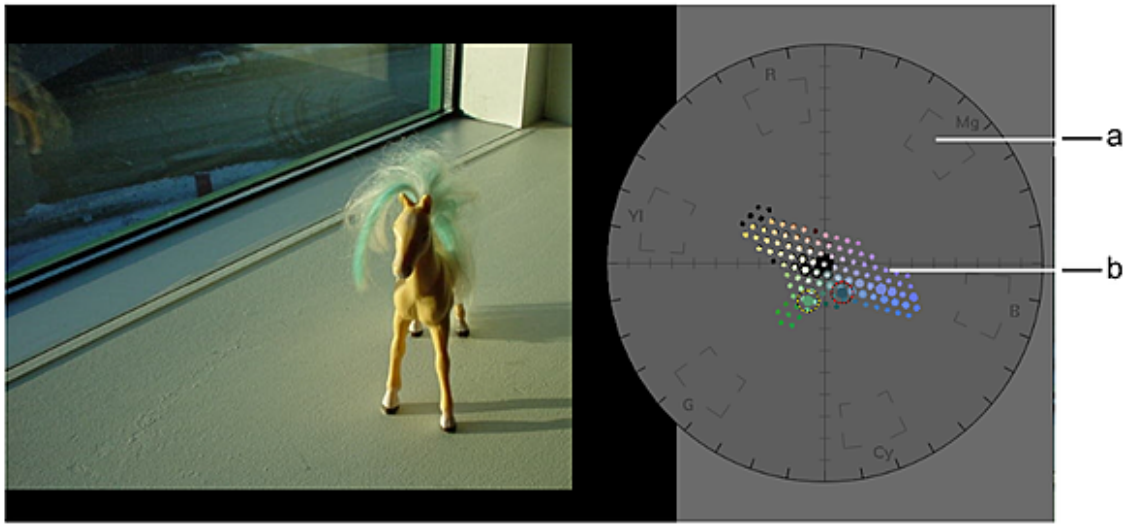
you see it in terms of hue and saturation. The size and placement of the colour squares in the 2D vectorscope show the distribution of all the colours in the clip—like a colour wheel; the distance from the centre to the perimeter of the scope maps colour saturation, with the outside edge of the scope delineating pure colour.

3DDisplays the 3D vectorscope. Use the 3D vectorscope to both analyse the colour content of the clip and to locate specific colours in terms of their chroma and luma values. When you view a clip in the 3D vectorscope, you see it in terms of HLS (hue, lightness, and saturation). Its X, Y, and Z axes represent red, green, and blue, and the size and placement of the colour cubes show the distribution of all the colours of the clip in RGB colour space.

The selected vectorscope is displayed in the image window. As you scroll through a clip, the histogram is updated to display the colour distribution of each frame.



(a) 3D vectorscope (b) 3D histogram



(a) 2D vectorscope (b) Histogram

4 Set the following options to customize the display of the 2D or 3D vectorscope.

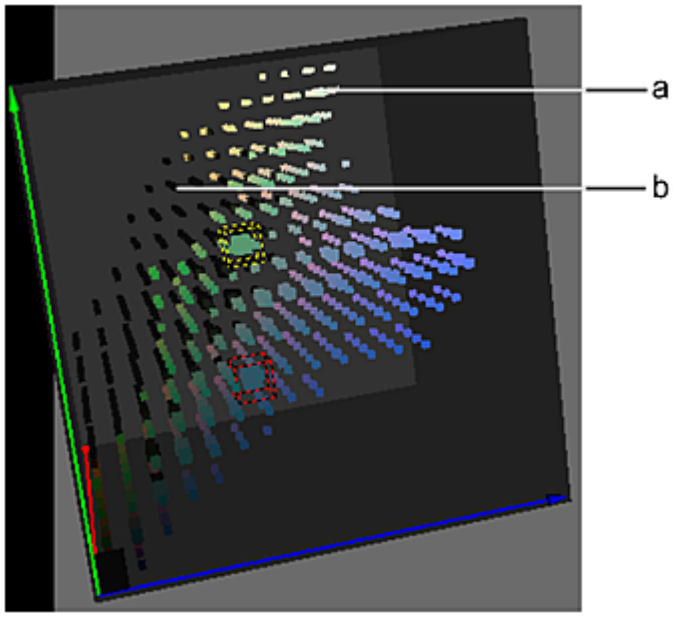
Enable:	To:
Canvas	Show the vectorscope canvas. When the canvas is off, the vectorscope is transparent, but can be outlined.

Enable:	To:
Lines	Show the vectorscope outline.

5 Set the following histogram options to show clip colour information.

Enable:	To:
Source	Show a histogram of the colour values in the front, or source clip. The source colour values are obtained from the current frame. If you input both front and matte clips, the colour values in the front clip that are defined by the white part of the matte are displayed.
Destination	Show a histogram of the colour values in the result clip. The destination colour values are obtained from the current frame. If you input both front and matte clips, the colour values in the front clip that are defined by the white part of the matte are displayed.

When both Source and Destination are enabled, you see both the source and destination colour values. The source clip's colour values are displayed in black and the result clip's colour values are displayed in colour. When only Source is enabled, the source clip's colour values are displayed in colour.



(a) Result clip (in colour) (b) Source clip (in black)

NOTE You can also toggle the Source and Destination buttons in the Basics menu to show or hide source and destination colours in the histograms.

6 From the Size box, select the size of the coloured squares or cubes that make up the histogram. You can display 8x8, 16x16, or 32x32 coloured dots.

7 To set SMPTE bars for the 2D vectorscope, select an option from the Bars box.

Select:	To match:
Bars 75%	75% SMPTE bars. If the current project is NTSC, the bars are set to 75% by default.

Select:	To match:
Bars 100%	100% SMPTE bars.

The parameters of the histogram of the 2D vectorscope update accordingly.

8 To toggle between the vectorscopes, double-click the vectorscope in the image window.

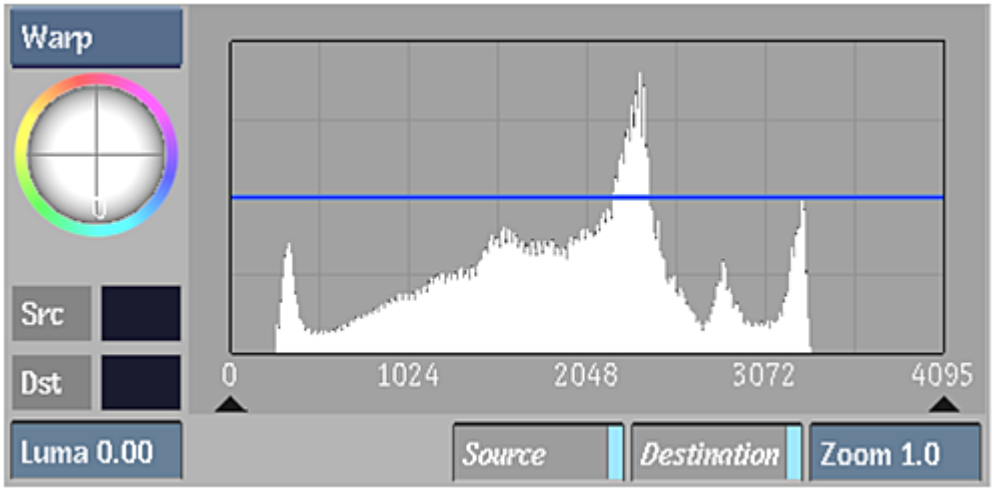
Use these other shortcuts to interact with the vectorscopes.

Press:	To:
`Alt`-drag	Move the 2D or 3D vectorscope
`Shift`-drag	Zoom the 2D or 3D vectorscope
`Ctrl`-drag	Rotate the 3D vectorscope. If you `Ctrl`-drag the 2D vectorscope, the 3D vectorscope appears and rotates.

9 To position a vectorscope at its default location and size, click Home.

Viewing Luma Information with the 2D Luma Histogram

The 2D luma histogram shows the changes you make to luma content in a clip. Use the 2D luma histogram when you want to adjust the lightness of a sample. The 2D luma histogram is displayed in the Basics menu of the Colour Warper.



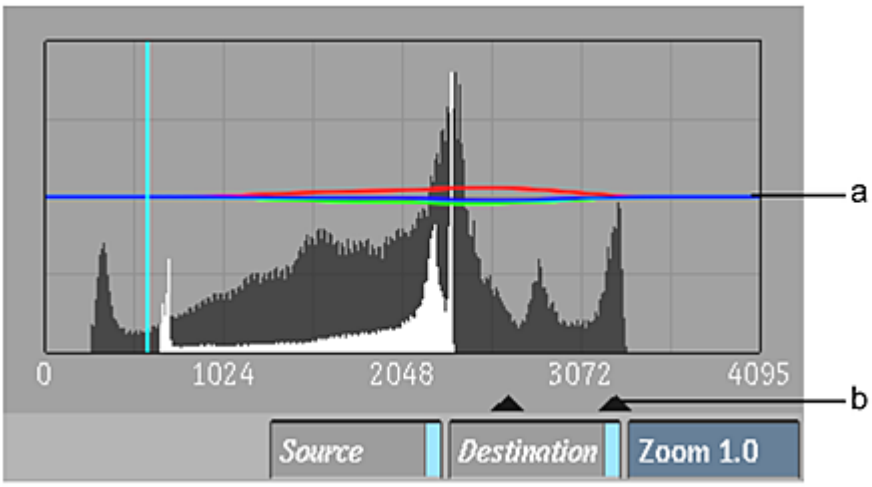
The 2D luma histogram can display:

- A histogram showing the distribution of image pixels across the luma range—the size and distribution of the vertical bars show the luma distribution.
- Source and destination colour values. Source colour values are the luma values in the front clip and destination colours are the luma values in the result clip.
- Plotted and reference colours that are obtained by sampling images in the image window. See Sampling Clips in the Image Window (page 1171).

To view luma information in the 2D luma histogram:

1 In the Colour Warper menu, click Basics.

The 2D luma histogram appears in the Basics menu. R, G, and B values that are modified by the Midtones trackball are displayed as curves in the histogram.



(a) RGB curves (b) Midtone triangle

2 Set the following options to show clip luma information.

Enable:	To:
Source	Show a histogram of the luma values in the front, or source, clip. The source luma values are obtained from the current frame.
Destination	Show a histogram of the luma values in the result clip. The destination colour values are obtained from the current frame.

When both Source and Destination are enabled, you see both the source and destination luma values in the histogram. Source clip colour values are displayed in black and result clip colour values are displayed in off-white.

NOTE You can also toggle the Source and Destination buttons in the Setup menu to show or hide source and destination colours in the histograms.

Colour Correcting All or Part of an Image

Use the options in the Work On box to apply colour corrections to all or part of an image. You can select an option from the Work On box at any time to further modify the image. Use Master to apply colour corrections to the entire image and use a selective to generate a matte and perform selective colour correction—apply the colour correction to a selected range of colour. The Work On box is available in every menu of the Colour Warper.

(a) Work On box

To colour correct all or part of a clip:

1 Select an option from the Work On box.

Select:	To:
Master	Use the Basics menu to modify the entire image. In Master mode, you cannot output a matte.
Sel. 1, Sel.2, or Sel. 3	Generate a matte from the front clip selective colour correction. You can then use the Basics menu to modify the range of colour defined by the matte. In a selective mode, you can output the matte, selective, or result. See Selecting Colour Ranges for Colour Correction (page 1173).

Interactively Updating Colour Information

When you modify the colours in a clip, create mattes, or scroll through the image window, you can either have the image and vectorscope update interactively as you move the cursor or have them update afterwards.

When interaction is slow, update the image afterwards to avoid lags in processing time, especially when working on high-resolution clips or images. Lags in processing time can also occur in Result view, where a large amount of processing is required to display the result.

- To update colour information interactively: In the Colour Warper menu, enable Regen. When you scroll through the clip, the 2D vectorscope or 3D vectorscope is dynamically updated to display the colour distribution of each frame. When you drag a trackball or field, the image is dynamically updated, showing the changing colour information.
- To update colour information after completing an operation: In the Colour Warper menu, disable Regen. After you scroll through a clip, the 2D vectorscope or 3D vectorscope is updated to display the colour distribution of the current frame. After you make an adjustment with a Colour Warper control, the image is updated

Resetting Colour Warper Effects

Resetting the Colour Warper

You can reset colour effects created with the Colour Warper.

To reset a colour correction:

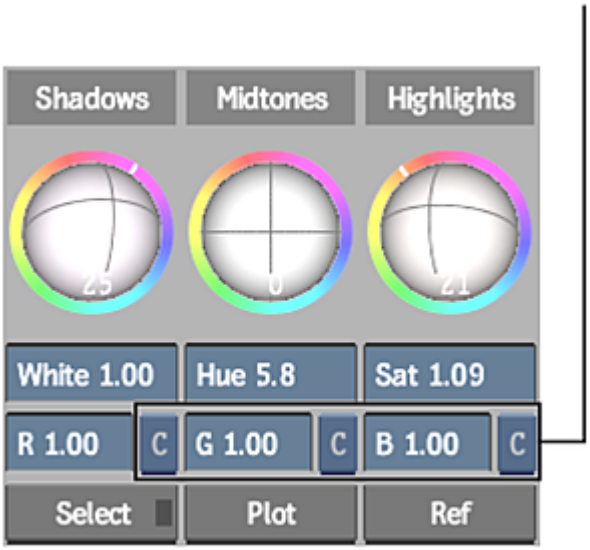
1 Do one of the following:
- In the Timeline FX or Tools Colour Warper editor, click Reset All. Click Confirm.
- In Batch, Batch FX or the Modular Keyer, right-click the Colour Warper node and select Delete.

Resetting and Clearing Values

You can clear or reset several Colour Warper values in the Basics menu using individual Clear/Reset boxes. C resets the value for the current frame by setting a keyframe. R resets the value for the entire animation curve.

To reset a Colour Warper value:

1 In the Basics menu, do one of the following:
- Select C from the Clear/Reset box next to a control.



The value is reset at the current frame.
- Select R from the Clear/Reset box.

The value is reset for the entire animation curve.

NOTE Fields that do not have Clear/Reset boxes next to them cannot be reset. Use the Undo box instead.

Saving Setups and Preferences

In the Colour Corrector or Colour Warper, you can save or load colour correction or colour warper setups using the Save or Load button. Any setup can be loaded from the file browser for use with another set of

clips using the same efffect. You can also save or load colour correction or colour warper preferences using the Load or Save button.

- To save setups or preferences, click Save.
- To load setups or preferences, click Load.

  TIP Click Revert to revert to the last saved setup. All changes made since the previous Save operation are undone.

Using Undo and Redo

Use the Undo and Redo boxes to remove or redo colour modifications. To undo or redo a modification, click a box and select an option from the list. Set the number of undo levels in the Setup menu.

NOTE You also set the number of undo levels for the Modular Keyer in the Setup menu of Batch or Batch FX.

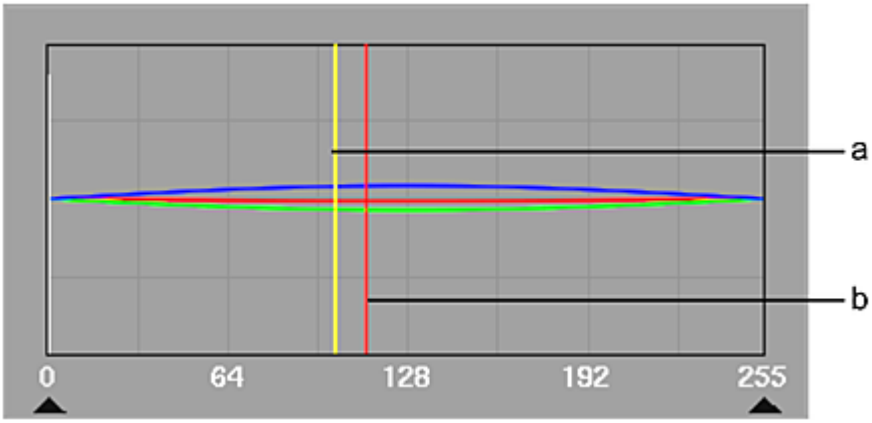
(a) Redo option box

Sampling Clips in the Image Window

When you generate mattes, modify clips, or match colours between clips, you can sample colours in both the result clip and a reference clip. Use Plot to sample a colour from the result clip and Ref to sample one from a reference clip. The Plot and Ref buttons appear on every menu in the Colour Warper.

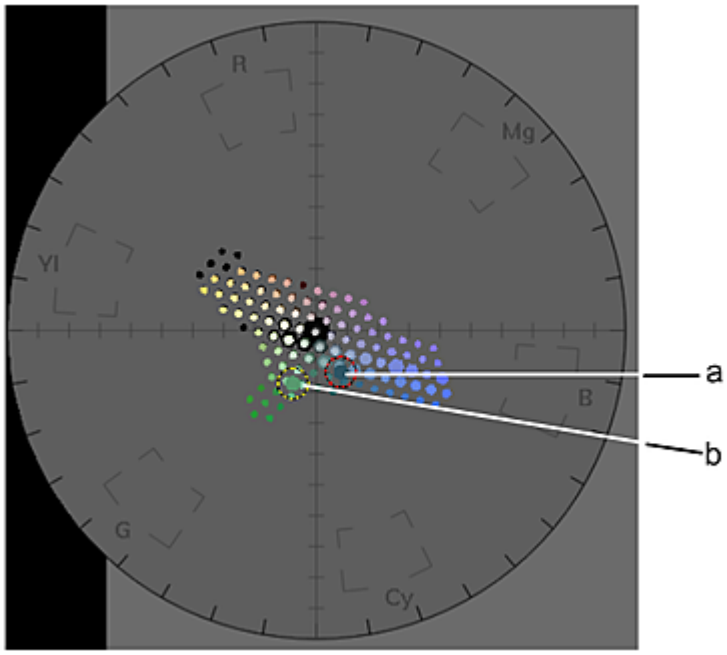
NOTE Ref samples cannot be adjusted.

Samples appear in the histograms. In the 2D luma histogram, the Plot sample is represented by a red bar and the Ref sample is represented by a yellow bar.



(a) Reference colour (yellow vertical line) (b) Plotted colour (red vertical line)

In the vectorscope, the Plot sample is outlined in red and the Ref sample is outlined in yellow.



(a) Plotted colour (outlined in red) (b) Reference colour (outlined in yellow)

You can use Plot to get more information on colours you want to change. When you are adjusting colours and levels and are not sure which controls to use, click Plot and select a colour in the image. You can then view the histograms to determine which control you should use. For example, if the plotted colour appears in the middle of the 2D luma histogram, you can modify the colour in the image using the Midtones trackball while viewing the plotted colour in the 2D vectorscope.

To sample clips:

1 Load a reference clip and position the split bar so that both the reference clip and the result clip appear in the image window.

2 Do one or both of the following:

- Enable Plot and then sample the result clip.

You can:	To:
Click-drag	Sample a single pixel in the image. You can drag through the image until you locate the pixel you want to sample.
Ctrl-drag	Sample an average taken from a range of colours in the image.

A red vertical line appears in the 2D luma histogram indicating the luma of the sample. The sampled colour is also outlined in red in the 2D or 3D vectorscope. In the 2D vectorscope, you see the sample in terms of hue and saturation. In the 3D vectorscope, you see it in terms of HLS. If you are plotting a colour in a 16-bit floating point image that is out of the 0:1 range, the red outline appears outside of the vectorscope.

- Enable Ref and then sample the reference clip.

You can:	To:
Click-drag	Sample a single pixel in the image. You can drag through the image until you locate the pixel you want to sample.
Ctrl-drag	Sample an average taken from a range of colours in the image.

A yellow vertical line appears in the 2D luma histogram and the sampled colour is outlined in yellow in the 2D or 3D vectorscope. If you are referencing a colour in a 16-bit floating point image that is out of the 0:1 range, the yellow outline appears outside of the vectorscope.

Selecting Colour Ranges for Colour Correction

With some images or clips, you may need to perform selective colour correction—where you modify a range of colour, as opposed to the entire clip or image. Use the tools in the Selective menu to create selectives. Selectives are generated mattes used to isolate colour ranges for selective colour correction.

In the Selective menu, you can:

- Create and refine a matte using tolerance and softness to define the range you want to modify.
- View a matte, a selected colour range, or the result.
- Enable mattes to apply the selective colour correction to the result image. Disable mattes to remove the selective colour correction from the result image.

TIP Set the image window to 2-Up view (Alt+2) so that you can create your matte in one viewport while simultaneously monitoring the result in the other.

Generating Mattes

Mattes can be used for several purposes. For instance, use a matte to remove colour spill, hue shift an object in an image for artistic effect, or match colours in a specific range. Use mattes to define the range of colour you want to modify in the result clip.

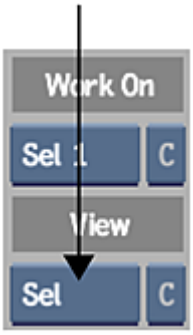
You can generate up to three mattes using the Selective menu.

TIP To get the best key, make sure you have enough processing speed to see the result in real time while interacting with the Tolerance and Softness boundary handles. In the Modular Keyer, you can use the Crop feature to improve interaction.

To generate a matte for selective colour correction:

1. Enable Plot and then sample the image.

   A black dot representing the sample appears on the hue cube. A red line also appears in the luma range and the sample is outlined in the 2D or 3D vectorscope. When you create the matte, you can modify the softness and tolerance boundaries to include or exclude the plotted colour.
2. From the Work On box, select a selective (Sel 1, Sel 2, or Sel 3). With each selective, you define a range for a matte by selecting colours in the front clip.
3. From the Selective View box that appears, select Sel.



   The front clip is displayed as a greyscale image.
4. Click one of the Define controls to set the initial softness and tolerance for the matte.

Click:	To define:
Pick Custom	The tolerance range based on a sample from the image. You must drag the cursor over the image to define initial tolerance.
R, G, B, C, M, or Yw	The tolerance range based on the selected colour channel. For example, click Yw to use the yellow channel to set the tolerance. The luma range is set to default tolerance and softness values.

Click:	To define:
Shadows, Midtones, Highlights	The tolerance range based on the selected luma range. These buttons expand the tolerance and softness boundaries to include all ranges of colours in the image.

The initial softness and tolerance is set for the matte. The range you use to define the matte becomes visible through the greyscale image. The unselected colours remain greyscale.

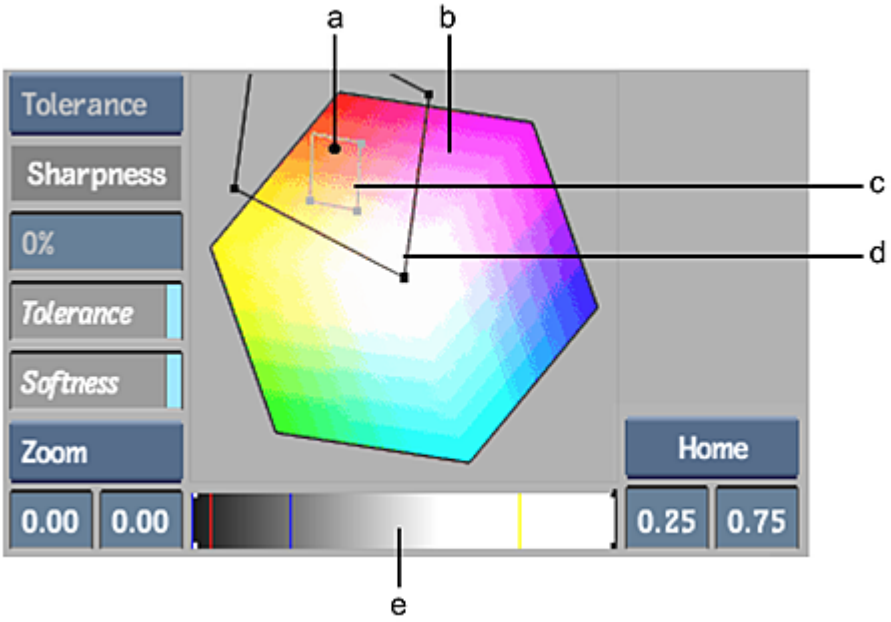
(a) Original colour visible through the greyscale display.

The Active button associated with the selective is enabled when you set the initial softness and tolerance. When an Active button is enabled, the selective's matte will be applied to the result clip. You can disable the Active button at any time if you do not want to apply this matte to the result clip.

TIP To redefine a matte based on a different Define control, click the control you want to use. The matte is reset according to your selection.

5 Enable Tolerance and Softness.

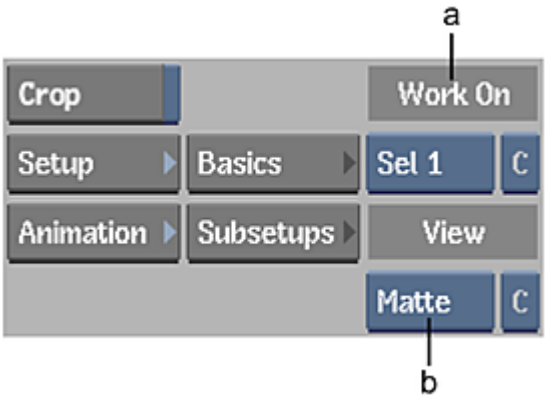
The range of colour used to define the matte is shown on the hue cube with tolerance and softness indicators. The light grey outline shows tolerance and the black outline shows softness. When you define a matte with a luma range, it is shown in the luma range—the white line indicates the tolerance and the yellow line indicates softness.



(a) Plotted colour (black dot) (b) Hue cube (c) Tolerance range (d) Softness range (e) Luma range

TIP Select Home from the Frame Options box to reset the hue cube to its original size and position.

6 To view the matte while you refine it, select Matte from the Selective View box.



(a) Work On box (b) Selective View box

The matte appears in the image window. The black and grey areas of the matte can be colour corrected. The white areas will remain unaffected.

TIP Select Matte view to output a matte for use in another tool in Batch or Batch FX. The selective must also be selected in the Work On box.

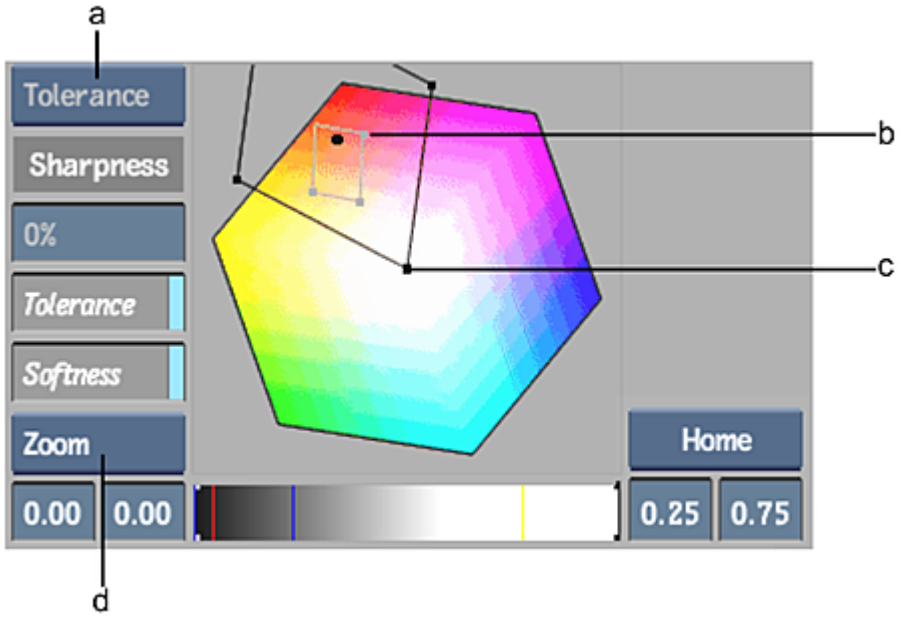
7 To refine the matte, do any of the following:

- Adjust softness and tolerance by selecting options from the Adjusting box and then sampling the result clip.

Select:	**To:**
Tolerance	Add tolerance to the matte.
+Softness	Add softness to the matte.

Select:	To:
-Softness	Remove softness from the matte.

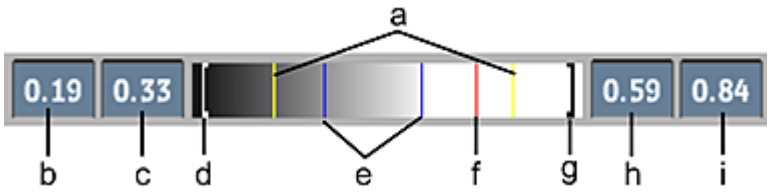
- Adjust softness and tolerance by selecting Move from the Move/Zoom box and then moving the handles of the tolerance or softness outlines on the hue cube. The Tolerance and Softness buttons must also be enabled.



(a) Adjusting box (b) Tolerance handle (c) Softness handle (d) Move/Zoom box

TIP You can zoom the hue cube by selecting Zoom in the Move/Zoom box and then dragging the cube. Alternatively, `Ctrl+spacebar`-drag to zoom. You can pan the hue cube by pressing `spacebar` and dragging.

- Adjust the softness and tolerance in the luma range by dragging the Softness and Tolerance fields.



(a) Softness range (b) Low Softness field (c) Low Tolerance field (d) Low bracket (e) Tolerance range (f) Plot line (g) High bracket (h) High Tolerance field (i) High Softness field

When working with 16-bit floating point images, you can press `spacebar` to pan the gradient bar, and `Ctrl+spacebar` to zoom the gradient. Select Home from the Frame Options box to reset the gradient to the default 0:1 position. If you have softness or tolerance values out of the 0:1 range, select Autoframe from the Frame Options box to view the complete gradient range. Select Plot Colour from the Frame Options box to enlarge the gradient to include the plot and reference colours.

- If the matte appears grainy, drag the Sharpness field to adjust softness and reduce noise.

8 To apply a Gaussian blur, enable G. To apply a box blur, disable G and set the width and height of the blur with the X and Y fields.

9 To invert the matte, enable Inv Selection.

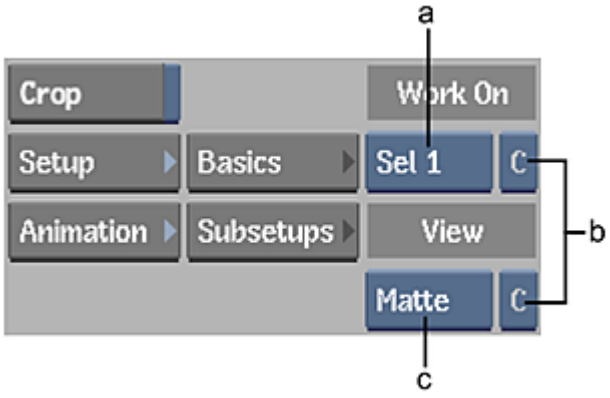
10 Continue fine-tuning the matte until you are satisfied.

You can select the selective in the Work On box from any menu in the Colour Warper to perform advanced colour corrections on the range defined by the matte. You can also change the view in the Selective View box.

TIP When you switch from the Selective menu to any other menu in the Colour Warper, Result appears in the Selective View box. Select Result to view the result clip.

Clearing and Deleting Mattes and Colour Corrections

Use the C/R boxes next to the Work On box and Selective View box to reset a matte or a colour correction. C deletes the matte or colour correction at the current frame by setting a keyframe. R deletes the matte or colour correction for the entire animation curve.



(a) Work On box (b) C/R boxes (c) Selective View box

NOTE If there is no matte or colour correction associated with the correction, the C/R button is disabled.

To delete a matte for the entire animation curve:

1 Select the matte from the Work On box (Sel 1, Sel 2, or Sel 3).

2 Select R from the C/R box next to the Selective View box.

The matte is deleted. All keyframes and colour corrections associated with the matte are also cleared.

NOTE Keyframes associated with the colour correction are not cleared.

To delete a matte at the current frame:

1 Select the matte from the Work On box (Sel 1, Sel 2, or Sel 3).

2 Select C from the C/R box next to the Selective View box.

The matte is deleted at the current frame and a keyframe is set.

To delete a colour correction for the entire animation curve:

1 Select a matte or the entire image from the Work On box (Master, Sel 1, Sel 2, or Sel 3).

2 Select R from the C/R box next to the Work On box.

The colour correction is deleted and all keyframes are cleared. If you selected a matte (Sel 1, Sel 2, or Sel 3) from the Work On box, the matte is unaffected and its keyframes are not cleared.

To delete a colour correction at the current frame:

1. Select a matte or the entire image from the Work On box (Master, Sel 1, Sel 2, or Sel 3).
2. Select C from the C/R box next to the Work On box.

   The colour correction is cleared at the current frame and a keyframe is set. If you selected a matte (Sel 1, Sel 2, or Sel 3) from the Work On box, the matte is unaffected and its keyframes are not cleared.

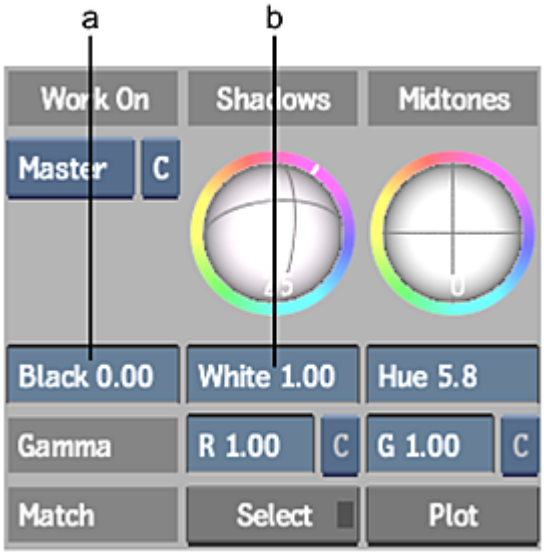
Correcting Colour Imbalances

During production, differences in lighting and equipment often create colour imbalances in and between clips. When you work on these clips in the Colour Warper, you should perform a basic colour correction to colour balance them. You can balance colours using the tools in the Basics menu. Perform your tasks in the following order:

- Set black and white levels.
- Remove unwanted colour.

Setting Black and White Levels

When you balance the colours in a clip, you should always start with the darkest and lightest parts of the image. You may, for example, need to establish parameters for the black and white (luma) content to ensure that the intensity of the image is in accordance with established broadcast parameters. These parameters are known as *black points* and *white points*. You use the Black and White fields to establish black and white points.



(a) Black field (b) White field

Black field Modifies the luma in the image's shadows without affecting the chroma.

White field Modifies the luma in the image's highlights without affecting the chroma.

To set black and white levels:

1. Sample the highlights in the image. To determine the white point, select a white that appears to be flat rather than reflective so that you obtain a more accurate reading. Do not use a highly reflective

surface such as a window or metal surface as they are often too bright. Enable Plot and then sample the whites in the result clip.

In the Basics menu, a red plot line is displayed in the 2D luma histogram indicating the luma value of the sampled whites in the result clip.

2 While viewing the 2D luma histogram and the image, adjust the white level to modify the luma value and set the white point—drag left to darken the highlights and right to lighten them. The Plot sample is updated as you drag the field.

TIP You can view both the 2D luma histogram and 2D vectorscope to determine what adjustments you need to make to the highlights. For example, if the white point is set to an acceptable level in the 2D luma histogram but the sampled chroma value shows an unwanted colour cast in the 2D vectorscope, use the Highlights trackball to modify the chroma value in the highlights. See Removing Unwanted Colour (page 1180).

3 Enable Plot and sample the shadows in the result clip to determine the black level. Sample the darkest part of the image to determine the black point.

A red plot line is displayed in the 2D luma histogram indicating the luma value of the sampled blacks in the result clip.

4 While viewing the 2D luma histogram and the image, adjust the black level to modify the luma value—drag left to darken the shadows and right to lighten them. The Plot sample is updated as you drag the field.

TIP Once the black level is set, you can use the Shadows trackball to adjust the chroma values in the highlights while viewing the plot sample in the 2D vectorscope. See Removing Unwanted Colour (page 1180).

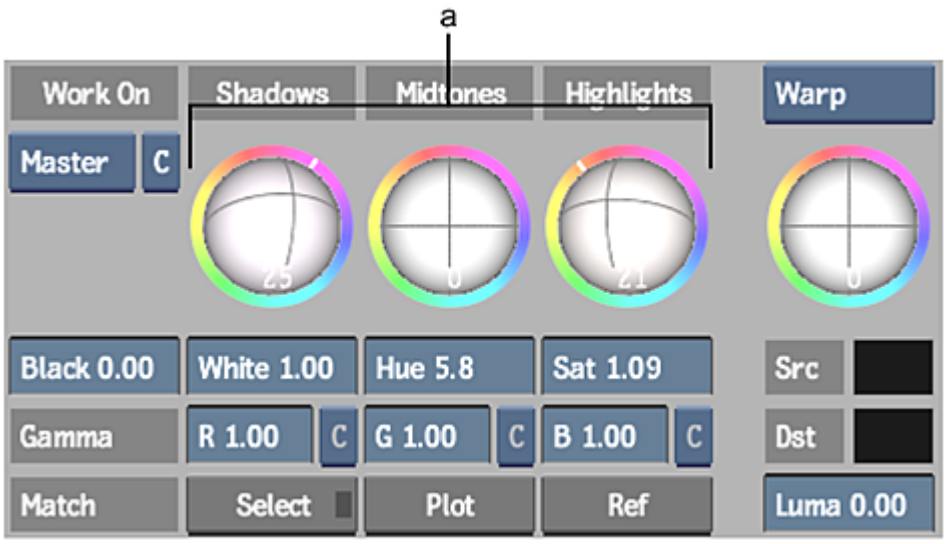
Removing Unwanted Colour

Unwanted colour can be caused by factors such as inconsistent lighting conditions during a shoot or incongruities between the white levels set on a video camera and the given lighting conditions. Factors such as these can result in clips or images that contain unnatural looking colours or one predominant colour, which gives the image an unwanted colour cast. After setting black and white points, you can remove the unwanted colour without affecting the black and white levels. You can:

- Use the Shadows, Midtones, and Highlights trackballs to remove colour from the shadows, midtones, and highlights ranges.
- Use the Suppress trackball to remove a range of colour, such as a colour cast that pervades the entire image.

Not all colour casts detract from the natural appearance of the image. Before you remove a colour cast, it is best to analyse the image or clip, both in relation to other clips in the project and with an eye on maintaining the atmosphere, or mood inherent to the clip. For example, images of sunny summer days should have a reddish cast to convey an atmosphere of warmth. Completely removing the reddish cast may result in a colder feel than you want. In cases where the cast is a complementary one, you may want to refine it to improve the overall effect. See Creating Colour Casts (page 1191).

The shadows, midtones, and highlights trackballs in the Basics menu are described as follows.



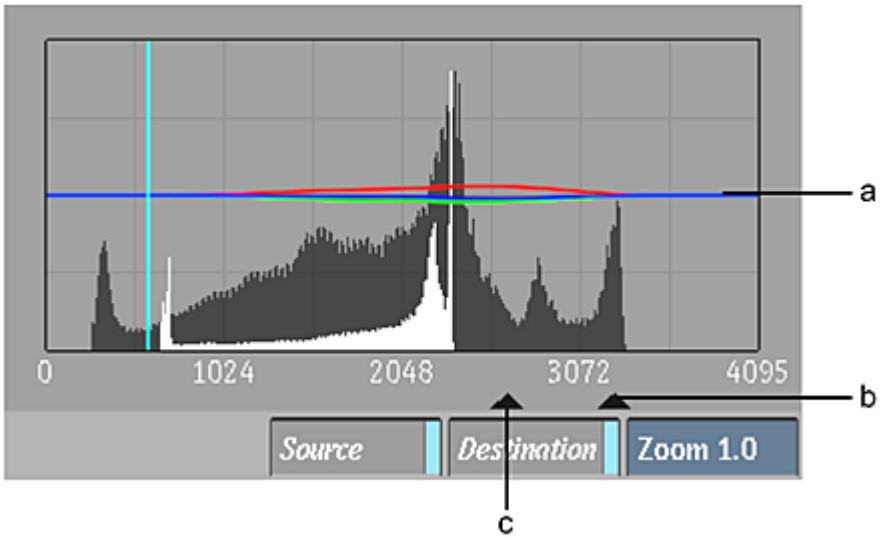
(a) Trackballs

NOTE To make the trackballs respond to changes in pressure, use the Pointer preferences in the Preferences menu.

Shadows trackball Modifies the chroma in the low luminance area without affecting the luma.

Midtones trackball Modifies horizontal RGB curves in the 2D luma histogram. These curves represent the amount of chroma tint you apply to the image. For example, if the red and green curves are above the middle line while the blue curve is below the middle line, you are adding a yellow tint to the midtones components of your image.

By default, the Midtones trackball affects all ranges except white and black, as shown by the histogram's RGB curves. To set the luma range affected by the Midtones trackball, use the two midtones triangles below the 2D luma histogram. Drag the triangles closer together to decrease the range and further apart to increase it. You can modify the range as many times as you like. When working with 16-bit floating point images, the midtones range is set to 0:1, and the midtones triangles can only be moved in this range.



(a) RGB curves (b) Midtones triangle (c) Midtones triangle

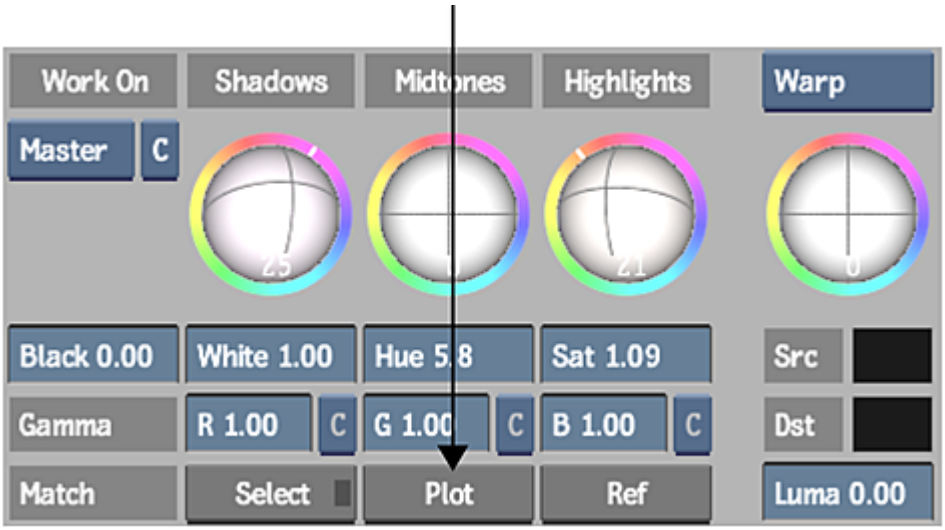
Because you can modify the range that the Midtones trackball affects, the trackball will always return to its default reference value—when you release the Midtones trackball and apply the change, it will return to 0.

NOTE You can animate the curve using the Shape channel under Midtone in the Channel Editor. Click Animation to view the Channel Editor.

Highlights trackball Modifies the chroma in the high luminance area without affecting the luma.

To remove unwanted colour using the Shadows, Midtones, and Highlights trackballs:

1. Click Basics to display the Basics menu.
2. Enable Plot and then sample the image's highlights.



   The sample (outlined in red) appears in the 2D or 3D vectorscope.

   TIP You can determine the colour of a sample by dragging right in the Saturation field to temporarily increase the saturation while monitoring the sample in the 2D vectorscope. When you release the mouse, the level that appears in the field resets to its default reference value. Once you determine the cast of the colour, click Undo to reset the saturation level and then resample the image.

3. If necessary, double-click the 3D vectorscope to display the 2D vectorscope.
4. While viewing the 2D vectorscope, drag the Highlights trackball to move the sample towards the centre of the 2D vectorscope.

   The chroma in the high luminance area of the clip is modified without affecting the luma.
5. Drag the Midtones trackball to make minor adjustments to the chroma content.

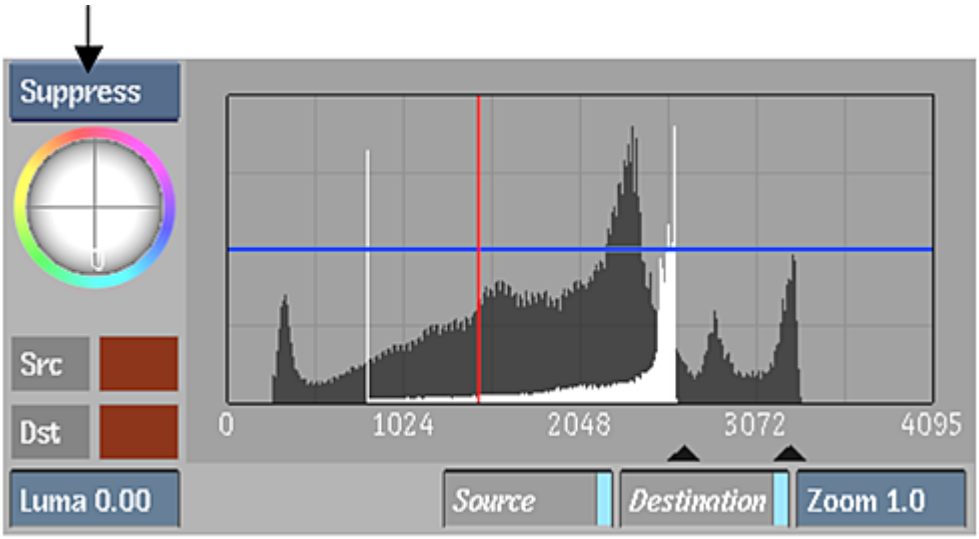
   TIP Drag the left midtones triangle to decrease the range affected by the Midtones trackball. Make sure the range includes the sampled colour.

   The RGB curves in the 2D luma histogram reflect the changes you make. More of the unwanted colour is removed from the image (while still preserving luma) without affecting the black and white points.

   TIP To achieve true white, drag the Highlights and Midtones trackball to move the plot sample to the centre of the 2D vectorscope.
6. Continue modifying the highlights and midtones until you are satisfied with the result.
7. Enable Plot and then sample the image's shadows.
8. Repeat steps 2 to 6 using the Shadows and Midtones trackballs to remove unwanted colour from the shadows (and midtones) in the image.

To remove unwanted colour using the Suppression trackball:

1. Click Basics to display the Basics menu.
2. Select Suppress from the Trackball option box.



3. To sample the colour you want to suppress, enable Plot and then sample the image.

   The sample (outlined in red) appears in the 2D or 3D vectorscope.
4. While monitoring the 2D vectorscope and image, drag the Suppress trackball towards the colour you want to suppress to move it closer to the centre of the 2D vectorscope. For example, to suppress yellows, move the trackball towards yellow.

   TIP Disable Source and Destination to only view the sample.
5. Release the trackball.
6. To further increase the suppression of the same colour, use the trackball to move the colour closer to the centre of the 2D vectorscope.

Improving Contrast

Poor image contrast can be caused by various factors. For example, when you remove a colour cast from a clip, it may become washed out. Washed out clips usually occur when the image contrast and saturation levels are too low. To complete the task of balancing the colours in the clip, use the tools in the Basics menu to correct the poor contrast. To improve contrast and the overall look of the clip, you can:

- Adjust the luma and chroma content in the shadows and highlights, including the black and white point.
- Increase colour saturation without changing luma.
- Modify the luma and chroma content using the Gamma controls.

Adjusting Shadows and Highlights to Improve Contrast

When there is not enough, or too much, black or white in a clip, it will lack contrast. If you want to improve the contrast by changing both the chroma and luma content, including the black and white points, use the controls in the Basics menu to adjust the shadows and highlights in the image.

To adjust the shadows and highlights to improve contrast:

1. Click Basics to display the Basics menu.
2. Enable Plot and then `Ctrl`-drag the cursor over the portion of the result clip containing black or white.

   In the Basics menu, a red plot line is displayed in the 2D luma histogram indicating the luma value of the sampled blacks. In the vectorscope, you can view the saturation and hue.
3. Use the 2D luma histogram and 2D vectorscope to determine which controls you need to use (such as the Black field, and Shadows and Midtones trackballs).
4. While viewing the 2D luma histogram, adjust the black level to modify the luma value. The Plot sample is updated as you drag the field.
5. While viewing the 2D vectorscope, drag the Midtones and Shadows (Highlights) trackballs towards the colour you want to increase in the blacks or whites. The Plot sample is updated as you drag the trackball.

   TIP To achieve true black or white, drag the Midtones and Shadows trackball to move the plot sample to the centre of the 2D vectorscope.
6. Continue modifying the shadows until you are satisfied with the result.

   TIP View the sample in the 3D vectorscope to make more refined modifications.

Increasing Saturation

When an image is desaturated, the colours can appear washed out. To improve contrast, you can increase the colour saturation globally, saturate specific hues, or do both. Increasing saturation levels increases the colour intensity and causes minor changes to the hue. The luma content is virtually unaffected when you modify saturation.

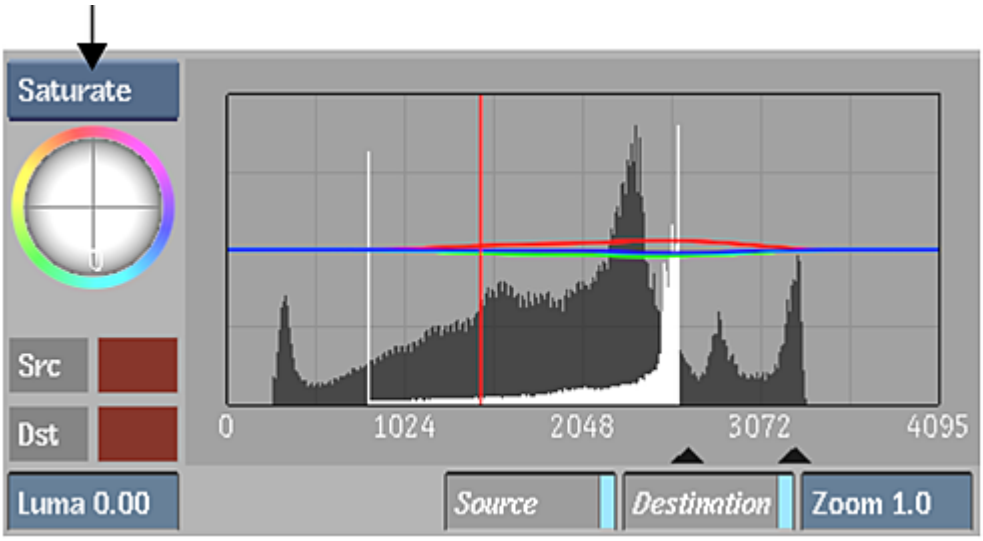
To increase saturation globally:

1. Click Basics to display the Basics menu.
2. Drag right in the Saturation field while monitoring the 2D or 3D vectorscope and the image.

The selected colour range (or entire image) moves towards the perimeter of the 2D luma histogram. There is also an increase in colour saturation in the image.

To saturate specific hues of an image:

1. Click Basics to display the Basics menu.
2. Select Saturate from the Trackball option box.



3. Drag the Saturate trackball towards the colour you want to saturate while monitoring the 2D vectorscope. Drag until the colour is closer to the edge of the 2D vectorscope. For example, to saturate reds, move the trackball towards red.

   NOTE To make the trackballs respond to changes in pressure, use the Pointer preferences in the Preferences menu.

4. Release the trackball.
5. To further increase the saturation of the same colour, use the trackball to move the colour closer to the edge of the 2D vectorscope.

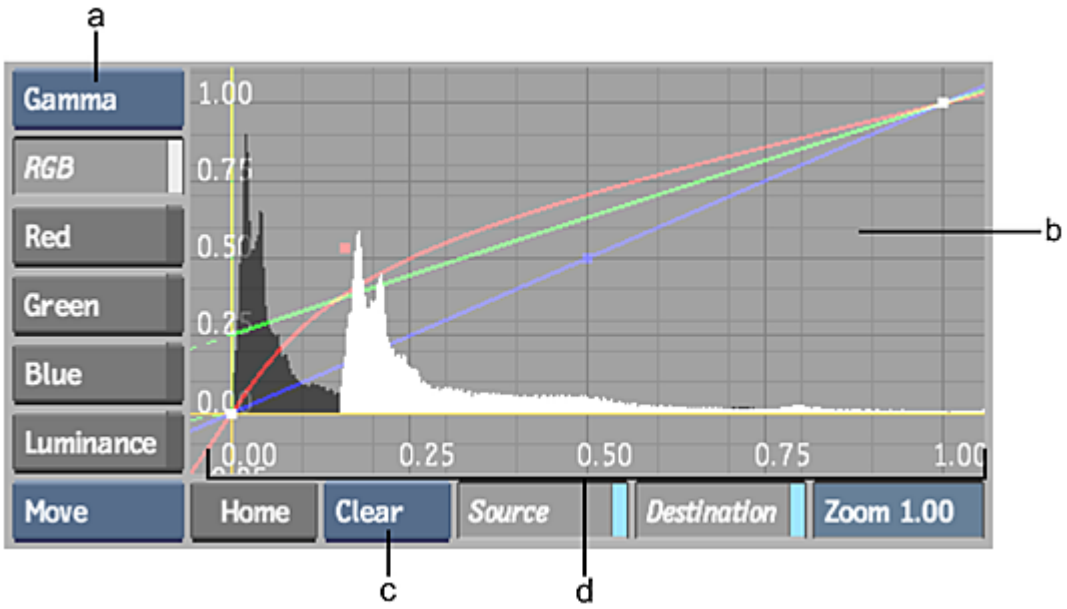
Improving Contrast Using Gamma

When you want to improve contrast by modifying both the chroma and luma content simultaneously without changing the black and white points, use the Gamma controls. You only need to make small adjustments to the RGB gamma curves to improve the contrast.

You can adjust the RGB gamma curves uniformly using the RGB Gamma fields.

You can also use a second set of adjustment curves to tweak each colour channel, as well as the luminance channel, regionally.

After making uniform or regional adjustments, if you are not satisfied with the result, you can reset each channel's gamma or adjustment curve, or clear edited curve values for individual frames.



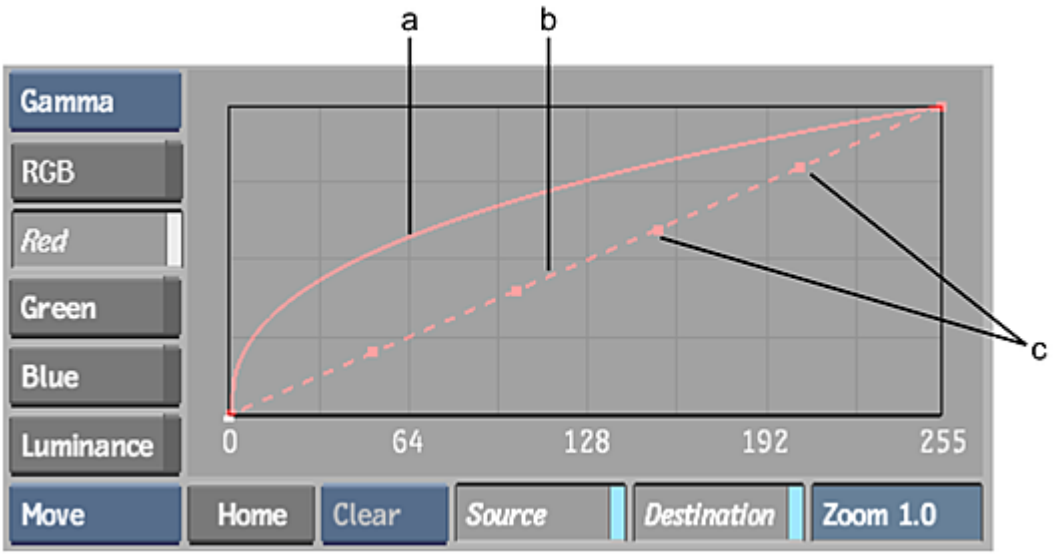
(a) Trackball option box (b) Gamma curves (c) Reset box (d) Luma range

To improve contrast using gamma:

1. Click Basics to display the Basics menu.
2. Select Gamma from the Trackball option box.
3. Display the gamma curve(s) you want to adjust.

Enable:	To display:
RGB	The R, G, and B gamma curves at the same time.
Red, Green, or Blue	The R, G, or B gamma curve individually.
Luminance	The Luminance curve. Use this curve to modify luminance levels locally without affecting the overall luminance.

Two curves are displayed for each channel, the original gamma curve, and an adjustment curve with control points for regional control. Initially these curves overlap.

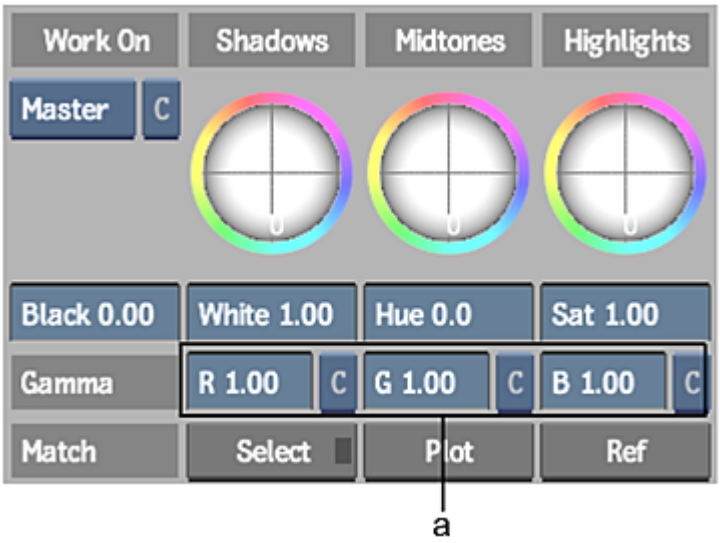


(a) Red channel's gamma curve (b) Red channel's adjustment curve (c) Added control points on adjustment curve

4 While monitoring the 2D vectorscope and image, make a minor adjustment to the gamma curves by doing one of the following:

- To modify R, G, and B gamma values individually across the entire luma range (from black to white), click and drag in the Gamma R, G, and B fields. Drag left to increase contrast or right to decrease contrast. You can also enter a value directly in these fields. Enter a low value to increase contrast or a high value to decrease contrast.

Only the field you modified is updated. The gamma curves are also updated to reflect your changes. The gamma curve is offset from its dotted adjustment curve. Use the adjustment curve to make regional adjustments to the channel.



(a) RGB Gamma Fields

- To modify R, G, and B gamma values proportionally across the entire luma range (from black to white), `Alt`-drag the Gamma R, G, or B field. Drag left to increase contrast or right to decrease contrast.

All three fields are updated. The gamma curves are also updated to reflect your changes. The gamma curves are offset from their dotted adjustment curves. Use the adjustment curve to make regional adjustments to the channel.

TIP To smooth the contrast, you can increase the gamma (raise the curves) and then adjust the Black and White levels in the Basics menu.

- To modify part of the RGB gamma or luminescence range rather than the entire range, use the Tools box to modify the adjustment curves.

Select:	To:
Add Points	Add control points to an adjustment curve. With Add Points selected, click either the red, green, blue, or luminance adjustment curve to add a control point.
Delete	Delete control points from an adjustment curve. With Delete selected, click a point on the red, green, blue or luminance adjustment curve to delete it.
Select	Select and move the control points (drag the points to move them).
Zoom	Zoom in on the curves. With Zoom selected, drag over the curves right or left to zoom in or out. You can also press `Ctrl+spacebar` and drag in the curves to zoom.
Rect Zoom	Zoom in on a section of the curves. With Rect Zoom selected, drag a selection box to zoom in on the area of the curves contained by the selection box.
Pan	Pan the curves. With Pan selected, drag over the curves to pan the curves in any direction. You can also press `spacebar` and drag in the curves to pan.

After dragging control points, the image and 2D vectorscope update accordingly.

To reset RGB gamma or luminance curves for all frames in the clip:

1. Do one of the following:
   - To reset individual RGB gamma curves, select R in one of the Channel Reset boxes next to the Gamma R, B, or C field.

   The gamma curve is reset along with its adjustment curve for all frames in the clip. Any control points added to the adjustment curve are removed.

   - To reset the R, G, or B gamma value for the current frame, `Ctrl`-click the GamR, GamB, or GamC field.
   - To reset RGB adjustment or luminance curve values for all frames in the clip, click one of the channel buttons (RGB, Red, Green, or Blue) or the Luminance button to display the curves you want to affect, then select Reset in the Reset box.

   The values for the displayed adjustment curves or luminance curve are reset. Any control points added to the adjustment curves or luminance curve are removed.

To clear RGB gamma or luminance curve values for the current frame:

1. Do one of the following:
   - To clear individual RGB gamma curve values for the current frame, select C in one of the Channel Reset boxes next to the Gamma R, B, or G field.

The gamma curve value is cleared for the current frame, and the gamma curve is updated without affecting its adjustment curve. A keyframe is set with the reset values.

- To clear RGB adjustment or luminance curve values for the current frame, click one of the channel buttons (RGB, Red, Green, or Blue) or the Luminance button to display the curves you want to affect, then select Clear in the Reset box.

The values for the displayed adjustment curves or luminance curve are reset for the current frame. The curves' control points are distributed equidistantly along each curve. A keyframe is set with the reset values.

Matching Colours in Clips

You can match a sample from the result clip to one from a reference clip. Matching clips can improve continuity between clips. You can:

- Match specific colours.
- Match shadows, highlights and overall saturation.

Matching Specific Colours

You can match plot samples to reference samples using the trackballs and by aligning the colours in the vectorscope. For example, you may want to match a red in one image to a different shade of red in another clip. For information on using Plot and Ref controls, see Sampling Clips in the Image Window (page 1171).

To match specific colours:

1. Select Result from the View box.

   The result clip is displayed in the image window.
2. If necessary, position the split bar so that both a reference clip and the result clip appear.
3. Enable Plot and then sample a colour in the clip you want to modify (the result clip).

   A red plot line is displayed in the 2D luma histogram indicating the luma value of the sample. In the 2D vectorscope, you can view the saturation and hue of the sample (outlined in red). In the 3D vectorscope, you can view the sample in terms of HLS.
4. Enable Ref and then sample the colour in the reference image you want to match.

   A yellow reference line is displayed in the 2D luma histogram indicating the luma value of the sample. In the 2D vectorscope, you can view the saturation and hue of the sample (outlined in yellow). In the 3D vectorscope, you can view the sample in terms of HLS.
5. Use the 2D luma histogram in the Basics menu and 2D vectorscope in the image window to determine which controls you need to use.

Drag:	When:
White or Black field	You need to adjust the luma values in the highlights or shadows without modifying the chroma. Drag the field so that the plot line in the 2D luma histogram moves towards the reference line.
Highlights, Midtones, or Shadows trackball	You need to adjust the chroma values in the image without modifying the luma. Drag the trackball(s) so that the plot sample in the 2D vectorscope moves towards the reference sample.

6 Use the necessary controls to modify the clip until the plotted sample is aligned with the reference colour in the 2D luma histogram and 2D vectorscope. If necessary, increase or decrease the saturation as well as shift the hue.

TIP View the sample in the 3D vectorscope to make more refined modifications.

Matching Shadows, Highlights, or Overall Saturation

Use the Match feature to match a range of colours in the result clip to a range of colours in a reference clip. You can match highlights, shadows, or the overall saturation of an image.

NOTE Match is not designed to match specific colours.

To match shadows, highlights, or overall saturation:

1 Select Result from the View box.

The result clip is displayed in the image window.

2 Position the split bar so that both a reference clip and the result clip appear.

3 Click Match.

The following message appears in the message bar: "MATCHING: Select an area to be modified."

4 Click and drag on the result clip to draw a box around a range of colours. You can select an area of the image that is predominantly black, white, or contains a wide spectrum of colours. To match the overall saturation of an image, select a larger area of the image.

The blue indicator on the Match button is enabled, indicating that you need to select a destination sample.

5 If necessary, position the split bar to show more of the reference image.

6 Click Match.

The following message appears in the message bar: "MATCHING: Select an area to match to."

7 Click and drag to draw a box around a range of colours in the reference clip. You should match whites with whites, blacks with blacks, or select a broad range of colours on both images to match saturation. Trying to match completely different colours will produce unpredictable results.

The range in the result clip is matched to the range in the reference clip.

8 Repeat these steps as often as necessary to obtain the best possible match between colours. You can use Match repeatedly to make your colour match more precise.

9 To refine the result, use other controls in the Basics menu (such as the Saturation field), and then reuse Match.

To clear a Match operation:

1 Click the Undo box.

To cancel a Match operation in progress:

1 If you begin using Match and want to cancel the matching process, adjust any value using the trackballs or fields.

The blue indicator on the Match button will be disabled, indicating that the matching process has been cancelled.

Performing Hue Shifts

You can use the Hue field in the Basics menu to perform hue shifts on the entire image or a range in the image defined by a selective's matte. Perform hue shifts when you want to change the colour of an object or create other artistic effects. Hue shifts do not affect the luma.

You can also perform hue shifts when you want to suppress colour spill. See Removing Colour Spill (page 1193).

To perform a hue shift:

1. If necessary, create a matte for the range of colour you want to change. See Selecting Colour Ranges for Colour Correction (page 1173).
2. Click Basics to display the Basics menu.

   If you are using a selective, the Selective View box should display Result when you display the Basics menu.
3. Sample the image to see the hue shift in the 2D luma vectorscope. Enable Plot and then sample the colour range in the image you want to hue shift.

   A red line also appears in the 2D luma histogram and the sample is outlined in the 2D or 3D vectorscope.
4. Drag the Hue field while monitoring the 2D or 3D vectorscope and the image.

   The selected colour content (or entire image) shifts through the colour range.
5. If necessary, you can change the saturation of the hue by dragging the Saturation field—drag right to increase saturation or left to decrease it.

   The selected colour range moves towards the perimeter or centre of the 2D luma histogram.

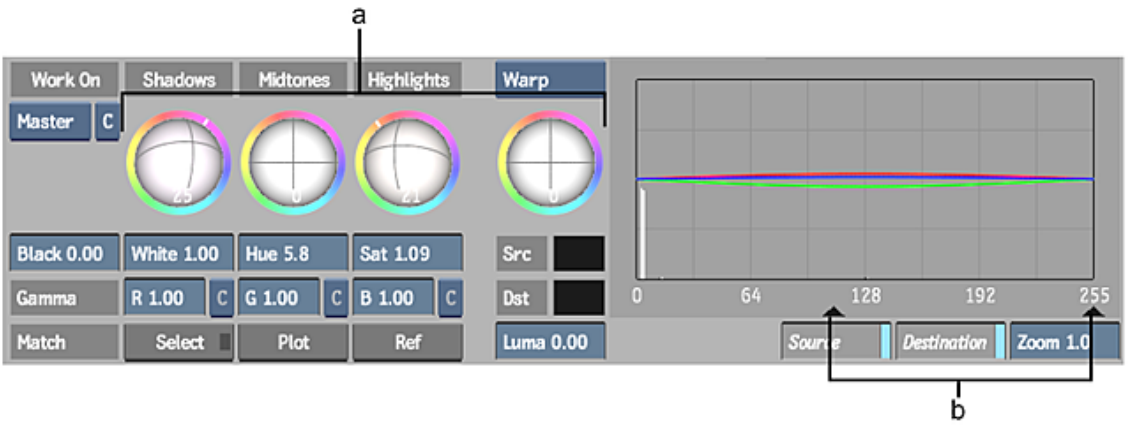
Creating Colour Casts

You can use the Midtones trackball to create a colour cast that affects the chroma in the entire image without affecting the luma. Create a colour cast when you want to emphasize a colour range to create an artistic effect or when you want to refine an existing cast. For example, create or refine a bluish cast to convey a cooler atmosphere.

NOTE Before you create a colour cast, you should correct colour imbalances in the image or clip. See Correcting Colour Imbalances (page 1179).

To create a colour cast:

1. Click Basics.



(a) Trackballs (b) Midtones triangles

TIP You can determine the colour of an existing cast by dragging right in the Saturation field to temporarily increase the saturation while monitoring the sample in the 2D vectorscope. Click Undo once you determine the colour of the cast.

2 To add a colour cast to the entire image, drag the midtones triangles so that they span the entire luma range.

3 While viewing a vectorscope and the image, drag the Midtones trackball towards the colour you want to increase in the clip or image. For example, to add more yellow to the image, drag the Midtones trackball towards yellow.

In the 2D or 3D vectorscope, the colours update to reflect your changes. In the image window, the colour in your clip changes according to the direction the trackball is being dragged.

NOTE When you drag the Midtones trackball, the RGB curves in the 2D luma histogram also reflect the changes you make.

4 To increase a range of colour in the highlights or shadows of the image, drag the Highlights or Shadows trackball.

5 To increase or decrease the saturation of the cast, drag the Saturation field—drag right to increase saturation and left to decrease it.

TIP You can make slight modifications to the hue of the cast using the Hue field.

Readjusting Colour After a Colour Correction

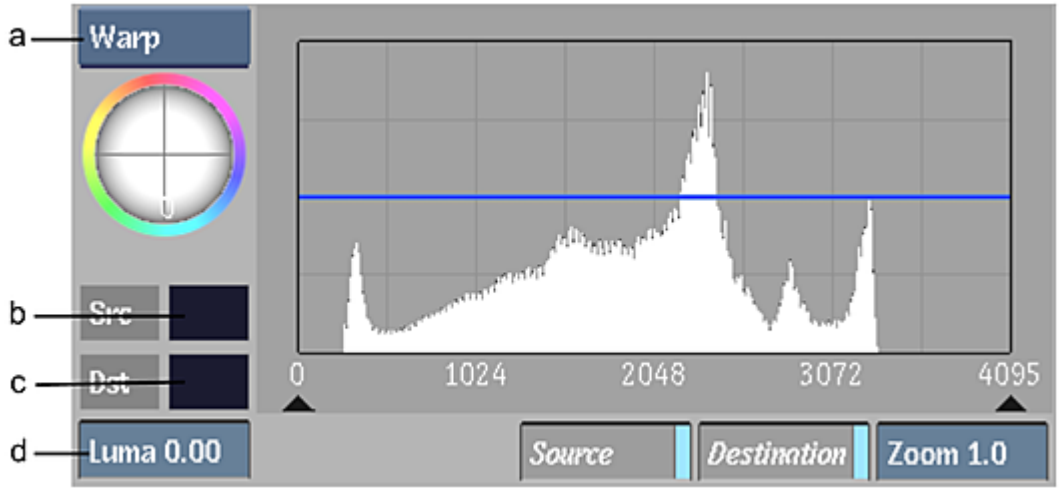
When colour correcting clips, colours that you do not want to alter may be modified in the colour correction process. Use the Warp trackball in the Basics menu to readjust a specific range of colour.

You can also adjust the luminance of the selected colour using the Luma field.

To readjust colours after a colour correction:

1 Click Basics.

The Basics menu appears.



(a) Trackball option box (b) Source Colour pot (c) Destination Colour pot (d) Luma field

2 Select Warp from the Trackball option box.

3 Click Pick and then sample a colour in the clip you want to modify.

The colour you select is outlined in white in the 2D or 3D vectorscope. A cyan bar representing the colour appears in the 2D luma histogram. The Src Colour pot displays the colour.

4 Click the Src Colour pot to set the source colour.

The Dst (destination) Colour pot is updated to match the source colour.

5 Move the Warp trackball towards the destination colour.

The Dst Colour pot dynamically updates to reflect your changes. When you release the mouse, the Src Colour pot is updated to match the destination colour. In the 2D or 3D vectorscope, the selected colour moves to reflect the changes you are making to the image.

NOTE To make the trackballs respond to changes in pressure, use the Pointer preferences in the Preferences menu.

6 To adjust the luminance of the destination colour, adjust the Luma field. When you adjust the luma of the destination colour, the entire luma content of the image is adjusted around the selection. The selected luma value changes the most.

The Dst Colour pot dynamically updates to reflect your changes. When you release the mouse, the Src Colour pot is updated to match the destination colour. In the 2D luma histogram, the cyan bar moves to reflect the changes you are making to the luma content.

7 Release the trackball or click the Luma field.

The Src Colour pot is set to the Dst box colour.

Removing Colour Spill

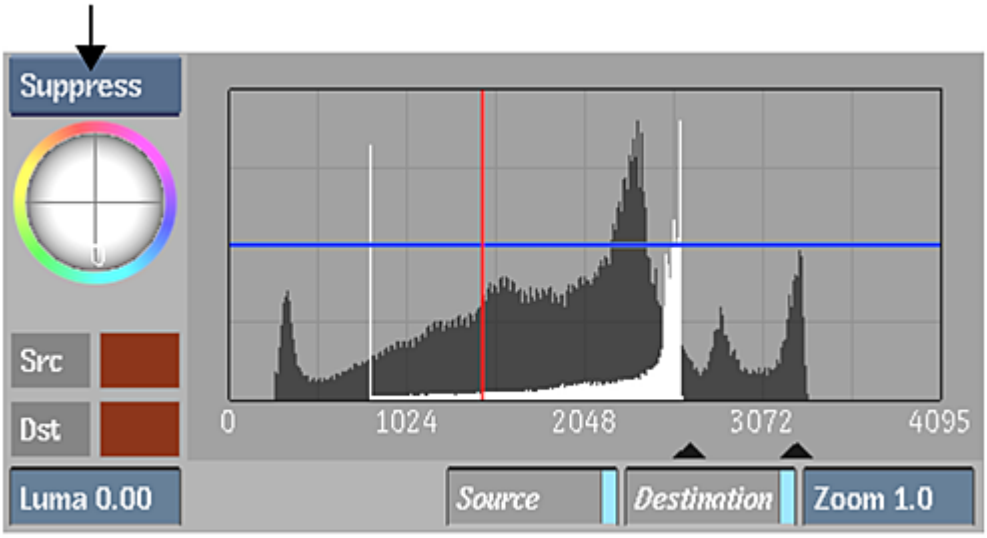
When you key clips in the Modular Keyer, there can be colour spill. You can refine the key by suppressing colour spill in the Colour Warper. Use the Saturate and Suppress trackballs in the Basics menu. Use the Suppress trackball to suppress the colour spill and the Saturate trackball to correct any unwanted colour suppression. When you use these trackballs to modify colour in the image or clip, a range of colour is affected.

In the Modular Keyer, place the Colour Warper node in the CBlend pipe.

To remove a colour spill:

1 Click Basics to display the Basics menu.

2 Select Suppress from the Trackball option box.



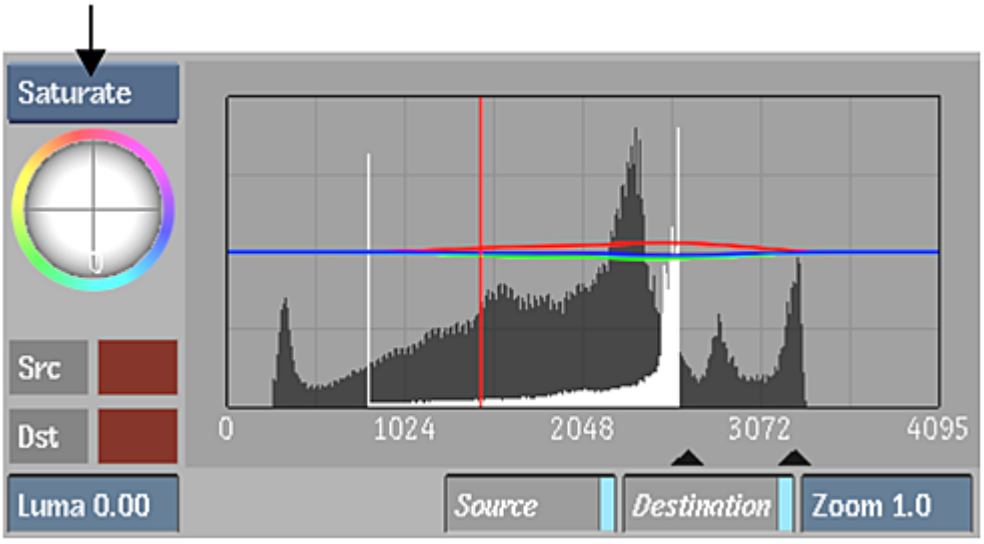
NOTE To make the trackballs respond to changes in pressure, use the Pointer preferences in the Preferences menu.

3 Move the Suppress trackball towards the colour that you want to suppress. For example, to remove a blue background, move the trackball so that the blue colours appear in the centre of the 2D vectorscope.

NOTE The colour you are suppressing appears in the middle of the 2D vectorscope at full suppression.

By comparing the Result clip and the Front clip, you may notice some unwanted colour suppression to the colour that is opposite (on the colour wheel) the one you just suppressed. You can fix such suppression with the Saturate trackball.

4 Select Saturate from the Trackball option box.



5 Move the Saturate trackball towards the colour that you want to saturate.

The colour that you are saturating moves closer to the edge of the 2D vectorscope.

6 It may be necessary to adjust colours in the image using the Suppress and Saturate trackballs several times to achieve suitable results. Continue adjusting the trackballs until you are satisfied with the result.

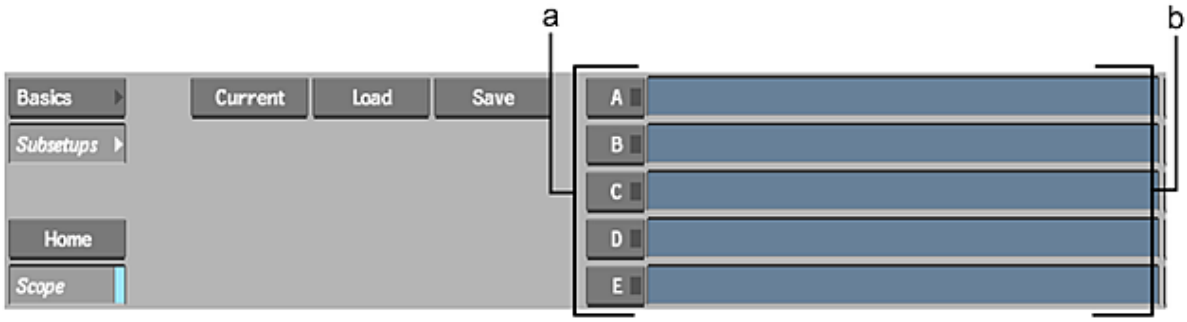
Creating Subsetups

Subsetups are Colour Warper setups selected at a specific moment at a specific frame. Subsetups are for the current frame only and do not include entire animations. You can select up to ten subsetups and compare colour corrections in other subsetups. You can save and load a group of subsetups.

NOTE The group of subsetups loaded affects all Colour Warper nodes in the current setup since subsetups are shared between Colour Warper nodes. Ten subsetups (A-J) constitute a group.

Use the Subsetups menu to:

- Selectively store intermediate setups.
- Compare any subsetup to the current setup.
- Share subsetups between Colour Warper nodes.
- Save and load a group of subsetups.



(a) Subsetup buttons (b) Subsetup fields

Storing and Retrieving Subsetups

You can store and retrieve subsetups using the Subsetup buttons.

To store and retrieve the current setup:

1 Press and hold one of the subsetups buttons until a message appears indicating "Storing subsetup <subsetup letter>" and release.

The LED indicator shows that the subsetup is stored. All active subsetups have a LED indicator. If you have not stored anything into a subsetup, the active light of the button will be off.

2 Use the setup at another frame or in another node by pressing the corresponding Subsetup button.

Naming Subsetups

You can name your subsetups. This allows you to store setups and later compare them to your current setup, or to apply the same correction to different frames in a clip.

To name a subsetup:

1. Click in a subsetup field, type in a name or description, and press `Enter` to store the setup.

Replacing Subsetups

You can replace or override a setup.

To replace an existing setup:

1. Click and hold a subsetup button to overwrite and replace an existing subsetup. A keyframe is set and the new subsetup is saved.

Comparing Subsetups

The current setup is accessed with the Current button and contains the most recently committed setup. Compare any of your subsetups to each other or to the contents of the current setup.

To commit the contents of the current setup:

1. Perform an operation such as advancing a frame or changing a value in the Basics menu. Your present settings will be displayed the next time you click Current.

To compare a subsetup:

1. Click a Subsetup button of the subsetup you want to compare.
2. Click Current to view the current result.

Saving and Loading Subsetups

Use the Save button in the Subsetups menu to save a group of subsetups. Then use the Load button in a later session to reload the same group of subsetups.

To save a group of subsetups:

1. In the Subsetups menu, click Save.
   The file browser appears.
2. Browse to a different directory path if necessary and then type a name in the Name field.
3. Click Save.
   The group of subsetups is saved in the specified directory.

To load a group of subsetups:

1. In the Subsetups menu, click Load.
   The file browser appears.
2. Browse to the appropriate path and select the name of the group of subsetups.
   The group of subsetups is loaded.

Painting 22

Paint Node

The Paint node is a system that provides a scalable matte painting, retouching, or restoration workflow in Batch or Batch FX.

Due to its underlying technology, the Paint node automatically scales strokes when changing the resolution, ratio, or bit depth of input clips, or when switching from Full Resolution to Proxy mode. The Paint node supports “clipless” setups, which can be applied to any image input, while accurately reproducing the sequence of painted strokes.

The Paint node accepts a front and matte clip as input, and creates a result and output matte clip, respectively. You can paint on the result and output matte, with a selection of brushes in different paint modes.

The Paint node also allows you to connect multiple sources and use them to paint the contents of source images onto the result. This paint operation, applied with the Reveal paint tool, can be used with in-context overlay over the result image. A front and matte clip can be connected as a source by connecting the clips to a source node. The content of source front and source matte input can be used to create brush strokes on the result and output matte.

Accessing the Paint Node

The Paint node can be accessed from

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

For a list of all Paint node menu settings, click here (page 1622).

Using Sources

Sources are additional image data that can be used in Reveal paint operations and displayed as an overlay on the result or output matte. You can scale, rotate, and offset a source. Any transformations applied to a source will be displayed in Reveal paint strokes.

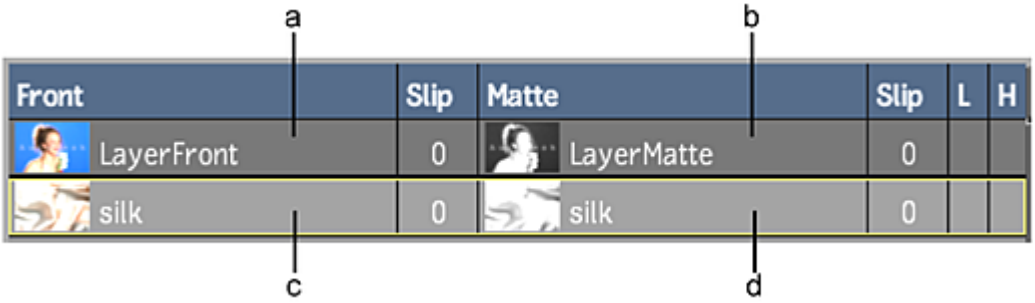
In the Paint Node menu, all sources appear in the Sources list. The Sources list allows you to select sources to be hidden, cleared, or displayed as an overlay. The first entry in the list is the front clip and matte clip.

In the schematic, source clips are connected to a Paint Node through a source node, which accepts a source front and source matte input.

A source can be a clip or the result of any process, and can include a front and/or matte clip. You can add a source node from the menu or the schematic.

If a clip with frames that are missing media is attached to a source node, the Paint Node processes incoming frames set to No Media as transparent. Strokes created with the source are stored, although they are not visible on the canvas at frames with No Media input.

When adding a source input, it is recommended that you use clips as input, or cache the node closest to the source node instead of parenting a large tree to a source node. This speeds up processing when changing frames, making for a more interactive experience.



(a) Front (b) Matte (c) Source front (d) Source matte

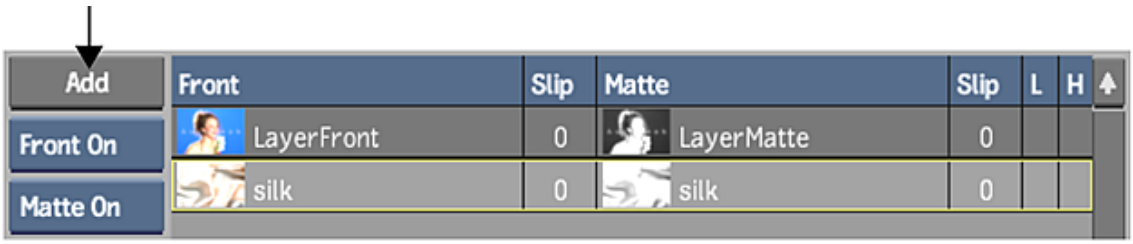
When you delete a source, it is removed from the Sources list, along with its associated strokes. When you clear a source, it remains in the Sources list, but all strokes are removed.

Adding a Source

When you add source, it includes both source front and source matte input. When you add a source front only, a black matte will automatically be created.

To add source node, front source and matte source:

1. In the Paint Node menu, click Add.
   The Viewing Panel appears.
2. Select a source front clip, and then a source matte clip.
3. Click Exit Clip Select to return to the Paint Node menu.
   In the Sources list, the source is added. In the schematic, the process tree is updated: a source node with the selected clips is connected to the Paint node.

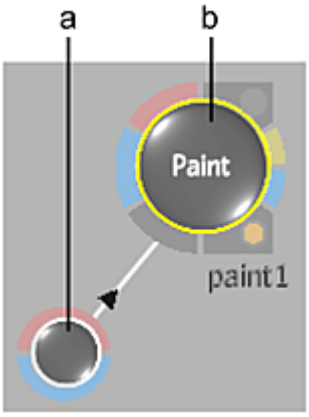


4 To help identify source nodes in the schematic, select a source in the Sources list to automatically highlight in orange the corresponding source node and link to the Paint Node. Conversely, selecting a source node also highlights the corresponding source in the Sources list.

To add a source node only:

1 Hold `Ctrl` and click Add to add a source node.

The source node is added and connected to the Paint Node.



(a) Source node (b) Paint Node

2 Connect clips or process tree input to the source node's front and matte inputs.

Removing a Source

You can remove a source clip from the Sources list. If you disconnected a source clip from the source node, the entry exists in the Sources list, but the clip name and proxy no longer appear. Because the stroke information is stored in the node, you can reconnect the same clip or a different clip to the source node, and the stroke information is preserved. All strokes in the canvas using the source are repainted using the new source clip.

When you delete a source node, the entry is removed from the Sources list, and the strokes stored in the node are deleted permanently.

To remove a source clip from the Sources list:

1 In the schematic, disconnect the source clip from the source node.

To delete a source node:

1 From the Tools box, select Delete (or use the keyboard shortcut), and click the source node in the schematic.

Clearing Strokes on a Source

You can clear strokes from the canvas that were created using a source front or source matte.

WARNING Clearing strokes for an entire sequence cannot be undone.

To clear the strokes associated with a source:

1. In the Sources list, select the source used to create the strokes you want to clear.
2. From the Selection Mode box, select Current Frame or Sequence.

3. Click Clear Strokes.

Hiding Strokes

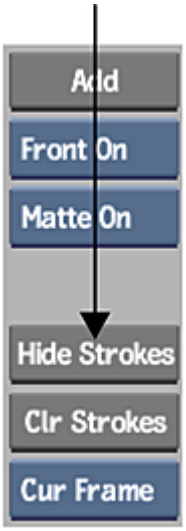
You can hide a group of strokes that were created from the same source clip.

Hiding strokes is useful when you need to focus on a particular component of your image: hide strokes on the canvas that do not require attention. This feature allows you to concentrate on the current stroke. It also improves the system's interactive performance.

To hide strokes on a source:

1. Select the source used to create the strokes that you want to hide.

2 In the Sources list, click Hide Strokes (H).



A check mark appears in the H column of each source.

3 Click Hides Strokes again to make the strokes visible.

Restricting Strokes with the Source Matte

You can limit the sections of the source that are painted onto the canvas using the source's matte. To limit painting on the canvas using the matte. See Restricting Brush Strokes (page 1209).

To limit source brush strokes using a source matte:

1 In the Paint Modes box, select Reveal (E).

2 In the Sources list, select a source.

3 From the source controls, select an option in the Matte Source box.

Select:	To:
Matte Off	Paint anywhere on the canvas.
Matte On	Limit painting to areas in the source matte.
Matte Invert	Invert the source matte and limit painting to areas outside the source matte.

4 Draw strokes.

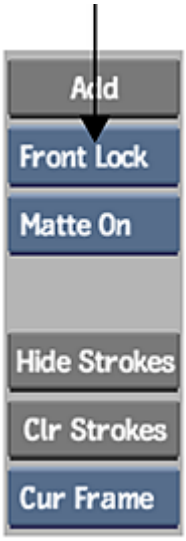
Brush strokes only use areas of the selected source that are delimited by its source matte.

Locking Sources

Lock a source to a specific frame. For the duration of the clip, only the source front and source matte at the selected frame is used.

To lock a source:

1. Select the source you want to lock.
2. Using the playback controls, choose the frame at which you want to lock the clip.
3. In the Source Front option box, select Front Lock.



   A check mark appears in the L column of each selected source.
4. In the Source Front option box, select Front On to unlock the source.

Displaying Sources

Paint Node offers an intuitive way of previewing the effects of a Reveal operation. In this view, the selected source is superimposed over the result, and the Transparency field is activated to allow for transparency adjustments. This feature allows you to see exactly what your brush strokes will reveal. See Previewing a Reveal Operation Using a Reference Image (page 1222).

Brush Attributes and Attribute Modes

You can set various brush attributes and attribute modes to determine how paint is applied to the image.

Setting Brush Attributes

The Brush Attribute fields set the size of the brush, the distribution of the paint, and the rate and direction of the paint application.

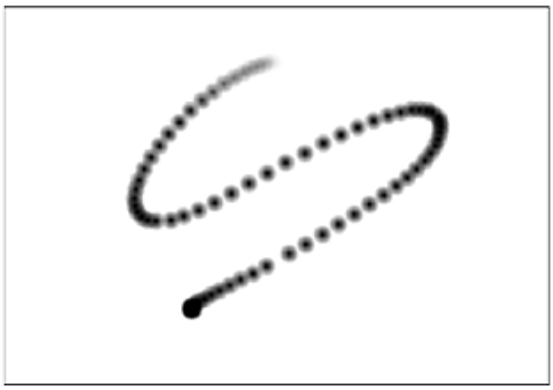
(a) Brush Attribute fields

Opacity Affects the transparency of the brush. A value of 100% applies a fully opaque colour. Use a lower value to apply a more transparent colour.

Size Determines the size of the brush, which is indicated by the diameter of the green dashed circle surrounding the cursor brush. To increase the brush size, press S and drag the brush to the right on the canvas. To decrease it, press S and drag to the left.

Rate Determines the rate at which brush strokes are applied to the canvas. Use a high value to produce a smooth continuous stroke, or a low value to produce a less continuous stroke with larger gaps between brush images. The C (Constant) button in the Rate field determines whether the consistency of the stroke is dependent on the speed at which you move the pen or mouse. For example, to create a stroke of evenly spaced dots, you could set the Rate field to 20 percent and enable Constant.

Rate attribute value = 100



Rate attribute value = 25

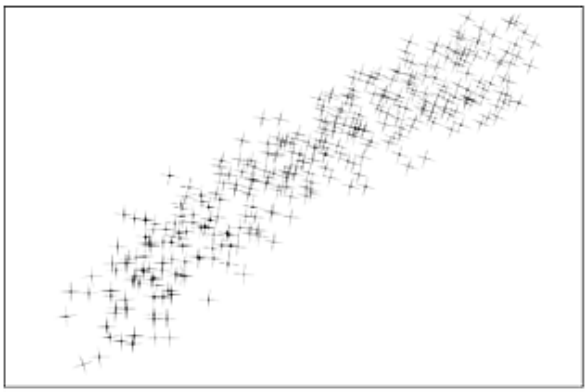
Pressure Affects the transparency of the paint applied to the image. To apply opaque paint, use a high percentage value. For more transparent paint, use a low value. The Pressure attribute differs from the Opacity

attribute in that you can set the Pressure attribute mode so that the paint transparency varies according to the pressure applied to the pen or the direction of the brush.

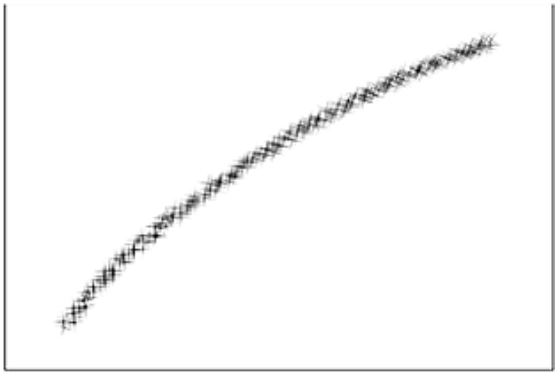
Pressure attribute value at 100%

Pressure attribute value at 50%

Jitter Randomizes the brush strokes applied to the image. A high value produces a greater dispersion of paint, while a low value produces a greater concentration.



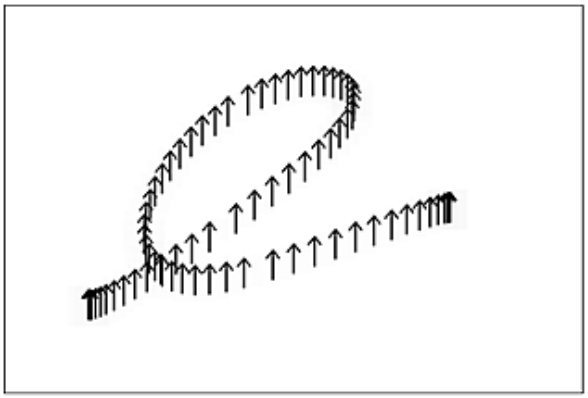
Jitter attribute value at 100%



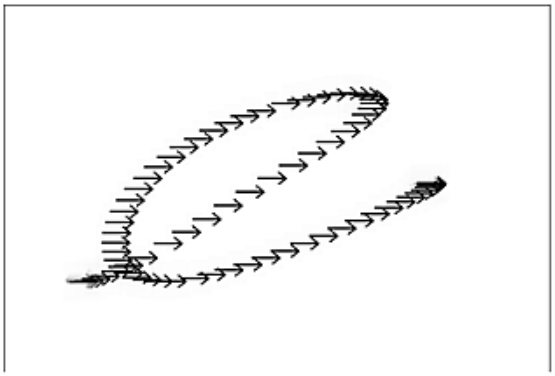
Jitter attribute value at 10%

Direction Causes the brush to rotate around the Z-axis and can be used to produce a calligraphy effect. The effect of the Direction attribute is most noticeable when used with one of the elliptical or star brushes. The

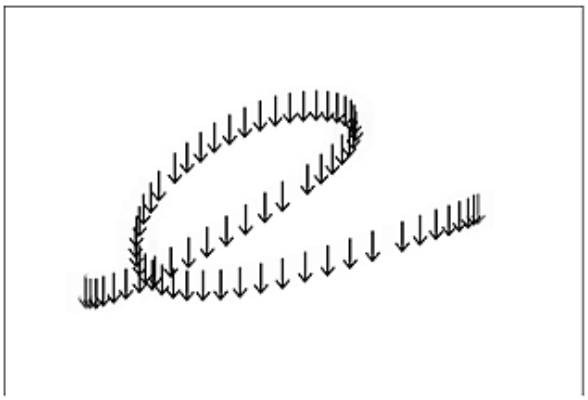
value of the Direction attribute causes the brush strokes to rotate by 90 degrees for each increment of 25 percent.



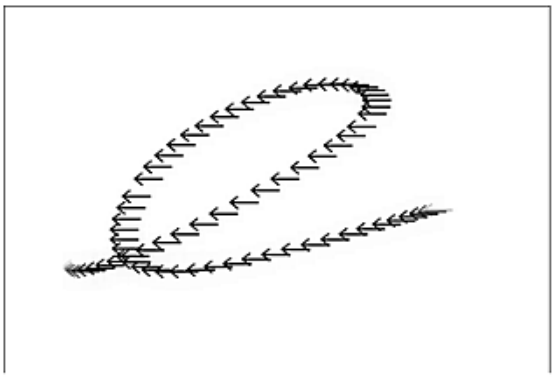
Direction attribute value at 100%



Direction attribute value at 75%



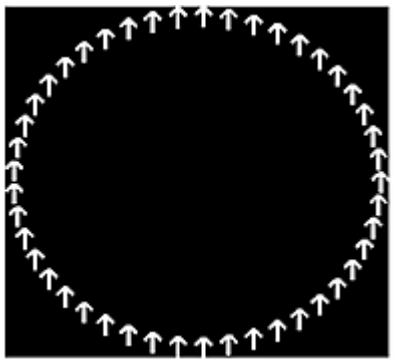
Direction attribute value at 50%



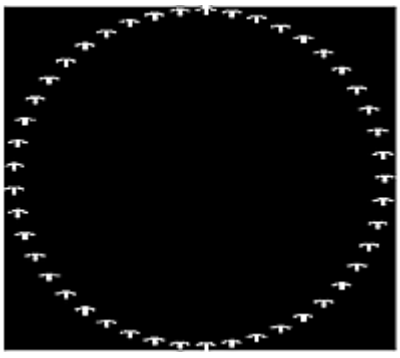
Direction attribute value at 25%

Roll Rolls the brush around the X-axis. The effect of the roll is most noticeable when used with one of the non-symmetrical brushes. For each increment of 25 percent, the Roll attribute value creates a rolled brush stroke of 90 degrees.

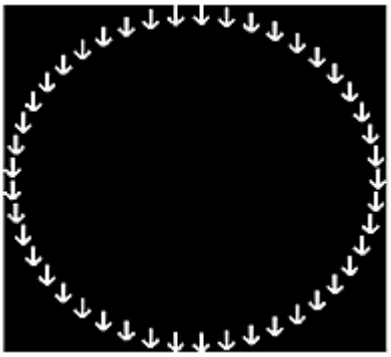
NOTE You can use any attribute mode with the Roll attribute.



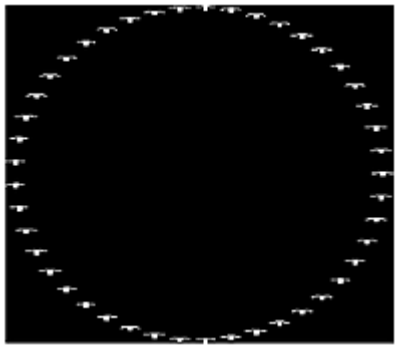
Roll attribute value at 100%



Roll attribute value at 75%



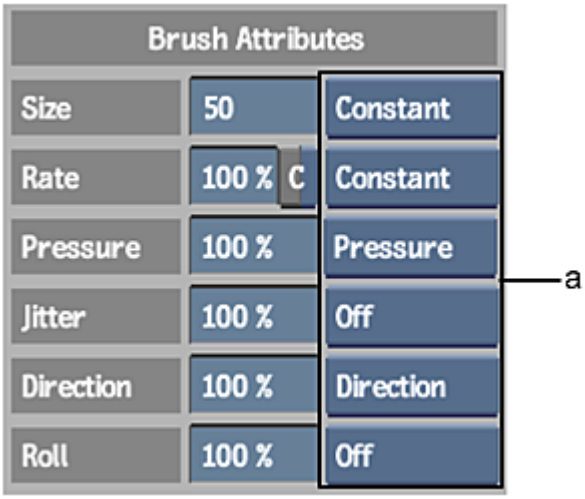
Roll attribute value at 50%



Roll attribute value at 25%

Setting Brush Attribute Modes

The value of a brush attribute depends on the selected attribute.



(a) Brush Attribute Mode boxes

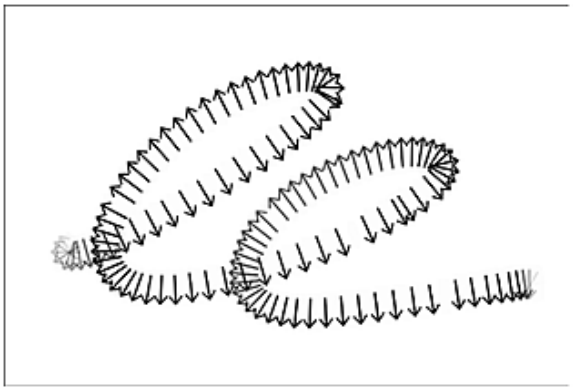
Off Disables the brush attribute.

Constant Keeps the brush attribute values constant.

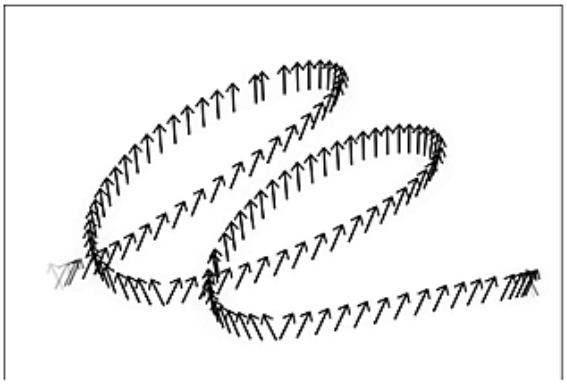
Pressure Uses the pressure exerted on the pen as the reference value. The harder you press on the pen, the greater the brush attribute value. The softer you press, the lower the brush attribute value.

Direction Uses the direction of the brush stroke as the reference. When used with the Direction mode, the Direction attribute causes the brush strokes to follow the trajectory of the brush. Increase the value to enhance

the effect on brush direction. Drag right to use 100% of the brush attribute value, left to use 0%, up to use 25%, or down to use 75%.



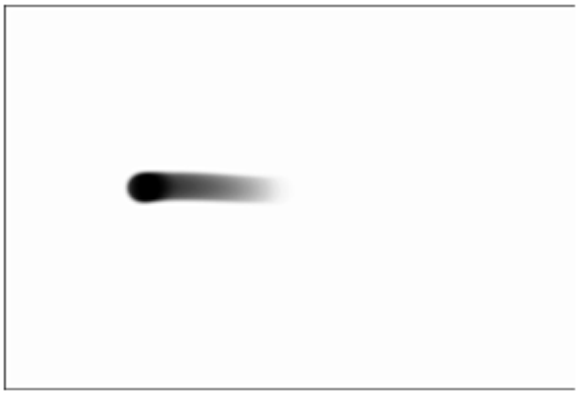
Direction attribute value at 100%, Direction attribute mode



Direction attribute value at 15%, Direction attribute mode

Fade Uses the percentage value entered in the Fade field to determine the length of time that elapses before the brush stroke fades. The greater the percentage, the faster the stroke fades. A value of 100 in the Fade field causes the stroke to fade quickly, whereas a value of 1 causes the stroke to last longer. Also, strokes applied quickly will be longer than those applied more slowly.

Pressure attribute value at 100%, Fade attribute mode at 35%



Pressure attribute value at 100%, Fade attribute mode at 85%

Front, **Matte**, **Source Front and Source Matte** Allows you to use the red channel in a reference image to set brush attributes. Source Front and Source Matte mode use the red channel of the selected source in the Sources list.

Painting Brush Strokes

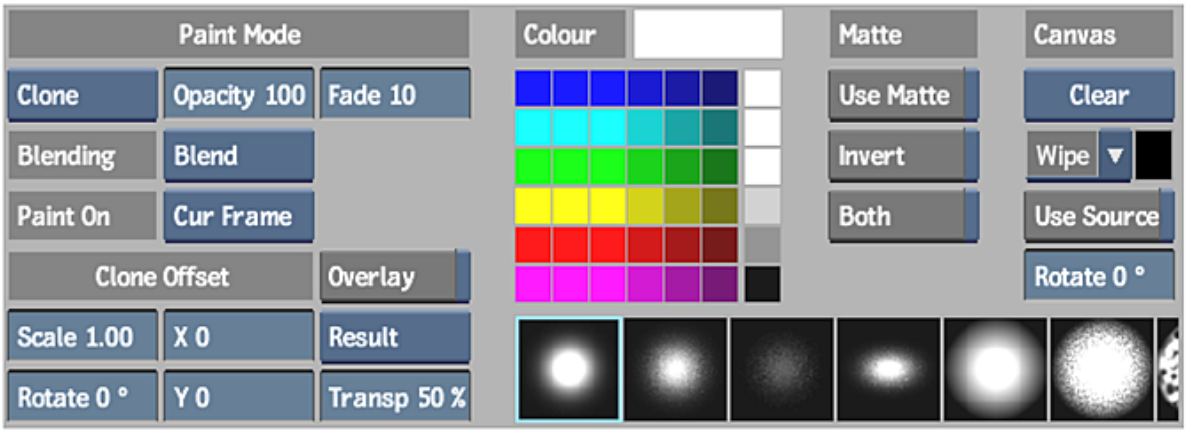
You can paint on the canvas using a variety of brush types and colours. Once you select a brush type, you can modify brush attributes to further refine your strokes. A stroke can be applied to a single frame, or simultaneously to multiple frames. You can paint on the result or output matte, or you can use matte paint mode controls to paint on the result and output matte at the same time.

You can restrict brush strokes to the area delimited by the matte. For example, assume that you have a front input that includes a person shot against a blue screen and a corresponding matte clip. If you opt to limit brush strokes to the matte, your brush strokes on the result or output matte appear only inside the matte

area and, therefore, over the person. If you then limit brush strokes to the existing strokes, new paint strokes are applied only over strokes that have already been applied.

Applying Brush Strokes

You can paint with either a pen or a mouse, and access various pressure settings.



To apply a brush stroke:

1. Select an option from the Paint Tools box.
2. Select an option from the Paint On box to apply the stroke to the current frame, all frames in the sequence, or the current frame and subsequent frames.
3. Select the Result (`F4`) or Output Matte (`F4 F4`) to set the view where you are painting.
4. If you want to paint on the result and output matte simultaneously, enable Both.
5. Click a brush to select it.
6. Set brush attributes. See Brush Attributes and Attribute Modes (page 1202).
7. If you will reveal a source on the image, select the source in the Sources list. See Revealing Sources (page 1214).
8. To change the colour, click a colour pot. If you are painting on the output matte, the brush colour is grey with an equivalent luminance value.

   TIP As an alternative to using the colour pots, you can designate the colour under the pointer as the current brush colour by pressing the left `Ctrl` key and clicking on the canvas.

9. Click and drag on the canvas to draw a stroke.

   NOTE Downstream context views are not automatically updated as you add strokes, since this would cause performance degradation. To force an update, click the Update button.

10. Use the eraser end of the pen if you are using the stylus to erase strokes that you have created.

Removing Brush Strokes

You can undo multiple brush strokes in Paint Node. You can use the Undo button to remove brush stroke operations, beginning with the most recent strokes. Set Undo level operations in the Preferences menu.

If you are using a stylus, remove brush strokes manually by using the back of the stylus to “erase” strokes. If the result is displayed, erasing reveals the front clip on the canvas. If the output matte is displayed, erasing reveals the matte.

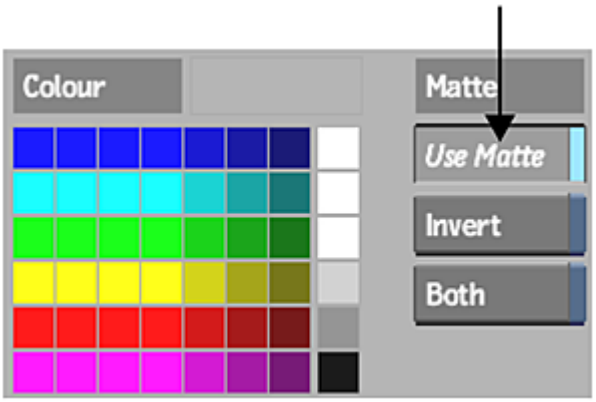
Restricting Brush Strokes

You can limit the area where brush strokes are applied on the canvas. Use the matte paint mode controls to limit painting on the canvas to the matte. You can also invert the matte to limit paint to only the areas outside the original matte.

To limit which regions of a source front can be used by its source matte, use the Matte Clip option box in the source controls. See Restricting Strokes with the Source Matte (page 1201).

To limit where brush strokes are applied using the matte:

1. Enable Use Matte.



2. If you want to paint outside of your selection, enable Invert.
3. Draw strokes.

   Brush strokes are applied only to the areas included in your selection.
4. Disable Use Matte to turn off matte restrictions.

Scaling Brush Strokes

You can set scaling options for brush strokes associated with the Paint Node. By specifying the previous resolution of an input clip before it was resized, the brush strokes applied to the clip can also be scaled based on these settings.

To display the stroke scaling options:

1. Double-click the Paint node.
2. Click Node Setup.

(a) Aspect Ratio Presets box (b) Project Resolution Presets box

Project Resolution Presets box Provides preset aspect ratio options and an option to use a custom resolution. Set this option to indicate the previous resolution of the clip.

Width and Height fields Displays the frame width and height of the selected resolution preset. If you select Custom from the Project Resolution Presets box, use these fields to enter the values you want to use.

Aspect Ratio Presets box Provides standard frame aspect ratio options and a w:h option to use a ratio based on the values entered in the Width and Height fields. Also provides a Custom option so you can enter a frame aspect ratio in the Ratio field.

Ratio field Displays the original aspect ratio of the clip. When Ratio is set to Custom, this field becomes active so that you can enter a custom frame aspect ratio.

Stroke Resize box Select a fit method option to be applied to the clip.

Select:	**To:**
Centre/Crop	Center the strokes over the destination frame. If the clip at the original resolution is larger than the destination, the strokes are cropped.
Crop Edges	Fit one edge of the original clip input into the destination frame without stretching or squashing the frame. Excess parts of the original clip after resizing are cropped.
Fill	Fit the original stroke width and height into the destination frame. If the clip at its original resolution and destination frames do not have the same aspect ratio, the brush strokes can become distorted.
Letterbox	Fit the original stroke to the destination frame without squashing or stretching it, and without cropping the source.

Keep Aspect button Enable to preserve the aspect ratio of non-square pixels. This button only appears if you selected Crop Edges or Letterbox in the Fit Method box.

Stroke Pan fields Enter an X and Y value to offset existing strokes on the result. You can also reposition strokes by holding down `Ctrl+Shift` and panning the image.

Current Resolution fields Displays details of the current resolution of the clip.

Active button Enable to activate smooth filtering of pixels for enhanced display

Using Paint Modes

You can select the type of special effects you want to apply to the brush from the Paint Modes box. Strokes applied with these brushes are processed by the graphics hardware, improving interactive performance. To paint sources onto images, see Revealing Sources (page 1214).

To select a brush:

1. Switch to either Result (F4) or Output Matte (F4 F4) view.
2. From the Paint Modes box, select a brush. Refer to the sections that follow for instructions on using each brush.

   TIP When a brush is selected, brush opacity and blend modes are disabled. To achieve an effect similar to modified brush opacity, use the Pressure attribute mode.

Blur

Apply a blur filter to the image with each stroke.

Original image

After using the Blur brush

To use the Blur brush:

1. From the Paint Modes box, select Blur (R).
2. Set the brush size according to the size of the area you want to blur.
3. Paint on the image.

Impression

Paint on colours from a reference clip. When you click on the canvas, the colour at the center of the brush is used to fill the entire brush stroke.

Original image

After using the Impression brush

Image courtesy of Buzz Image Group, Inc., and Les Films Stupefiants

Image courtesy of Buzz Image Group, Inc., and Les Films Stupefiants

To use the Impression brush:

1. From the Paint Modes box, select Impression.
2. Set the brush size.
3. Paint on the image.

Smear

The Smear brush smudges areas of the image, creating a fingerpainting effect.

Original image

After using the Smear brush

To use the Smear brush:

1. From the Paint Modes box, select Smear (T).
2. Set the brush size according to the size of the area you want to smear.
3. Paint on the image.

Clone

Use the Clone tool to paint a copy of the result image to any position on the result image. The Clone tool can also be applied to an output matte image. For example, if the result image has a tree that you would like to copy to another position on the image, you can offset a copy of it to a different position and then paint it onto the result.

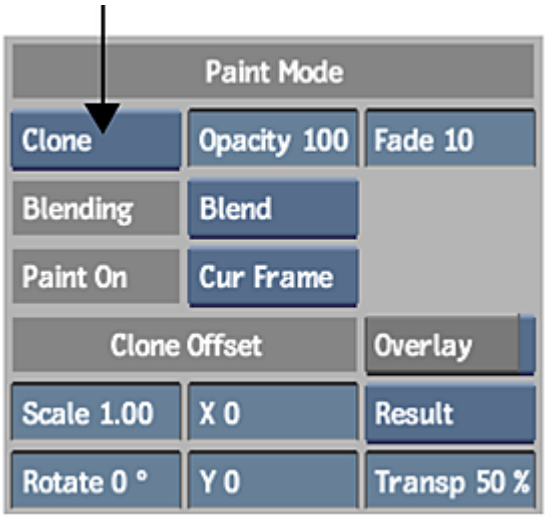
To paint a source onto the canvas, use the Reveal tool. See Revealing Sources (page 1214).

To clone an image and include cloned data in the brush strokes, use the Recursive Clone tool. See Recursive Clone (page 1214).

NOTE If changes are made upstream of the Paint Node, cloned strokes are preserved. However, the strokes will reflect the "old" input unless you force an update by clicking the Update button or by pressing U. Updates are not performed automatically because they can affect system performance.

To clone an image:

1 Select Clone from the Paint Modes box.



The Clone Offset parameters appear.

2 Set brush attributes. See Brush Attributes and Attribute Modes (page 1202).

3 To limit the cloned regions to the areas delimited by the matte, select an option in the Matte Source box.

4 To transform the reference image, do one of the following:

- To manually offset the image, enable the Overlay button (`Tab`), then hold down `Ctrl+Shift` and drag the overlaid reference image.
- To scale the image, enter a value in the Scale field.
- To rotate the image, enter the angle of rotation in the Rotate field.

NOTE To select an area for cloning, Overlay must be turned off. If you offset an image, verify that the Overlay button is disabled after use.

5 To select the area you want to clone, press `Ctrl` and click the canvas.

The cursor turns red. When you click the canvas, the red cursor is anchored and a green cursor appears.

6 Position the green cursor over the destination area and click the canvas.

The green and red cursors are now locked into positions relative to each other and move in tandem.

7 To set precise coordinates for the clone offset, use the X and Y fields.

8 Click and drag on the canvas to clone to the destination area.

Recursive Clone

Use the Recursive Clone brush to paint the contents of the result image to another position on the result. Unlike the Clone tool, when brush strokes created while using the Recursive Clone tool are used as a reference when the tool is applied elsewhere, the updated image data will be used.

Original image

Image courtesy of Buzz Image Group, Inc., and Les Films Stupefiants

After using the Recursive Clone medium

Image courtesy of Buzz Image Group, Inc., and Les Films Stupefiants

To use the Recursive Clone brush:

1. From the Paint Modes box, select Recur Clone (`Y`).
2. Set the brush size according to the size of the area you want to clone.
3. To select the area you want t clone, press `Ctrl` and click the canvas.

   The cursor turns red when you press `Ctrl`. When you click the canvas, the red cursor is anchored and a green cursor appears.
4. Position the green cursor over the destination area and click the canvas.

   The green and red cursors are now locked into positions relative to each other and move in tandem.
5. To set precise coordinates for the offset, use the X and Y fields.
6. Paint on the image.

   The image contained within the red circle brush is copied to the region defined by the green circle.

Revealing Sources

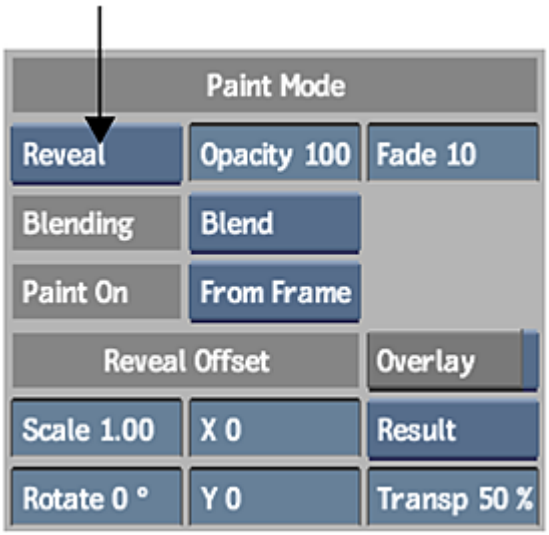
The Reveal tool allows you to paint the contents of one or more source front images directly onto the result. You can also use the Reveal tool to copy source matte images onto the output matte. For example, if one source has a tree that you would like to include on the canvas, you can select the source with the tree and then paint it onto the result.

To paint with a copy of the result or output matte image, use the Clone tool. See Clone (page 1212).

NOTE If changes are made upstream of the Paint Node, revealed strokes are preserved. However, the strokes will reflect the "old" input unless you force an update by clicking the Update button or by pressing `U`. Updates are not performed automatically because they affect system performance.

To reveal a front or matte input:

1 Select Reveal from the Paint Modes box.



The Reveal Offset parameters appear.

2 Set the brush attributes. See Brush Attributes and Attribute Modes (page 1202).

3 In the Sources list, select the source you want to reveal.

NOTE You can reveal sources that are marked as hidden in the Sources list. You will only be able to view the strokes created with a source when you disable Hide Strokes.

4 To limit the revealed sources to the areas delimited by their mattes, select an option in the Matte Source box.

5 To display the source as an overlay of the result image, enable Overlay (`Tab`). You can apply transformations to the source before you apply strokes:

- To interactively offset a source image, press `Ctrl+Shift` and drag the source. See Previewing a Reveal Operation Using a Reference Image (page 1222).
- To scale the source, enter a value in the Scale field.
- To rotate the source, enter the angle of rotation in the Rotate field.

6 Click and drag on the canvas to apply strokes that reveal the transformed contents of the selected source.

Drawing Shapes

You can draw circles, squares, rectangles, triangles, and freeform shapes. You can set the colour of the shape before you draw it. After you draw the shape, you can change its colour and determine its offset and X and Y position. You can also change a shape by moving its control points.

To draw a shape, from the Shape box, select a shape for the selection. Use the Bezier option to draw a freeform shape. Click Add Shape. The way in which you draw your shape depends on the shape that you are drawing.

To move a shape: in the Paint Tools box, select Move (keyboard shortcut M). Click the axis of the shape you want to move, and drag the shape to a new position.

Rectangle Click the canvas and drag until you reach your desired shape. Click again to complete the shape.

Square Use the Rectangle tool. Hold Shift and click to make the lower left anchor point of the square. Drag until you reach your desired shape. Click again to complete the square.

Ellipse Click the canvas and drag until you reach your desired shape. Click again to complete the shape.

Circle Use the Ellipse tool. Hold Shift and click to determine the centre of the circle. Drag until you reach your desired shape. Click again to complete the circle.

Triangle Click the canvas to make your first point. Drag and click the canvas a second time to complete the first side of the triangle. Drag and click once more to make the final corner of the triangle.

Bezier Click to make your starting point. Drag and click to draw subsequent points. Double-click the last point to complete the shape.

Creating an Offset of a Shape

To create an offset of a shape:

1. Click Selection.
   The blue enabled LED is displayed on the Selection button when the selection mode is turned on.
2. Click the axis of the shape you want to offset.
3. Set a pixel width for the offset in the Soft field.
   Pixels fade from the original edge of the shape to the edge of the offset.

Changing the Shape of a Shape

To change the shape of a shape:

1. In the Paint Tools box, select Move.
2. Click the axis of the shape that you want to change.
3. Click and drag an anchor point to move it.
4. Optional: Drag on a handle to change the angle of the side of the shape.

Drawing Selections

Use selections in layer-based Paint to limit the affected area of your paint strokes. Additionally you can select objects to change the object properties such as colour and opacity.

Selection LEDs

The following table summarizes the meaning of the coloured LEDs on the Selection button.

LED Colour:	Meaning:
Blue	The Selection tool is on. You can use the Add Shape tool to make a selection. Any existing selections are active.
Yellow	The Selection tool is temporarily off. Any existing selections are temporarily inactive.

LED Colour:	Meaning:
None	The Selection tool is not in use.

Making Selections

Make a selection to define the objects or area of the selected layer that you want to affect.

To make a selection:

1. Click Add Shape.
2. Enable the Selection button to turn on the selection mode.

   The blue enabled LED is displayed. If the blue LED is not displayed, click the Selection button again until it is.
3. From the Shape box, select a shape for the selection.

   NOTE Use the Bezier option to draw an irregular shape.
4. The way in which you draw your shape depends on the shape that you are drawing.

For a:	Do the following:
Rectangle	Click the canvas and drag until you reach your desired selection. Click again to complete the selection.
Square	Use the Rectangle tool. Hold Shift and click to make the lower left anchor point of the square. Drag until you reach your desired selection. Click again to complete the selection.
Ellipse	Click the canvas and drag until you reach your desired selection. Click again to complete the selection.
Circle	Use the Ellipse tool. Hold Shift and click to determine the centre of the circle. Drag until you reach your desired selection. Click again to complete the selection.
Triangle	Click the canvas to make your first point. Drag and click the canvas a second time to complete the first side of the triangle. Drag and click once more to make the final corner of the selection.
Bezier	Click to make your starting point. Drag and click to draw subsequent points. Double-click the last point to complete the selection.

   TIP You can hide the borders and axes of selections by clicking Hide Icons.
5. Optional: You can temporarily turn off your selection by clicking Selection a second time.

   The Selection button LED turns yellow.

Painting with Selection Masks

Use selection masks in layer-based Paint to paint shapes or to isolate a selection of the front or matte clip that you want to paint.

To paint inside or outside a selection mask:

1. With your selection mask enabled, select Paint from the Paint Tools box.
2. Optional: Enable Invert to paint outside of the selection mask.
3. Paint a stroke.

   The stroke is only applied in the selection mask (or outside the selection mask if Invert is selected).

Copying, Pasting, and Deleting Shapes, Selections, and Strokes

You can copy selections and then paste them on the same or a different layer.

1. Click Selection.
2. Click the axis of a shape, selection, or stroke on the canvas to select it.
3. Click a button or press a hot key to copy. Ctrl+C selects the current layer, Ctrl+Shift+C selects all layers. Ctrl+Shift+X cuts the selected contents.
4. Paste with Ctril+V. Tap `U` to update if you do not see your pasted object.

To delete a selection:

1. Click Selection.
2. Select the items you want to delete.
3. Click Delete. All selected items are deleted. The corresponding matte information is also deleted.

Clear shapes strokes and selections from a layer

1. Click a layer in the layers list to select it.
2. Click Clear. All of the shapes, strokes, and selections for the selected layer are deleted. The front and matte for the layer are not deleted.

Erasing Shapes and Strokes

If your Wacom tablet support it, you can use the eraser end of the pen if you are using the stylus to erase shapes and strokes. To use the eraser end of the pen: enable E then use the eraser end of the pen as you would and ordinary pencil eraser.

Using Blending Modes

Blending modes are Boolean operations that can be applied to the brush's colour components. A mode is applied separately to each of the red, green, and blue components of images. You can apply blending modes to combine the RGB channels of corresponding pixels from a stroke, described as follows.

Lighten Increases the RGB channel values of each pixel of the brush stroke.

Darken Reduces the RGB channel values of each pixel of the brush stroke.

Exclusion Adds the RGB channel values of the brush stroke, then subtracts twice the product of these channels.

Hard Light Multiplies or screens the selected colour of the brush stroke onto the image, depending on the colour. The effect is similar to shining a harsh spotlight on the image, and greatly reduces the contrast levels in the image.

If the blend colour (light source) is lighter than 50% grey, the image is lightened as if it were screened—this is useful for adding highlights to an image. If the blend colour is darker than 50% grey, the image is darkened, as if it were multiplied—this is useful for adding shadows.

Multiply Multiplies the RGB channel values of corresponding pixels of the stroke and the current image and normalizes the result by dividing by 255 in 8-bit mode, or 4095 in 12-bit mode. The resulting RGB channel values are assigned to the corresponding pixels in the generated clip.

Divide Divides the RGB channel values of pixels of the stroke and the current image and normalizes the result by multiplying by 255 in 8-bit mode, or 4095 in 12-bit mode. The resulting RGB channel values are assigned to the corresponding pixels in the generated clip.

Overlay Multiplies or screens the colours, depending on the RGB channel values of the colour of the stroke and the current image. Patterns or colours overlay the existing RGB channel values while preserving the highlights and shadows of the stroke's colour. The colour of the stroke is not replaced, but is mixed with the colour of the current image to reflect the lightness or darkness of the original colour.

Screen Multiplies the inverse of the current image with the colour of the brush stroke. The resulting colour is always lighter. The colour remains unchanged when you screen with black. Screening with white produces white. The effect is similar to projecting multiple photographic slides on top of each other.

Soft Light Shines a soft, diffuse light on the image. If the blend colour (light source) is lighter than 50% grey, the image is lightened. If the blend colour is darker than 50% grey, the image is darkened.

Using this mode with a black brush stroke results in a very dark effect; with white, a very bright one.

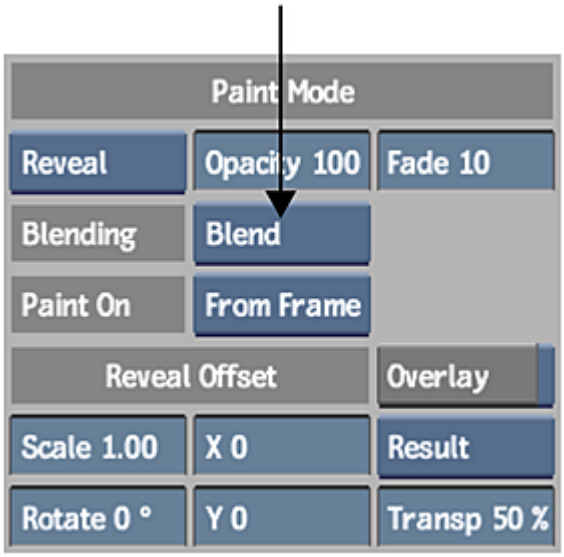
Add Adds the luma values of corresponding pixels of the brush stroke and the current image. The resulting value is assigned to the corresponding pixels in the generated clip. If the result is greater than 255 (in 8-bit mode), the pixel in the destination clip is clamped at a value of 255 (white). In 12-bit mode, the maximum colour value is 4095. The resulting clip is always brighter.

Subtract Subtracts the RGB channel values of the pixels of the current image from the RGB channel values of the pixels of the brush stroke and assigns the result to the RGB channel values of the pixel.

If an RGB channel value of the current image is larger than the corresponding channel value of the brush stroke, yielding a negative result, that result is clamped at 0 (black). The resulting clip is always darker.

To blend a stroke with the current image:

1 Select a Blend mode from the Blend option box.



2 Select a brush and colour.

3 Draw a stroke.

The stroke with the selected blend is applied. The current blend mode only affects new strokes. However, if you draw over existing strokes, the blend is created using the pixel values for the two different strokes.

Using the Canvas

Use the canvas controls to control clearing and wiping of the image.

Clearing the Canvas

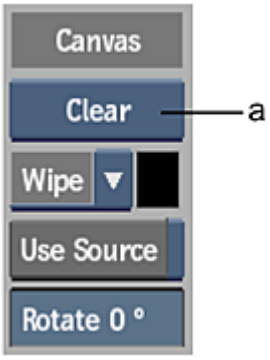
You can clear all strokes from the result image and output matte at the current frame or all frames. To limit the operation to a source, see Clearing Strokes on a Source (page 1199).

WARNING Any strokes that have been applied will not be maintained when you clear strokes.

To clear strokes from the canvas:

1 Select one of the following from the Clear Canvas option box:

- Clear to remove strokes from the current frame.
- Clear All to remove strokes from all frames.



(a) Clear Canvas option box

The strokes are removed from the result image.

Wiping the Canvas

You can wipe the result image when you want to apply a uniform colour at the current frame in a single brush stroke. Alternatively, you can use a source, to wipe over an image.

To wipe the result image with a colour or a source:

1. From the View box, select Result (`F4`).
2. Do one of the following:
   - Pick the wipe colour by clicking the colour pot to the right of the Wipe button.
   - Select the source in the Sources list and enable Use Source.

3. To display the source as an overlay of the result image, enable Overlay (`Tab`). You can apply transformations to the source before you apply the wipe:
   - To interactively offset a source image, press `Ctrl+Shift` and drag the source. See Previewing a Reveal Operation Using a Reference Image (page 1222).
   - To scale the source, enter a value in the Scale field.
   - To rotate the source, enter the angle of rotation in the Rotate field.
4. Do one of the following:
   - Click Wipe to wipe the front and matte.
   - Click the Wipe dropdown list and select Wipe F to wipe the front only.
   - Click Wipe dropdown list and select Wipe M to wipe the matte only.

Rotating the Canvas

During the painting process, you can rotate the canvas, making it easier to paint on any part of your image.

To rotate the canvas:

1. Do one of the following:
   - Set the angle of rotation in the Rotation field.
   - Press `Alt+spacebar` and drag in your image.

(a) Rotation field

Previewing a Reveal Operation Using a Reference Image

Overlaying a reference image on the result offers an intuitive way of previewing paint operations.

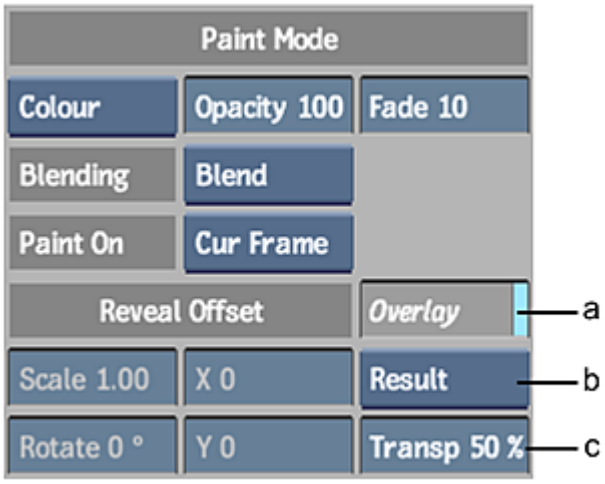
When you overlay a source onto the result, you can preview the effects of a Reveal operation. The selected image is superimposed over the result, and the transparency can be adjusted. This allows you to see exactly what your brush strokes will reveal.

When you overlay a result reference clip onto the result, you can slip the overlaid frame to display previous or next frames, creating an onion skin effect.

The overlay of the front and the matte can also be shown on the result, allowing you to rotoscope more easily, for example.

To display the reference image:

1. Enable the Overlay button (`Tab`).



(a) Overlay button (b) Reference button (c) Transparency field

The reference image appears by default at 50% transparency.

2 Select the reference image you want to display from the Reference box.

3 If you selected Source Matte or Source Front, select an image from the Sources list.

4 With the reference image, you can:

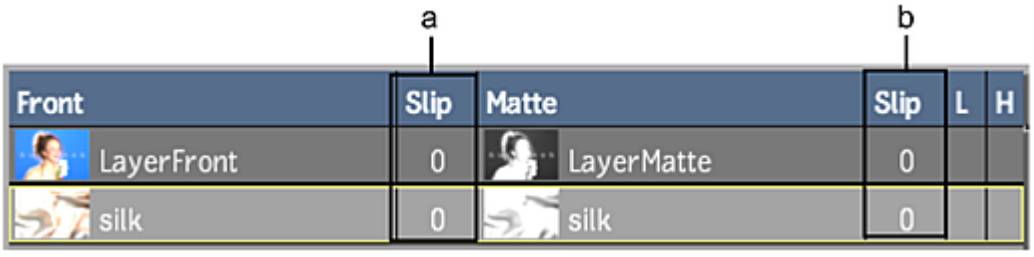
- Hold down `Ctrl+Shift` and drag the source front or source matte to offset it to preview a Reveal operation.
- Scale the image, by entering a value in the Scale field.
- Rotate the image, by entering the angle of rotation in the Rotate field.
- Set the Slip value in the Sources list to show a reference image at a different frame relative to the actual frame position. See Slipping a Reference Image (page 1223).
- Set the transparency for the reference image in the Transparency field.

NOTE You can also drag the result image to offset it for a Clone operation by holding down `Ctrl+Shift`.

Slipping a Reference Image

A reference image that is offset in time can be overlaid on the canvas. You can slip a front, matte, result, output matte, source front, and source matte clips. In the Sources list, use the Slip field to indicate the offset amount.

Each slip value can be set independently. To set the same front and matte values for a source, hold down the `Alt` key and edit either value.



(a) Front and Source Front Slip fields (b) Matte and Source Matte Slip fields

Saving Setups and Preferences

Paint Node setups are saved as XML files with the *.bpaint* extension. The procedures for saving and loading Paint Node setups are the same as for other setups.

Paint Node Keyboard Shortcuts

Front (`F1`)
Matte (`F3`)
Source Front (`F1 F1`)
Source Matte (`F3 F3`)
Result (`F4`)
Output Matte (`F4 F4`)

Add a source node (`Ctrl`-click the Add button)

Hide Strokes (`H`)

Clone and Recursive Clone selection (`Ctrl`-click)

Overlay button (`Tab`)

Offset reference image(`Ctrl+Shift` and drag)

Increase the brush size (`S` and drag right)

Decrease the brush size (`S` and drag left)

Rotate Canvas (`Alt+spacebar` and drag)

Paint (`Q`)

Clone (`W`)

Reveal (`E`)

Blur (`R`)

Smear (`T`)

Recur Clone (`Y`)

Paint Tool

Use the Paint Tool (also known as Desktop Paint) to create graphics, paint on images, and retouch clips. Use a matte to protect areas of the front clip during painting. Record, animate, and apply a series of brush strokes to a clip. Use the Graphic and Cut/Paste tools to create rotoscoped sequences and shape animations.

You can access the Paint Tool through the Tools tab.

Loading Clips into Paint

When accessing the Paint Tool, you first select the clips that you want to use. Any of the following combinations of clips can be used in Paint:

- A front clip only
- A front clip and a back clip
- A front clip and a matte clip
- A front clip, a back clip, and a matte clip
- None (a blank canvas)

The front clip appears on the Paint canvas, which you can use to apply colours and effects to the clip. The matte clip delimits the area of the front clip affected by painting. The back clip can be revealed or brushed through onto the front clip. The colour of the blank canvas is defined by the wipe colour.

NOTE The image window in Paint is referred to as the *canvas* in the following sections.

To load clips into the Paint Tool:

1. Select **Tools ➤ Paint**.
   The cursor changes to Pick Front.
2. Select the front clip.
   The cursor changes to Render Here.
3. Click on any free (or a grey) area on the workspace.
   You are now in the Paint Tool.
   The Paint menu appears.

4. Access one of the following Paint menus.

Click:	To:
Paint	Use the brushes or geometric shapes to paint on the canvas, record and apply strokes to the canvas, fill areas of the image with a reference colour or image, and move the image on the canvas.
Graphics	Create, edit, and animate shapes and apply them to the canvas.
CutOut	Create and add effects to cutouts and apply them to the canvas.
Setup	Set preferences, rendering options, grid guides, and colour correction options.

TIP See the tooltips for hints on all the Paint settings.

To access the Paint menu with a blank canvas:

1 Select **Tools ➤ Paint**.
The cursor changes to Pick Front.

2 From the Input Mode box, select None.
The Resolution Parameters controls appear.
The cursor changes to Render Here.

3 Choose a resolution, width, height, pixel aspect ratio, bit depth, and scan mode to apply to the background in Paint.

NOTE The default parameters are the project resolution parameters.

4 Set the frames per second and drop-frame values for the clip from the Frame Code Mode box.

5 Click on any free (or a grey) area on the workspace.
You are now in the Paint Tool.
The Paint menu appears.

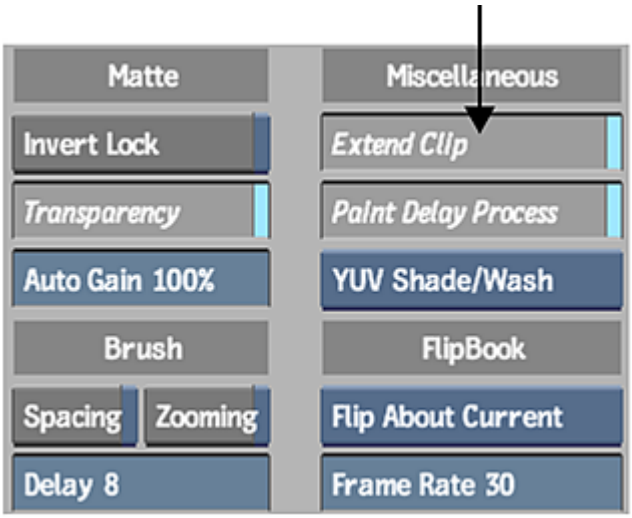
Locking and Unlocking the Duration of a Clip

You can determine whether or not frames are added to the end of a clip.

To add frames to the end of the clip:

1 Click Setup.

2 Enable Extend Clip.



3 Click Next Frame.

A frame is added. You can continue to add frames in the same manner with Extend Clip enabled.

TIP To maintain the length of the original clip, disable Extend Clip. No frames are added to the clip when you click Next Frame.

Using the Mouse

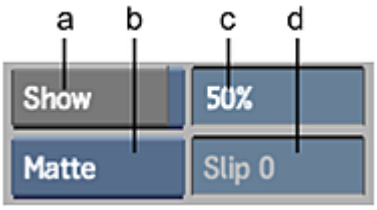
You can use either the mouse or the tablet and pen to paint. Press and hold the right mouse button to apply paint at 80% pressure, the middle button to paint at 40% pressure, and the left button to paint at 20% pressure.

Displaying a Reference Image

You can display a reference image in the background of the result image to use as a guide for rotoscoping.

To display the reference image:

1. Enable the Show button.



(a) Show button (b) Reference box (c) Transparency field (d) Slip field

The reference image appears by default at 50% transparency.

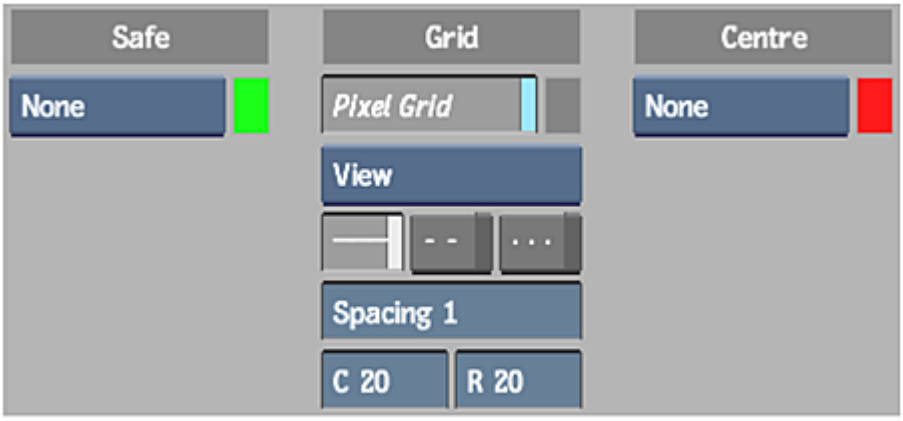
2. Select the reference image you want to display from the Reference box.
3. Set the transparency for the reference image in the Transparency field.
4. Set the Slip field value to show different images from the reference clip.
   This option only works if the reference image you selected is from a clip with more than one frame.

   NOTE The Slip field is disabled if you select Matte or Saved from the Reference box.

Using Overlays

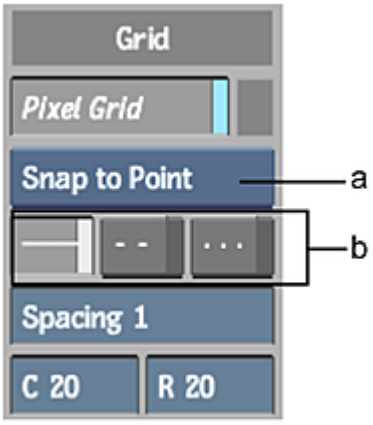
Click the Overlays button to enter the Overlays menu. The Overlays menu contains the Safe area guide controls, the Grid controls, and the Centre controls.

Use the field guides to provide reference points when painting, and the grid to help you accurately place strokes on the image. Neither the field guides nor the grids appear on the processed clip.



Grid Controls

You can set the behaviour of the grid so that paint strokes snap or lock to points on the grid. Use the Grid controls to configure the appearance and behaviour of the grid.



(a) Grid Behaviour box (b) Style buttons

To toggle the grid on and off, click Pixel Grid. When Pixel Grid is on, use the following controls to configure the appearance and behaviour of the grid.

Grid Behaviour box Displays the behaviour of the grid with respect to paint strokes:

- Snap to Point snaps each point of a stroke to the nearest intersection of a horizontal and a vertical grid line.
- Snap to Line snaps the current point of a paint stroke to the nearest point on a horizontal or vertical grid line.
- Lock to Point locks each point of a paint stroke to the nearest intersection of a horizontal and a vertical grid line.
- View uses the grid without the snap to or lock to options.

Style buttons Set a style for the lines of the grid. The style is either a solid line, a dashed line, or a dotted line.

Spacing field Sets the number of pixels between the dashes in a dashed line style, or between the dots in a dotted line style.

C, R fields Set the number of columns and rows respectively in the grid.

Using the Player in the Paint Tool

You can access the Player from the Paint Tool. To do so, click Render in the Paint Tool and then click Play once the clip is processed. Each time you access the Player, a clip is created in the Viewing Panel. This clip cannot be removed from the Paint tool. You must delete it from the Viewing Panel instead.

Selecting Colours

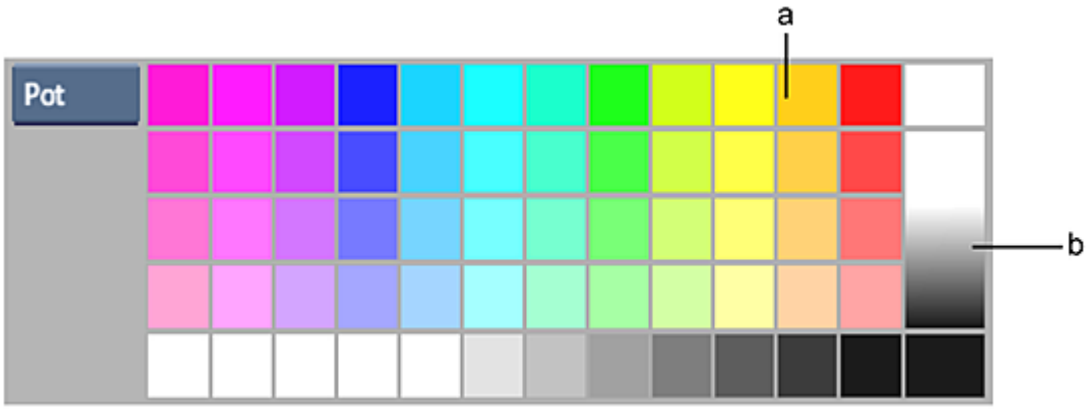
In Paint, you select colours using the colour picker. You store colours in the colour palette, mix or try out colours on the scratch pad, and create colour or greyscale gradients using the colour gradient.

Using the Current Colour

The current colour is used when you paint. It is also used to set the colour for the wipe function and the matte colour. Select the current colour from the colour palette, scratch pad, or colour gradient. Or, click the Current Colour pot to use the colour picker.

Using the Colour Palette

The colour palette appears in the Paint, Graphics, and CutOut menus. It contains the scratch pad and colour gradient. A series of colours are stored in the colour palette in colour pots. To select the current colour in the colour palette, click a colour pot. To set the current colour, press and hold a colour pot.



(a) Colour pot (b) Colour gradient bar

Build your own colour palette by storing the current colour in the colour pots. You can also save and load colour palettes.

To store a colour in the colour palette:

1. To display the colour palette, swipe the bar at the bottom of the screen.
   The colour palette appears.

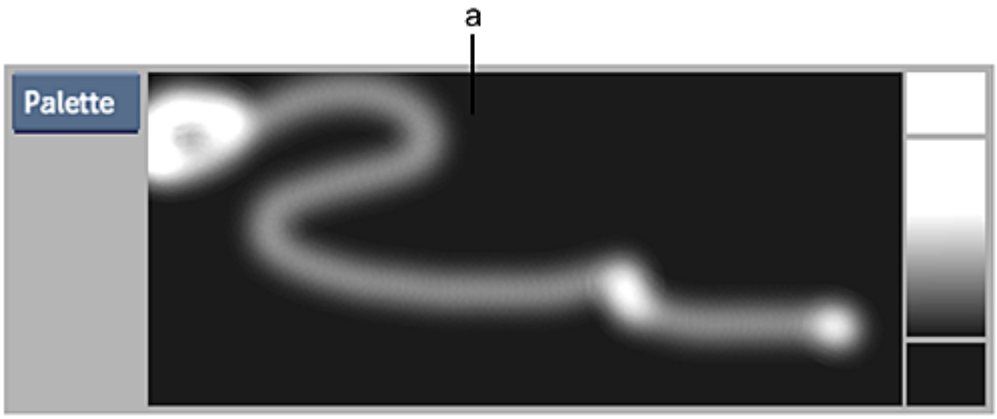
   TIP If you do not see the palette, click the Palette button. To hide it, swipe the bar a second time.

2. Set the current colour.

3 Click a colour pot in the colour palette and hold the cursor down momentarily.
The colour is saved in the colour pots.

Using the Scratch Pad

Use the scratch pad to mix colours selected from the image or colour pots and to test the selected brush type. You can also paste cutouts into the scratch pad to use when mixing colours.



(a) Scratch pad

To mix colours on the scratch pad:

1 Click the Pot button.
The scratch pad appears.
2 Paint on the scratch pad.
3 Change the current colour and paint over the previous brush strokes.
The colours are mixed.

You can also use the Wash, Shade, Smear, Drag, Warp, Impressionist, Recursive Clone, Stamp, and Blur Special Effects media in the scratch pad.

To use one of the media types in the scratch pad:

1 Select the Special Effects medium you want to use.
2 Enable Medium.
3 Use the brush cursor to apply the medium in the scratch pad.

To set the current colour using the scratch pad:

1 Click the Current Colour pot.
2 Use the colour picker to select the mixed colour in the scratch pad.
3 Click the Current Colour pot to use the selected colour.

Using the Colour Gradient Bar

The colour gradient bar is used to set the gradients for graphics you create in the Graphics menu. You can also set the current colour by selecting a colour on the gradient using the colour picker.

To set the gradient:

1 Set the current colour.

2 Click either the upper or lower colour pot on the gradient bar.

NOTE You must hold the cursor down for a moment to set the Gradient colour pot.

To set the current colour using the colour gradient bar:

1 Click and drag the cursor across the gradient bar.
The selected colour appears in the Current Colour pot.

2 Select a colour.
The selected colour becomes the current colour.

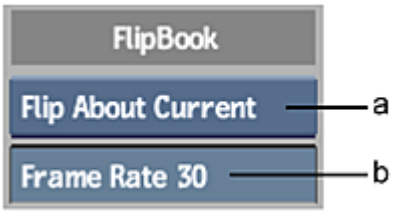
Previewing an Animation

Use the Flipbook command to preview an animation by playing a sequence of either five or nine frames.

To set up the Flipbook command:

1 Click Setup.
The Setup menu appears.

2 In the Flipbook area, select an animation mode from the Flipbook box.



(a) Flipbook box (b) Frame Rate field

Select:	To:
Flip About Current	Play two (or four) frames before the current frame and two (or four) after the current frame. This is the default setting.
Flip From Current	Play five (or nine) frames beginning at the current frame.
Flip To Current	Play five (or nine) frames ending at the current frame.

3 Enter the frame rate in the Frame Rate field.

NOTE If the system is running at high resolution, it may not be able to achieve a flip rate of 30 frames per second.

4 Press `F` to play five frames, or press `Shift+F` to play nine frames.
If you selected Flip About Current, pressing `F` plays two frames before and after the current frame. Pressing `Shift+F` plays four frames before and after the current frame.

Painting on Full-Resolution Film Images

Use the Zoom Mode box to paint on a film resolution image at full resolution from a zoomed out perspective. With this feature, you can perform complex operations like full-frame rotoscoping without having to constantly zoom in and out from the image on which you are painting.

In the Paint menu, the Zoom Mode box shows the zoom mode in which you are working. This mode is set automatically. If you zoom out from the image, the Zoom mode automatically switches from Raster to Tiled. This allows you to paint on the image from a zoomed out perspective.

(a) Zoom Mode box

NOTE If you manually switch the zoom mode to Raster while zoomed out from the image, you will not be able to paint on it.

You can also manually select either Raster or Tiled mode if you want to override the default mode. In Raster mode, if you apply paint to the edge of the image while zoomed in, the paint is applied in an abruptly sharp straight edge along the border of the canvas; the brush stroke is not completed on the image. In Tiled mode, if you attempt the same operation, the paint is applied from the full diameter of the brush you are using to the area of the image that is not visible on the canvas.

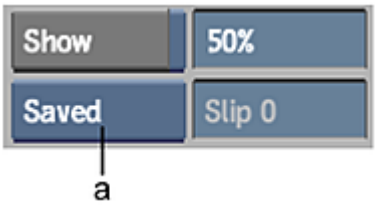
Saving and Restoring the Image

You can save the current frame on the canvas. If you want to reverse a paint application, you can restore the image without having to recreate it.

Save (Image Buffer) Saves the image that currently appears on the canvas. A single frame is stored in the Save buffer. The next time you click Save, the current frame replaces the frame stored in the Save buffer.

Restore Replaces the current image on the canvas with the contents of the Save buffer.

Preview Displays the contents of the Save buffer. Select the Saved option in the Reference box and click and hold Preview to view the contents of the Save buffer.



(a) Reference box

Exiting Paint

When exiting Paint, you may keep or discard the changes you have made to the front clip. Click Exit and select an option.

Select:	To:
Exit	Keep the changes you made to the front clip. The modified front clip appears in the Viewing panel.
Keep One	Keep only the current frame of the front clip. When this option is selected, a Confirm button appears to the right of the canvas. Click Confirm to keep only the current frame of the front clip. The current frame appears in the Viewing panel.
Cancel	Quit Paint without saving changes to the front clip. When this option is selected, a Confirm button appears to the right of the canvas. Click Confirm to quit without saving your changes, or click elsewhere to cancel.

About Canvas Mode

When you first open Paint, you are in Canvas mode. In Canvas mode, you can use brushes to modify your images. Brushes apply colour, filters, and Special Effects media to the image on the canvas. The brush cursor appears as a green cross surrounded by a circle when placed over the canvas. Canvas mode features are not available from multiple menus, such Canvas, Geometry, Fill, and Roll.

You can also paint on the canvas using geometric shapes to define the path of the brush.

To ease the painting of canvas edges, you can roll the canvas in the image window.

To paint on the canvas:

1. In the Paint menu, click Canvas.
   If the scratch pad is covering this button, swipe down to hide the scratch pad.
2. From the Paint Mode controls, click Paint.
3. Set a colour in the Current Colour pot. See Selecting Colours (page 1229).
4. Select a brush from the Brushes window. See Selecting a Brush (page 1233).
5. Set the brush attributes and modes in the Brush Attributes fields. See Brush Attributes (page 1239) and Brush Attribute Modes (page 1243).
6. Set the brush opacity in the Opacity field. Set the opacity to 100% to apply a fully opaque colour.
7. Stroke the brush over the canvas. To paint straight horizontal and vertical lines, press `Shift` and drag the brush up and down or left and right.
8. Click Undo to erase the strokes applied to the canvas since the last time you zoomed, panned, or changed a brush attribute.

Selecting a Brush

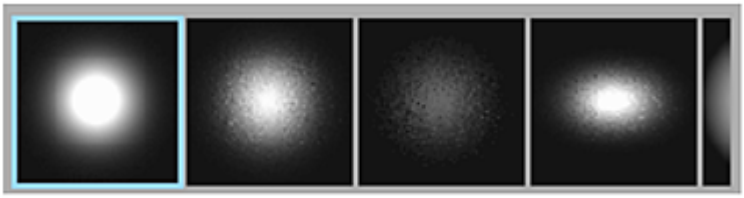
A number of predefined brush types are available in Paint including round, square, and elliptical brushes of various sizes and edge softness. The cursor has the same appearance regardless of which brush type you use.

Each brush has its own icon in the Brushes window. The white portions of the brush icon indicate where the paint will be applied on the image.

See Creating a Custom Brush (page 1313).

To select a brush:

1. Scroll through the Brushes window.



   To scroll the selections, click the Brushes window and drag left or right.

2. Click the brush icon you want to use.

   The selected brush is highlighted by a blue outline.

   NOTE Only one brush can be active at a time.

Using the Large Canvas

In Large Canvas mode, almost the entire image window is available for painting, but not all Paint options are available. You can still change the brush characteristics, medium, and show options. You can also save, restore, and wipe the canvas.

To display the large canvas:

1. Click Paint.
2. Click Canvas.
3. Swipe your cursor anywhere on the right edge of the screen, or press `Esc` to toggle between the large canvas and Paint menu.

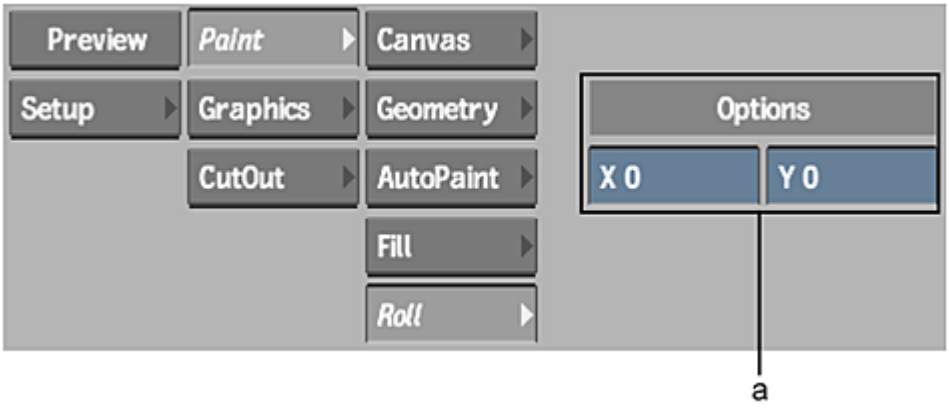
Rolling the Image

Use the Roll menu to reposition the current frame on the canvas to paint its edges.

To roll the image:

1. Click Paint.
2. Click Roll.

   The Roll menu appears.



(a) Coordinate fields

3 Zoom in on the image if necessary.

4 Drag the image using the pan cursor.

You can also enter the roll values in the Coordinate fields.

5 To recentre the frame, click Reset.

NOTE The frame is automatically recentred when you go to another frame or exit Paint.

Painting with Geometry

Use the Geometry feature to draw lines, rectangles, circles, and triangles on the canvas.

Painting with geometry is different from drawing objects in Graphics mode. In Graphics mode, you create objects that can be edited. When you paint with geometry, you define paths for the brush to follow. Paint uses the current brush setup to apply the stroke to the canvas.

Painting Lines

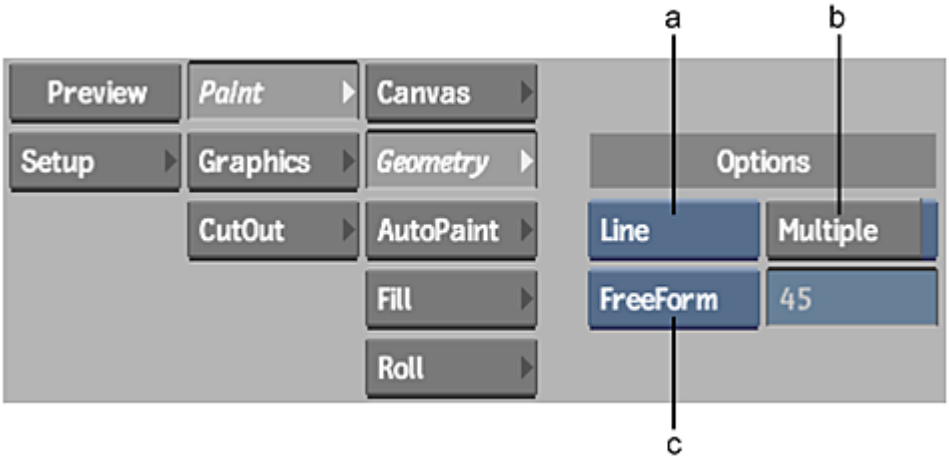
Use the Line option to paint straight lines. Paint single lines one at a time, or draw connected multiple lines. You can constrain the lines to vertical or horizontal paths or specific angles.

To paint a line or multiple lines:

1 Click Geometry to display the Geometry menu.

2 Select Line from the Geometry box.

The Multiple Line button and the Line Type box appear.



(a) Geometry box (b) Multiple button (c) Line Type box

3 Select the type of line you want to draw from the Line Type box.

Select:	To:
Angle	Paint a line at a specific angle. This option displays a field in which you enter the angle in degrees.
Vertical	Paint vertical lines.
Horizontal	Paint horizontal lines.
FreeForm	Paint lines at any angle with no constraint.

4 If you selected Angle, enter a degree in the field.

5 To draw single lines, move to the canvas and click, drag, and release.

A brush stroke is painted along the line.

6 To draw multiple lines, enable Multiple, move to the canvas, and click to place the start point of the first line. Click again to draw the end point. Continue clicking to place additional points and draw more lines.

7 To end multiple lines, click below the timebar or on the menu panel to turn the option off.

The brush strokes are painted along the lines.

Painting Rectangles

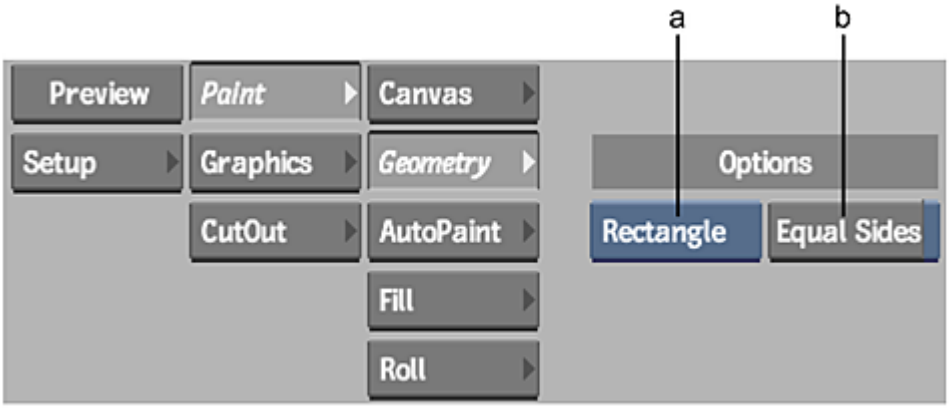
Use the Rectangle option to draw rectangles or squares.

To paint a rectangle or square:

1 Click Geometry to display the Geometry menu.

2 Select Rectangle from the Geometry box.

The Equal Sides button appears.



(a) Geometry box (b) Equal Sides button

3 To draw a square, enable Equal Sides or press and hold `P`.

4 Press the cursor on the canvas to anchor the first corner of the rectangle. Drag the cursor to size the rectangle.

5 When the rectangle is the required size, release the cursor.
A brush stroke is painted along the sides of the rectangle.

Painting Circles

Use the Circle option to draw circles of any size.

To paint a circle:

1 Click Geometry to display the Geometry menu.

2 Select Circle from the Geometry box.

3 Press the cursor on the canvas to anchor the centre point of the circle. Drag the cursor to size the circle.

4 When the circle is the required size, release the cursor.
A circular brush stroke is painted.

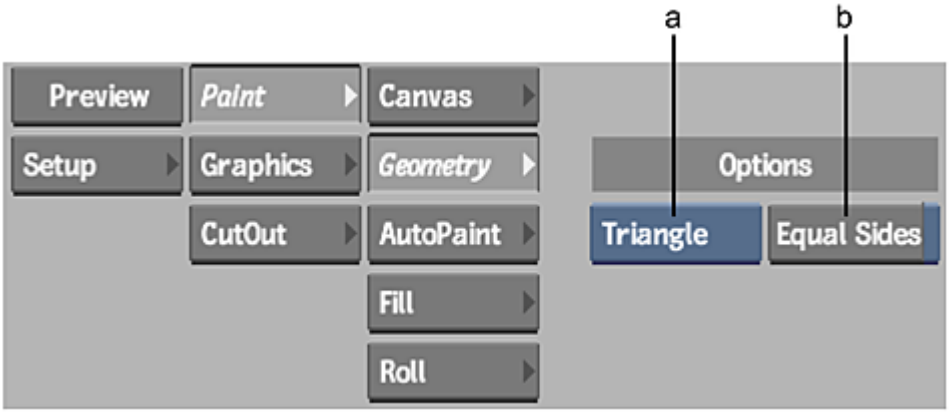
Painting Triangles

Use the Triangle option to draw equilateral or asymmetric triangles.

To paint a triangle:

1 Click Geometry to display the Geometry menu.

2 Select Triangle from the Geometry box.
The Equal Sides button appears.



(a) Geometry box (b) Equal Sides button

3 To draw an equilateral triangle, enable Equal Sides or press and hold P.

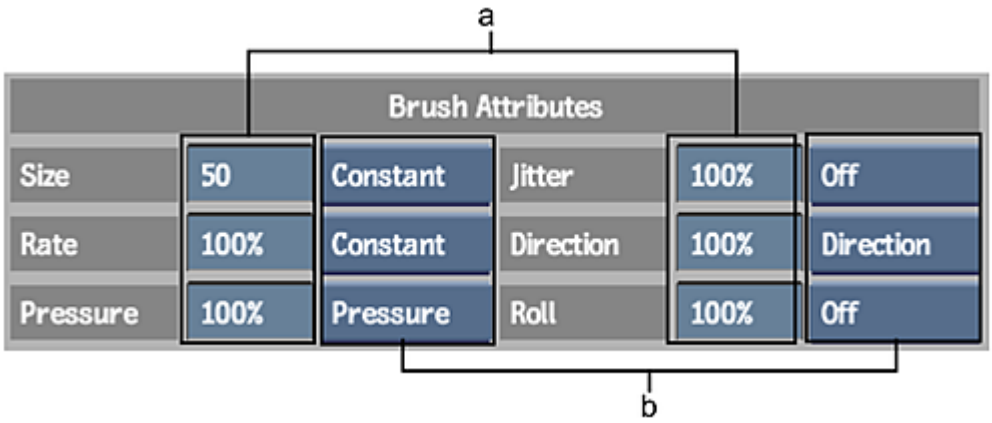
4 Move to the canvas and click to anchor the first vertex of the triangle.

5 Click to place the second vertex and then again to place the third vertex. If you are drawing an equilateral triangle, drag the cursor until the triangle is the required size.

A brush stroke is painted along the sides of the triangle.

About Brush Attributes and Modes

You can set various brush attributes and attribute modes that determine how paint is applied to the image. Each brush attribute has a corresponding Attribute Mode control. You use the Preferences menu to affect the way paint is applied to the image.



(a) Brush Attribute fields (b) Attribute Mode controls

In the Paint menu, the Brush Attributes and Attribute Mode controls share the same space as the colour palette. To display the brush attributes and modes in the Paint menu, swipe the cursor at the bottom of the screen.

In the Graphics Edit menu, the Brush Attributes and Mode controls appear only when the object attribute is set to Outline or Fuzzy. To display the brush attributes and modes in the Graphics menu, swipe the cursor twice at the bottom of the screen.

NOTE The brush attribute and attribute mode values set in the Paint menu are independent from the values set in the Graphics menu.

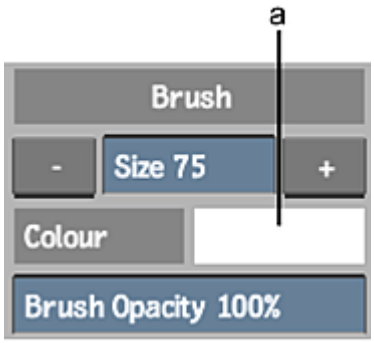
Brush Attributes

The Brush Attribute fields set the size of the brush, the distribution of the paint, and the rate and direction of the paint application. Brush attributes are listed as follows:

- Size
- Rate
- Pressure
- Opacity
- Jitter
- Direction
- Roll

Brush Opacity

The brush opacity affects the transparency of the brush. A value of 100% applies a fully opaque colour. Use a lower value to apply a more transparent colour. Enter a value in the Opacity field.



(a) Current Colour field

NOTE You can only use the Front, Back, Result, and Saved attribute modes with the Opacity attribute.

Brush Size

The brush size is indicated by the diameter of the green dashed circle surrounding the cursor brush. To increase the brush size, set a value in the Size field and drag the brush to the right on the canvas. To decrease it, set a value in the Size field and drag to the left.

You can also use the Size buttons to increase or decrease the brush size. Click the + button to increase the brush size. Click the - button to decrease the brush size. You can also click and drag in the Size field or click and enter a value.

NOTE You can use any attribute mode with the Size attribute.

Brush Rate

The brush rate is the rate at which brush strokes are applied to the canvas. Use a high value to produce a smooth continuous stroke, or a low value to produce a less continuous stroke with larger gaps between brush images.

NOTE You can use any attribute mode with the Rate attribute.

Rate attribute value = 100

Rate attribute value = 25

Brush Pressure

The brush pressure affects the transparency of the paint applied to the image. To apply opaque paint, use a high percentage value. For more transparent paint, use a low value.

The Pressure attribute differs from the Opacity attribute in that you can set the Pressure attribute mode so that the paint transparency varies according to the pressure applied to the pen or the direction of the brush.

NOTE You can use any attribute mode with the Pressure attribute.

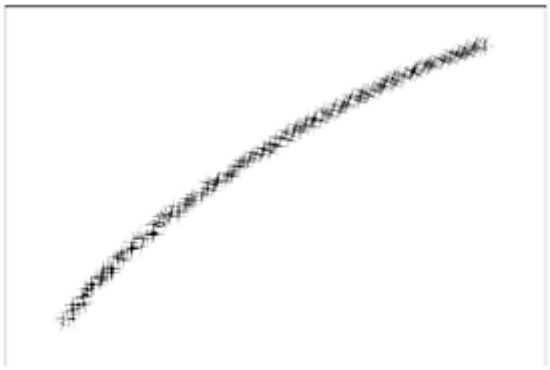
Pressure attribute value at 100%

Pressure attribute value at 50%

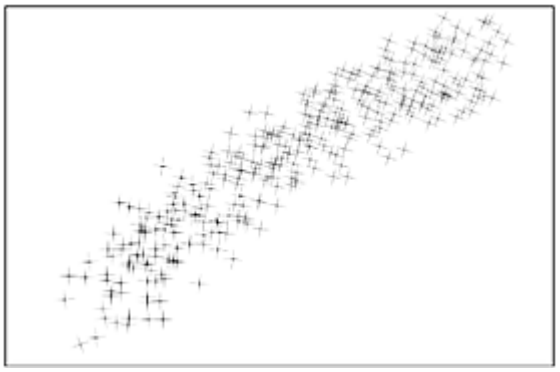
Brush Jitter

The Jitter attribute randomizes the brush strokes applied to the image. A high value produces a greater dispersion of paint, while a low value produces a greater concentration.

NOTE You can use any attribute mode with the Jitter attribute.



Jitter attribute value at 10

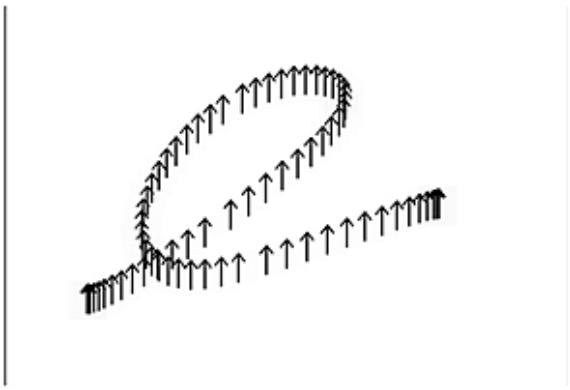


Jitter attribute value at 100

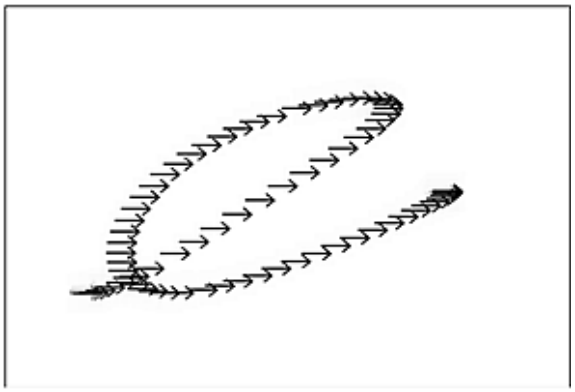
Brush Direction

The Direction attribute causes the brush to rotate around the Z-axis and can be used to produce a calligraphy effect. The effect of the Direction attribute is most noticeable when used with one of the elliptical or star brushes. The value of the Direction attribute causes the brush strokes to rotate by 90 degrees for each increment of 25 percent.

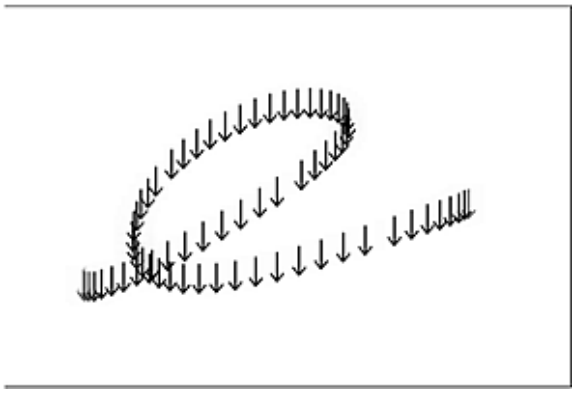
NOTE You can use any attribute mode with the Direction attribute.



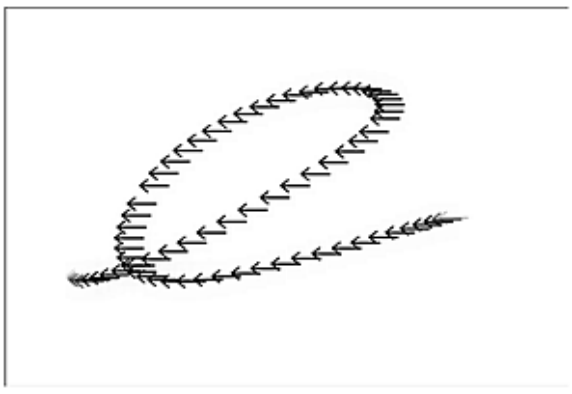
Direction attribute at 100%



Direction attribute at 75%



Direction attribute at 50%

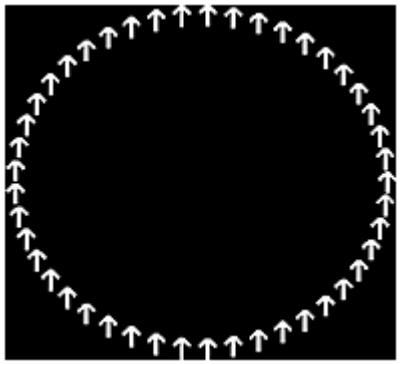


Direction attribute at 25%

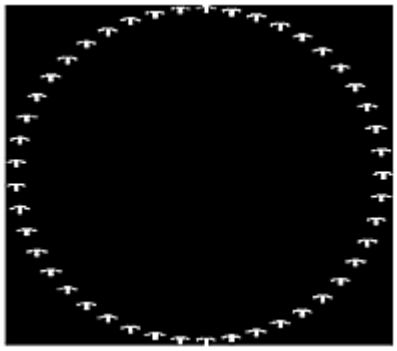
Brush Roll

The Roll attribute rolls the brush around the X-axis. The effect of the roll is most noticeable when used with one of the non-symmetrical brushes. For each increment of 25 percent, the Roll attribute value creates a rolled brush stroke of 90 degrees.

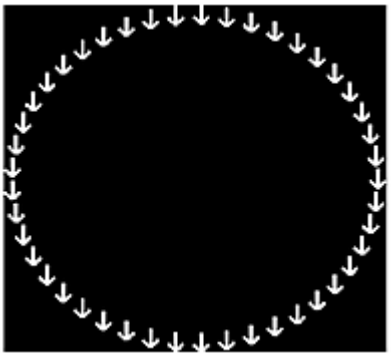
NOTE You can use any attribute mode with the Roll attribute.



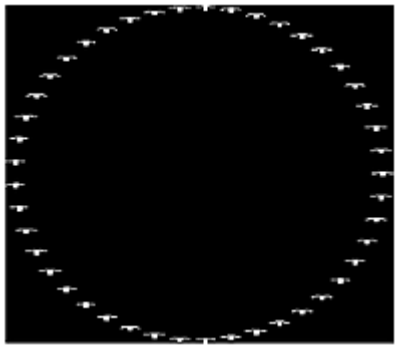
Roll attribute value at 100%



Roll attribute value at 75%



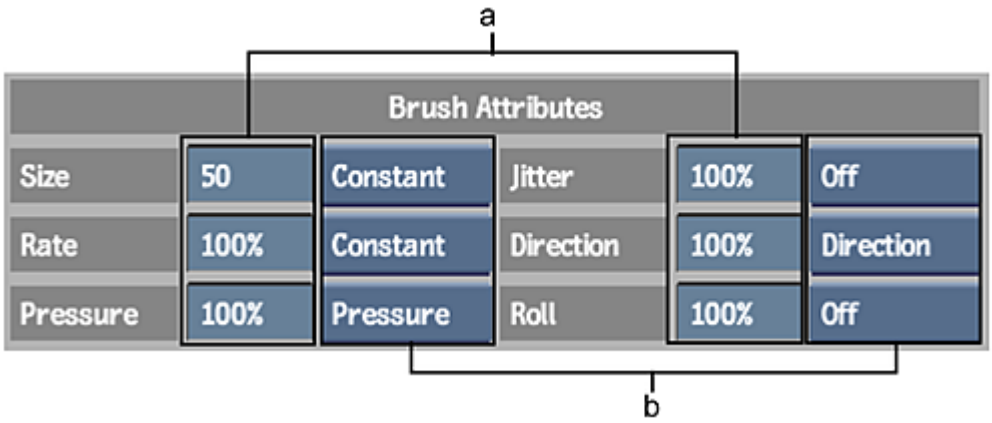
Roll attribute value at 50%



Roll attribute value at 25%

Brush Attribute Modes

The value of a brush attribute depends on the selected attribute mode. You can choose Constant, Off, Front, Back, Result, Saved, Pressure, or Direction.



(a) Brush Attribute fields (b) Attribute Mode controls

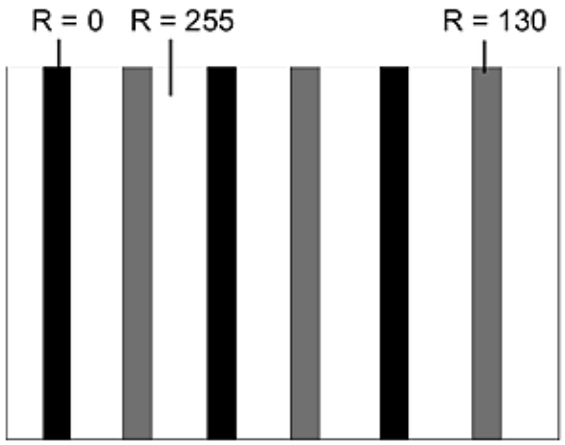
In Constant mode, the brush attribute values remain constant. In Off mode, the attribute is disabled.

The Front, Back, Result, and Saved modes use the colour values in a reference image to vary the brush attribute value. Pressure and Direction modes affect how paint is applied by causing the brush attribute value to vary in relation to the pressure exerted on the pen and the direction of the brush, respectively.

NOTE Do not set the Size, Rate, or Pressure attributes to Off mode.

Using Reference Images

The Front, Back, Result, and Saved modes use the red channel in reference images to set the brush attribute values. Front mode uses the front clip as the reference, Back mode uses the back clip, Result mode uses the result clip, and Saved mode uses the image in the Save buffer.



Back image

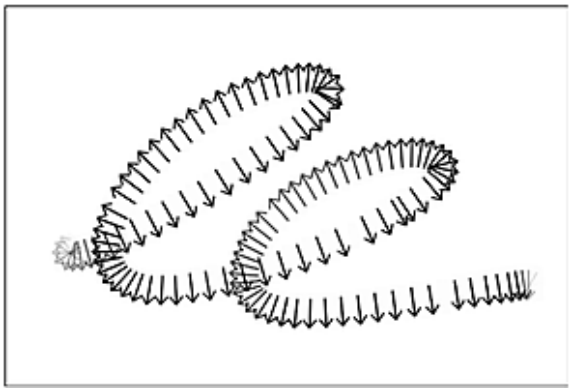
Using Size attribute with mode set to Back

Direction of the Brush

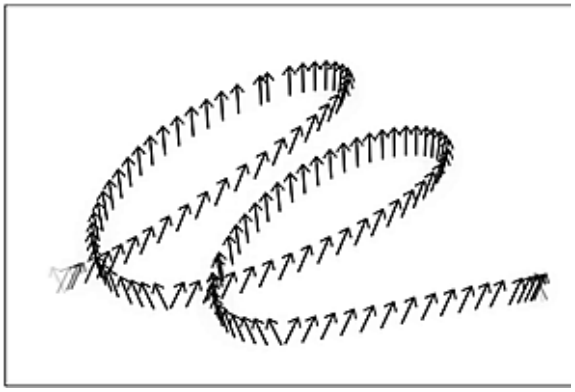
The Direction mode uses the direction of the brush stroke as the reference.

When used with the Direction mode, the Direction attribute causes the brush strokes to follow the trajectory of the brush. Increase the percentage value to enhance the effect on brush direction.

Drag:	To:
Right	Use 100% of the attribute value.
Left	Use 0% of the attribute value.
Up	Use 25% of the attribute value.
Down	Use 75% of the attribute value.



Direction attribute value at 100%, Direction mode



Direction attribute value at 15%, Direction mode

Pressure Exerted on the Pen

The Pressure mode uses the pressure exerted on the pen as the reference value. The harder you press on the pen, the greater the brush attribute value. The softer you press, the lower the brush attribute value.

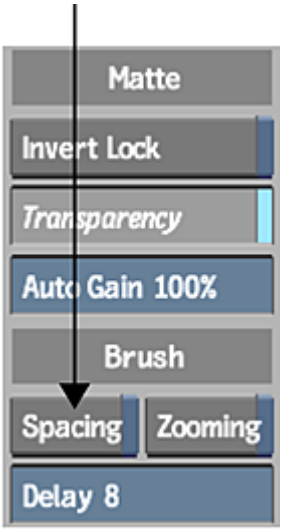
NOTE The Pressure attribute mode cannot be used in the Graphics menu.

Preferences Affecting Brushes

The Spacing setup preference affects how paint strokes are applied to the canvas.

To enable Brush Spacing:

1. Click Setup in the Paint menu.
2. Enable Spacing.



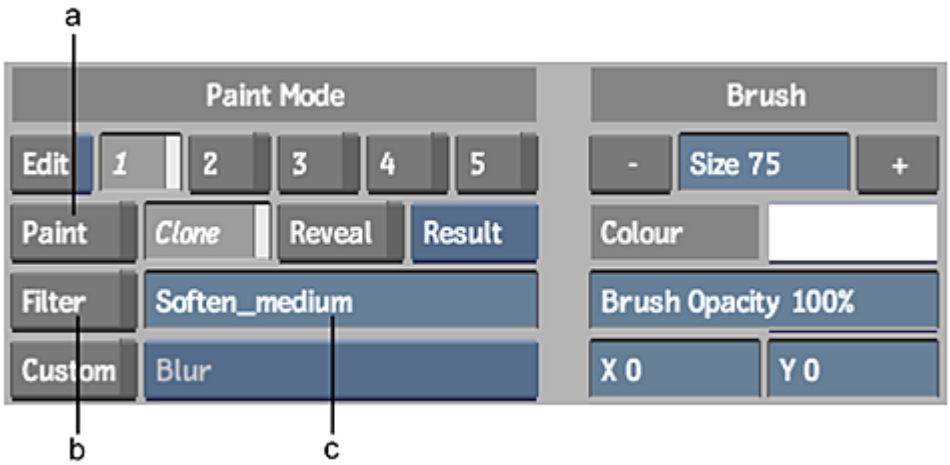
This option sets a uniform distance between paint strokes. No matter how fast you move the brush, the brush spreads the paint evenly.

TIP Use a high brush rate when using stamps with the Spacing option enabled. See Using the Stamp Medium (page 1253).

Applying Filters

When applying filters, it is important to try different brush types and change the brush attributes to create different effects.

Apply filters to the canvas using the brush, Wipe command, or Wash and Shade media. Paint uses the same filter library as the Filter command in the Processing menu.



(a) Paint Media button (b) Filter button (c) Filter field

To brush a filter onto the image:

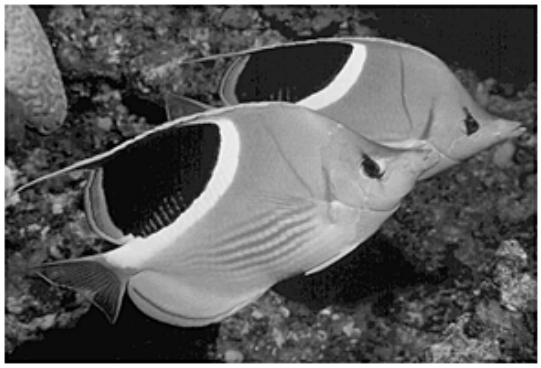
1. In the Paint menu, click the Filter field.
   The filter library appears.
2. Select the filter you want to use.

You are returned to the Paint menu and the filter name appears in the Filter field.

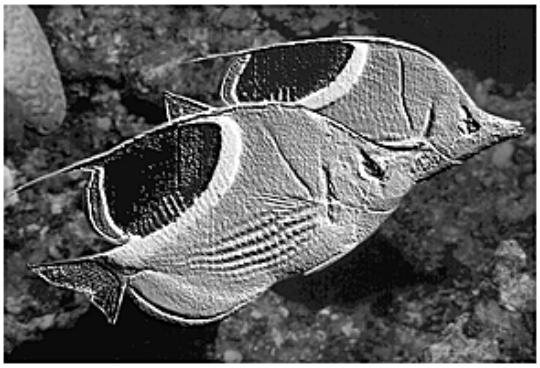
3 Click Filter to enable the selected filter.

4 Set the brush opacity. The opacity determines the level of filtering. Reduce the opacity value to reduce the level of filtering.

5 Paint on the image.



Original image

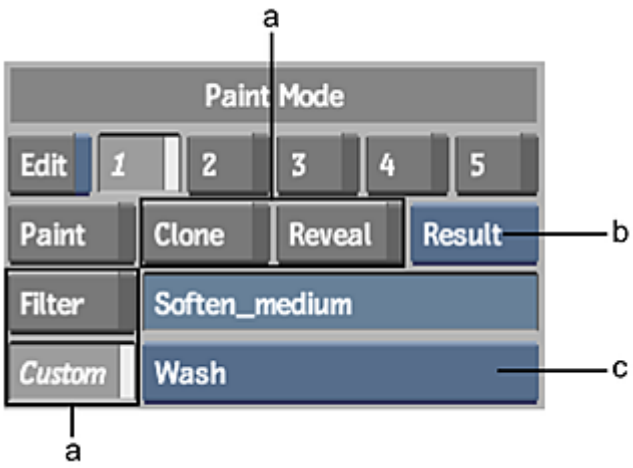


After using the emboss filter

Using Special Effects Media

Special Effects media can be brushed onto the image in Canvas mode. You can also apply the Wash, Shade, and Reveal media to the entire image using the Wipe command. See Wiping Using Special Effects Media and Filters (page 1257). Only one media type can be used at a time.

Use the media buttons to enable some Special Effects media, for example, the Clone and Reveal media. Other media types, such as Drag and Smear, are enabled using the Custom Media box.



(a) Special Effects Media buttons (b) Reference box (c) Custom Media box

The following Special Effects media are available.

Use:	To:
Blur	Apply a blur filter to portions of the image. See Blurring the Image (page 1247).

Use:	To:
Clone	Copy a portion of the image to a new location. See Cloning the Image (page 1248).
Drag	Create an image trail from a selected region of the screen. See Dragging the Image (page 1249).
Impressionist	Paint on colours from a reference clip. See Using the Impressionist Medium (page 1250).
Recursive Clone	Make a number of copies of a selected area of the image. See Using the Recursive Clone Medium (page 1251).
Reveal	Brush a reference image onto the current image. See Revealing a Reference Image (page 1251).
Shade	Darken or lighten the image depending on the luminance value of the current colour. See Washing and Shading the Image (page 1255).
Smear	Smudge areas of the image. See Smearing the Image (page 1252).
Stamp	Apply a captured image to the image. See Using the Stamp Medium (page 1253).
Warp	Stretch and distort a region of the image. See Warping the Image (page 1254).
Wash	Apply a transparent wash of the current colour to the image. See Washing and Shading the Image (page 1255).

Blurring the Image

Use the Blur medium to blur portions of the image.

To blur the image:

1 Click Custom and select Blur from the Custom Media box.
The Blur option boxes appear.

(a) Filter Type box (b) Blur Strength box

2 Click Canvas.

3 Set the brush size according to the size of the area you want to blur.

4 Select either a Box or Gaussian filter from the Filter Type box.

5 Set the density of the Blur brush from the Blur Strength box.
You can choose Light, Medium, or Heavy.

6 Drag the brush on the image.

Original image

After using the Blur medium

Cloning the Image

Use the Clone medium to copy a region of the image and paint it on a destination point. The result image is used as the source for the Clone medium. The offset between the origin point and the destination point is set in the Offset fields.

To paint using the Clone medium:

1 Click Clone.
The Clone controls appear.

(a) Offset fields

2 Click Canvas.

3 Set the brush opacity. The brush opacity determines the transparency of the clone. When the opacity value is set to 100%, the clone is completely opaque.

4 Set the brush size.

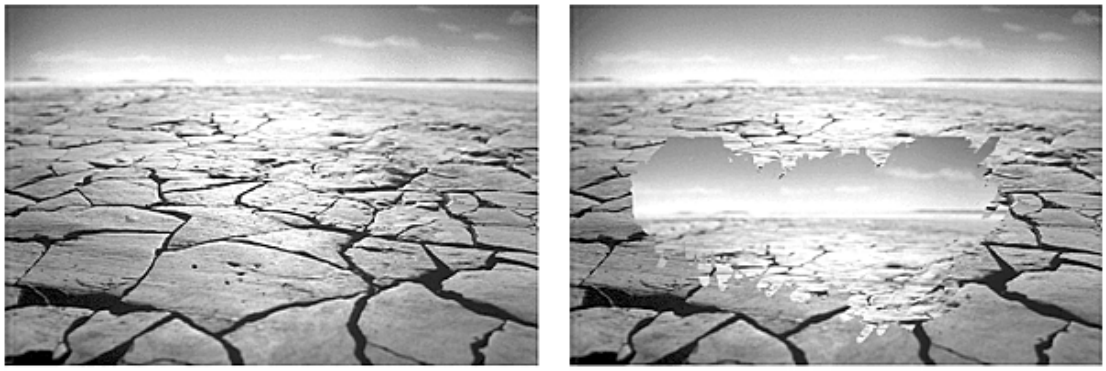
5 Set the offset between the origin point and the destination point in the Offset fields.

A tracking circle appears at the offset co-ordinates you specified; this is the destination point for the cloned image.

TIP Press `Ctrl` and drag the cursor to set the destination point.

6 Paint on the image.

The image contained within the red circle brush is copied to the region defined by the green circle.



Original image

After using the Clone medium

Dragging the Image

Use the Drag medium to drag a selected area of the image across the canvas. The selected area is painted on the canvas as you drag the brush, creating an image trail.

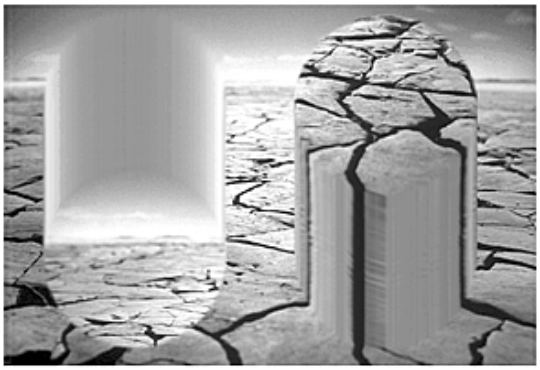
NOTE The Direction brush attribute cannot be used with the Drag medium.

To use the Drag medium:

1. Click Custom and then select Drag from the Custom Media box.
2. Click Canvas.
3. Set the brush size.
4. Position the brush over the region of the image that you want to use. Hold down the cursor to select that region.
5. Drag the brush.

The selected region is painted onto the image as you drag. The pressure of the brush determines the density of the image trail. See Brush Pressure (page 1240).

Original image



After using the Drag medium

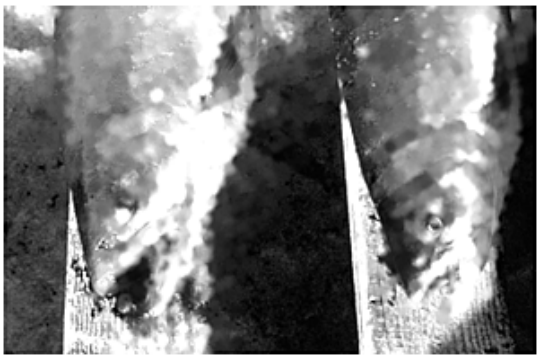
Using the Impressionist Medium

Use the Impressionist medium to brush on colours from a reference clip. The brush opacity determines how much colour is taken from the reference clip. When the opacity value is set to 100%, all the colour is taken from the reference image. At 50%, the colour applied is a 50/50 blend of the reference colour and the result image.

To use the Impressionist medium:

1. Click Custom and then select Impressionist from the Custom Media box.
2. Click Canvas.
3. Set the brush opacity.
4. Select a reference image from the Reference box.
5. Paint on the image.

Original image



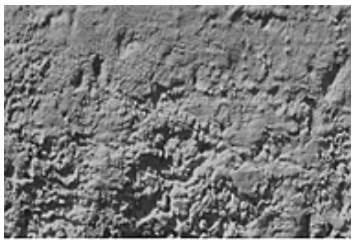
After using the Impressionist medium

Revealing a Reference Image

Use the Reveal medium to reveal portions of an image or an entire reference image. You can reveal specific areas using the brush or the entire image using the Wipe command. The opacity of the brush determines how much of the reference image is revealed on the image. When the opacity is set to 100%, the reference image applied is completely opaque. At 50%, the reference image applied is a 50/50 blend of the reference image and the result clip.

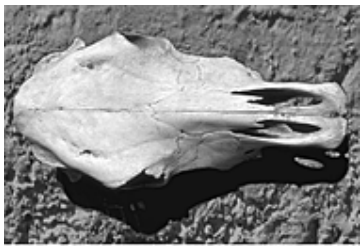
To reveal a reference image on the canvas:

1. Click Reveal.
2. Click Canvas.
3. Set the brush opacity.
4. Select a reference image from the Reference box.
5. Paint on the image.



Back image

Result image



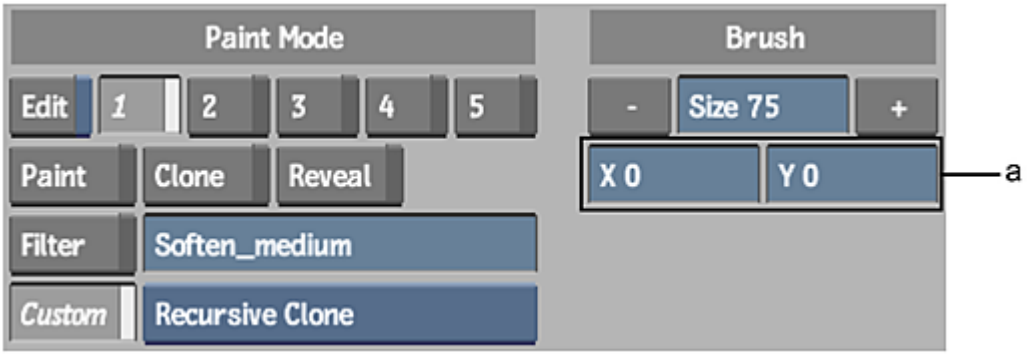
The Back image revealed on the Result image

Using the Recursive Clone Medium

Use the Recursive Clone medium to make multiple copies of an area of an image. The result image and the paint applied to it is used as the source. Each copy is a degraded version of its predecessor. The offset between the origin point and the destination point is set in the Offset fields.

To paint using the Recursive Clone medium:

1. Click Custom and select Recursive Clone from the Custom Media box.
   The Recursive Clone options appear.



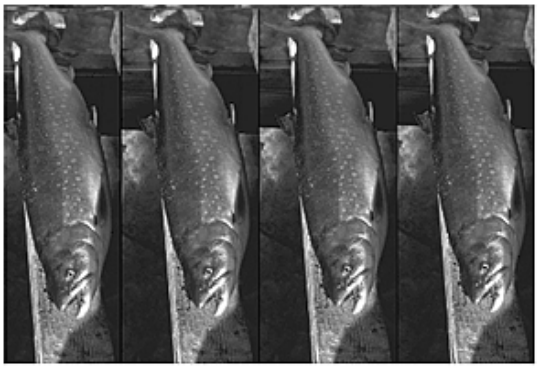
(a) Offset fields

2 Click Canvas.

3 Set the offset between the origin point and the destination point in the Offset fields.
A red tracking circle appears around the destination point.

4 Paint on the image.
The image contained within the green circle brush is copied to the region defined by the red circle.

Original image



After using the Recursive Clone medium

Smearing the Image

Use the Smear medium to smudge portions of the image.

To smear the image:

1 Click Custom and select Smear from the Custom Media box.

2 Click Canvas.

3 Drag the brush over the area you want to smear.

Original image

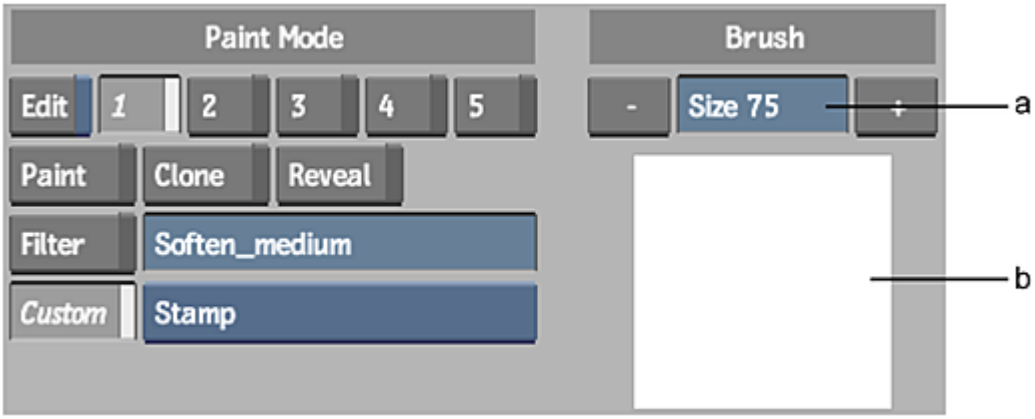
After using the Smear medium

Using the Stamp Medium

Use the Stamp medium to capture a portion of the image and apply it to the canvas.

To capture and apply a stamp:

1 Click Custom and select Stamp from the Custom Media box.
The Stamp window appears beneath the Brush Size field.



(a) Brush Size field (b) Stamp window

2 Click Canvas.

3 Set the brush size. Use a small brush to isolate a specific detail of the image. Use a large brush to capture a bigger sample.

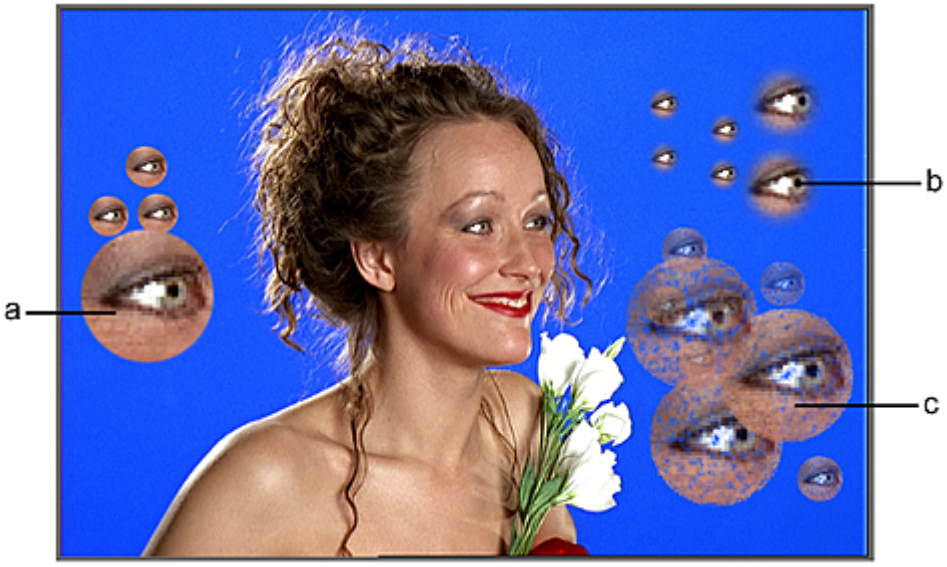
4 Click and hold the cursor on the Stamp window.

5 Without releasing the cursor, move it over the image.
The Stamp window is updated as you move the cursor across the image.

6 Release the cursor when the Stamp window contains the part of the image you want to capture.
The brush cursor appears.

7 Paint on the image.

(a) The captured stamp



(a) Stamp applied using a circular brush (b) Stamp applied using an air brush (c) Stamp applied using a chalk brush

Saving and Loading Stamps

You can save a stamp and load it in another session to use with a different clip. See Saving Setups (page 1310).

Warping the Image

Use the Warp medium to stretch and distort regions of the image.

To warp the image:

1. Click Custom and select Warp from the Custom Media box.
2. Click Canvas.
3. Set the brush size.
   The area that can be warped is determined by the brush size.

4 Select a portion of the image and hold down and drag the cursor to warp the selection.

Original image

After using the Warp medium

Washing and Shading the Image

Use the Wash medium to apply a transparent wash of the current colour to the image. Use the Shade medium to darken or lighten the image. You can wash or shade specific areas using the brush or the entire image using the Wipe command.

With the Shade medium, you darken images using a colour with a low luminance value, and lighten them using a colour with a high luminance value. The opacity of the brush affects the transparency of the paint applied to the image. When the opacity value is set to 100%, the paint applied is completely opaque. As you decrease the opacity, the paint becomes more transparent.

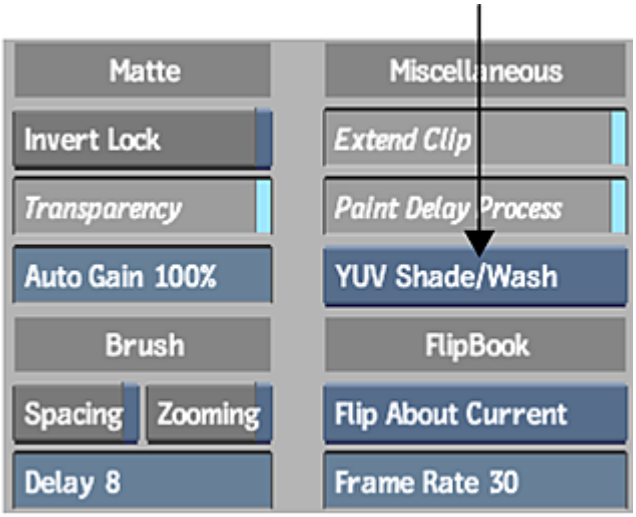
Selecting a Colour Model

You can apply the colour using either the YUV or HLS.

To select a colour model:

1 Click Setup.

2 Select either YUV Shade/Wash or HLS Shade/Wash.



To use Wash or Shade:

1. Click Custom, and select Wash or Shade from the Custom Media box.
2. Click Canvas.
3. Set the current colour.
4. Set the brush opacity.
5. Paint on the image.

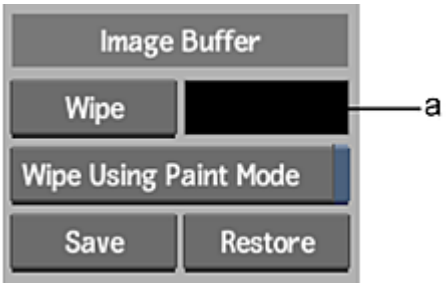
A paint stroke using the Paint medium, current colour red.

A paint stroke using the Wash medium, current colour red.

A paint stroke using the Shade medium, current colour red.

Wiping an Image

Use the Wipe command to apply colours, filters, and Special Effects media to the entire result image in a single stroke.



(a) Wipe colour pot

NOTE You can also wipe an image with the AutoPaint Wipe mode. See Wiping the Canvas in AutoPaint (page 1263).

Wiping Using a Colour

Use the Paint medium to wipe the image with a selected colour. The colour used is set in the Wipe colour pot.

To wipe the image using a colour:

1. Set the current colour you want to use for the wipe. See Selecting Colours (page 1229).
2. Click the Wipe colour pot.
   The current colour is transferred to the wipe colour.
3. Set the brush opacity.
   A value of 100% wipes the image with a completely opaque colour.
4. Click Wipe.

Wiping Using Special Effects Media and Filters

You can use a filter, or the Reveal, Clone, Wash, and Shade Special Effects media.

To wipe with Special Effects media and filters:

1. Set the wipe colour.
2. From the Paint Mode controls, select the Special Effects medium to be applied. See Using Special Effects Media (page 1246).

NOTE If you are using a filter for the wipe, you must enable the Filter button.

3 Set the brush opacity.

A value of 100% wipes the image with the full effect of the Special Effects Media or filter.

4 Enable the Wipe Using Paint Mode button.

5 Click Wipe.

Using the Wipe Command in Graphics

You can also use the Wipe command in the Graphics menu. The Wipe command wipes over any objects tacked down on the image. Objects not tacked down are not part of the image and are not wiped over.

Filling an Image

Use the Fill command to fill areas of an image with similar colour values or areas delimited by a colour. These areas can be filled with either a solid colour or a reference image. You can choose how far the filled area extends by specifying how similar the pixels must be in order to be filled. This enables you to fill only the dark areas of an image, or include slightly lighter areas.

Use the Fill controls to set the colour model, range, and softness of the fill.

To display the Fill controls:

1 Click Fill in the Paint menu. If the Fill button is hidden by the colour palette, swipe the bottom of the screen.

Defining the Range for the Fill

When you use the Fill command, you specify a range for the colour comparison. The Fill command compares the colour values of adjacent pixels to determine if the values are within the specified colour range. Adjacent pixels within the range are filled.

The range is determined using two values: the colour value of the pixel you select to begin the fill, called the *selection point*, and the value you set in the Range field.

Selecting Colour Channels

You can use either the RGB or YUV model. Within each colour space, you can work with any combination of colour channels. For example, if you select the R, G, and B channels in the RGB colour model, the Fill command considers the red, green, and blue values. Adjacent pixels with red, green, and blue values within the specified range are filled.

Adjusting the Softness

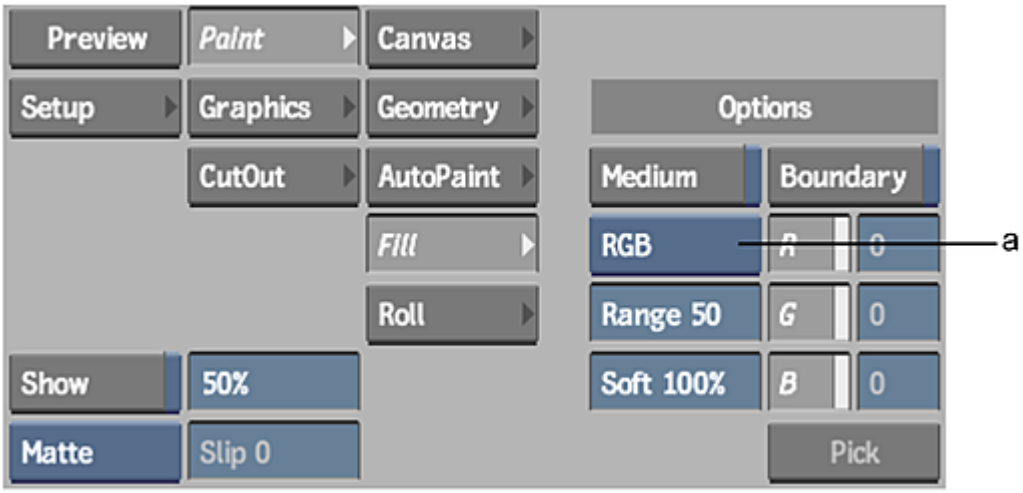
The softness value for the fill determines the amount of colour diffusion applied at the edges of the fill. This value can be adjusted to modify the transition between the filled and unfilled areas. A softness value of 100% produces the greatest amount of diffusion. A softness value of 0% produces a filled area with well-defined edges.

Filling a Region with a Colour

You can choose to fill an image with the current colour.

To fill a region of the image with a colour:

1. Click Fill.
   The Fill controls appear.



(a) Colour Model box

NOTE To fill a matte, click From Matte. When this button is enabled, the pixels in the matte are used for the colour comparison. If this button does not appear at first, click Matte on the right side of the menu panel.

2. Select a colour model from the Colour Model box.

Select:	To:
RGB	Display the R, G, and B channel buttons. Enable each channel you want to use.
YUV	Display the Y, U, and V channel buttons. Enable each channel you want to use.

3. Set the colour range in the Range field.
   You can also set a tolerance by enabling the Boundary button and selecting a distinct boundary colour from the image.
4. Set the softness in the Soft field.
5. Set the current colour.
6. Select a point in the area you want to fill. This is the selection point.

The pixels that fall within the specified range are filled with the current colour.

Original image

Fill with range 75 and softness 100

Fill with range 75 and softness 50

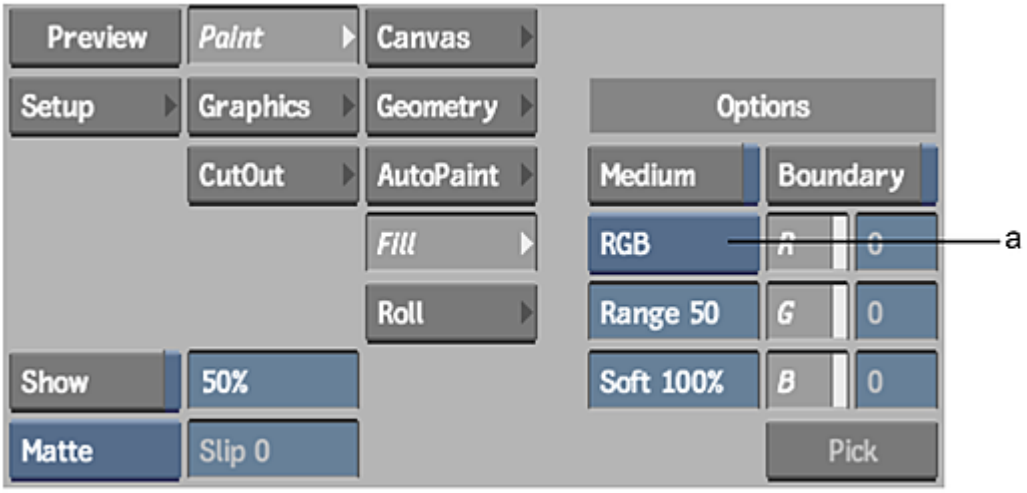
Fill with range 50 and softness 100

Filling a Region with an Image

You can fill a region of an image with a reference image instead of a colour. The front, back, or result image can be used as the reference image. The Wash, Shade, Clone, or Reveal media can also be used to perform the fill.

To fill a region with a reference image:

1 Click Fill.
The Fill controls appear.



(a) Colour Model box

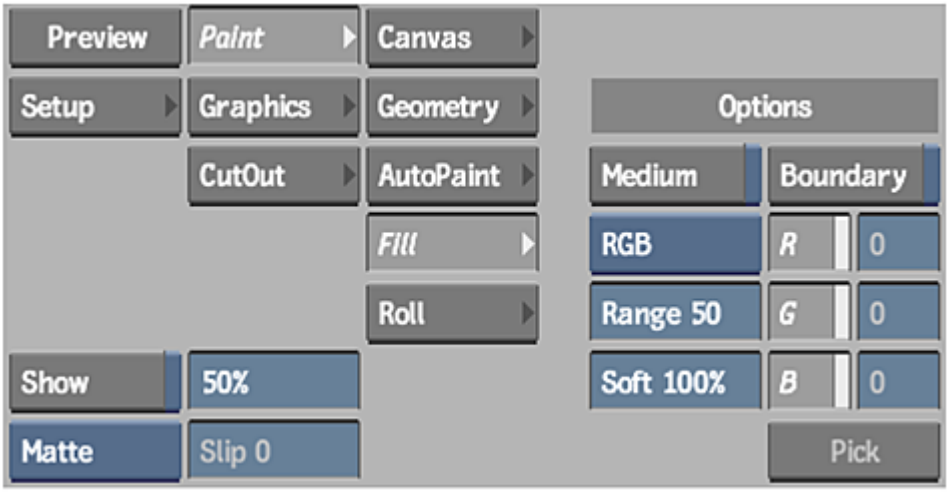
2. Select the colour model and channels you want to work with.
3. Set the range and softness in the Range and Softness fields.
4. Select the Special Effects medium you want to use (Clone, Reveal, Wash, or Shade). See Using Special Effects Media (page 1246).
5. Set the Reference box to Front, Back, Result, or Saved, and set the brush opacity.
   A value of 100% fills the area completely with the reference image.
6. Enable Medium.
7. Select a point in the area you want to fill.

Filling a Boundary

Use the Boundary option to fill an area of the canvas delimited by another colour.

To use boundary fill:

1. Click Fill.
   The Fill controls appear.



2 Select the colour model and channels you want to work with.

3 Set the range for the fill in the Range field.

4 Enable Boundary.

The Pick button is no longer greyed out.

5 Click Pick.

The cursor changes to a colour picker when dragged over the image.

6 Drag the colour picker over the canvas without clicking.

The various colour channel values in areas of the image are displayed as you move the colour picker around the canvas.

7 Click a point to select the colour for the boundary. You can also select a colour by entering the RGB values directly in the colour channel fields.

The cursor changes to a paint bucket.

8 Click the area inside the boundary to fill that region.

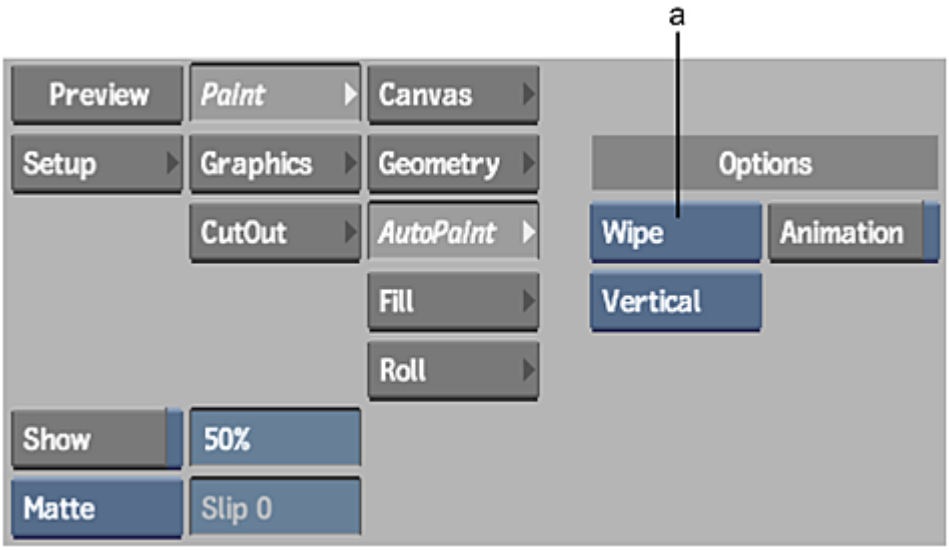
NOTE To fill a boundary with a reference image, set the Reference box to Front, Back, or Saved, and enable Medium.

Using AutoPaint

Use the AutoPaint controls to apply paint strokes to each frame in the result clip, a range of frames, or the current frame.

To display the AutoPaint controls:

1 Click AutoPaint in the Paint menu. If the controls are hidden by the colour palette, swipe the bottom of the screen.



(a) Autopaint Mode box

The AutoPaint Mode options are described as follows.

User Records and plays back a series of manual paint strokes.

Random Applies a number of random strokes to the image.

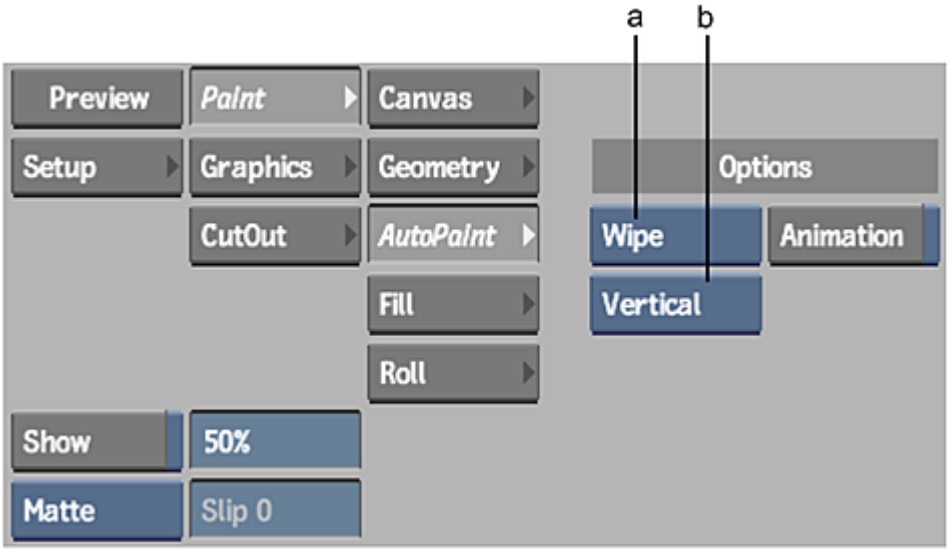
Wipe Wipes the entire canvas with the current colour or medium.

Wiping the Canvas in AutoPaint

Use Wipe mode to automatically apply paint strokes from left to right, top to bottom, or diagonally across the entire canvas. You can use a filter or any of the Special Effects media except Warp, Drag, and Smear. The size of the brush determines the number of strokes applied. Use a small brush to apply several strokes or a large brush to apply fewer strokes.

To wipe the canvas in AutoPaint:

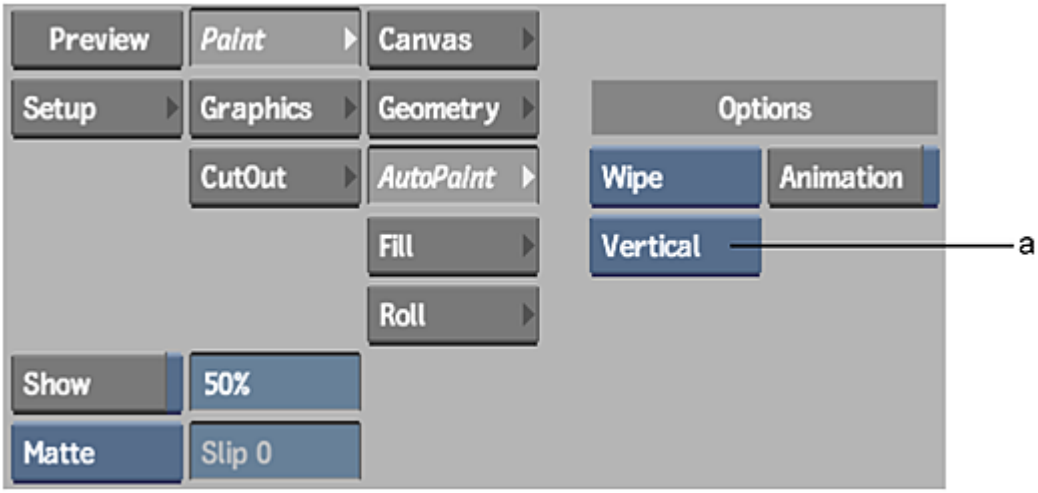
1. Click AutoPaint, and select Wipe from the AutoPaint Mode box.
   The Wipe menu appears.



(a) AutoPaint Mode box (b) Wipe Mode box

2. Set the current colour.
3. Select the Special Effects medium or filter, and set the brush attributes. See Using Special Effects Media (page 1246), and Brush Attributes (page 1239).
4. Select the Wipe mode in the Wipe Mode box. If this box is not visible, make sure that the attribute mode for the Direction attribute is set to Direction.

Select:	To:
Diagonal	Apply diagonal strokes.
Vertical	Apply strokes from top to bottom.
Horizontal	Apply strokes from left to right.



(a) Wipe Mode box

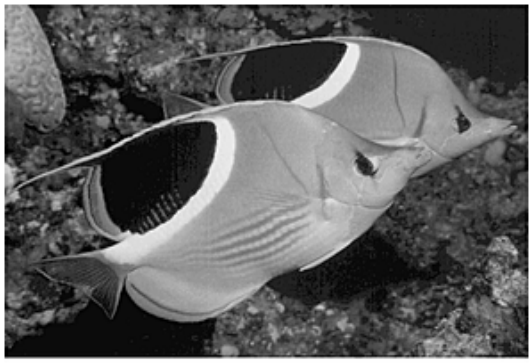
5 Enable Animation to display the Channel Editor.

If the Channel Editor is not visible, swipe the bar below the menu.

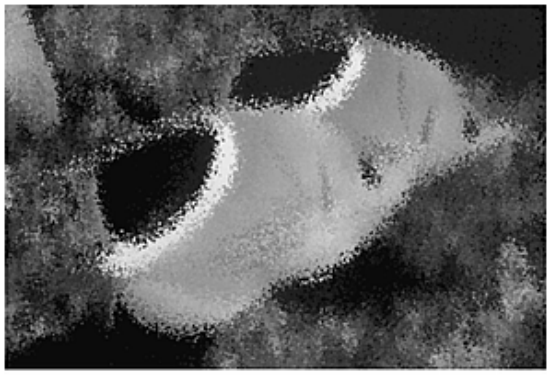
You can animate all brush attributes as well as the current colour.

6 Enable Wipe if you are using a filter or colour.

7 Click the image to apply the paint strokes only to the current frame. Click Render to apply the paint strokes to each frame in the front clip.



Original image



After using the Wipe command with Jitter attribute at 35% and Colour attribute mode set to Front

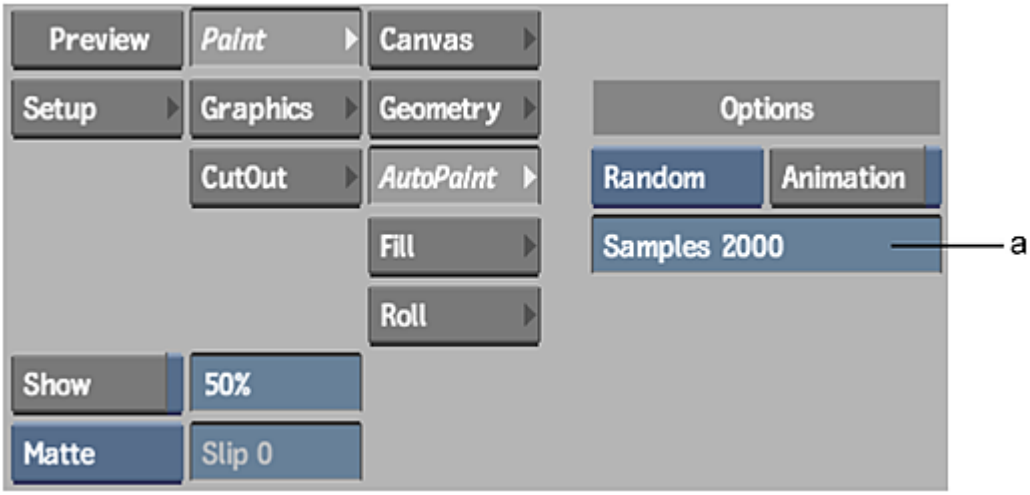
Applying Random Strokes

Use Random mode to automatically generate a number of paint strokes with random orientation on the image. Use the Samples field to specify the number of random strokes you want to generate. You can use any Special Effects media in Random mode except Warp and Drag.

To apply random paint strokes to the image:

1 Click AutoPaint and select Random from the AutoPaint Mode box.

The Samples field appears.



(a) Samples field

2 Enter the number of strokes you want to apply in the Samples field.

3 Set the current colour.

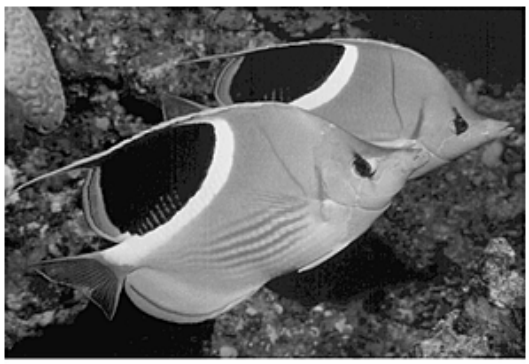
4 Select the Special Effects medium, and set the brush attributes. See Using Special Effects Media (page 1246), and Brush Attributes (page 1239).

5 Enable Animation to display the Channel Editor. If the Channel Editor is not visible, swipe the bar below the menu.

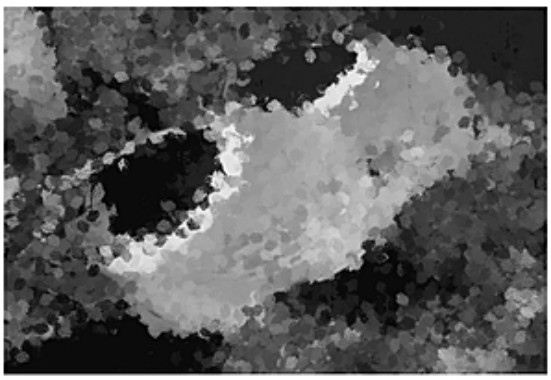
You can animate the following parameters:

- Number of strokes
- All the brush attributes
- Colour

6 Click the image to apply the paint strokes to the current frame only. Click Render to apply the paint strokes to each frame in the front clip.



Original image



After using the Random command with Jitter attribute at 30% and Colour attribute mode set to Front

Recording Brush Strokes

Use User mode to record and play back a series of paint strokes. Only the positions of the brush strokes are recorded. This means that you cannot change the brush type, current colour, or brush attributes while

recording the strokes. You can, however, change or animate these parameters after you finish recording and before you play back the strokes.

TIP You can also play paint strokes created by converting objects (write-ons).

To record a series of brush strokes:

1. Click AutoPaint and select User from the AutoPaint Mode box.
2. Set the brush characteristics so that you can see the recorded brush strokes on your image.

   NOTE The strokes you paint are recorded as a series of points or stamps of the brush. The number of strokes is determined by dividing the number of points by the number of frames. The quality of the rendered strokes depends on the number of recorded points. To record a larger number of points, select a small brush size.
3. Enable Record.
4. Begin painting on the image.

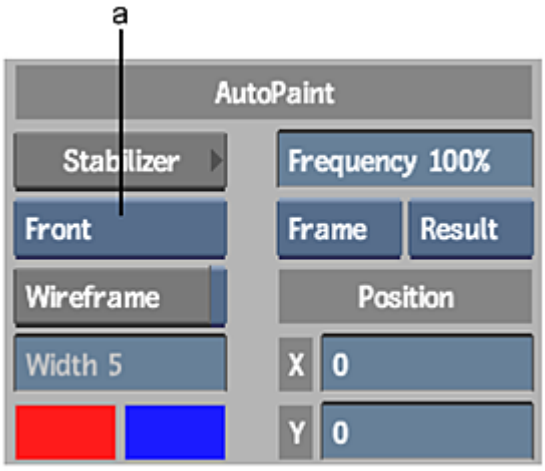
   The position of each brush stroke is recorded, and the stroke count appears in the message bar.
5. To stop recording, click below the timebar.

   All paint strokes applied to the image while recording are removed from the image and the number of strokes is recorded.

   NOTE AutoPaint stores only one set of recorded paint strokes at a time. If you record another set of paint strokes, you lose your previously recorded strokes.

Tracking with AutoPaint

You can apply tracking data to the painted strokes.



(a) Tracking box

To apply tracking data to recorded strokes:

1. Enable Animation.
2. Select Front, Back, or Result from the Tracking box and click Stabilizer to track a point and apply the offset information to the recorded strokes.

   NOTE You can only track after you record paint strokes.
3. Use the X and Y field to offset the painted strokes.

Playing Recorded Brush Strokes

After recording brush strokes, you can apply them to the front clip using the Play option, which is activated automatically when you finish recording the paint strokes.

To play recorded strokes:

1 Enable Play.

The Play options appear.

2 Define the brush characteristics using the Brush Attribute fields.

3 Define the duration of the stroke sequence using the second timebar. Drag the indicator to modify the duration.

(a) Indicator

The number of strokes is divided by the length of the stroke, defined by the second timebar.

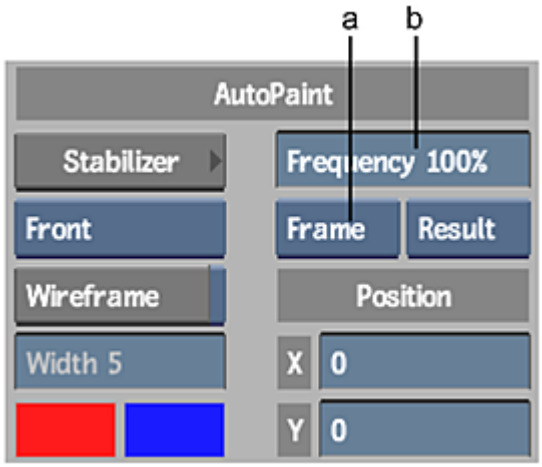
The indicator in the second timebar is identical to a track in the Channel Editor.

4 Enable the Play options.

Enable:	To:
Part	Play only a part of the recorded strokes. AutoPaint applies the strokes to a frame, erases those strokes, and moves to the next frame. This has the effect of creating streaks on your rendered clip.
Backward	Play the paint strokes backward.
Distance	Play paint strokes based on distance. AutoPaint divides the length of paint strokes by the number of frames to determine what to render in each frame.
Stroke	Play the paint strokes simultaneously.

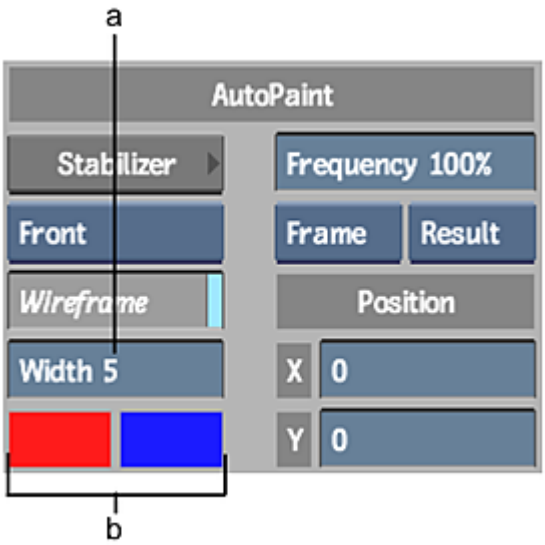
5 Enable Animation.

The Animation menu appears.



(a) Animation Type box (b) Frequency field

6 Enable Wireframe to preview the path(s) of the painted strokes.



(a) Width field (b) Wireframe colour pots

The wireframe preview shows the complete path(s) of the painted strokes. It also shows what will be painted in each frame as set using the second timebar.

You can set the width of the wireframe using the Width field. You can also change the colour of the path and progression wireframes using the colour pots beneath the Width field.

For Path animation, you define how often AutoPaint reads the Channel Editor values when rendering the points or stamps that make up a given stroke. At a frequency of 100%, AutoPaint reads the Channel Editor values the most frequently.

For example, assume you set your AutoPaint sequence of 100 stamps to run over 10 frames using Path animation. If you set the Frequency field to 100%, AutoPaint reads the Channel Editor values 10 times every frame. If you set the Frequency field to 50%, AutoPaint reads the Channel Editor values 5 times every frame.

TIP Use the Frequency field to lower the processing time for your AutoPaint sequence. Use a low frequency to render a quick preview of the result.

7 Swipe the bar at the bottom of the menu to display the Channel Editor.

You can animate the following parameters:

- Sampling amount
- X and Y values
- Brush size, rate, pressure, jitter, direction, roll, opacity, and colour
- Tracker translation

NOTE The Channel Editor only appears if Animation is enabled. Also, animation data in the Channel Editor is not erased when you record new strokes.

8 To apply the paint strokes, click Render.

NOTE If you are zoomed in on the image and are in the Raster zoom mode, only the visible portion of the image will be processed. This is much faster than using Tiled mode, but will not apply your modifications to the entire frame. For more information on zoom modes, see Painting on Full-Resolution Film Images (page 1232).

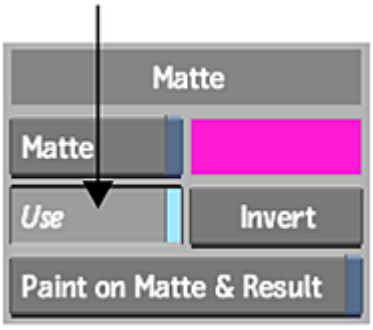
About Mattes

Mattes protect specific areas of the image when you apply paint, filters, or Special Effects media to the canvas. You can also use mattes to limit the area of a cutout.

For instructions on loading mattes into Paint, see Loading Clips into Paint (page 1225), and Loading Setups (page 1311).

To use the matte:

1 Click Use to enable the matte.



2 Paint on the image.

3 Click Use again to disable the matte.

Using Mattes with Cutouts

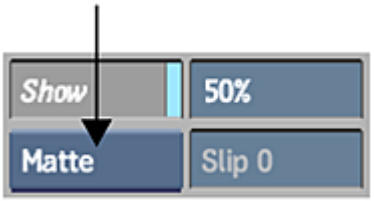
If you enable your matte when creating a cutout, the cutout is limited by the matte. Only objects outside the matte will show up in the cutout. See Creating a Cutout (page 1301).

Viewing Mattes

You can view the matte as you use it.

To view the matte:

1. Load the matte into Paint.
2. Enable Show.
3. Select Matte in the Reference box.



The matte appears.

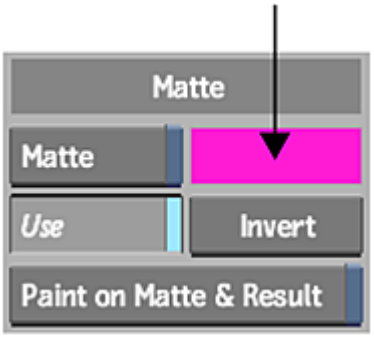
Changing the Matte Colour

If the colour used to display the matte blends with the image, you can change its colour.

NOTE The matte is always created using shades of grey.

To change the matte display colour:

1. Set the current colour.
2. Click the Matte colour pot.



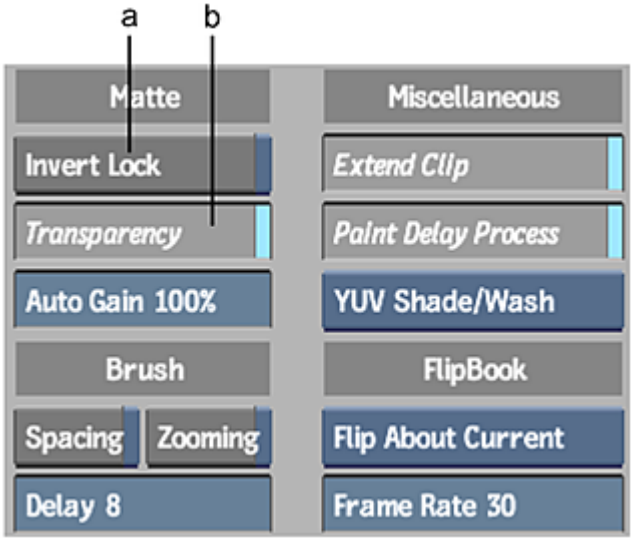
The matte colour display changes to the current colour.

Changing the Matte Display Transparency

When viewing the matte, use the Transparency field to vary the matte display transparency from 0% (completely transparent) to 100% (fully opaque). Press `Alt+T` or click Transparency in the Setup menu to toggle between the last set value and 100%.

(a) Transparency field

NOTE The transparency value only affects the display of the matte. It does not affect how the matte is used.



(a) Invert Lock button (b) Transparency button

Inverting the Matte Clip

Click Invert to invert the matte in the current frame. Enable Invert Lock in the Setup menu to invert the matte in every frame of the clip.

Creating or Modifying Mattes

You can create or modify a matte by painting directly on the matte using any brush, Special Effects media, or graphic tool. For information on using graphic and cutout tools to create mattes, see Using the Cutout Commands (page 1302).

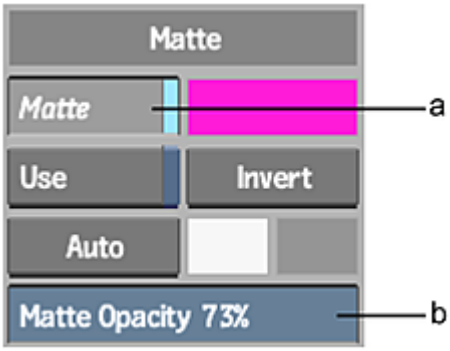
To create or modify a matte:

1. Load the matte into Paint.

   NOTE You must load a matte clip in order to process a matte in Paint. If you want to create a matte from scratch, load a black source as the matte clip.

2. Click Matte.

   The Matte menu appears.



(a) Matte button (b) Matte Opacity field

3 Set the paint transparency in the Matte Opacity field.

The value in the Matte Opacity field determines the transparency of the paint applied to the matte. Use a value of 100 to apply fully opaque paint or objects to the matte. Reduce the Matte Opacity value to increase the transparency of the paint.

4 Paint or place graphics or cutouts on the matte.

The paint and graphics are applied to the matte. If you use graphic tools, use Tack to tack the graphic to the matte. See Tacking Down Selected Objects (page 1300).

5 You can save the matte in the matte library. See Saving Setups (page 1310).

6 To process the matte clip, click Exit.

The new matte clip is saved to the rendered destination.

Painting on the Matte and Image

When applying paint strokes or graphics to the image, enable Both to paint the image and the matte simultaneously.

Resetting the Matte

If you do not load a matte, Paint displays the last matte that was loaded.

To erase the matte:

1 Click Matte.

2 Set the Matte Opacity to 0%.

3 Click Wipe.

The matte is erased.

Creating a Gradient Matte

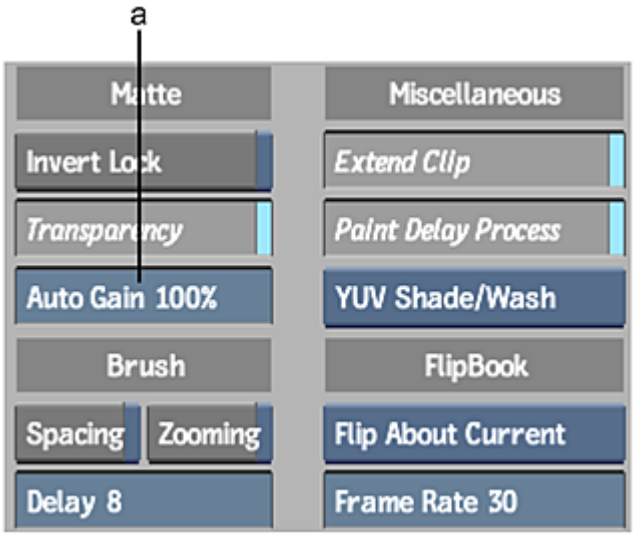
You can use graphics that contain gradients to create a matte that blends from one level of the matte colour to another level. A gradient matte is useful for blending paint strokes or creating soft edges.

To create a gradient matte, draw a graphic with a gradient on the matte, and then use the Channel Editor to change the "a" channel of the start or end colour of the gradient. See Creating Graphics (page 1274), and Changing the Object Gradient (page 1288).

Creating High-Contrast Mattes

Use the Auto command to create a high-contrast matte. This command works like the Auto Matte command in the Processing menu. The minimum and maximum luminance values for the matte are set using the two colour pots in the AutoMatte menu. Any pixel with a luminance value below the minimum is set to black, and any pixel with a luminance value above the maximum is set to white.

Unwanted grey areas can be removed from the matte by increasing the percentage in the Auto Gain field in the Setup menu.

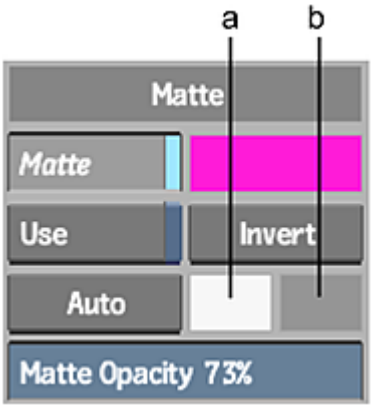


(a) Auto Gain field

Pixel values between the minimum and maximum values are multiplied by the Gain value. The resulting values are clipped at the specified maximum luminance value. The Gain is expressed as a percentage value. The default value of 100% has no effect on the image since the luminance values are multiplied by 1.

To use the AutoMatte command:

1. In the Paint menu, click Matte.
   The AutoMatte menu appears.



(a) Minimum luminance (b) Maximum luminance

2. Set the minimum luminance value for the matte in the colour pot on the left. To set the value, click the field and use the colour picker to select the lightest colour from the image.
3. Set the maximum luminance value for the matte in the colour pot on the right.

4 Set the Auto Gain for the matte. The Auto Gain field is in the Setup menu.

5 Click Auto to generate the matte.

6 Click Use to enable the generated matte.

Using Graphics in Paint

Accessing the Graphics Menu

To access the Graphics menu, click the Graphics button in the Paint menu. Some options may be hidden if the colour palette is open. Swipe the cursor across the bars at the bottom of the screen to hide the colour palette.

Creating Graphics

Graphics can be used to create cutouts or mattes, or to apply geometrical shapes, text, and fills to the result clip.

To create a graphics object:

1 In the Paint menu, click Graphics.

2 Click Add.

3 Select the type of object to add by selecting an icon in the Object Tools window.

4 Draw the object on the image. See Using the Object Tools (page 1275).

5 Set the resolution to be used when adding or editing objects. See Setting the Object Resolution and Display (page 1285). Objects appear at full resolution when they are drawn.

6 Set the object's attributes and gradient. See Setting the Object Attribute (page 1287), and Changing the Object Gradient (page 1288).

7 Set the object's colour in the Current Colour pot. See Selecting Colours (page 1229).

8 Resize and move the object. See Changing the Size or Position of an Object (page 1290).

9 Use the Transformation box or controls to move, rotate, or scale the object. See Changing the Shape of an Object (page 1291).

10 Use the Animation controls to animate the object. See Animating Graphics (page 1295).

11 Click the Tack button to permanently place the object on the result clip.

Once the object has been tacked down, it cannot be moved, rotated, deleted, resized, or copied. See Tacking Down Objects (page 1299).

Auto Edit After Add

After you add an object to the image, Paint automatically switches from Add mode to Edit mode. You can turn off this default setting by disabling the Auto Edit After Add button in the Setup menu. You can then add objects one after another without interruption.

(a) Auto Edit After Add button

Using the Object Tools

A number of predefined graphic tools are available in Paint. The cursor looks the same for all graphic types. When you move the cursor onto an image, it appears as a small green cross with crosshairs that extend the width and height of the canvas. The crosshairs help align objects on the image.

Each tool has its own icon in the Object Tools window.

To select an object tool:

1 Scroll through the Object Tools window.

Click the Object Tools window and drag left or right. Use the left mouse button to scroll slowly, the middle button to scroll faster, and the right button to scroll the fastest.

2 Click the object tool icon you want to use.

The selected object tool is highlighted by a blue outline.

NOTE Only one object tool can be active at a time.

Drawing a Line

Use the Line object tool to draw a single straight line or a multi-line object (a series of lines joined end-to-end).

To draw a line or a series of lines:

1. In the Graphics menu, click Add.
2. Select the Line tool in the Object Tools window.
   The Create Multiple Lines button appears.
3. Click Create Multiple Lines.
4. To draw multiple lines, move to the canvas and click to place the start point of the first line. Click again to draw the end point. Continue clicking to draw more lines.
5. To end a multiple line object, click Create Multiple Lines.
6. To draw single lines, move to the canvas and click, drag, and release.
   Paint draws the line as you drag.

Drawing a Rectangle

Use the Rectangle object tool to draw a rectangle or square in one of two ways:

- Corner-to-corner, with the two vertices located at diagonally opposite corners of the rectangle
- Centre-to-corner, with the first vertex at the centre of the rectangle and the second at one corner

To draw a rectangle from corner-to-corner:

1. In the Graphics menu, click Add.
2. Select the Rectangle tool in the Object Tools window.
3. Press the cursor to anchor one corner of the rectangle. Do not release the cursor.
4. Drag the cursor diagonally. You can adjust the width and height of the rectangle as long as you hold down the cursor.
5. Release the cursor to anchor the second vertex.
   The rectangle is drawn on the image.

To draw a rectangle from centre-to-corner:

1. In the Graphics menu, click Add.
2. Select the Rectangle tool in the Object Tools window.
3. Press `Alt`.
4. Position the cursor where you want to place the centre of the rectangle and press down.
5. Drag diagonally, and release the cursor when the rectangle is the correct size.

Drawing a Square

Press `P` while you draw and repeat the procedure for drawing a rectangle from corner-to-corner.

Drawing a Triangle

Use the Triangle object tool to draw a triangle or equilateral triangle. The vertices are located at the three corners of the triangle.

To draw a triangle:

1. In the Graphics menu, click Add.
2. Select the Triangle tool in the Object Tools window.
3. Position the cursor and press to anchor the first vertex on the image. Do not release the cursor.
4. Drag the cursor horizontally to draw the base of the triangle. You can continue to adjust the length of the base as long as you press down on the cursor.
5. Release the cursor to anchor the second vertex.
6. Move the cursor to where you want to place the third vertex. You can continue to adjust the position of the third vertex until you press down on the cursor. Press to anchor the third vertex.

   The triangle is drawn on the image.

Drawing an Equilateral Triangle

An equilateral triangle has three sides of equal length. To draw an equilateral triangle, repeat the procedure for drawing a triangle and press `P` while drawing the base of the triangle. This also establishes the height of the triangle. Release the cursor to anchor the second and third vertices on the image.

Drawing an Ellipse

Use the Ellipse object tool to draw an ellipse defined by three vertices. The first vertex determines the centre of the ellipse. The second vertex determines the horizontal radius of the ellipse. The third vertex determines the vertical radius.

To draw an ellipse:

1. In the Graphics menu, click Add.
2. Select the Ellipse tool in the Object Tools window.
3. Position the cursor at the centre of the ellipse and press to anchor the point on the image. Do not release the cursor.
4. Drag the cursor horizontally to establish the width of the ellipse. You can continue to adjust the width as long as you press down on the cursor.
5. Release the cursor to anchor the second vertex.
6. Move the cursor along the vertical axis to position the third vertex. You can continue to adjust the height of the ellipse until you press down on the cursor. Press to anchor the third vertex.

   The ellipse is drawn on the image.

To draw a circle with the Ellipse tool:

1. In the Graphics menu, click Add and select the Ellipse tool.
2. Press `P`.
3. Position the cursor at the centre of the circle and press to anchor the point on the image. Do not release the cursor.

4 Drag the cursor horizontally to establish the radius of the circle. Notice that there are two vertices at the edge of the circle, as there are for an ellipse.

5 Release the cursor and the `P` key to anchor the second and third vertices.

The circle is drawn on the image.

Drawing a Circle

Use the Circle object tool to draw a circle in one of two ways:

- Centre-to-edge, with one vertex at the centre of the circle and a second on the circumference
- Edge-to-edge, with two vertices at opposite points on the circumference

To draw a circle from edge-to-edge:

1 In the Graphics menu, click Add.

2 Select the Circle tool in the Object Tools window.

3 Position the cursor at the centre of the circle and press to anchor that point on the image. Do not release the cursor.

4 Drag the cursor to establish the radius of the circle. You can continue to adjust the size of the circle as long as you press down on the cursor.

5 Release the cursor to anchor the second vertex.

The circle is drawn on the image.

To draw a circle from centre-to-edge:

1 In the Graphics menu, click Add and then select the Circle tool.

2 Press and hold `Alt`.

3 Drag the cursor to establish the radius of the circle.

When the circle is the required size, release the cursor.

Drawing a Polygon

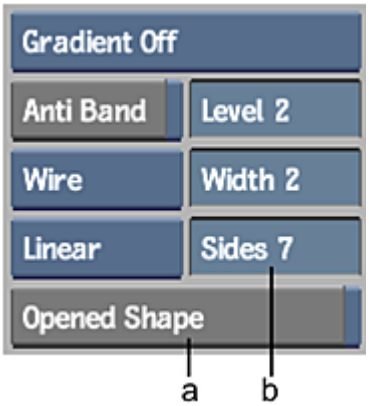
Use the Polygon object tool to create a series of connected lines that form a closed or open object. You can specify how the vertices of the polygon should be connected by selecting one of the options from the Polygon Mode box.

To draw a polygon:

1 In the Graphics menu, click Add.

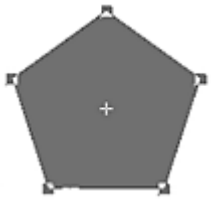
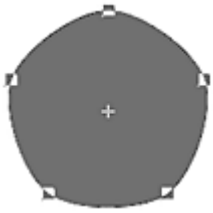
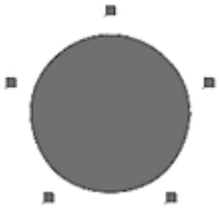
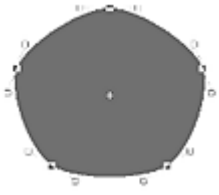
2 Select the Polygon tool in the Object Tools window.

The Polygon menu appears.



(a) Polygon Mode box (b) Sides field

3 Select an option from the Polygon Mode box to set the type of curve used to join the vertices of the polygon.

Select:		To:
Linear		Use straight lines to join the vertices of the polygon.
Cardinal		Use a smooth curve that passes through the vertices of the polygon.
Bspline		Use a very smooth curve that passes on the inner side of the vertices of the polygon.
Bezier		Use Bezier curves. Each vertex of the polygon has a tangent with two tangent handles. In Edit mode, you can move the tangent handles to adjust the slope of the polygon. See Editing a Bezier Curve (page 1292).

The minimum number of vertices for a polygon depends on the type of interpolation used. A polygon should have at least three vertices when using Linear interpolation, or four when using Cardinal or Bspline interpolation. The maximum number of vertices is 255. A polygon can be concave.

4 To create an open-ended polygon, click Opened Shape.

5 Position the cursor at one vertex of the polygon, and click to anchor that point on the image.

6 Release the cursor and move it to where you want the next vertex of the polygon. You can continue to adjust the positioning of the next vertex until you press down on the cursor. Press to anchor the point on the image.

7 Repeat steps 4 and 5 for each vertex.

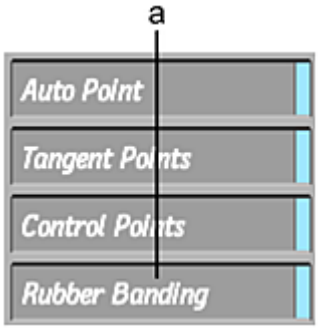
8 To close the polygon, press anywhere outside the canvas.

The polygon is drawn on the image.

Rubber Banding

Rubber banding is the default setting for drawing polygons and lines. This means that a vertex is added to the polygon only when you press and release the cursor.

To draw a polygon with multiple vertices, disable the Rubber Banding button in the Setup menu. When you drag the cursor while drawing a polygon or line, vertices are drawn on the image.



(a) Rubber Banding button

Drawing Regular Polygons

Draw regular polygons using the Sides field and the P key. A polygon can have three or more sides. This enables you to draw a triangle or a square based on the centre point of the object.

To draw a regular polygon:

1 In the Graphics menu, click Add.

2 Select the Polygon tool in the Object Tools window.

3 Enter the number of sides for the polygon in the Sides field.

4 Press P.

5 Position the cursor over the image and press down.

This is the centre of the polygon.

6 Drag the cursor away from the centre point. The polygon is drawn on the image. You can continue to adjust the size of the polygon as long as you press down on the cursor.

7 When the polygon is the correct size, release the cursor.

The polygon is drawn on the image.

Drawing a Fill Object

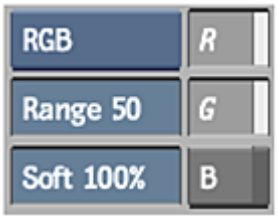
Use the Fill object tool to fill areas of an image with colour. While this is similar to using the Fill command in the Paint menu, you can only use this tool with colour, Shade, and Wash, and not to fill areas with a reference image. However, you can edit fill objects after you draw them.

NOTE A fill object cannot have a gradient.

To draw a fill object:

1. In the Graphics menu, click Add.
2. Select the Fill object in the Object Tools window.

   The Fill object controls appear.



3. Select a colour model from the Colour Model box (RGB or YUV).
4. Select the colour channels you want to use (R, G, and B, or Y, U, and V).

   The choice of colour channels determines how Paint evaluates the area you want to fill. For example, if you select R, G, and B, Paint fills areas whose adjacent pixels have red, green, and blue values in the specified range. If you select only R, Paint fills only areas whose adjacent pixels have red values in the specified range.
5. Enter the Range and Softness values for the fill.

   The range determines how far the fill spreads from the point you click on the image; if you increase the range, you increase the fill area. The softness determines how much diffusion occurs at the edges of the fill; if you increase the softness, you increase the diffusion at the edges of the fill.
6. Move the cursor to the canvas and click inside the area to fill.

   Paint fills the area with the current colour starting at the point you click.

   NOTE The colour of the image and the channel, range, and softness settings determine the extent of the fill. As you move the fill object, it changes size and shape in response to the different colours in the image.

Drawing a Text Object

Use the Text object tool to add text to the image. You can change the size, kerning, and inclination of the text string.

To add a text object:

1. In the Graphics menu, click Add.
2. Select the Text tool in the Object Tools window.

   The Text object controls appear.

(a) Text field (b) Font field

3 Click the Font field and select a new font from the font library.

4 Click the Text field, type the text string, and click Enter.

5 Click the image to place the text object.

You can adjust the size, kerning, and italics of the text in Edit mode.

Use:	To:
Size	Adjust the size (in pixels) of the text.
Italic	Incline the text. Enter a positive value to slant the text to the right, and a negative value to slant it to the left.
Kern	Adjust the space (in pixels) between all letters in the text.

Selecting Objects

You must select an object before you can edit it. In Edit mode, you can select objects in three ways:

- Click an object to select it.
- Use the Selected field to select individual objects.
- Use the All command to select all objects at once.

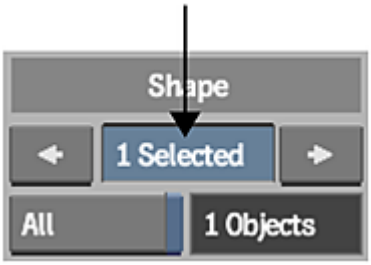
Using the Selected Field

As you add objects to the image, they are numbered sequentially, starting at 1. You can select an object by specifying the number of the object in the Selected field.

As you change the number in the Selected field, the corresponding object is highlighted by a selection box.

To select a single object using the Selected field:

1 Set the number in the Selected field to the number of the object you want to edit.



You can also click the < and > buttons beside the Selected field to move through the sequence.

2 When the selection box outlines the object, release the cursor. You can now edit it.

Selecting More than One Object

To select an additional object, hold down `Shift` and click another object. Repeat this step for each object you want to select.

Deselecting Objects

To deselect an object or objects, select a different object or click the Add button.

Selecting All Objects

To select all the objects, click the All button. All is shown in the Selected field. Click it again to deselect them.

Editing Objects

Use the object controls to copy, delete, hide, or layer objects.

Copying Objects

Use the Copy command to create multiple copies of an object. Once a copy is created, it can be selected and edited like any other object.

To copy an object:

1 In the Graphics menu, click Copy.

2 Click the object you want to copy. Do not release the cursor.

A copy of the object is superimposed on the original image. You cannot see the copy until you drag it to a new location.

3 Drag the copy to a new location.

4 When the copy is positioned correctly on the image, release the cursor.

Deleting Objects

Use the Delete command to remove one or more objects from the image.

To delete one or more objects:

1. Select the objects you want to delete. Click an individual object or use the Selected box to select an object. To select multiple objects, click an object, press `Shift`, and click any other objects you want to delete.
2. Click Delete.
   All selected objects are removed from the image.

To delete all objects from the image:

1. Enable All.
2. Click Delete.
   All objects are deleted from the image.

Hiding Objects

Use the Hide/Unhide commands to hide or unhide objects before they are tacked onto the image.

To hide or unhide one or more objects:

1. Select the objects you want to hide. Click an individual object or use the Selected box to select an object. To select multiple objects, click an object, press `Shift`, and click any other objects you want to hide.
2. In the Graphics menu, click Edit and then click Hiding.
   The Hide options appear.

Select:	To:
Hide Selected	Hide all selected objects.
Hide Unselected	Hide all objects that are not selected.
Unhide All	Show all hidden objects.

Changing the Order of Overlapping Objects

Objects can be drawn so they overlap other objects to produce a stack of objects.

To send an object to the bottom of the stack:

1. Select the object.
2. In the Graphics menu, click Push and then click the object until it is at the bottom of the stack.

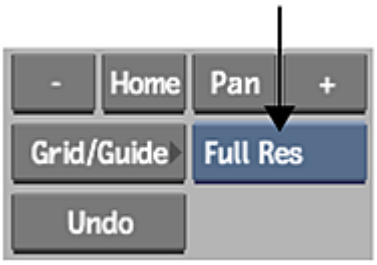
   NOTE To bring the object back to the front, click the object until it moves to the front.

Setting the Object Resolution and Display

Use the Object Resolution box to set the resolution while drawing or editing objects. The objects are regenerated at high resolution once you release the cursor at the end of each editing operation. Use the options in the Setup menu to affect the display of the graphics before they are tacked down onto the image.

To set the object resolution:

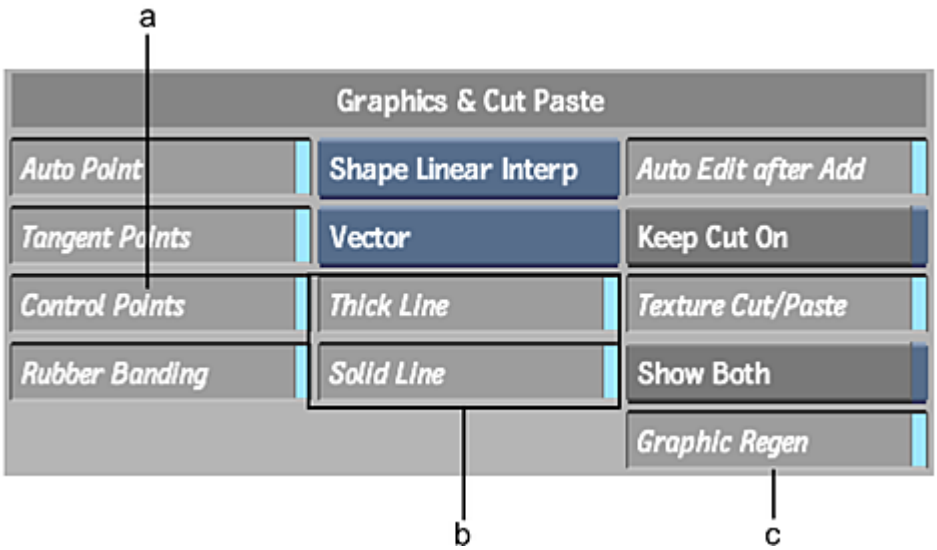
1 Select an option from the Object Resolution box.



Select:	To:
Full Res	Display the object at full resolution when updating. This is helpful for displaying gradients, but slows down the rate at which image display is refreshed.
Low Res	Display the object at low resolution when updating.
Wireframe	Display the object as a wireframe when updating.

Wireframe Display Options

The Thick Line and Solid Line buttons in the Setup menu determine the thickness and continuity of the wireframe.



(a) Control Points button (b) Thick Line and Solid Line buttons (c) Graphic Regen button

Enabling and disabling the Thick Line and Solid Line buttons affects the wireframe as follows:

- When Thick Line is enabled, the wireframe has a thickness of 2 pixels.
- When Thick Line is disabled, the wireframe has a thickness of 1 pixel.
- When Solid Line is enabled, the wireframe is unbroken.
- When Solid Line is disabled, the wireframe is dashed.

Increasing Editing Speed

To speed up editing, disable the Graphic Regen button in the Setup menu. Instead of waiting for the graphics objects to be regenerated at the end of an editing operation, the objects appear at the display resolution. This can be a useful time-saving feature.

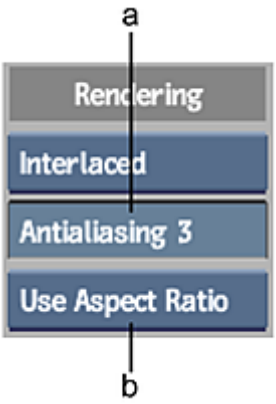
Hiding the Object Vertices

Each object is defined by two or more vertices. These vertices, or tangent handles, appear while you are drawing or editing the object.

Disable the Control Points button in the Setup menu to turn off the display of the vertices while you are drawing or editing the object. This feature is useful if you want to trace the outline of a figure.

Displaying Anamorphic Geometry

Select Use Aspect Ratio in the Rendering area of the Setup menu to draw and display anamorphic geometry. For normal display, click Use Aspect Ratio and switch to Use Square Pixels.



(a) Anti-alias field (b) Use Aspect Ratio option

Anti-Aliasing for Objects

The jagged edges that can occur along diagonal and curved lines in geometry are caused by aliasing, or insufficient spatial sampling of the image. The process of minimizing jagged edges by increasing the sampling rate is called *anti-aliasing*.

Use the Anti-alias field in the Rendering Setup menu to specify an anti-aliasing factor for geometry. When the value is set to 1, no anti-aliasing occurs. Best results are obtained with a value of 4.

Setting the Object Attribute

Use the Object Attribute box to set the display mode for the object you add to the image.

To change the object attribute:

1. In the Graphics menu, click Edit.
2. Select the object you want to edit.
3. Click Attributes.
4. Select an option from the Object Attribute box.

Select:	To:
Solid	Draw a filled object with a well-defined edge. Use the Brush Opacity field to set the object's transparency. No other brush attributes or types are available. Line objects cannot be set to Solid.
Outline	Draw an outline of the object. Use the Brush Attribute buttons to set the appearance of the outline. See Changing the Object's Appearance (page 1287).
Fuzzy	Draw a solid object with a soft edge. Use the Brush Attribute fields to set the appearance of the outline.
Wire	Draw a wireframe object. Use the Brush Opacity field to set the object's transparency. No other brush attributes or types are available.

Changing the Object's Appearance

You can change the appearance of an object by changing any of the following display attributes:

- Current colour
- Brush attribute values or modes
- Special Effects media or filters
- Colour gradient

Changing display attributes affects all currently selected objects. See Selecting Objects (page 1282).

Setting Brush Attributes and Modes

In the Graphics menu, you can only change brush attributes and modes when using the Outline or Fuzzy attribute. See Brush Attributes (page 1239).

If the Brush Attribute controls are not visible, swipe the bar at the bottom of the menu. Use the Current Colour pot to set the graphic's colour, and the Opacity field to set the opacity of the object, or the effect a graphic has on the image.

Using Filters and Special Effects Media with Graphics

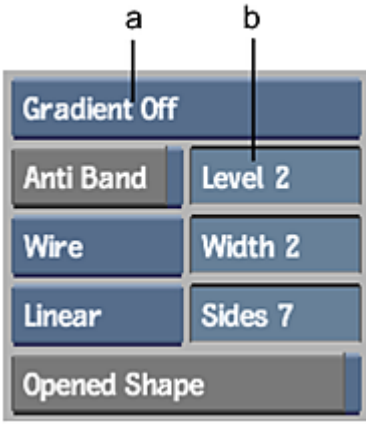
Any filter can be used with objects. Only the Paint, Clone, Reveal, Wash, and Shade Special Effects media can be used. See Using Special Effects Media (page 1246).

Changing the Object Gradient

All objects, except the fill object, can have a gradient. You can edit a gradient or apply a gradient to an object that does not have one. A selected object with a gradient displays a gradient control bar that you can use to edit the orientation of the gradient.

To edit a colour gradient:

1. In the Graphics menu, click Edit.
2. Select the object you want to edit.
3. Click Attributes.
4. Select an option from the Gradient box.



(a) Gradient box (b) Noise Level field

Select:	To:
Rectangular Gradient	Use a rectangular gradient in which colour changes from top to bottom.
Circular Gradient	Use a circular gradient in which colour changes from centre to edge.
Gradient Off	Turn off the gradient and use a solid colour.

5 Set the colours for the gradient. See Using the Colour Gradient Bar (page 1230).

6 To eliminate banding, enable Anti Band.

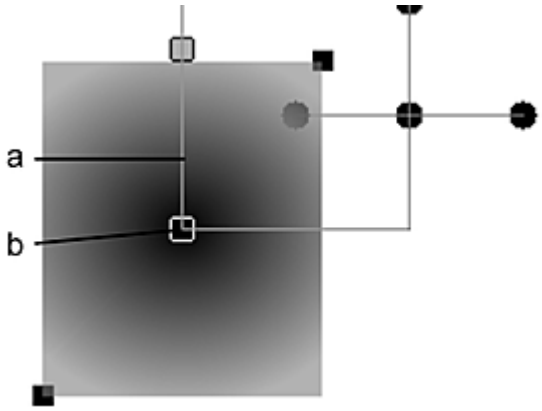
Banding normally appears only if you are working at 8-bit colour depth and if the gradient that you apply is stretched.

TIP You can also set a noise level to the anti-band function. Noise may help when colour bands are very large. Set the value in the Noise Level field to 7 to apply the maximum noise, or lower it to apply less dither/noise. If you set the value to 0, no noise is applied.

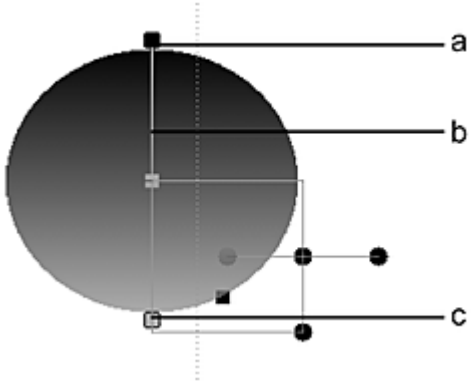
7 Use the gradient control bar to change the object's gradient.

Using the Gradient Control Bar

When you select an object with a gradient, Paint displays a gradient control bar in addition to the object transformation box. The gradient control bar has two handles at each end that show the colours of the gradient. In a rectangular gradient, the bar also indicates the direction of the gradient.



(a) Gradient control bar (b) Centre point of gradient



(a) Start point of the gradient (b) Gradient control bar (c) End point of the gradient

NOTE If you want to move the transformation box, press M and click the location where you want to move the box.

To use the gradient control bar:

1 Select an object with a gradient, or add a gradient to an object.

Paint displays the transformation box and the gradient bar.

2 Move the handles of the gradient control bar to change the orientation and location of the gradient.

In a rectangular gradient, this affects the gradient orientation. For example, the default orientation for a rectangular gradient is top to bottom. If you drag one of the handles left or right, you change the orientation to diagonal. If you drag the handles so the gradient bar is horizontal, you get a rectangular gradient with the colours blended from side to side.

If you select the centre colour handle in a circular gradient, you can move the gradient's centre inside the object or even outside it.

3 To decrease the amount of one colour visible in an object, move the handle to the edge or outside the object.

Changing the Size or Position of an Object

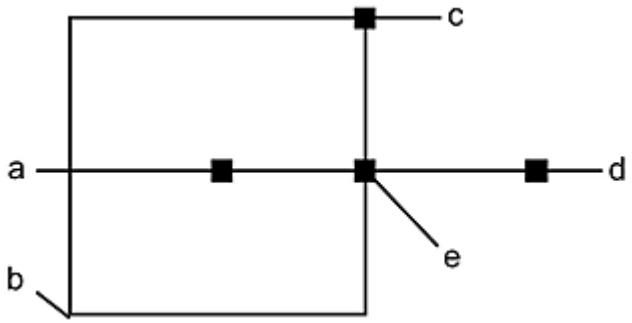
Use the transformation box, or the transformation controls in the Axis Transformation menu, to change the size or position of an object. You can use the transformation box and controls to modify the object in the following ways:

- Move the object in any direction on the image.
- Rotate the object about its centre point.
- Change the size of the object.

You can translate, resize, or rotate more than one object at the same time by selecting all the objects you want to edit.

Using the Transformation Box

To display the transformation box, click the Edit button in the Graphics menu and click the object you want to edit.



(a) Reset handle (b) Centre point (c) Resize handle (d) Rotation handle (e) Translation handle

The position of the transformation box is saved for each object. To move the transformation box, press M and click the destination.

Resize handle To change the size of an object, drag the resize handle in any direction. To maintain an object's proportions, press PE as you drag the resize handle.

Centre point To change the point around which the object rotates, move the centre point.

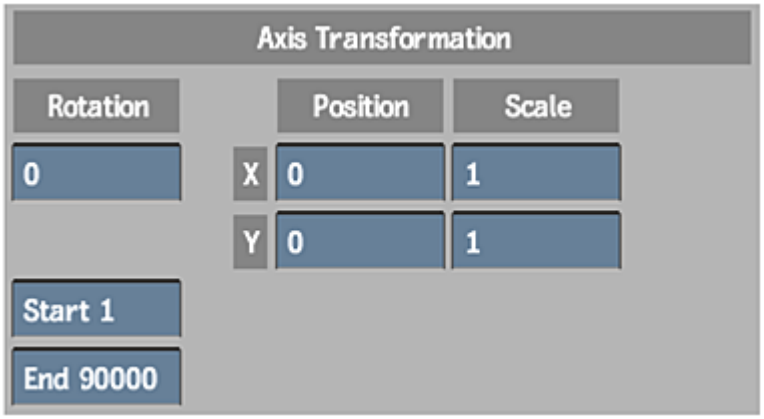
Rotation handle To rotate the object about its centre point, drag the rotation handle.

Translation handle To move the object in any direction on the image, drag the translation handle. The transformation box moves with the object.

Reset handle To restore the object to its initial size and position on the image, click the reset handle.

Using the Transformation Controls

Use the transformation controls to translate, rotate, and resize a selected object. To display the transformation controls, select an object and click the Transform button.



X (Position) Move the object along the horizontal axis (X-axis).

Y (Position) Move the object along the vertical axis (Y-axis).

X (Scale) Change the size of the object along the horizontal axis.

Y (Scale) Change the size of the object along the vertical axis.

Rotation Set the object's angle of rotation along the Z-axis. The object is always rotated around its centre point.

Reset Use the Reset button to reset the object to its initial size and position.

Start and End Numeric Fields Use these fields to set when the object appears in the clip.

Changing the Shape of an Object

To change the shape of an object, such as a polygon or a line, you must add, delete, or move a vertex on the object.

To change the shape of an object:

1. In the Graphics menu, click Edit.
2. Select the object you want to edit.
   The object vertices and the transformation box appear on the selected object.
3. Press the vertex you want to edit and drag it to a new location. When you are satisfied with the new position, release the cursor.

Adding and Deleting Vertices on a Polygon or Line

Change the shape of a polygon or line by changing its number of vertices.

To add a vertex to a polygon or line:

1. In the Graphics menu, click Edit.

2 Select the polygon or line you want to edit.

   The vertices for that object are selected.

3 Press A.

4 Press on one of the existing vertices and drag outward. A vertex is added counterclockwise to the selected point.

   To add a point clockwise to the selected point, press Ctrl+A.

5 Repeat steps 3 and 4 for each vertex you want to add to the object.

To delete a vertex from a polygon or line:

1 In the Graphics menu, click Edit.

2 Select the polygon or line you want to edit.

   The vertices for that object are selected.

3 Press D.

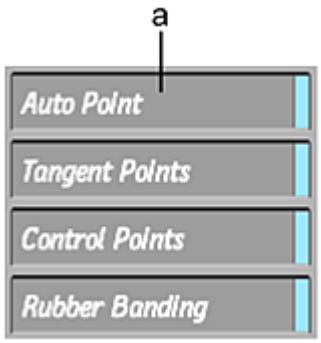
4 Click the vertex you want to delete.

   The selected vertex is deleted from the object.

5 Repeat steps 3 and 4 for each vertex you want to delete on the object.

The Auto Point Command

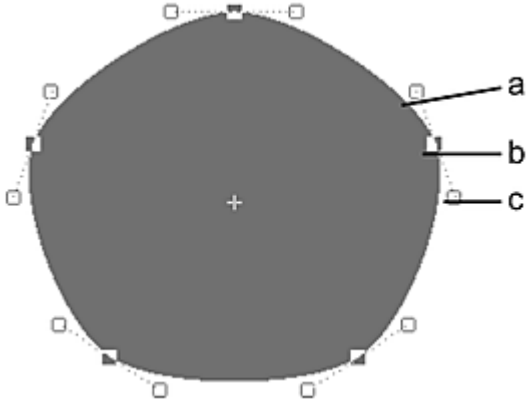
When you add or delete a vertex on a polygon or line, the same point is added or deleted on all shapes for that object. You can disable this option in the Setup menu. When the Auto Point button is disabled, any point you add or delete on a shape is added/deleted in the current shape key only.



(a) Auto Point button

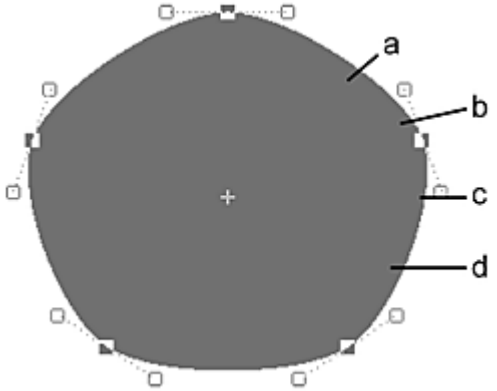
Editing a Bezier Curve

When you use the Bezier option to draw a polygon, each vertex of the polygon has two tangents. Each tangent ends with a handle.

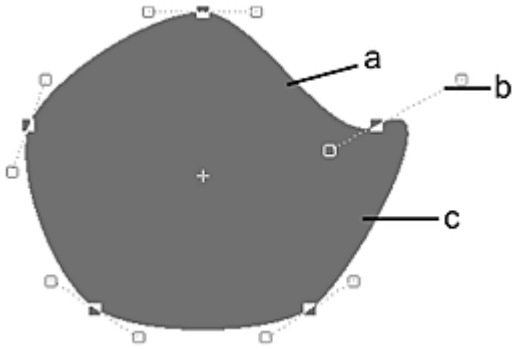


(a) Tangent (b) Vertex (c) Handle

Use Edit mode to move a tangent handle and adjust the slope of the adjacent side of the polygon. When you move one tangent handle, the tangent's reciprocal handle moves in the opposite direction because handles A and B are joined to keep the joint at the intersection of the two edges smooth.

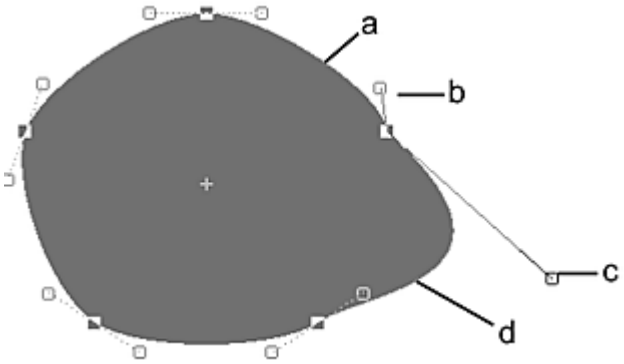


(a) Edge A (b) Tangent A (c) Handle B (d) Edge B



(a) Edge A (b) Handle B (c) Edge B

To adjust the slope on one side of the vertex only, press `B` and click the vertex. This breaks the tangent so that you can move its handles independently. To reset the tangent, press `B` and click the vertex again.



(a) Edge A (b) Handle A (c) Handle B (d) Edge B

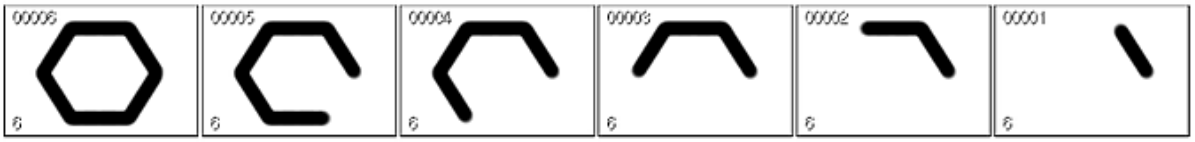
Converting Graphics to AutoPaint Strokes

Use the Convert command to convert objects to AutoPaint strokes.

(a) Convert button

You can play the paint strokes using the AutoPaint controls in the Paint menu. See Using AutoPaint (page 1262).

The following shows how a hexagon is drawn over six frames. One edge is drawn in each frame. After the object is converted to AutoPaint strokes, it is always drawn as an outline regardless of its attributes in the Graphics menu.



To determine how much of an object to draw in each frame, the number of edges in the object is divided by the number of frames in the clip. The polygons are drawn one after the other in the order in which they were drawn in the Graphics menu.

To convert an object to AutoPaint strokes:

1. Select the object.
2. Click Convert and Confirm.

Animating Graphics

Use the Channel Editor and Shape Animation controls to animate Paint graphics. You can animate the position, display, and shape of a graphic. To display the Channel Editor, click the Animation button and swipe the cursor across the bottom of the screen.

Animation Parameters

The following parameters can be animated for every object:

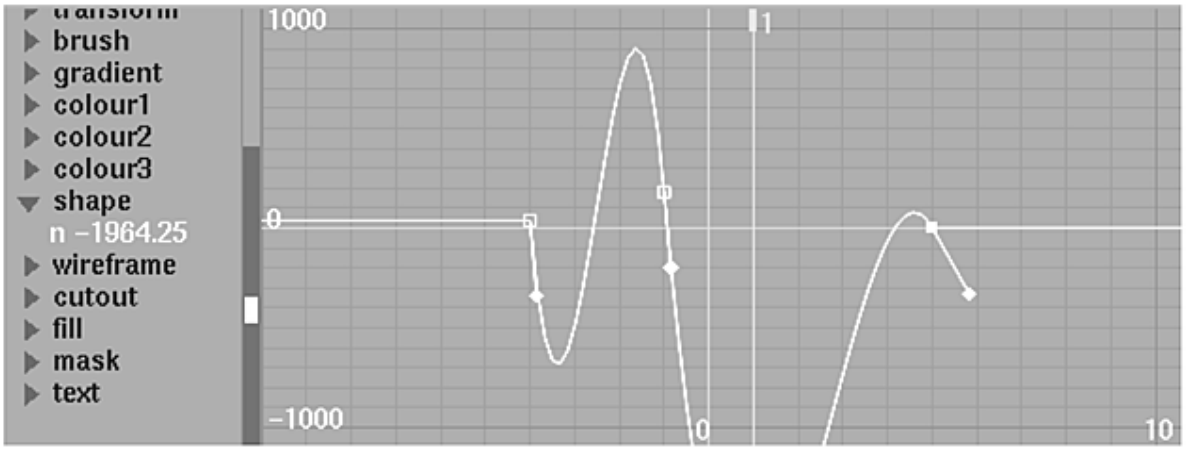
- Position, rotation, and size
- Brush attributes
- Gradient orientation, transparency, and colour
- Colour (red, green, and blue channels)
- Wireframe width
- Range and Softness of a fill object
- Transparency
- Size, Kerning and Italics for a text object

NOTE In Paint, the commands in the Tools box can be performed on animation curves only. To modify a keyframe by editing the object in the image window, you must use the editing commands in the Graphics menu.

Shape Animation

To animate the shape of an object, you must create keyframes with the Shape Animation controls. Each shape you define becomes a keyframe in the Shape channel of the Channel Editor. The difference between the keyframes is interpolated and the shape animation is created.

The Shape channel is used to identify the number and location of shape keyframes in a clip. Use the shape curve to control the rate at which an object changes to a new shape. The following figure shows the Shape channel for an object that changes from shape 1 in frame 1, to shape 2 in frame 8, and shape 1 in frame 15.



To animate the shape of an object:

1 Create an object.

This original shape becomes shape keyframe 1 in the Channel Editor.

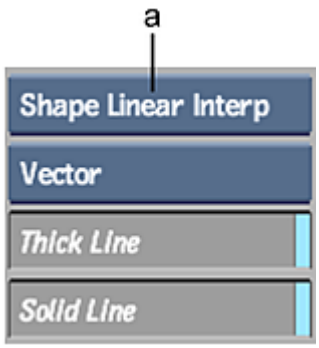
2 Advance to another frame in the clip.

3 Select the object, and change its shape.

To change its shape, you must move, add, or delete a vertex on the object. This new shape is shape keyframe 2 in the Channel Editor.

Shape Interpolation

Use the Shape Interpolation box in the Setup menu to specify the interpolation between shapes in the animation.



(a) Shape Interpolation box

Select:	To:
Shape Linear Interp	Produce sudden transitions between shapes.
Shape Cardinal Interp	Produce smooth transitions between shapes.

NOTE The type of shape interpolation you use is independent of the type of polygon interpolation you use.

Moving between Shape Keyframes

Use Previous and Next to move between shape keyframes in an animation.

To move between keyframes:

1 Select the object you want to modify.

2 Click Edit and then Shapes.

The Shapes menu appears.

3 Click Next to advance to the next shape keyframe.

4 Click Previous to go to the previous shape keyframe.

Deleting Keyframes

Use Delete to delete a shape keyframe from the animation.

To delete a shape keyframe:

1. Select the object.
2. Click Shapes.
   The Shapes menu appears.
3. Select the keyframe you want to delete.
4. Click Delete.
   The selected shape is deleted.

Adding Keyframes

Use Add to create a shape keyframe. This can be useful for creating animations that start and end with the same shape.

To add a shape key using Add:

1. Select the object you want to animate.
2. Go to the frame where you want to add the shape keyframe.
3. Click Shapes.
   The Shapes menu appears.
4. Click Add.
   A new keyframe is added to the Shape channel in the Channel Editor.

Gradient Animation

Use the Channel Editor to animate the colours, orientation, and direction of a colour gradient. The Gradient folder includes Start and End folders for the two colours in the gradient. The Start and End folders each contain channels for the X and Y position of the colour, as well as the R, G, B, and alpha values.

Although you can animate the gradient independently, by default, the gradient will follow any transformations of the object.

To animate a gradient:

1. Select the object with the gradient you want to animate.
2. Click Animation. If necessary, swipe the cursor across the bottom of the screen to display the Channel Editor.
3. Click the Geometry and Stroke folders to expand them.
   If you selected more than one object, a Stroke folder appears for each selected object. Click the Stroke folder for the object you want to animate.
4. Click the Gradient folder to expand it, and click the Start and End folders:
   - The Start folder contains the X and Y channels and a Colour folder for the gradient's colour.
   - The End folder contains the X and Y channels and a Colour folder for the gradient's colour.

5 Click to expand each Colour folder.

Use:	To:
x	Move colour origin left or right. This is the same as moving the gradient bar handle.
y	Move colour origin up or down. This is the same as moving the gradient bar handle.
r	Change percentage of red in the colour.
g	Change percentage of green in the colour.
b	Change percentage of blue in the colour.
a	Change percentage of the colour's alpha channel. Use this channel to create a gradient matte or to change the opacity of the colour.

6 Change the values for the channels in different frames to create a gradient animation.

Rendering the Animation

To render the animation, click Render in the Graphics menu. Make sure you are at the first frame of the animation since the clip will be processed from the currently displayed frame until the end of the clip.

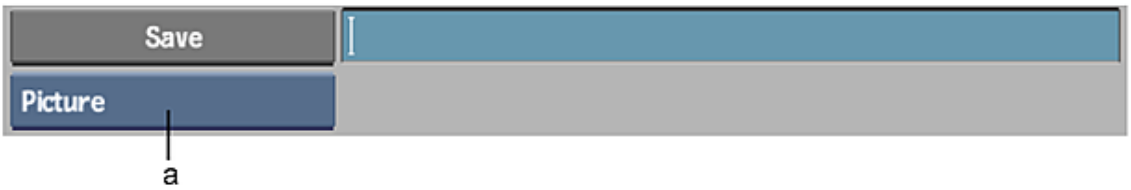
Saving and Restoring Objects

Use the Store and Recall commands to save and restore objects to the image. Use the Library menu to save or load graphics from the graphics library.

To save or load a graphic:

1 In the Paint menu, click Load or Save.

2 Select Geometry from the Load or Save option box.



(a) Save option box

3 Select a file or type in a name for the graphic.

NOTE If you are using Save and have typed in a name for the file, you must press `Enter` to finish saving the file.

The Store and Recall Commands

To save the selected graphic in the buffer, click Store in the Graphics menu. The current object replaces the graphic saved in the buffer.

The Store command does not save the result image. This means that if you tacked the objects onto the image, they are not saved using the Store command. To restore the geometry to the image, click Recall.

(a) Store button (b) Recall button

TIP Use Store and Recall to add graphics to clips loaded at a later time.

Tacking Down Objects

An object can be selected, deleted, edited, and transformed until it is tacked down on the image. As soon as it is tacked down, the object becomes part of the image and can no longer be manipulated.

There are several ways to tack down an object:

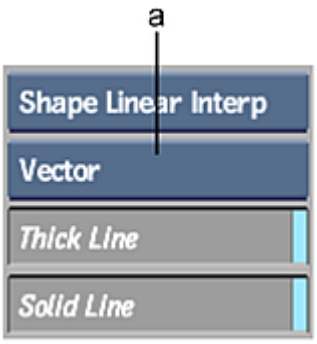
- Select a Tack mode so that an object is either tacked down as soon as it is drawn or as soon as the next object is drawn.
- Tack down selected objects only.
- Tack down all objects simultaneously using the Tack All command.

Tack Modes

The Tack mode is selected with the Tack Mode box in the Rendering Setup menu.

To set the Tack mode:

1 In the Paint tool, click Setup.
The Setup menu appears.



(a) Tack Mode box

2 In the Rendering area, select an option from the Tack Mode box.

Select:	To:
Vector	Explicitly tack down the objects using the Tack command.
Edit Last	Edit the last object drawn until the next object is added to the image. As soon as the next object is added, the last object is tacked down.
Raster	Tack down an object as soon as it is added to the image.

Tacking Down Selected Objects

When the Tack mode is set to Vector, objects must be explicitly tacked down on the image using the Tack button.

(a) Tack button

To tack down one or more objects:

1 Select the objects you want to tack down.

2 Click Tack.

All selected objects are tacked down on the image.

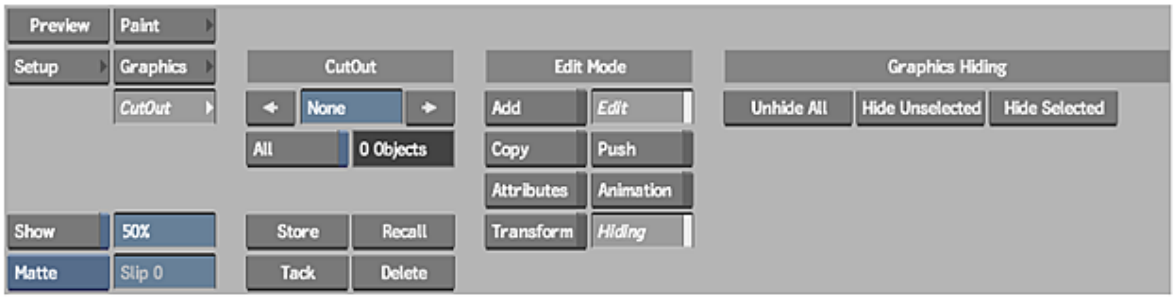
To tack down all objects on the image:

1. In the Graphics menu, Enable All.
2. Click Tack.
   All objects are tacked down on the image.

Cutting and Pasting in Paint

Accessing the CutOut Menu

To access the CutOut menu, click the CutOut button in the Paint menu. Some options may be hidden if the colour palette is open. Swipe the cursor across the bar at the bottom of the screen to hide the colour palette.

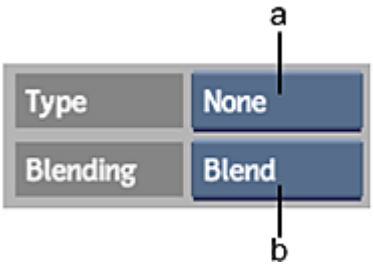


Creating a Cutout

Creating a cutout is similar to creating a graphic. In the CutOut menu, you use the same controls as you would in the Graphics menu to select, copy, hide, move, rotate, resize, delete, store, recall, push, and tack cutouts.

To create a cutout:

1. Click CutOut.
2. Click Add.
   The CutOut Shadow options appear.



(a) Shadow Type box (b) Blending Mode box

3. To limit the area of the cutout with a matte, enable Use in the Matte controls.
4. Select an option from the Shadow Type box.

The Shadow Type box determines how the cutout will be pasted onto the result clip. See Pasting Cutouts (page 1305).

5 Click Attributes.

6 To create a cutout that spans the entire clip, enable Sequence.

7 Click one of the four Cutout commands. See Using the Cutout Commands (page 1302).

NOTE To use the GeoCut option, select all the objects you want to use in the cutout in the Graphics menu before clicking the GeoCut button.

8 Draw the cutout on the image.

9 Set the resolution to be used when updating the object.

The object appears at full resolution when it is drawn. See Setting the Object Resolution and Display (page 1285).

10 Set the cutout's position using the transformation box or controls.

11 Set the cutout's colour using the Current Colour pot.

12 Click Tack to permanently place the object on the result clip.

Once the object is tacked down, it cannot be moved, rotated, deleted, resized, or copied.

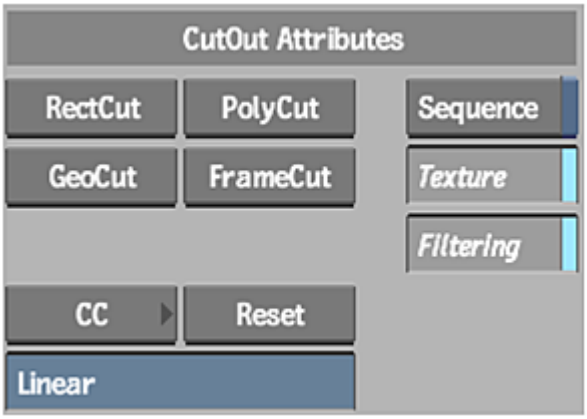
Keep Cut On

Once you add a cutout to the image, the default setting automatically switches from Cut mode to Edit mode. To prevent the switch to Edit mode, enable Keep Cut On in the Setup menu.

(a) Keep Cut On button

Using the Cutout Commands

Four commands are used to create cutouts: FrameCut, RectCut, PolyCut, and GeoCut. FrameCut, RectCut, and PolyCut create cutouts using shapes you define in the CutOut menu. Geocut creates cutouts using objects you select in the Graphics menu.



The FrameCut Command

Use the FrameCut command to make a cutout of the entire frame.

To create a cutout of an entire frame:

1. Click Add in the CutOut menu.
2. Click FrameCut.
   The entire frame is cut out and pasted onto the image.

The RectCut Command

Use the RectCut command to cut and paste a single rectangular area of the image.

To use the RectCut command:

1. Click Add in the CutOut menu.
2. Click RectCut.
3. Draw a rectangle over the area of the image that you want to cut out.
   The image area defined by the rectangle is copied and pasted onto the image.

The PolyCut Command

Use the PolyCut command to cut and paste a single polygonal area of the image.

To use the PolyCut command:

1. Click Add in the CutOut menu.
2. Click PolyCut.
3. Draw a polygon over the area of the image that you want to cut out.
   The image area defined by the polygon is copied and pasted onto the image.

The GeoCut Command

Use the GeoCut command to create a cutout with objects you selected in the Graphics menu.

To create a cutout using GeoCut:

1. Click Graphics in the Paint menu.
2. Create any number of objects.
3. Select all the objects you want to use to make the cutout.
4. Set the opacity of the object(s) using the Brush Opacity field.

   The opacity of the object(s) used in the cutout determines the transparency of the cutout. Use objects with a low opacity value to create an opaque cutout. Use objects with a high opacity value to create a transparent cutout.
5. Click CutOut in the Paint menu.
6. Click GeoCut.

   The image area defined by the selected objects is copied and pasted onto the image.

Displaying Graphics and Cutouts

To display both the cutouts and the graphics objects in the CutOut menu, enable Show Both in the Setup menu. Although the graphics objects appear, they cannot be edited. Enabling Show Both also displays both the graphics objects and the cutouts in the Graphics menu. Although the cutouts appear, they cannot be edited.

(a) Show Both button

Increasing Editing Speed

To streamline cut and paste functionality, use Graphic Regen and Texture Cut/Paste in the Setup menu, and Texture in the CutOut menu.

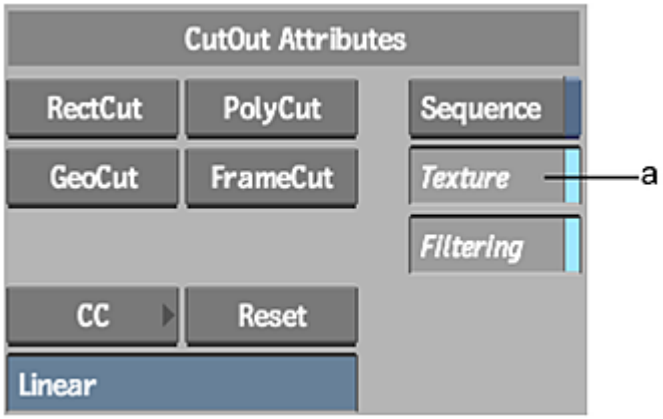
The Graphic Regen Button

Disable Graphic Regen (regeneration) in the Setup menu to ensure the cutout always appears at the display resolution.

(a) Texture Cut/Paste button (b) Graphic Regen button

Texture Cut/Paste and Texture Button

Enable Texture Cut/Paste in the Setup menu or Texture in the CutOut menu to speed up cut and paste operations. These preferences are designed for platforms that support fast texture mapping.



(a) Texture button in CutOut menu

The Filtering button can be used when Texture is enabled. Use Filtering to avoid image degradation as a result of recursive cutting and pasting. When Filtering is enabled, the image should not degrade.

Pasting Cutouts

Four options in the Shadow Type box are available for pasting a cutout onto the image.

Select:	To:
Emboss	Create an embossed cutout. See The Emboss Option (page 1306).
Extrude	Create cutout extrusions. See The Extrude Option (page 1307).
None	Use the blend functions available for pasting cutouts. See The Blend Options (page 1307).

Select:	To:
Shadow	Add a drop shadow to the cutout. See The Drop Shadow Option (page 1308).

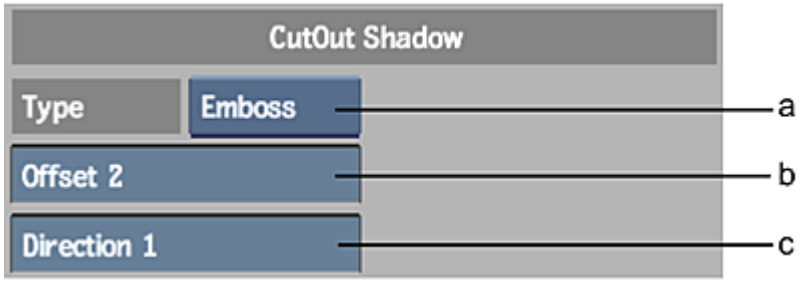
The Emboss Option

The Emboss option makes the cutout appear etched into the image. Shadows and highlights are added to simulate the effect of light shining on a raised object. Three layers are used to produce this effect:

- The top layer is a copy of the cutout in which the luminance values are increased by 50%. The top layer adds the highlights to the cutout.
- The middle layer is the cutout.
- The bottom layer is a copy of the cutout in which the luminance values are decreased by 50%. The bottom layer adds the shadow to the cutout.

Offset Field

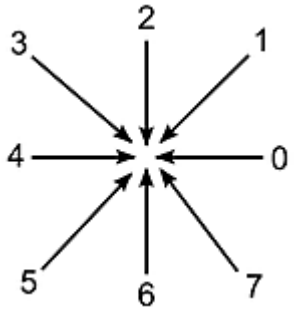
Use the Offset field to specify the offset value in pixels for the three layers. The best results are obtained with values of 1 or 2.



(a) Paste Mode box (b) Offset field (c) Direction field

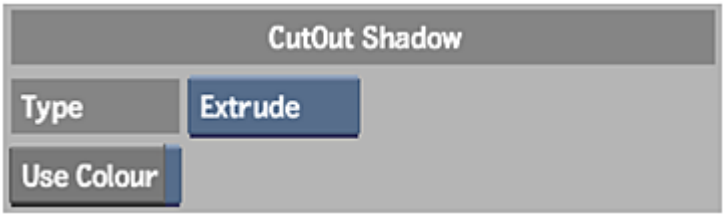
Direction Field

Use the Direction field to specify the angle of the simulated incident light. The values range from 0 to 7 and specify the following angles of incident light.



The Extrude Option

The Extrude option makes the cutout appear to pop out of the image by redrawing the cutout one layer on top of another. Each layer in the stack is offset slightly from the one below so only the edges of the layer are visible.



Editing the Layers

When you create a cutout using the Extrude option, two layers are created to produce the effect. There are vertices at the centre of each layer for manipulating the layer. You can change the depth and direction of the extrusion by moving either layer.

Changing the Colour of the Layers

Enable Use Colour to apply the current colour to the extrusion layers. The cutout image is used for the top layer only. When Use Colour is enabled, use the Brush Opacity field to set the transparency of the extrusion layers.

The Blend Options

The Blend options are similar to the commands in the Processing menu. They use the RGB values of the cutout and those of the result image. Select a blend option from the Blend Mode box.

(a) Blend Mode box

Blend Pastes the cutout onto the result image.

Add Adds the RGB values of the corresponding pixels in the cutout and the result image. Values greater than 255 are clipped.

Subtract Subtracts the RGB values of pixels in the image from those of the cutout. Values less than 0 are clipped.

Black Pastes a black object in the shape of the cutout on the result image.

White Pastes a white object in the shape of the cutout on the result image.

Colour Pastes a coloured object in the shape of the cutout on the result image. The current colour is used for the object.

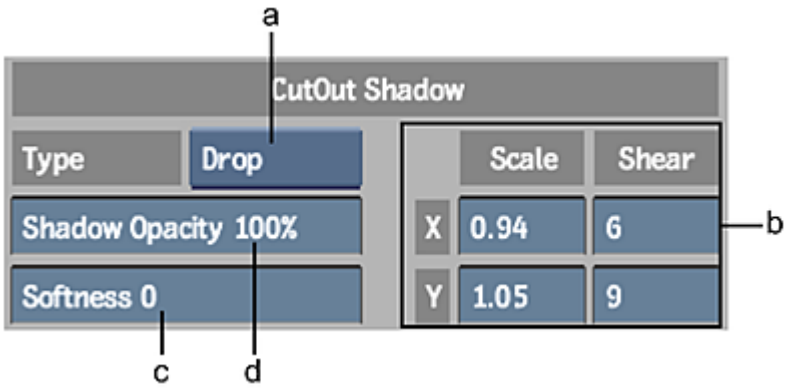
NAdd Min Compares the brightness values of corresponding pixels in the cutout and the result image, and uses the pixel with the smaller value in the tacked down cutout.

NAdd Max Compares the brightness values of corresponding pixels in the cutout and the result image, and uses the pixel with the greater value in the tacked down cutout.

Multiply Multiplies the RGB values of corresponding pixels in the cutout and the result image. The value is normalized by dividing the result by 255.

The Drop Shadow Option

You can add a drop shadow to a cutout by selecting the Drop option from the Paste Mode box. The drop shadow's colour is taken from the Current Colour pot.



(a) Paste Mode box (b) Scale and Shear fields (c) Softness field (d) Shadow Opacity field

Shadow Opacity field Adjusts the transparency of the drop shadow. Decrease the value to increase the transparency.

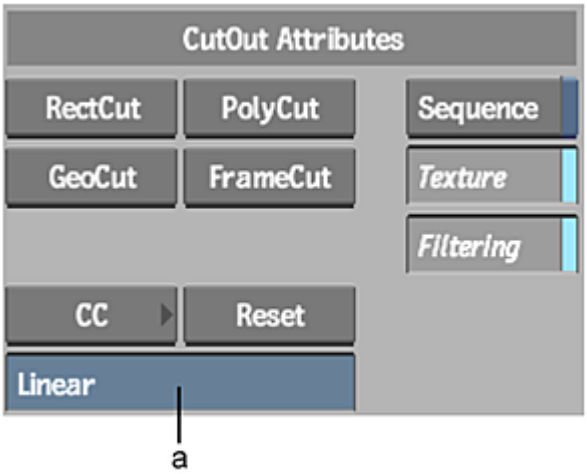
Softness field Adjusts the edge softness of the drop shadow. Increase the value to increase the softness of the shadow.

Scale and Shear fields Shears and scales the drop shadow along the X- or Y-axis. These fields can be used to add lighting perspective to the image.

Colour Correcting Cutouts

Apply colour correction to cutouts using the Colour Correction field and CC in the CutOut menu. Click the Colour Correction field to load a setup directly from the colour correction library. Click CC to load the cutout into the Colour Corrector.

The default colour correction setup is Linear. When you load a different setup, the name of the setup appears in the Colour Correction field. To reset the colour correction setup, click Reset.



(a) Colour Correction field

Animating Cutouts

Use the Channel Editor to animate cutouts. To display the Channel Editor, click Animation and swipe the cursor across the bottom of the screen.

The following parameters can be animated for every cutout:

- Position, rotation, and shearing
- Brush attributes
- Colour (red, green, and blue channels)
- Emboss options
- Shadow options

Loading and Saving Cutouts

Use the Cutout Library to load and save cutouts.

To save or load a cutout:

1. In the Paint menu, click Save or Load.
2. Select CutOut from the Save or Load option box.
3. Select a file or type in a name for the cutout.

   NOTE If you are using Save and have typed in a name for the file, you must press `Enter` to finish saving the file.

About Paint Setups

A setup is a file that contains a record of all changes you make to a clip in a particular tool. This record includes references to clips used. Setups let you save your work separately from the clips, so you can load and work on them any time, or apply the setup to other clips. Both the procedure and the interface involved

in saving Paint setups and preferences differ slightly from those involved in saving setups in most other tools

NOTE Custom brushes, brush sets, colour pots, and palette setups that you create in Paint are stored in your user directory. These resources are loaded when you specify a user in the Project Management menu.

Saving Setups

You can save pictures, preferences, mattes, palettes, brush groups, brushes, and stamps. You can also save cutouts, cut sequences, recorded strokes in AutoPaint, as well as geometry created in the Graphics menu.

To save a setup:

1. In the Paint menu, click Save.
2. Select the item you want to save from the option box.

Select:	To:
Picture	Save an image created in Paint.
Matte	Save the matte.
Palette	Save the current colour palette.
BrushSet	Save the current set of brushes.
Brush	Save the brush that is currently active.
Stamp	Save the Stamp currently in the Stamp window.
Geometry	Save the geometry created in the Graphics menu. This also includes all cutouts.
CutOut	Save the cutouts created in the CutOut menu.
CutSequence	Save the cutout sequence created in the CutOut menu.
AutoPaint	Save the last series of recorded strokes.
Preferences	Save current preferences.
Defaults	Save current default values.

   The file browser appears, listing any existing setups for that item.
3. Type the name of the setup in the Save field.
4. Press Save `(Enter)` to save the setup.

Cropping a Setup

Use the crop box to limit the area of the picture, matte, cutout, or paint graphics to be saved in the setups directory.

To save a cropped matte, image, or cutout:

1 In the Paint menu, click Save.

2 Click Picture, Matte, or CutOut to specify the element from the image you want to save.

3 Draw the crop box on the canvas by pressing and dragging the cursor diagonally across the screen. Alternatively, use the left, right, top, and bottom Crop fields to set the boundaries of the crop box.

NOTE Click Reset to reset the crop box values.

4 Click Save.

5 Select the item you are saving from the option box and name the setup in the keyboard display.

If you want to overwrite an existing setup, select the filename from the file browser.

6 Press Save (`Enter`) to save the setup.

Loading Setups

Use the Load button to load any type of Paint setup.

To load a setup:

1 Click Load.

The Load Paint menu appears.

2 Select the type of setup you want to load from the Load option box.

A list of existing setups appears in the file browser.

3 Click the title or proxy of the setup you want to load.

Deleting Setups

Use the Delete button to remove setups from the setup directory.

To delete a setup:

1 Click Load.

The Load Paint menu appears.

2 Enable Delete.

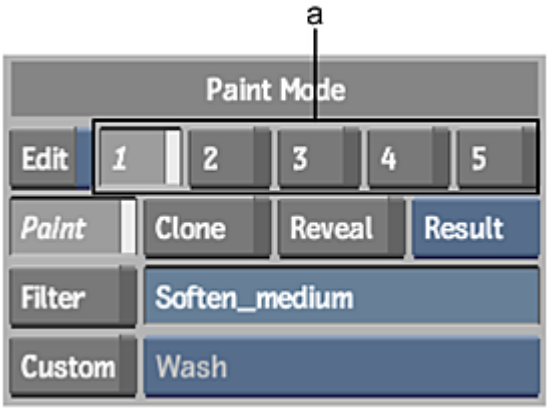
3 Select the type of item you want to remove from the Load option box.

A list of existing setups appears in the file browser.

4 Click the title or proxy of the setup you want to remove and click Confirm.

Creating and Saving Brush Setups

Within a given brush set, there are five individual brushes. You can create and save five brush setups using the buttons numbered 1 to 5 in the Paint menu. Each brush setup saves the brush type, attribute values and modes, Special Effects medium, filter, fill parameters, and colour correction setups.



(a) Brush Setup buttons

To create and save a brush setup:

1 In the Paint menu, click one of the five Brush Setup buttons.

2 Enable Edit next to the Brush Setup buttons.

3 Set the brush attributes. See Brush Attributes (page 1239).

4 Disable Edit to save the changes to the selected brush setup.

NOTE To save a set of five brush setups, use BrushSet in the Save menu.

Copying Brush Setups

You can copy brush setups using the setup buttons.

To copy brush setups:

1 Select a setup using the Brush Setup buttons and enable Edit.

2 Press `Ctrl` and click another Brush Setup button to copy the setup.

3 Disable Edit to save the setup.

Creating a Custom Brush

With Paint, you can create your own custom brushes for specialized applications. You can modify an existing brush, draw your own brush shape, or grab a part of the image to use as a custom brush.

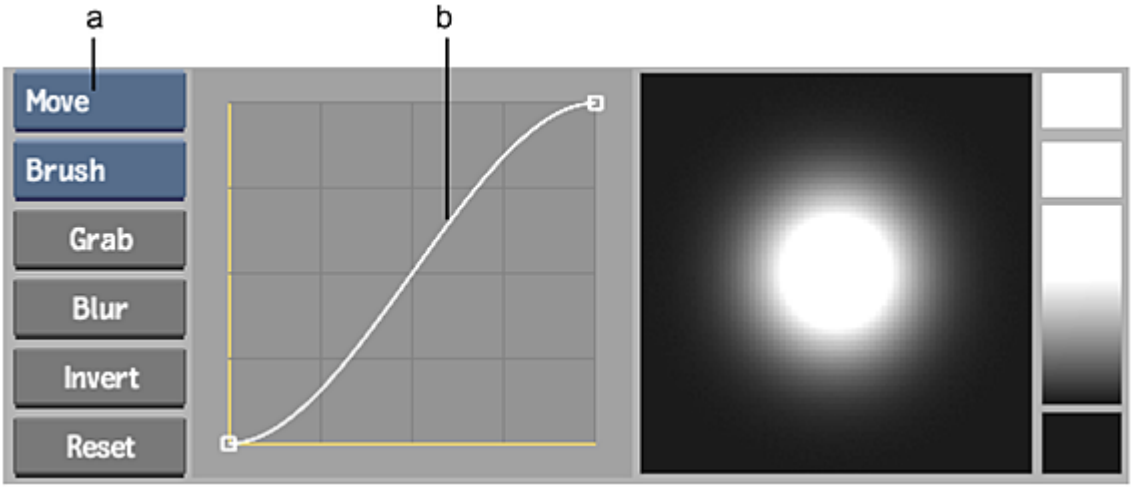
Use the Custom Brush menu to create your own brush. To display the Custom Brush menu, double-click one of the brush icons in the Brushes window. An enlarged view of the selected brush appears in the editing panel.

There are three ways to create a custom brush:

- Using the brush profile
- Drawing a brush shape in the editing panel
- Grabbing an area of the image

Changing the Brush Profile

The brush profile describes the size and edge softness of the custom brush. The default S-curve defines a circular airbrush. You can change the curve by manipulating the two points that define the curve, or you can add points to the curve.



(a) Profile box (b) Brush profile

The profile does not affect the brush in the editing panel until you modify the curve or click the Update button.

Changing the Curve

By default, there are two points that define the brush profile. For the following table and examples, these points will be named A and B as shown in the preceding illustration. To move the points that define the curve, select the Move option from the Profile box.

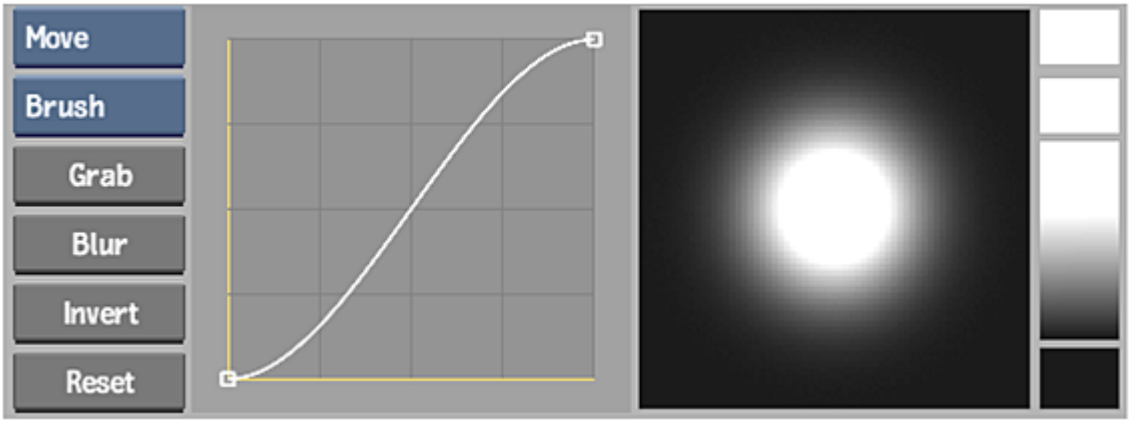
Move:	To:
Point A up	Lighten the brush.
Point B down	Darken the brush.

Move:	To:
Point A right	Harden the outer edge of the brush.
Point B left	Soften the middle of the brush.

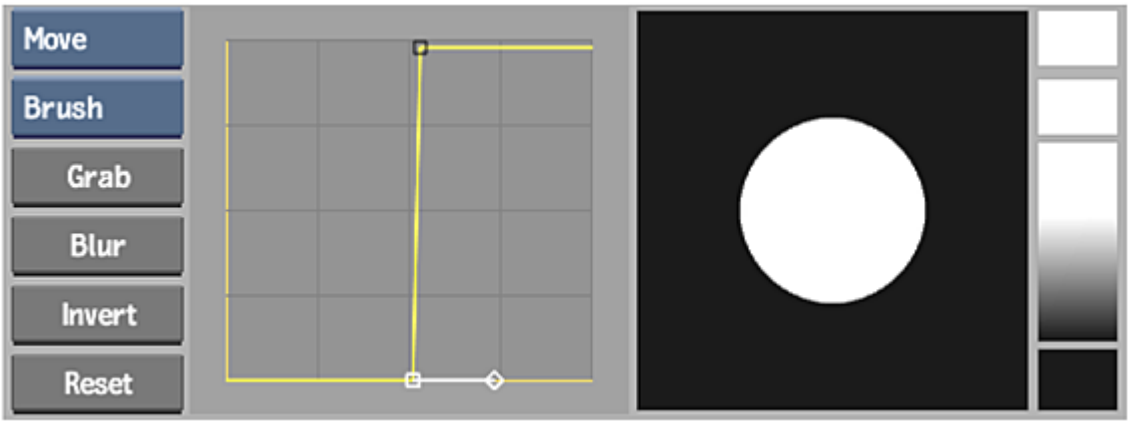
To move the points only on the X-axis, select the Xscale option from the Profile box. To move the points only on the Y-axis, select the Yscale option. To break a point and manipulate its tangent, select the Break option and click on a point.

Adding Points to the Curve

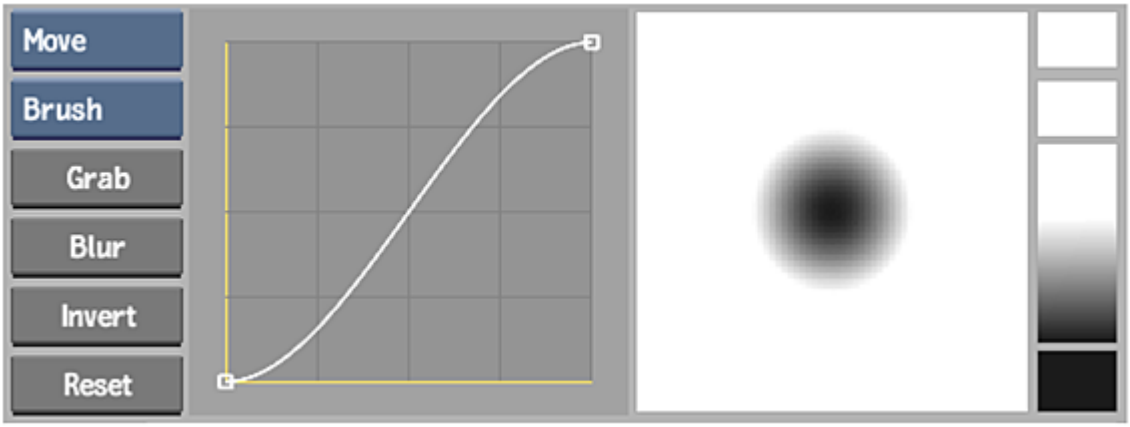
To add points to the curve, select the Add option from the Profile box and click on the curve. To delete points from the curve, select the Delete option and click on one of the points on the curve.



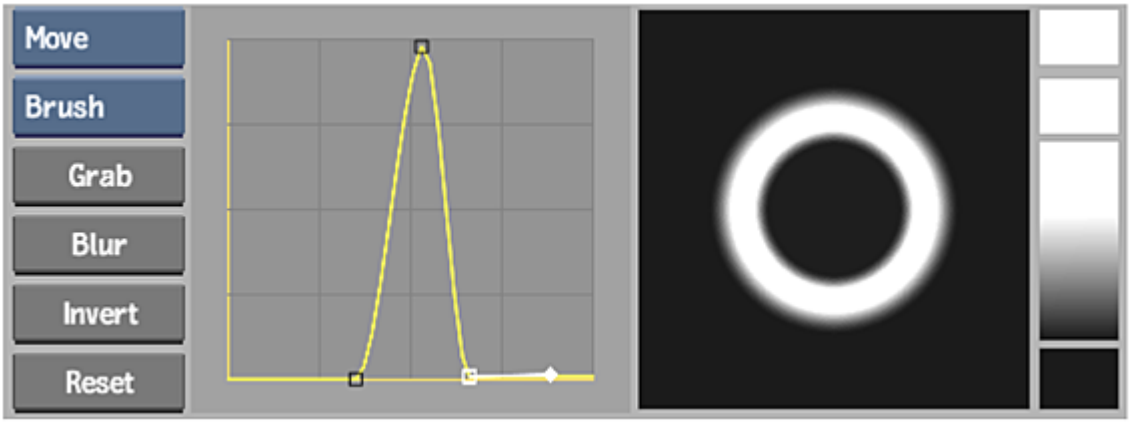
Default S-shaped curve



Hard edge curve



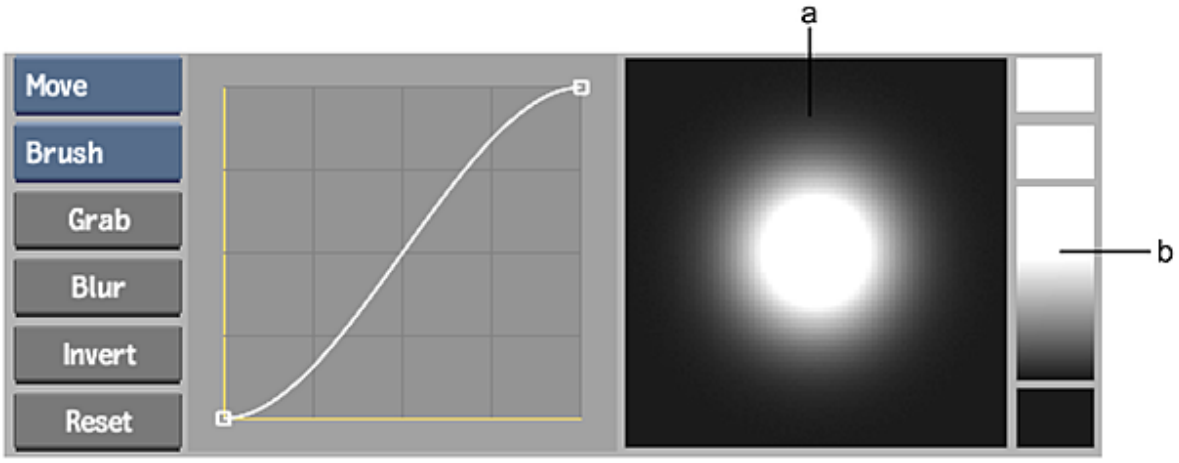
Invert brush curve



Ring-shaped brush curve

Drawing and Updating a Brush Shape

You can draw a new brush shape or update the current brush shape in the editing panel. The new shape is drawn using the brush that you selected to open the Custom Brush menu.



(a) Editing panel (b) Gradient bar

NOTE You cannot use the Undo command when drawing or updating a custom brush.

To draw a brush shape:

1. To create a new shape, erase the existing brush shape by painting over the editing panel.
2. Draw the new shape.
3. To invert the brush shape, click Invert.
4. To blur the brush shape, click Blur. Each time you click Blur, a 3 x 3 filter is applied to the brush image.

Selecting an Area of the Image

You can select an area of the image to use as the custom brush. The brush created from the selected area will be a square monochrome brush.

To grab an area of the image:

1. Click Grab in the Custom Brush menu.
2. Click on the image and drag across the screen to define the selection box.

   To select a square area, press `P` as you sweep out the selection box.

   The selected area appears as a monochrome image in the editing panel. If the area that you select is not square, then the selected area is resized to fit in the editing panel.

Updating an Existing Brush

To update an existing brush, click the Update button. The changes are applied to the brush icon that you selected to open the Custom Brush menu.

Creating a New Brush

To create a new brush, click the New button. This creates an icon for the new brush shape and adds it to the Brushes window.

Saving the Custom Brush

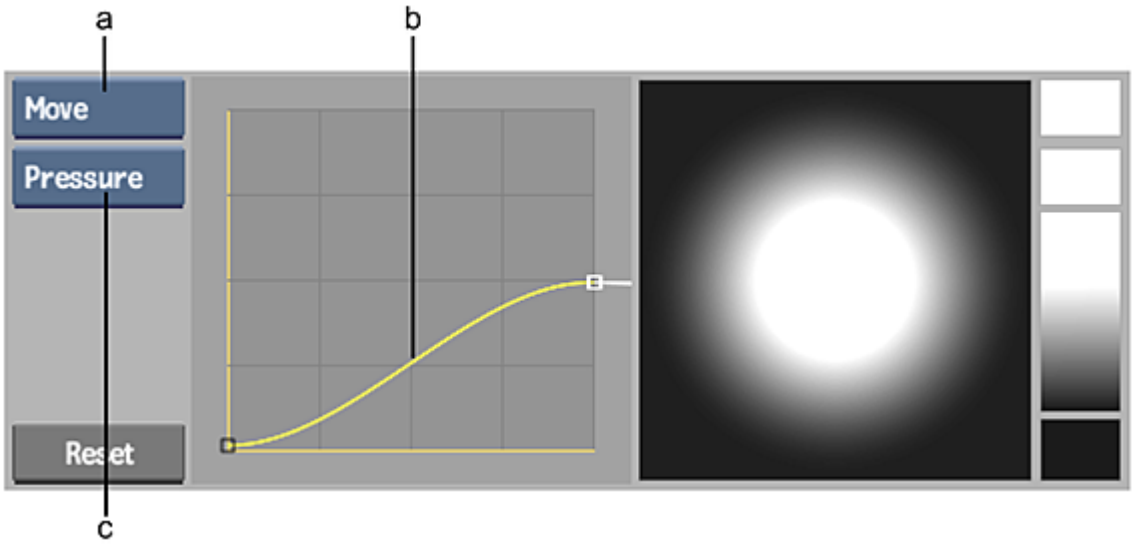
Custom brushes are not saved automatically when you exit Paint. To save the updated or new brush for use in another work session, click the Save button. This opens the Brush setups directory. Use the keyboard to enter the name of the brush, and click the Enter button. The new brush is saved in the Brush setups directory.

Loading a Custom Brush

Click the Load button to load a custom brush from the Brush setups directory.

Customizing the Pressure Profile

Adjust the pressure profile to change the amount of pressure needed to paint with a given brush. The slope of the curve indicates how quickly paint is applied as you press on the tablet. If the curve is steep, a small amount of pressure applies full paint to the canvas. If the curve is soft and rounded, you must press harder and longer to apply full paint.



(a) Profile box (b) Pressure curve (c) Brush/Pressure box

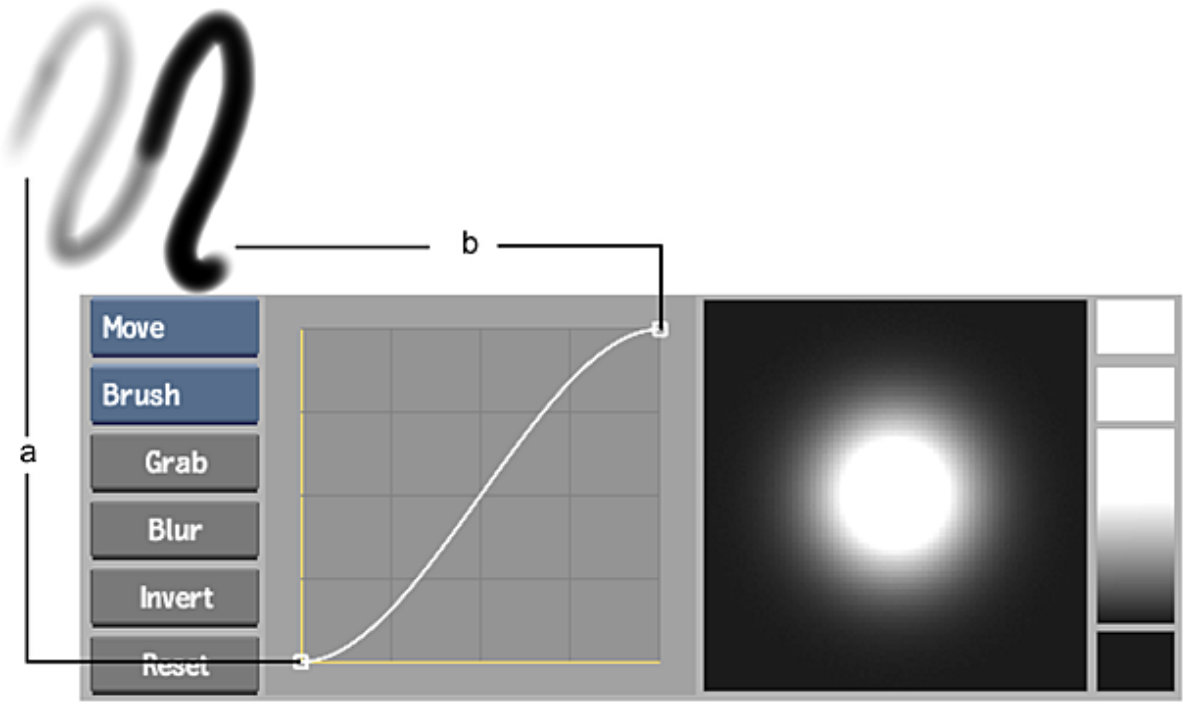
You use the options in the Profile box to change and add points to the pressure curve. Click Load to load a Pressure profile. Click Save to save a custom pressure profile.

To edit the pressure profile:

1. Toggle the Brush/Pressure box to Pressure.
2. Double-click a brush icon in the Brushes window to display the Custom Brush menu.
3. Click the Brush/Pressure box to display the pressure profile.
4. Select Move from the Tools box.
5. Press the left point (A) and drag it to its new position.
   This changes the amount of paint the brush applies when you press lightly on the tablet.
6. Press the right point (B) and drag it to its new position.
   This changes the amount of paint the brush applies when you press down firmly on the tablet.
7. Click Update to update the pressure profile for the brush.
   The pressure settings will be used when you paint with the brush during the current work session.

Default Pressure Profile

Pressure is mapped along the horizontal axis (X-axis) of the curve, and the amount of paint applied is mapped along the vertical axis (Y-axis). If you are using the default S-shaped curve and you press lightly on the pen, very little paint is applied. As you press harder, more paint is applied.



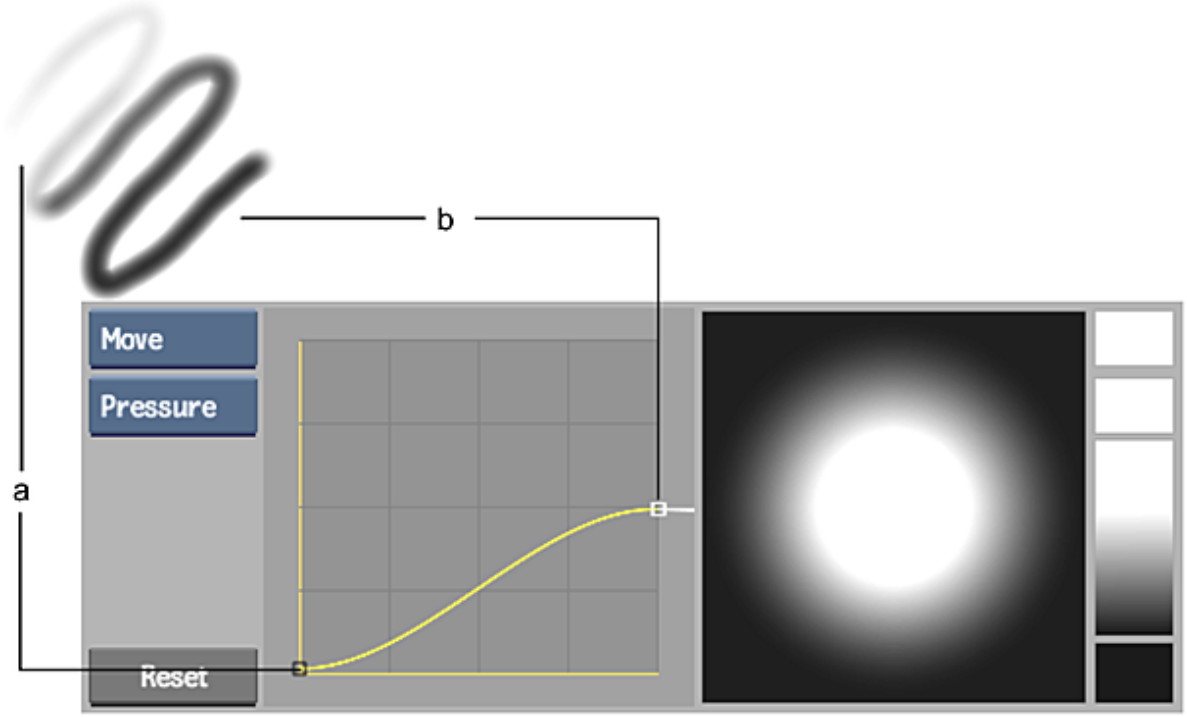
(a) Point A: Low paint at low pressure (b) Point B: Full paint at full pressure

Maximum Pressure Curve

If you move point A in the previous figure to the maximum position on the Y-axis, the curve results in full paint applied all the time, no matter how much or how little pressure you use.

Soft Airbrush

If you move Point B in the previous figure to a point midway on the vertical axis, the curve gives you a soft airbrush, even at full pressure.



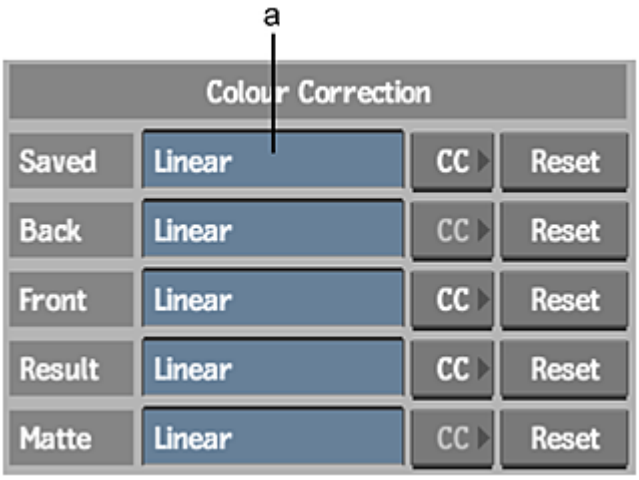
(a) Point A: Low paint at low pressure (b) Point B: Soft paint at full pressure

Using Colour Correction Setups

Use the Colour Correction Setup menu to apply colour correction setups to the clip loaded into Paint. You can bring the clip into the Colour Corrector or you can load a setup from the Colour Correction setups directory into Paint.

To load a clip into the Colour Corrector:

1 In the Paint menu, click Setup.
The Colour Correction menu appears.



(a) Setup Name field

2 Click CC next to the clip you want to load into the Colour Corrector.

The Colour Corrector appears.

3 Use the Colour Corrector as described in About Colour Correcting (page 1141).

4 Save the setup to the Colour Correction setups directory.

5 Click Exit.

The name of the Colour Correction setup appears in the Setup Name field beside the clip label for the selected clip.

To load a setup directly from the Colour Correction setups directory:

1 Click the Setup Name field for the clip you want to colour correct.

The file browser appears, displaying the contents of the Colour Correction setups directory.

2 Select the Colour Correction setup you want to use.

To reset the colour correction for a clip:

1 Click Reset beside the clip you want to reset.

2 Click Confirm.

Text and Titling

23

Text is comprised of layers, paragraphs, and characters. You can make text spin, dance, and change colour over time. You can also create effects such as a text roll of credits, text crawls, bumpers, and text that moves on a motion path. You can use logo images in a text roll so that the logo of a sponsor appears in the credits list.

You use the Text Tool to add text and text effects to a clip. Once text effects are applied to a clip, you can then use the clip in Action or Batch or Batch FX depending on the final effect you want to achieve. Use Action to create 3D text deformation effects, including sliding, magnifying, and oscillating effects. For example, to create an effect of text rolling on a cylinder in a composite, you first render a text roll in Text and then, in Action, project it as a texture onto a cylinder. See Diffuse Mapping (page 800).

Accessing the Text Tool

The Text Tool can be accessed from the following locations:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

Setup and Rendering Options

Before rendering a text clip and rendering text effects, you must set blur settings, text animation, softness, and whether or not you want to use prerendering. The prerendering feature accelerates the rendering process for a text layer.

Set text rendering options in the Text Setup menu. To access the Setup menu, click Setup. The Setup controls are described as follows.

Software anti-aliasing values can range from 1 to 64. A higher value increases the sampling rate, but also increases processing time. An anti-aliasing of 1 means no anti-aliasing is performed and the result is a hard edge.

TIP A higher anti-aliasing value is usually needed with outlined text.

Softness box Defines the softness of the anti-aliasing samples.

Blur box Sets Gaussian Blur or Box Blur. Gaussian Blur uses subpixel resolution and creates a subtle effect with the Blur Shadow attribute. Gaussian has rounded, smoother edges and is better for animation; however, processing time is increased.

Box Blur has rectangular, rougher edges, but is more economical with rendering time, especially if you are working on a rough draft.

Rendering Text Clips

You can render clips in progressive or interlaced mode. Interlaced rendering produces a better result, especially if you use keyframes that are far apart, but it takes longer to render. You should preview the quality of the image and the text before you process the final clip.

To render a clip with text:

1. From the Rendering box, select Interlaced, Progressive, or Auto (to render using the mode of the back clip).
2. From the Render Mode box, select which clip gets processed.

Select:	To:
RGB	Render only the text on the back layer.
Matte	Render only the matte of the text. The text fill transparency attribute is transferred to the matte. By generating a matte, you can easily composite the text onto another clip.
Both	Render the text on the back layer and the matte of the text. The fill transparency is removed from the text and transferred to the matte.

3. Click Setup.
4. From the Setup menu Rendering section, select the sampling level from the Auto-Softness box.

5 Set other options as required.

6 Position the clip at the first frame or the specific frame where you want to start processing.

7 Click Preview.

TIP You can zoom (`Ctrl+spacebar`) or pan (`Ctrl+Shift+spacebar`) the image window while in Preview mode. To disable Preview, make a modification to the text or processing options, or click anywhere in the image window.

8 Make modifications to the text and rendering options and preview until you are satisfied with the result.

9 When you are ready to process the text clip, click Render.

The clip is processed from the currently displayed frame until the end of the clip.

Rendering Considerations

Rendering is contingent upon PreRender, Blur, and Animation settings. Depending on which options you choose, you can increase the rendering speed:

- When more than one non-static layer has blur shadows, Global Blur is faster, although all shadows are blurred with the same unique colour and appear under all other layers.
- When there is a combination of animated and non-animated layers, speed depends on the blur shadows. If animated text has blurred shadows, Global Blur is faster. If you want a static blurred shadow, and a non-blurred animated layer, Layer Blur is faster.
- If you have static layers only (no animation), speed depends on the number of layers and whether PreRender can be used. Global Blur is faster since it uses PreRender. The processing speed of Layer Blur is the same with one or two layers. However, it becomes progressively slower with a higher number of layers and if the layers use blur shadows.

Saving and Loading Files

You can save, load, and import text files, text setups, and images for specific use with the Text tool. You can save text, text effects, text preferences, layers and new default text settings.

You can load text files from third-party word processing packages, existing Text tool setups, and logos. Loading existing text files is particularly convenient if you are creating a long text roll of credits as it saves time that you would otherwise spend typing and spell checking.

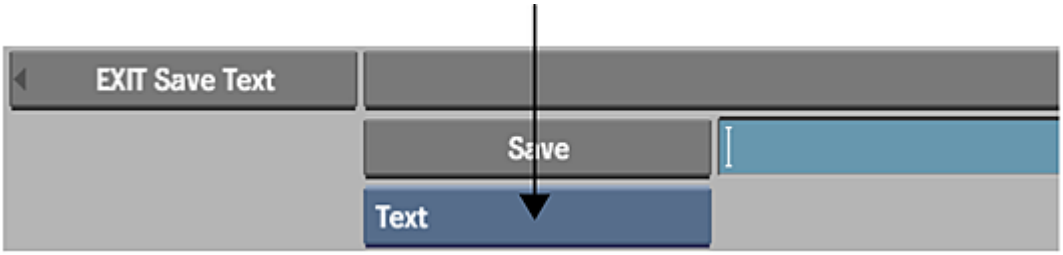
Saving Text Files

When saving files, you can save text setups, preferences, and default settings.

To save a file:

1 Click Save.

2 In the Save menu, select an option from the Save box.



Select:	To:
Defaults	Override the default menu options in the Setup menu and replace them with your specifications.
Preferences	Save Setup menu specifications only and omit text and text attributes.
Selected Path	Save any selected text paths.
Selected Logos	Save any selected logos.
Selected Layers	Save selected layers, their text, text attributes and Setup menu options.
Text	Save text files with text attributes and Setup menu options.

3 Navigate and choose the directory where you want to save the file.

4 Type a name for the file and press `Enter` or click the Save button.

The file is saved and you return to the Text menu.

Loading Text Files

You can load text files, preferences, and ASCII files. You can also reset the Text menu settings with its factory defaults. To load a file:

1 Click Load, and in the Load menu, select an option from the Load box.

Preferences Previously saved setup menu options.

Path Previously saved text path setups. Setups between text paths and garbage masks are interchangeable. So, you can also load a garbage mask shape as a path. Preset paths are in `opt/Autodesk/<product home>/path/default`.

Logo Previously saved logos.

Text File Text files. When you select Text File, you can select the encoding of the loaded file. This applies the correct conversion from the selected file's encoding to the application's internal encoding. For example, select ISO8859-1 to import Latin-1 ASCII files, or UCS-2 to import 2-byte Unicode encoded text files. The encoding options reflect the character encodings available with your system. The load mechanism supports the same encodings as the iconv utility. Optional iconv converters are available by installing national language support options.

Multiple Text Setup Several layers or text files at a time with preferences. Text tool preferences that were saved with the selected file are loaded as well.

Text Setup A layer or text file created in the Text tool. Text tool preferences that were saved with the selected file are loaded as well.

Factory Defaults The original Text tool settings that were delivered with the application.

2 Choose a directory in either Proxies or Titles mode.

3 Select the file(s) to load. To select multiple files you must already have selected Multiple Text Setup, otherwise only the last file of multiple files is loaded.

4 While loading a text file or text setup, click anywhere in the user interface to abort the load before it is completed.

5 To discard unwanted files, enable Remove.

6 If necessary, click Exit Load Text to exit the file browser.

Adding Text to Clips

You add text to a clip by first creating a layer for entering the text, setting the layer attributes, and then typing the text. You can create several layers of text that overlap each other.

Creating a Layer

Text is contained in a layer called a text layer. When you create a text layer, a coloured border called the *crop border* appears in the image window indicating the region the text will occupy.

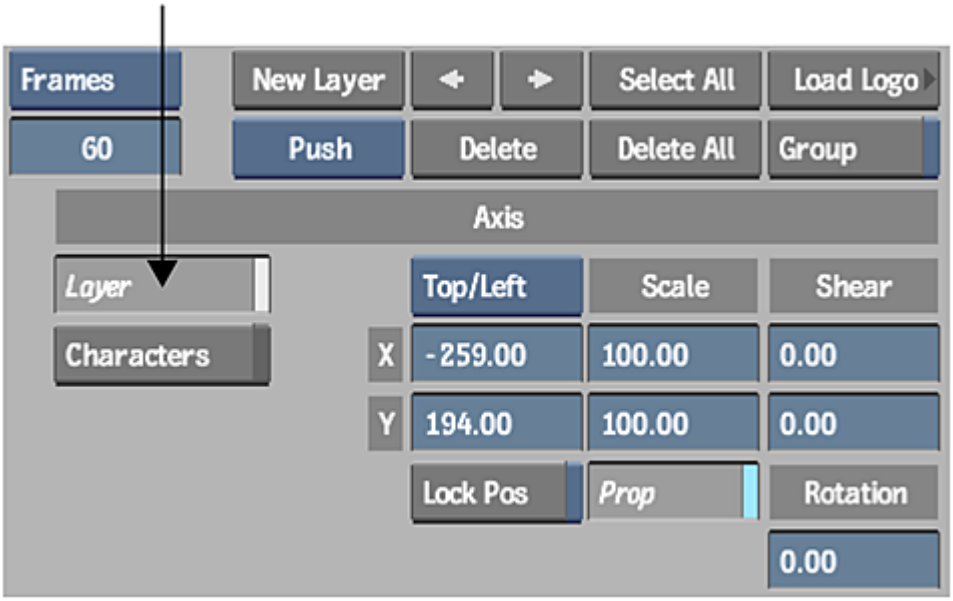
To create a layer:

1 Click New Layer. If this is the first layer you are creating, you can click directly in the image window.

The text layer is positioned in the upper-left corner of the safe title area by default. The text in the layer is left justified, while the width is equal to the safe title overlay. You can modify the safe title overlay using the Grid and Guide controls. You can also modify the boundary colour. See Rendering Text Clips (page 1322).

2 Click Axis to make sure you are in the Axis menu.

3 To set the text layer dimensions, enable Layer.



Each layer has its own parent axis for translating, rotating, resizing, and shearing. The axis for the layer can be in the upper-left or centre of the layer. See Creating Text Effects (page 1330).

Entering Text

The text you type may be as short as one letter or word or as long as several paragraphs. If the text is long or exists in a word processor, you can load the text file into the current layer.

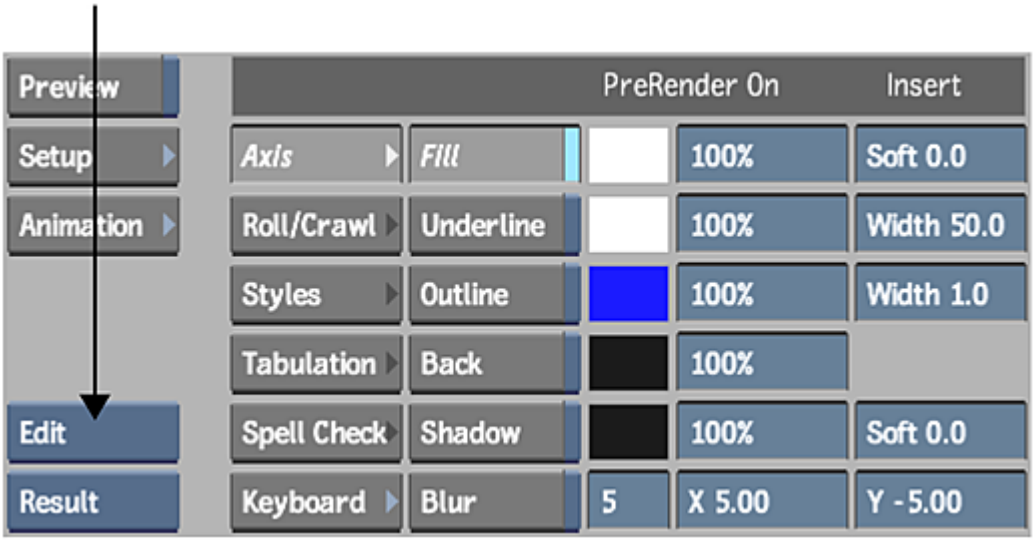
You enter and edit characters much like in a word processor. You type characters using the keyboard and many known text editing conventions, such as `Shift+arrow keys` to select text lines. You can also click once to insert the cursor and select the adjacent character, double-click to select the whole word, and triple-click to select the entire line. Quadruple-click to select all characters on the layer and use the middle mouse button to paste selected text.

Use either the workstation or on-screen keyboard to enter characters in a layer. You can also paste the current text selection (from a shell or any other application) into any keyboard input field by pressing the middle mouse button or by using the pen button.

You can also input Asian characters using their corresponding ASCII codes with the numeric keypad.

To enter text in a layer:

1 From the Text Mode box, select Edit.



TIP Use Edit mode to type text strings, select text, edit text in layers, and modify text attributes. Press `Esc` to switch between Edit and Move modes. The message bar displays the current mode.

2 In the image window, create a new layer by doing one of the following:

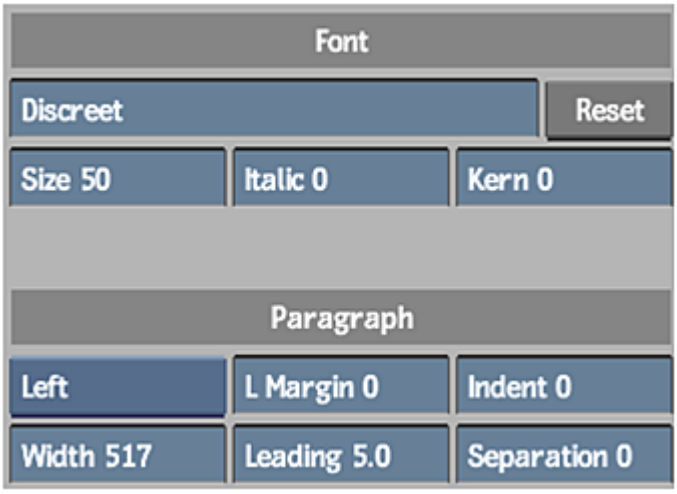
- Click New Layer.
- Press `Alt+N`.
- Click directly in the image window (if this is the first layer you are creating).

3 To begin a new paragraph within the layer, press `Enter`.

Notice that each paragraph has its own coloured boundary and a paragraph tag that is a coloured rectangle in the upper-left corner of the paragraph. In Edit mode, a selected paragraph's boundary is white.

(a) Blue indicates paragraphs in text layer (b) Red indicates currently selected paragraph, which has a paragraph boundary (c) Layer boundary is shown in red

4 In the Paragraph and Font menus, set properties.



Using the Text Keyboard

When you use the on-screen keyboard, consider the following:

- Special characters use the ISOLatin1 encoding vector. You specify the encoding vector in the *init.cfg* configuration file using the FontMapping keyword.
- The extended keyboard uses the Standard encoding vector.

- Asian character sets may not display all characters on the on-screen and extended keyboards. Use the numeric keypad or load an ASCII file that contains unavailable characters for the selected font. See Loading Text Files (page 1324).

To use the Text keyboard:

1. In the Text menu, click Keyboard.
2. On the on-screen keyboard that appears, enable Up ASCII to display extended characters contained in the selected font set.

NOTE Enabling Up ASCII also affects your workstation keyboard, so make sure you disable the button again before exiting the Text menu.

3. If the selected font has special symbol characters, enable Symbols to see them.
4. Type the characters in the text layer.

   The keyboard characters appear in the current font.

Word Processing Keyboard Shortcuts

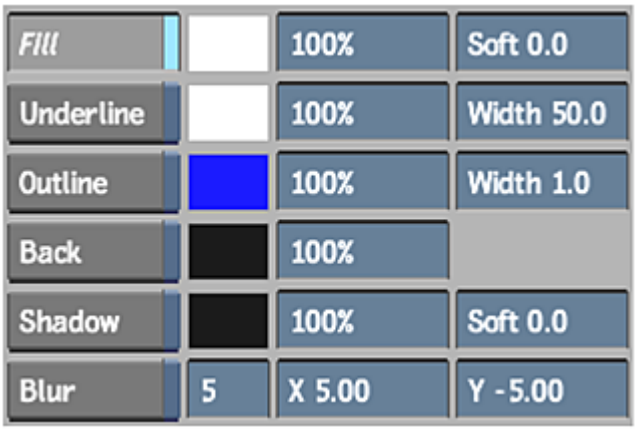
Typical word processing tasks include text selection, cutting, copying, and pasting (use the shortcuts based on your User profile). The Text tool includes several character manipulation keyboard shortcuts for these tasks.

Press:	To:
Alt+A	Select all characters in the selected text layer.
Alt+P	Select all characters in the selected paragraph within a layer.
Alt+Shift+A	Select all characters in all layers. This operation is the same as using the Select All Layers button.
Shift+up arrow	Extend the selection one line up.
Shift+down arrow	Extend the selection one line down.
Shift+left arrow	Extend the selection one character to the left.

Press:	To:
Shift+right arrow	Extend the selection one character to the right.
Shift+Home	Extend the selection to the beginning of the current line.
Shift+End	Extend the selection to the end of the current line.
Ctrl+Shift+PgUp	Extend the character selection to the beginning of the text layer.
Ctrl+Shift+PgDn	Extend the character selection to the end of the text layer.
Insert	Toggle between Insert and Overwrite text entry modes.

Creating Text Effects

Use the Attributes controls to change the appearance of individual characters, paragraphs, and layers. You set similar properties for both layers and characters. You can also load a logo in a paragraph and then format it using the Attributes menu (see Loading Logos (page 1334)). If you create text formats that you plan to reuse, save them using the Styles menu (see Defining Styles (page 1345)).

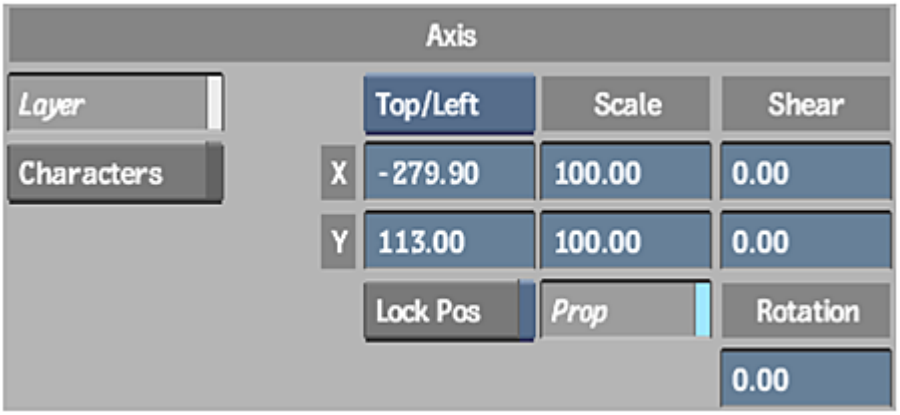


These text attributes have a cumulative effect on the selected characters. For example, if you enable the Fill and Outline buttons, the text appears as solid characters with a coloured outline.

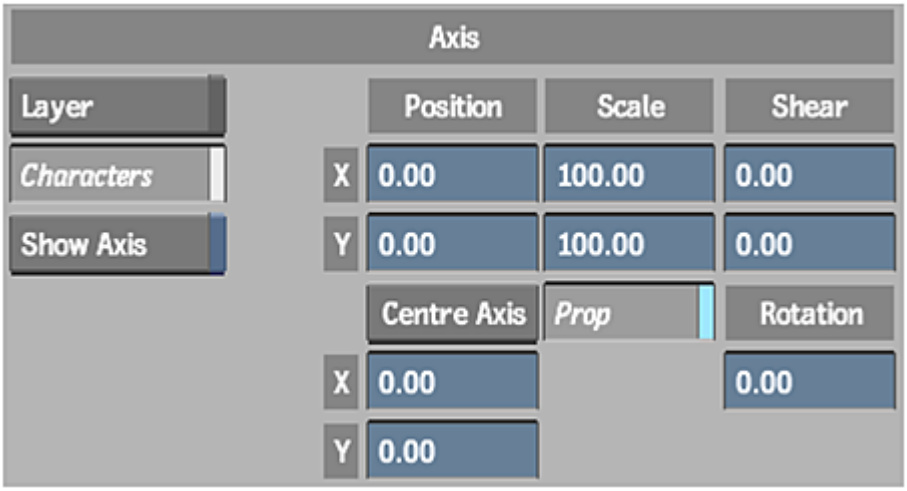
NOTE For a reference of the complete Text menu, see Text Menu Settings (page 1665)

Modifying Layer and Character Properties

Use the Layer controls in the Axis menu to rotate, translate, scale, or shear a selected layer and to apply an effect uniformly to all characters.



Use the Character controls in the Axis menu to view and position the axis of each character. You can use the position, rotation, scale, and shear settings for each character in a layer to create an effect of scrambled letters.



To edit text characters, you must be in Move or Edit mode and some text must be selected. The Layer and Character controls are described as follows.

Layer Order box (in the Layer Attributes section) Moves a layer in front of or behind another layer.

Select:	To move the layer:
Push	One position down in the stack behind another layer.
Pop	One position up in the stack in front of another layer.
Bottom	To the bottom of the stack behind all other layers.

Select:	To move the layer:
Top	To the top of the stack in front of all other layers.

Aligning Layers

Use Align Sel mode to align multiple layers and specify the direction for the alignment.

To align text layers:

1. In the image window, select two or more layers to be aligned. To select multiple layers, `Ctrl`-click the layers (or click the pen button).
2. In the Text Mode box, select Align Sel.
3. On the numeric keypad, press the number that corresponds with the direction in which you want to align the selected layers.

Press:	To:
4	Align left.
6	Align right.
8	Align to the top.
2	Align to the bottom.
5	Centre align.

The text mode returns to Move when the Align Sel operation is done.

Adjusting Text Leading

Use Leading mode to adjust the spacing between lines of text in a paragraph. The selected lines are adjusted in proportion to the leading values already specified. Use this mode to adjust the leading among paragraphs with various leading values.

Rekerning Text

Kerning refers to the space between characters that you can either increase or decrease. Use Rekern mode to change the kerning of all selected characters in relative proportion to the current kerning values.

Use Rekern to adjust the kerning among paragraphs with various kerning values.

To rekern text:

1. Select the string of characters you want to rekern.
2. In the Text Mode box, select Rekern.

   In the Paragraph menu, the Kern field changes to the Rekern field.
3. Enter a new value in the Rekern field or use the `up` and `down arrow` keys to rekern the text in single increments. Press `Shift+up arrow` or `Shift+down arrow` to rekern in increments of 10 pixels.

Resizing Text

Use Resize mode to change the font size of all the characters in a selection in relative proportion to the current font sizes. Use this mode if you have several font sizes represented in a selection.

The Size field allows you to apply an absolute font size value to selected text.

To resize text:

1. Select the string of characters you want to resize.
2. In the Text Mode box, select Resize.

   The Size field you used to specify the original size of the text changes to the Resize field.
3. Enter a new value in the Resize field or use the `up` and `down arrow` keys to resize the text in single increments. Press `Shift+up arrow` or `Shift+down arrow` to resize in increments of 10.

Offsetting Text Layers

Use Y Offset mode to shift selected layers along the vertical axis—the Y-axis. Use the `up` and `down arrow` keys to offset the selection.

Using Safe Title

Use Safe Title mode to align selected text within the safe title overlay. Use the numeric keypad (0-9) to specify the direction of alignment. Safe title is used to define the boundary for positioning text and how it appears in a rendered clip. By default, new layers appear in the upper-left corner of the safe title area.

To align to the safe title:

1. Select the layer you want to align with the safe title.
2. In the Text Mode box, select Safe Title.
3. Press the number on the numeric keypad that corresponds with the direction in which you want to align the selected layers. Press `1`, `2`, `3`, `4`, `6`, `7`, `8`, or `9` to determine the direction in which to align; press `5` to align the layer in the centre of the safe title area. Press `0` to align the layer horizontally in the safe title area.

Copying and Pasting Character Channels

You can copy character attributes, transformations, or both, from one character and paste them onto a selection of others.

To copy and paste character channels:

1. Select the text from which you want to copy the attributes or transformations.
2. In the Character Channels section of the Text menu, click Copy Attributes, Copy Transformations, or Copy All.

3 Select the text to which you want to paste the attributes or transformations.

4 Click Paste.

Grouping Text Layers

You can group two or more layers together to create a montage of text and move text layers together. Grouped layers can be saved and loaded as one unit. You can add more text layers to an existing group, resulting in a new single group.

NOTE When a group of layers is selected, you cannot switch to Edit mode.

To create a text layer group:

1 In the Text menu, select Move from the Tools box.

2 `Ctrl`-click (or click the pen button) to select two or more text layers, or click Select All to select all text layers in the scene.

3 In the Layer Attributes section, enable Group.

Loading Logos

You can use a logo so that it appears on every frame in a clip. Also, you can use logos in text rolls to credit sponsors, contributors, and products. Place a logo directly in a paragraph and then set text formatting properties for the logo.

To insert a logo in a paragraph:

1 In the layer Attributes section, click Load Logo.

The main workspace appears.

2 Select a front clip, and optionally, a matte clip for the logo.

The logo is inserted at the cursor position in the text layer.

3 In the Font and Paragraph menus, increase or decrease the size of the logo in the text layer by adjusting the font size, leading, and kerning. See Entering Text (page 1326).

To save a logo:

1 Select the logos you want to save.

You can select and save multiple logos.

2 Click Save.

3 From the Save box, select Selected Logos.

4 Use the File field to name the logo.

5 Click Save.

The logo is saved as a *.tif* file.

Tabulating Text

You can organize text into columns by setting tabs in the Tabulation menu. In a text layer, you can also set tab stops to align text at specific locations in a paragraph.

To access the Tabulation menu:

1 Create a text layer.

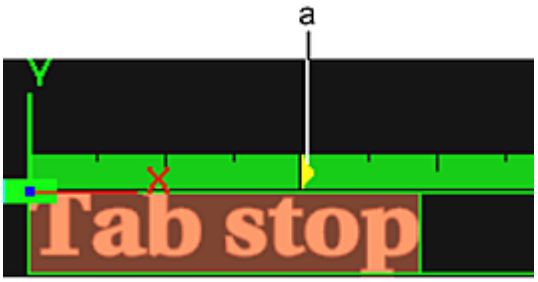
2 In the Text menu, click Tabulation.

The Tabulation menu appears.

The Tabulation controls are described as follows.

To set a tab stop:

1 In the Tabulation menu, click Add to add the tab stop in a text layer.



(a) Tab stop shown in the ruler

2 Click the tab stop and drag it to its new location on the ruler. Alternatively, enter a pixel value in the Position field.

Once you set tab stops for a paragraph, press `Tab` in subsequent paragraphs to type text in the location of the next tab stop. In a text layer, each time you press `Enter`, the tab stops you set in the previous paragraph are carried to the next paragraph.

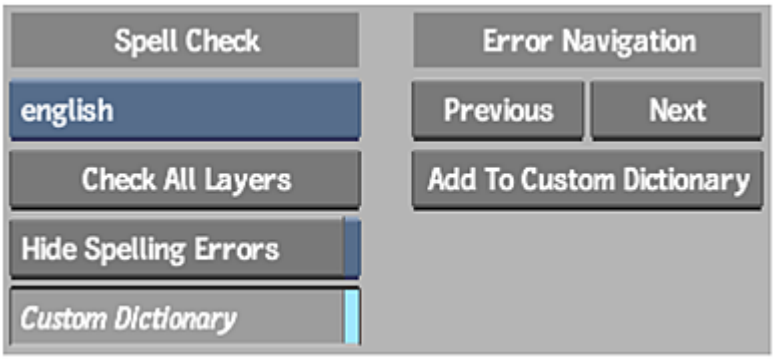
Spell Checking

In the Spell Check menu, you can check the spelling of the text in a text layer. When spelling errors are encountered, a red line is drawn through words not in its dictionary. By default, the spell checker uses the language and dictionary installed with the operating system to verify the spelling of words. You can also create your own custom dictionary.

NOTE If the spell checker is not installed on your system, the Spell Check menu controls are disabled.

To access the Spell Check menu:

1. In the Text menu, click Spell Check.
   The Spell Check menu appears.



To spell check a text layer:

1. In the Text menu, click Spell Check.
2. Enable Check All Layers.
   Lines appear through misspelled words.
3. To correct misspellings, in Edit mode, click Next or Previous to navigate to each misspelled word.

Creating Text Rolls and Text Crawls

You can create text rolls, which are commonly used for credit rolls. A text roll is a layer of text that rolls from the bottom to the top of a clip for a series of frames. In a text roll, you can use special fonts as well as coloured, animated, and tabulated text.

You can include logos of sponsors, contributors, and products in text rolls. For example, you can create a list of credits that uses a green, sheared font for all contributors' names and includes a single, larger capitalized letter spinning on an axis at the beginning of each title line. The contributors' names can appear in a white, semi-transparent, Courier-type font.

A text crawl scrolls text horizontally across the frame. For example, you can create a list of phone numbers that scroll from left to right across the bottom of the screen during a telethon broadcast.

To access the Roll/Crawl menu:

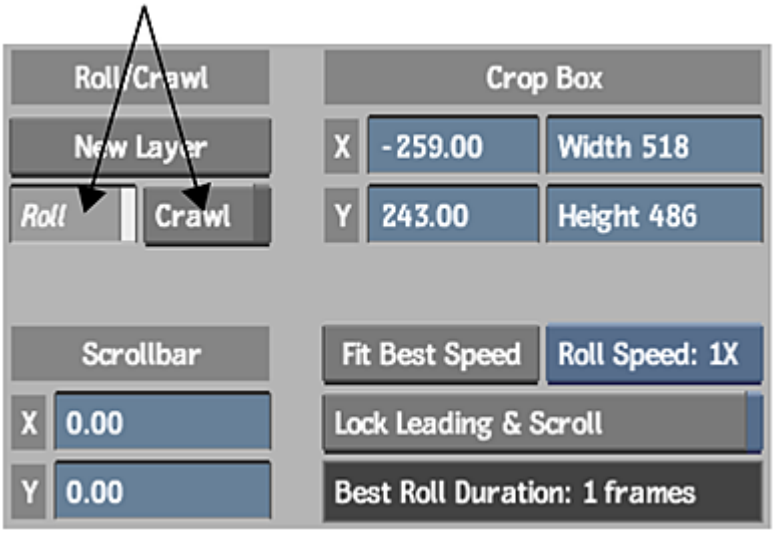
1 In the Text menu, click Roll/Crawl.
The Roll/Crawl menu appears.

The Roll/Crawl controls are described as follows.

To create a text roll or crawl:

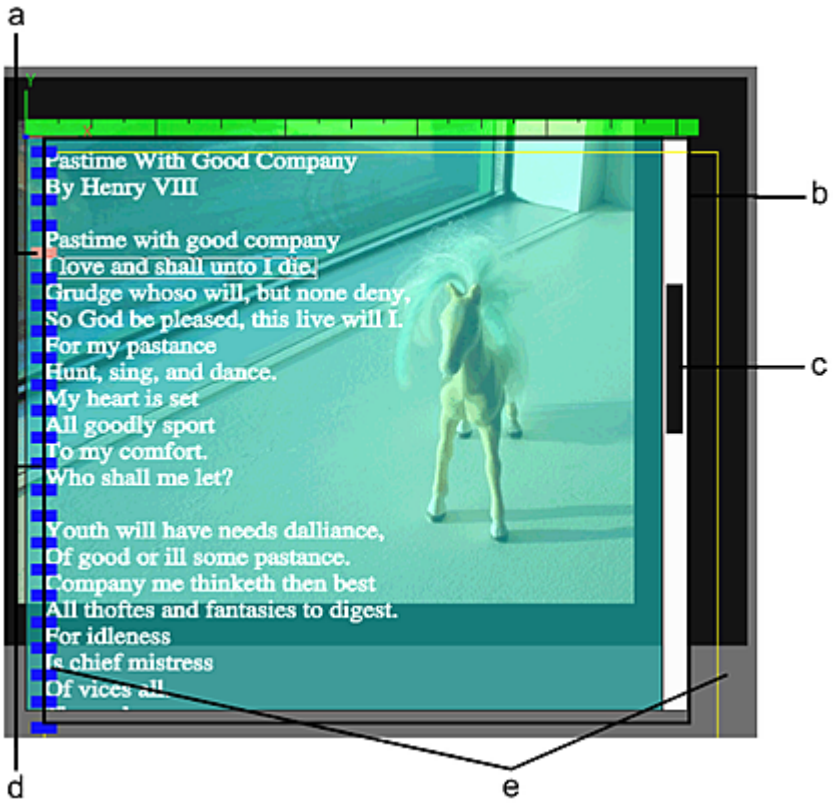
1 In the Text Setup menu Rendering section, enable PreRender, and then select Interlaced from the Render box.

2 In the Text menu, select Interlaced from the Render box.
3 Enable Auto Key.
4 Click Roll/Crawl to enter the Roll/Crawl menu.
5 Go to the first frame of the clip, and enable Roll or Crawl.



6 Click New Layer and add the text you want to use for the text roll or crawl.

A shaded layer appears in the image window with a vertical scrollbar for a roll or a horizontal scrollbar for a crawl. The shaded area represents the crop box area. The following is an example of a text roll.



(a) Red indicates currently selected paragraph (b) Crop box (c) Scrollbar (d) Blue indicates paragraphs in text roll layer (e) Yellow border delimits text

7 To set the start position of where the text appears in the text roll or crawl, use the Scrollbar X and Y fields or drag the scrollbar up or down (roll), or left or right (crawl).

8 Go to the frame where you want the text roll or crawl to finish and set the end position by dragging the scrollbar or using the Scrollbar X and Y fields.

9 If you want to calculate the ideal number of frames for the text roll, choose a roll speed: 1X, 2X, 3X, or 4X.

The suggested duration appears in the Best Roll Duration field. You can either change the number of frames in your clip to match the one in the Best Roll Duration field, or you can leave the clip as is.

10 If the suggested duration is acceptable, enter it in the Duration field and then click Fit Best Speed.

The roll is created and the Scrollbar position and Leading are changed (if required) and locked.

11 If the suggested duration is not acceptable, click Fit Best Speed.

The roll is created and the Scrollbar position and Leading are changed (if required) and locked.

12 Click Render to render the text roll and view the results.

When you play the clip, the text rolls or crawls through the image area from the first position to the final position.

NOTE When rendering text rolls and crawls, use PreRender in the Text Setup menu to render the roll or crawl relatively fast while bypassing the animation. Check the message bar to see whether PreRender is enabled.

TIP When creating credit rolls, the anti-aliasing on the text should be turned on.

Animating Text

You can animate text properties including colour, softness, size, shadow, and transformation effects such as rotation, scale, and shear. For example, you can animate text channels to create an effect where the text appears gradually over time and then disappears from view.

Use Auto Key to create keyframes automatically as you make changes to text attributes. The movement between keyframes is interpolated according to the interpolation mode you set.

NOTE PreRender must be disabled when creating text animations. See Rendering Considerations (page 1323). Check the message bar to verify the PreRender status.

To animate text automatically:

1 In the Text Animation menu, enable Auto Key.

2 Go to the first frame in the clip.

3 With the Attribute controls, set the attributes and transformation properties.

4 Navigate to another frame in the clip and add more attributes and transformation properties according to the effect you want to produce.

5 Click Preview.

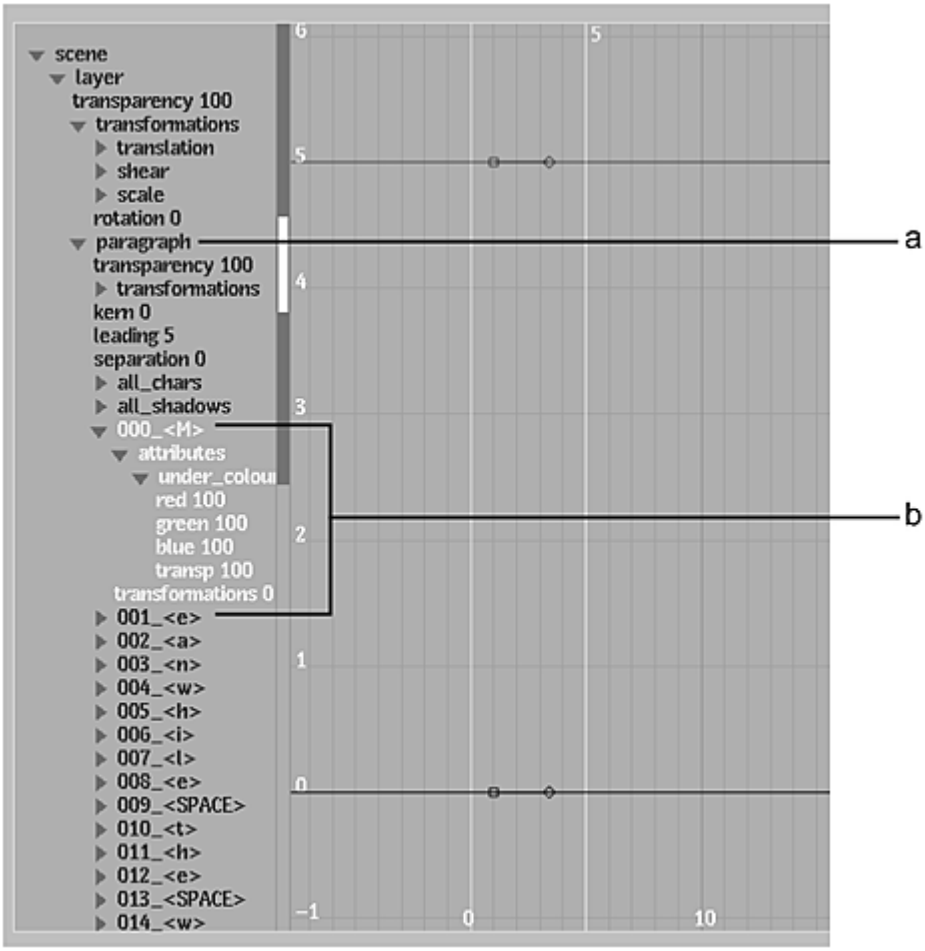
6 Make modifications as required.

7 Click Play to view the result.

When you play the clip, the animation moves from the first frame to the last.

Animating Paragraph Channels

In the Text Tool, the Animation controls include a Paragraph Channel View for viewing text channels in the Channel Editor. Use the attributes and transformation channels to animate paragraphs and characters and fine-tune animations created using the Attributes menu.



(a) Paragraph folder contains all the characters, including spaces between words (b) You can animate specific channels for each character

To animate paragraph channels:

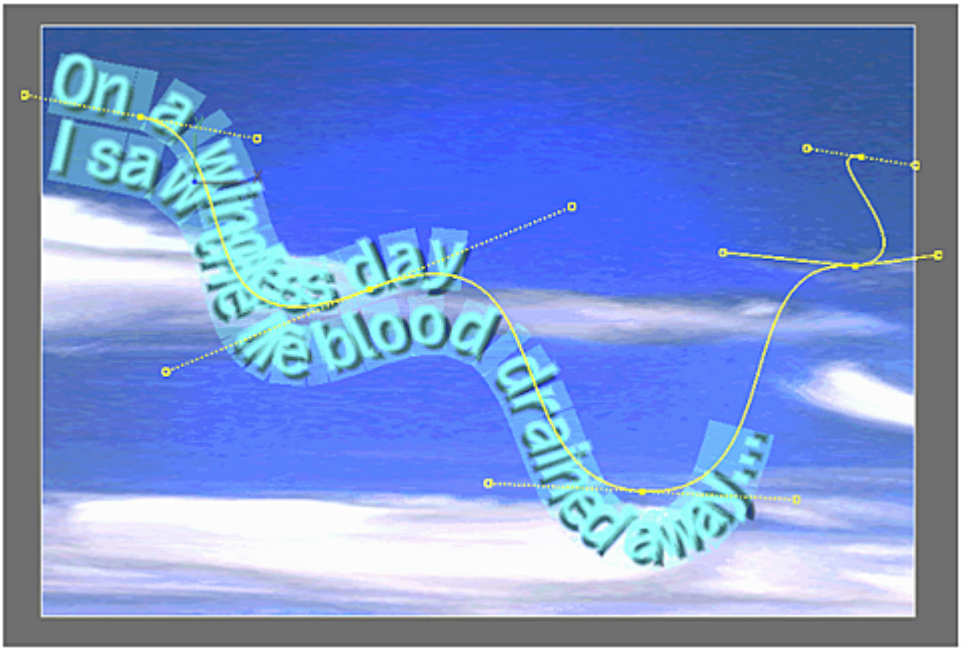
1. In the Text menu, click Animation to display the Channel Editor.
2. From the Paragraph Channel View box, select the paragraph channels you want to view in the Channel Editor.

Select:	To view:
Current	The channels for the selected paragraph—the current cursor location.
Layer	The channels for all paragraphs in the selected layer.
All	All channels for all paragraphs in all layers.

3 Animate the selected channels as required.

Animating Text on a Motion Path

You can create a text layer on a spline and animate it over time. In this way, you create marquee text that twists and turns over an image or in time with other objects in the scene. The spline is fully editable; you can add control points and manipulate their tangent handles.



You can set custom colours for the text path wireframe and vertices in the Text Setup menu. See Setup and Rendering Options (page 1321).

To animate text on a motion path:

1. Create a layer or select a layer in the image window.
2. From the Text On Path section of the Text menu, enable On Path.

By default, the text appears on a path with three vertices.

3. Adjust and animate the shape of the path using the Text On Path controls.

The Text On Path controls are described as follows.

Select:	To:
Move	Modify the shape of the spline by changing the position of the vertex or tangent.
Add	Add more vertices to the spline to create a more complex shape with text. Click anywhere on the path to add a vertex.
Delete	Remove a vertex from the spline.
Break	Separate two tangent handles and move them independently. Click a tangent; the tangent handle changes from a dashed to a solid line to indicate it is broken.

Select:	To:
Auto	Connect broken tangents automatically. Click a tangent to reconnect it.

In the Channel Editor, the Path group contains a Shape channel to animate the path and an Offset channel to animate the offset value.

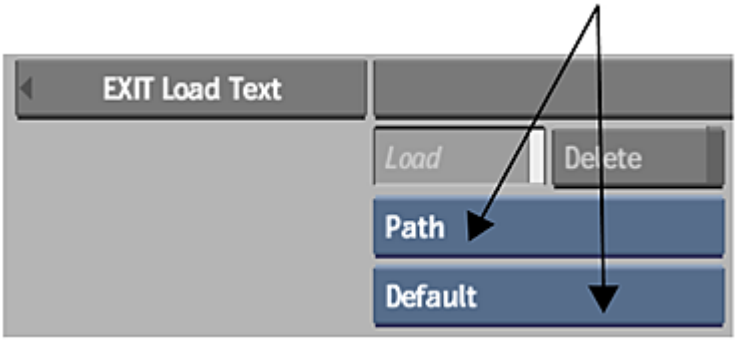
Saving and Loading Path Setups

You can use a garbage mask spline as a shape for the motion path of text on a path. You can save and load the Shape channel as a list in raw shape data (*.raw* file), which is compatible with garbage mask raw data. Raw setups between text paths and garbage masks are interchangeable.

To load a raw shape data file:

1. In the Text On Path controls, enable On Path.
2. Click Load.

   The Text load options and file browser appear.
3. In the file browser, select Path and then select Default from the Text Path loading boxes.



   A set of path files is visible when the Path and Default options are selected in the Load Text menu.
4. Browse to find the file you want to load.

   TIP Gmask setup files are typically located in the */opt/Autodesk/project/<project_name>/path/*directory.
5. Click Load.

   The spline appears with its new shape in the image area.

Defining Styles

Styles are sets of text attributes and specifications that can be replicated when you need to use the same format again. You create and modify styles, and save and load them from the file browser using the Styles menu. The following character attributes are saved with the style: font, colour, outline, shadow, blur, scaling, rotation and shear. In addition, the layer's background colour and opacity are saved as well.

To access the Styles menu, click the Styles button in the Text menu.

(a) Style Mode box (b) Style Option box

To define a style:

1. Select a character with the attributes you want to save.

   You can select multiple characters, but it is the first one that defines the style.

2. From the Style Option box, select Define Style and then click the style button to which you want to assign the style.

3 Click Name and enter a name for the style. Then click the style button to which you want the name to apply.

The new name appears on the button.

NOTE You can use `Ctrl+Shift` and `F1` to `F9` to assign text attributes to Style buttons 1 through 9 from any menu.

To modify a style:

1 Select the character with the attributes you want to save.

2 From the Style Option box, select Define Style and then click the style button to which you want to assign the style.

NOTE You can use `Ctrl+Shift` and `F1` to `F9` to assign a style to Style buttons 1 through 9 from any menu.

To clear a style:

1 From the Style Option box, select Clear Style and then click a style button.

NOTE It is sometimes convenient to clear all styles just prior to using AutoStyle to save multiple styles automatically. AutoStyle assigns up to nine style buttons at once. See Saving Multiple Styles Automatically (page 1347).

To apply a style:

1 Select a paragraph or string of characters to which you want to apply a style.

2 Press `Ctrl+F1` to `F9`, or press the style button, to apply the style to the selected text.

Saving and Loading Styles

After you create styles, you may want to save one or all of them. You can load any of them in future sessions.

To save a style:

1 Do one of the following in the Styles menu:
- Select All Styles from the Style Mode box.
- Select One Style from the Style Mode box, and the style you want to save (Style 1 to Style 9).

2 Click Save Style.

The Save Styles menu and file browser appear.

3 Choose a directory for the style.

NOTE You can click the Create Dir button to create a new directory in which to save your styles.

4 Enter the filename.

The style is saved. The file browser automatically closes and the Styles menu reappears.

To load a style:

1 Select an option from the Style Mode box.

Select:	To:
All Styles	Load files saved with multiple styles. The file browser lists setups saved with multiple styles.
One Style	Load files saved with one style only. The file browser lists setups saved with a single style.

2 Click Load Style.
The Load Styles menu and file browser appear.

3 Navigate to the appropriate directory and select the style(s) you want to load.

4 If necessary, click Exit Load Text to exit the file browser.

Saving Multiple Styles Automatically

If you create a section of text with multiple sets of attributes, you can assign each set as a separate style with a shortcut and then save them. The Text Tool automatically detects the different sets of attributes used in the section and assigns each set to a Style button. You can then use shortcuts to apply them.

To assign styles from an existing layer using shortcuts:

1 Select the section of text.

2 From the Style Option box, select AutoStyle.

NOTE If you used more than nine sets of attributes in the selection, only the first nine are assigned to a Style button.

3 Save the styles.

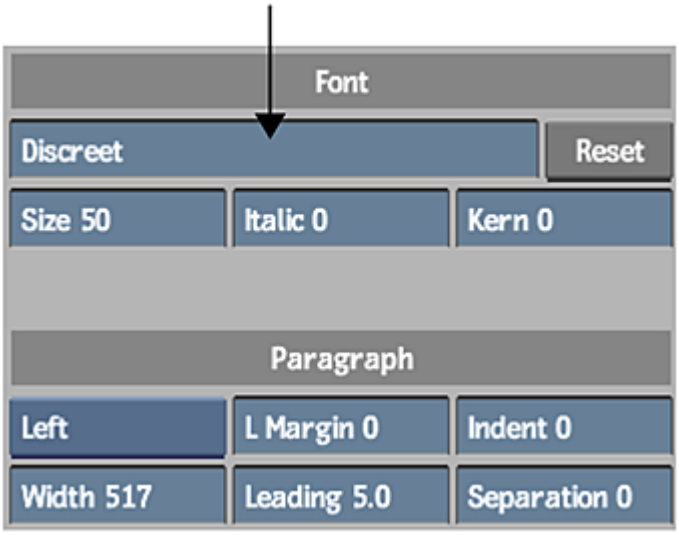
Changing Fonts

When you open the Text tool, the default font is loaded and ready to use. When you choose a different font, it becomes the current font and all text you type appears in the current font. If you edit text that has a different font, the current font is replaced by the font of the text at the cursor position. For example, if you are working with Carta font and you edit text that uses Helvetica, Helvetica becomes the current font and all subsequent text you type appears in Helvetica.

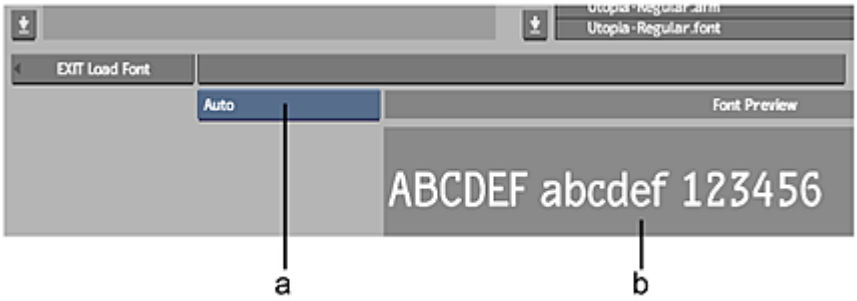
You specify the default font in the **Timeline FX ➤ BFX** tab of the Preferences menu. Also, you can install additional fonts for use in the Text Tool.

To change the current font:

1 In the Font section of the Text menu, click the Font field.



The font library appears.



(a) Font Type box (b) Font Preview window

2 In the Font Type box, select the font type you want to load.

Type	Description
Type 1	A single-byte font type designed for Roman language fonts. Soft-links to the Adobe® Postscript® Type 1 fonts installed with the system or supplied by Autodesk are contained in */opt/Autodesk/font* directory.
CID	A multi-byte adaptation of Adobe Postscript Type 1 fonts, well-suited to representing the large character sets of Asian languages such as Japanese, Korean and Chinese.
TrueType	A widely used cross-platform font format.
OpenType®	A font developed by Adobe and Microsoft® that supports expanded character sets and layouts.
Auto	Displays all font types, detecting the type automatically.

3 In the file browser, navigate to the directory that contains the font you want to load.

4 Preview a font. Do one of the following:

- Select a font to see it with sample text in the Font Preview window.

The sample text can be the text you selected in the text layer, or text that you enter by clicking the Font Preview window to display the on-screen keyboard.

- In the Proxies/Titles box, select Proxies to preview fonts. If no proxy appears, click Generate Proxies to generate a font proxy.

  This process may take a while, but once you create the proxies, the settings are retained. Each time you return to the font library, you can toggle between viewing font titles and font proxies by selecting Proxies or Titles.

5 Select the font.

6 Click Load.

The Text menu reappears with the name of the new font in the Font field.

Accessing OTF Fonts

In order to access symbols of the OTF font type, access the on-screen keyboard, enable the Symbols button (in the lower right of the keyboard), then use the arrow keys to browse through the pages of symbols until you find the character set you need.

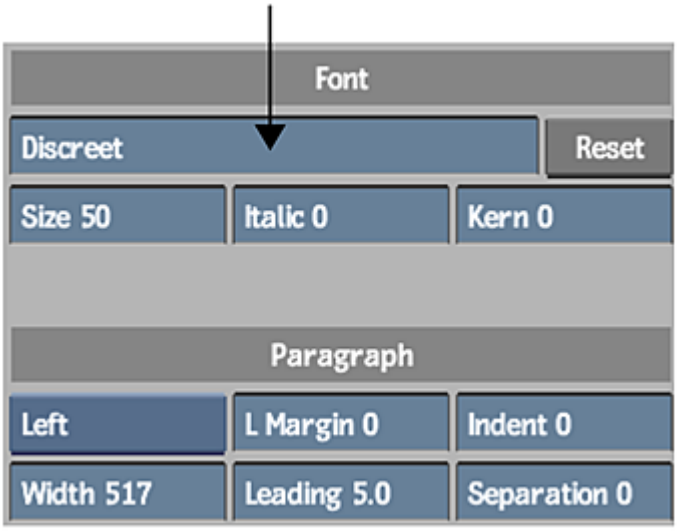
Pasting Text from an External Source

You can copy text from a terminal or another application and paste it directly in a text layer.

To paste text to the Text tool:

1 In the Text tool, create a layer for the text you want to paste.

2 Click the Font field and select a font.



3 Stitch to the other application and copy the text.

4 In the Text tool, paste the text in a text layer (using the default system shortcut for Paste).

Entering Non-Latin Text

When entering non-Latin text characters, consult your operating system documentation to make sure how to install and access the required input method packages.

The following example illustrates entering Japanese text in a text layer on a Linux Red Hat system.

To enter Japanese text in a text layer:

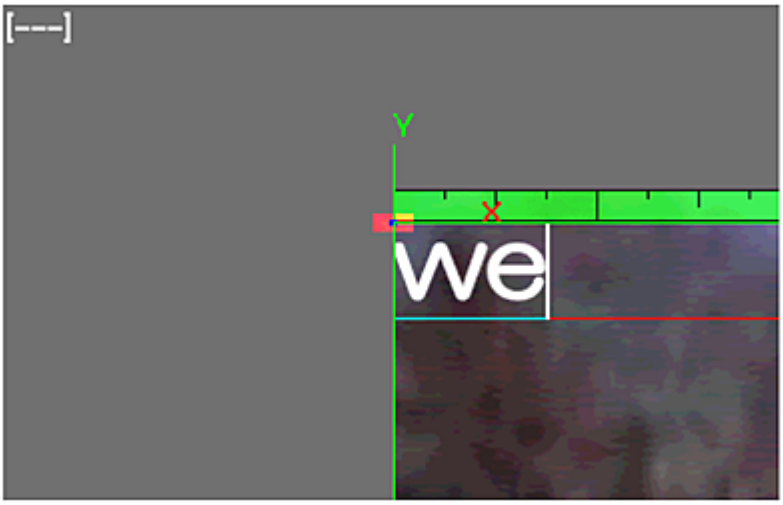
1. Open the Linux IM Chooser in **Applications ➤ Settings ➤ Input Method**.
2. Select Use IBus as the Input Method, then click Input Method Preferences.
3. In the General Tab, make sure that "Embed preedit text in application window" is enabled.
4. In the Input Method tab, select your language and input method from the list.
5. Start your Autodesk application.
6. Load a clip in the Text tool.

   Notice the message bar that appears at the top of the screen. This indicates you are in Latin input mode.

(a) Message bar

7. Create a layer and click the Font field to select a Japanese font from the file browser. From the Text Mode box, select Edit and then select the layer in the image window.

   When you type text, notice that you are still in Latin input mode.



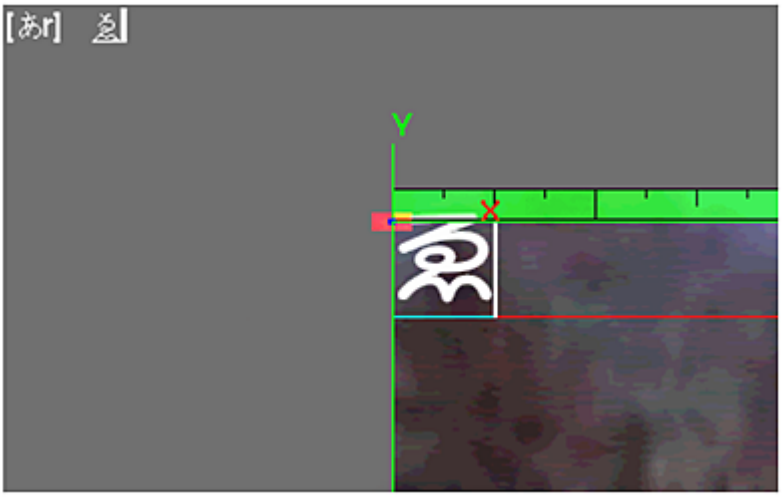
8. Press your Input Method keyboard shortcut (by default, this is `Ctrl+spacebar`).

   The message bar changes to indicate that you are in Japanese text mode.

(a) Message bar

9 Type text, as required.

10 Press `Enter` to accept the characters and add them to the text layer.



11 To revert back to Latin input mode, press the Input Method keyboard shortcut again.

Using Shaders to Create Your Own Effects

24

Matchbox and Lightbox are interactive development tools that allows you to run OpenGL Shading Language (GLSL) shader code, to add specific functionality, or create custom effects. GLSL is a high-level shading language that is part of the OpenGL specification.

You can create your own Matchbox and Lightbox shaders to use in Flame. You can also create preset looks for your shaders by exporting settings into a shell directly from Flame. Tools are provided to build and test your shaders, and a Shader Builder API Guide is available to help you at *http://www.autodesk.com/shader_builder_api_guide* .

TIP You can find many more shaders by visiting the community-driven shader repository at *https://logik-matchbook.org/*. You can download free shaders created by fellow Flame users, or share your own custom shaders with other users.

About Matchbox

Because of the nature of GLSL fragment shaders, Matchbox works well on image processing effects. You can however, create simulated 3D effects using a number of image processing techniques, such as a Z-depth pass, for example.

The Matchbox node populates the user interface dynamically, based on the parameters required by the shader. By default, Matchbox has six physical inputs, but you are not limited to the amount of actual inputs you can use in the effect, since you can use the same image for more than one input. You can also specify colours as inputs, such as black or white; in this case, you may need to select User Defined in the Output Resolution box in the Matchbox menu to be able to view the colour result.

Re-purposing of existing effects is easy, since Matchbox shaders use simple generic GLSL fragment shader code, with no required customization. Included are a number of useful examples and presets, that can be used as is, or serve as starting points for you to develop your own tools. If the loaded shader has any preset starting points built into it (such as in the Duotone shader), you can find them in the Presets list in the Shader tab of the Matchbox menu.

Matchbox shaders can also be used as timeline transitions. Flame includes some Matchbox transition presets (see Using Matchbox as a Transition (page 1363)). The Matchbox XML schema contains tags to identify if the shader is designed as an Action node, a Timeline FX or a Transition. The tags *ShaderType* and *SoftwareVersion* are added automatically to the XML by the *shader_builder* script.

Matchbox can be accessed from multiple places:

- Batch, then select a node from the Node bin.

- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- The Modular Keyer, then select a node from the Node bin.
- Action or GMask Tracer, then select a Matchbox node. See Matchbox in Action (page 821) for specific information.
- Tools, then select from the menu (See Accessing Tools (page 493)).

NOTE In Batch, Batch FX and the Modular Keyer, the Matchbox node's input tab colours are based on the type of input (for example, yellow for composite, blue for matte). For some Matchbox shaders, you may need to connect all inputs before you can see a result.

When selecting a Matchbox shader from the file browser, you can use the File Format box to display shaders in .glsl format or encrypted Matchbox .mx format. Some more efficient, complex, or sophisticated effects can be split into multiple passes. Make sure to select the parent .glsl shader from the browser if you want to load a multipass Matchbox shader (the .mx format packages all passes into one file).

NOTE When accessing Matchbox in Batch, Batch FX, and Action, you can also use a dynamic Matchbox node bin to quickly access all Matchbox shaders housed in a folder of your choice. See Matchbox Node Bin Tab (page 550) and Matchbox and Lightbox Node Bin Tabs (page 550).

Matchbox in Action

While you can use interactive Matchbox shader effects elsewhere in Flame (such as from the timeline or Batch), using Matchbox in Action presents some unique benefits:

- Matchbox effects in Action can be parented to a texture node (or multiple texture nodes). Multiple Matchbox effects can also be added to a texture node. Supported texture maps are Diffuse, Normal, UV, Displacement (HW Only), Parallax, IBL, Specular, Emissive, Lens Flare textures, and Substance textures. These Matchbox effects occur in texture space, after Media list effects but before Action objects, and can be seen as pre-processing effects.
- You can also connect a Matchbox shader to an Action camera, to create a Camera FX that is applied to the whole scene. Camera FX are scene-based post-processing effects.

Keep in mind that the Matchbox XML schema contains tags to identify if the shader is designed as an Action node, a Timeline FX, or a Transition. The tags *ShaderType* and *SoftwareVersion* are added automatically to the XML by the shader_builder script.

The Action Priority Editor (page 565) has a Matchbox Only mode to view and change the drawing order of Matchbox effects.

See the sections below for information on how to add, and things to watch for, when working with each type of Action Matchbox effect.

Matchbox Shaders Connected to Texture Nodes

To add a Matchbox node in Action connected to a Texture node:

1. Do one of the following:
   - With a supported texture node selected in the schematic (Diffuse, Normal, UV, Displacement (HW Only), Parallax, IBL, Specular, Emissive, Lens Flare texture, or Substance texture), double-click the Matchbox node in the All Nodes bin.

     The file browser opens to allow you to select a Matchbox effect.
   - With an Action surface or 3D Geometry node selected in the schematic, double-click a Matchbox preset node from the dynamic Matchbox node bin (page 550).

The Matchbox node with a parented Diffuse node (with its own parent axis) are added to the scene as children of the selected surface or geometry.

TIP If you add a Matchbox node to the schematic without a supported texture node present, you can parent the Matchbox to an image, and a Diffuse Map node (with accompanying Axis) is automatically added to create a proper Matchbox connection. Matchbox nodes are not supported on Stereo objects.

Matchbox nodes and their links are displayed as black in the schematic to help you identify them. If you added the Matchbox from the dynamic node bin, the name of the effect also appears on the node in the schematic.

2 Double-click the Matchbox node in the schematic to display its menu.

The Matchbox node populates the user interface dynamically, based on the GLSL (and XML) code, but there are a few UI elements that are constant, located in the Shader tab.

3 If you added a Matchbox effect from the All Nodes bin, you can change the effect by clicking Change Shader in the Shader tab of the Matchbox menu.

4 Use the Action Object Solo (`F8`) view on the parent texture to cycle through three Matchbox views: Context, Result, and Result Matte. The `F8` view also allows you to interact directly in the image window with icons, if they are present in the shader.

TIP Instead of using a source node for a simple front/matte transform, consider using the Front Matte Offset Matchbox (available as a preset, along with many other useful Matchbox effects).

Matchbox Shaders Connected to Cameras (Camera FX)

You can connect Matchbox shaders to the Action camera and have the result of the shader applied to the result of the Action scene. Similar to using Texture Matchbox shaders, the priority editor is used to handle the pipeline order.

NOTE Post-processing (Camera FX, Lens Flares, and Rays) effects are only visible in the Live Preview view. Camera FX can have 2D and 3D widgets visible in the Live Preview view. Only the widget(s) of the currently selected Matchbox are shown. Lens Flares and Rays are rendered at the end of the Camera FX pipeline, when enabled in the Comp output.

GMasks can be connected to a Camera to limit their effect to the post-processing pipeline exclusively. GMask output passes have an option to only show GMasks connected to the current camera, so you can set GMask to not interact with the scene, and only be created to be used with the post-processing pipeline. You can also use a GMask input Matchbox shaders (if they have been implemented to use this input).

The Action post-processing pipeline can connect the Matchbox shader automatically to the appropriate Action output. To support this, shader developers need to set the *InputType* tag properly in the shader XML (each Action output has its equivalent *InputType* tag). This includes RGBA shader rendering. The following input types are supported:

- Front (this defines the Primary Output render layer selection).
- Back
- Matte
- 3D Motion Vectors
- Albedo
- Ambient Occlusion
- Comp (this defines the Action Comp output of the render layer, in case the Primary is another type).
- Emissive
- GMask
- Lens Flare
- Motion Vectors
- Normals
- Occluder
- Position
- Reflection
- Roughness
- Shadows
- Specularity
- UV
- Z-Depth
- Z-Depth HQ (32-bit Z-Depth output packed in two 16-bit channels; you must use the Matchbox API to leverage the data).

Matchbox Camera FX also have access to the Acton output list in their patch panel (in the Shader tab on the Matchbox menu), so you can manually set the proper connection if the Matchbox shader hasn't been tagged properly in the XML.

To add a Matchbox node in Action connected to a Camera:

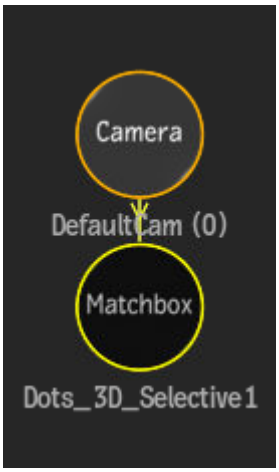
1 Do one of the following:

- With a Camera node selected in the schematic, double-click the Matchbox node in the All Nodes bin.

  The file browser opens to allow you to select a Matchbox effect. Some Matchbox shaders supplied with Flame are already optimized for use as Camera FX; navigate to the ACTION_CAMERA_FX sub-folder.

- With a Camera node selected in the schematic, double-click a Matchbox preset node from the dynamic Matchbox node bin (page 550).

  The Matchbox node is added to the scene as a child of the selected camera. Once a Matchbox node is parented to a camera, it becomes a Camera FX, and you are not able to connect the Matchbox node to any other node in the schematic (other than other camera nodes).



Matchbox nodes and their links are displayed as black in the schematic to help you identify them. If you added the Matchbox from the dynamic node bin, the name of the effect also appears on the node in the schematic.

NOTE Since a Matchbox connected to a Camera node (Camera FX) is considered a post-processing effect in Action, make sure that your image window is set to Live Preview mode to be able to see the results of the shader.

2 Double-click the Matchbox node in the schematic to display its menu.

The Matchbox node populates the user interface dynamically, based on the GLSL (and XML) code, but there are a few UI elements that are constant, located in the Shader tab.

3 If you added a Matchbox effect from the All Nodes bin, you can change the effect by clicking Change Shader in the Shader tab of the Matchbox menu.

Rendering Camera FX

When rendering a Camera FX pipeline, Action uses Multi-Render Targets (MRT) to allow outputs to be much more efficient. Depending on the Rendering settings, the scene is rendered only once or twice instead of as many times as there are output passes, leading to a significant speed increase in the case of Camera FX requiring many input connections. The biggest speed improvement can be found when the Accumulation Rendering (page 603) box is set to Accumulation On or Accumulation Off, since all passes can then be done using a single MRT. In the case of Accumulation RGBA, two MRTs are needed, which reduces the speed improvement compared to the other Accumulation settings, but still offers a significant boost compared to not using MRT at all.

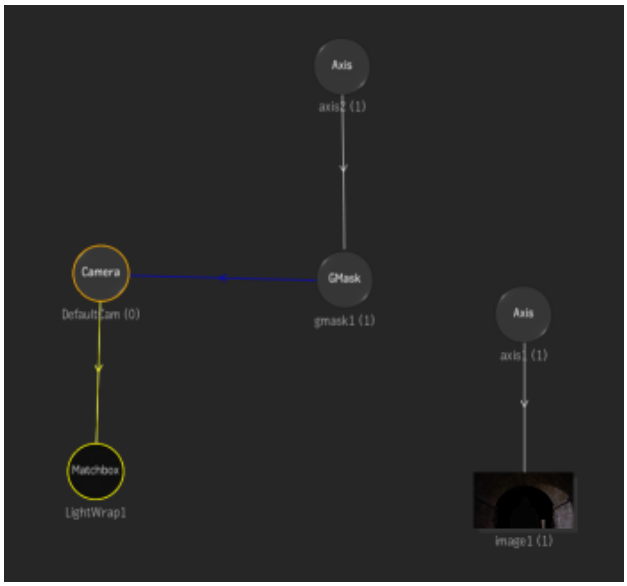
Matchbox in Action Examples

The following examples can help you to get a feel for the workflow of using different types of Matchbox effects in Action.

TIP Post-processing (Camera FX, Lens Flares, and Rays, Blooming, and Ambient Occlusion) effects are only visible in the Live Preview view.

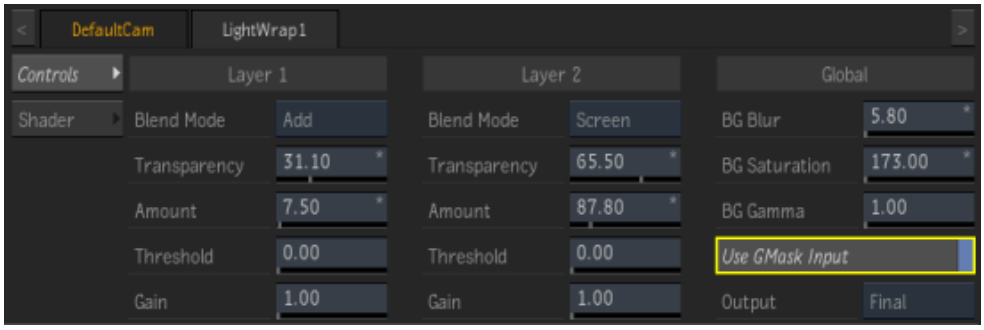
Using a GMask to Limit a Camera FX Matchbox

A GMask connected to the Camera with a GMask link can be used to limit the effect of a Camera FX Matchbox.



In this example, a Gmask is used to limit the effect of a Lightwrap effect that works well for the roof, but is too strong for the character. The Gmask limits the effect and the gradient gradually softens out the Lightwrap, so the top of the head of the character still has a little bit of Lightwrap.

To be able to use a GMask in this manner, you need to used a compatible Matchbox shader and enable Use GMask iInput in the menu. GMask output passes have an option to only show GMasks connected to the current camera, so you can set GMask to not interact with the scene, and only be created to be used with the post-processing pipeline.



Without Lightwrap

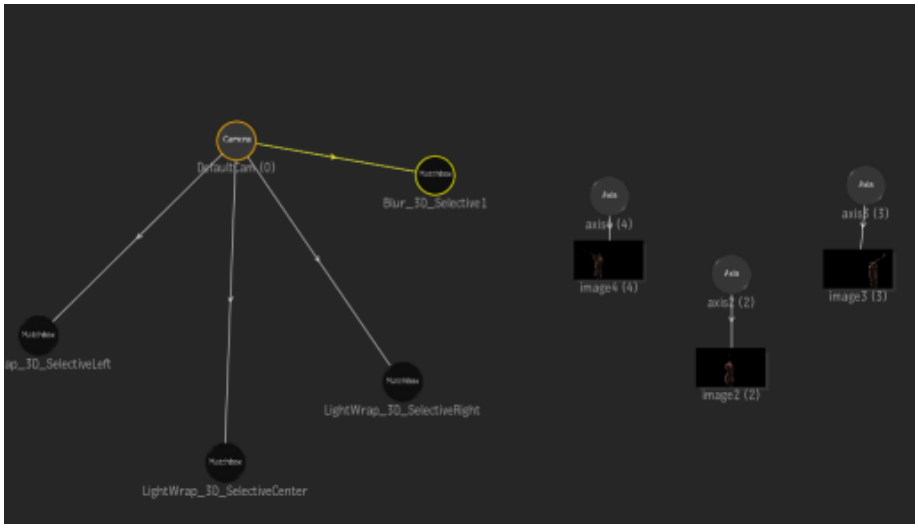
Without GMask

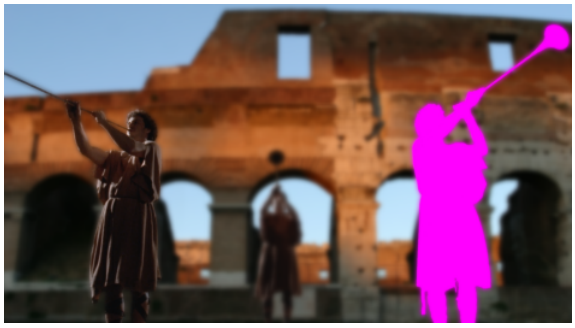
With GMask and Lightwrap

3D Selectives as Camera FX

Some Matchbox shaders include 3D Selective functionality, allowing you to segment the render of Action using different 3D techniques, applying the given effects to a specific portion of the image.

The following example shows the usage of 3D Selective with both the Lightwrap and Blur 3DSelective shader. The Blur is used to create a DepthOfField effect using the Distance mode, while three different Lightwrap3DSelective shaders are used to create independent Lightwrap effects on each of the trumpeters, as single values for all three didn't worked out in this case.

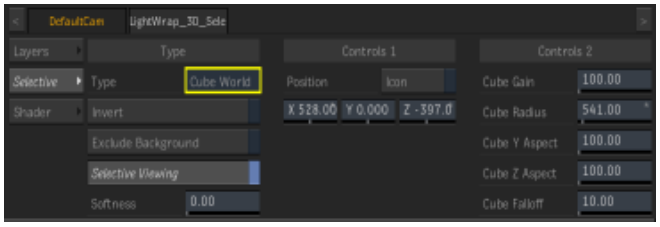




3D Selective on one trumpeter

Result

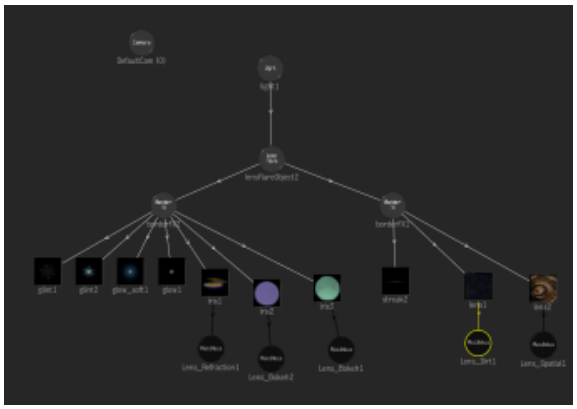
For each Lightwrap, the Cube World Selective type was used to isolate each trumpeter.



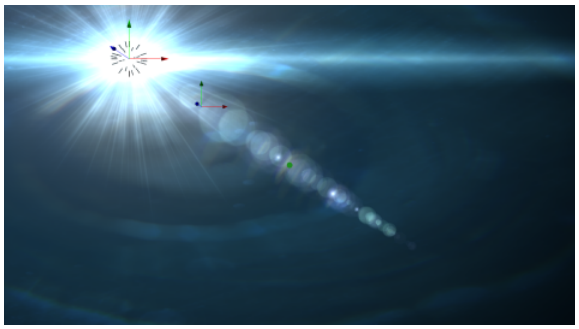
Replacing Lens Flare Elements with Matchbox Effects

You can use the Matchbox Lens texture grid shaders with Action Lens Flares. You can use the LensBokeh, LensRefraction, and LensOptic Matchbox to replace Glint Texture, Glow Texture, Iris Texture, Ring Texture

and Streak Texture nodes to create more organic Lens Flare results; and you can use LensDirt and LensSpatial Matchbox to replace the Lens Texture to create more organic lens dirt and light leak type of effects.

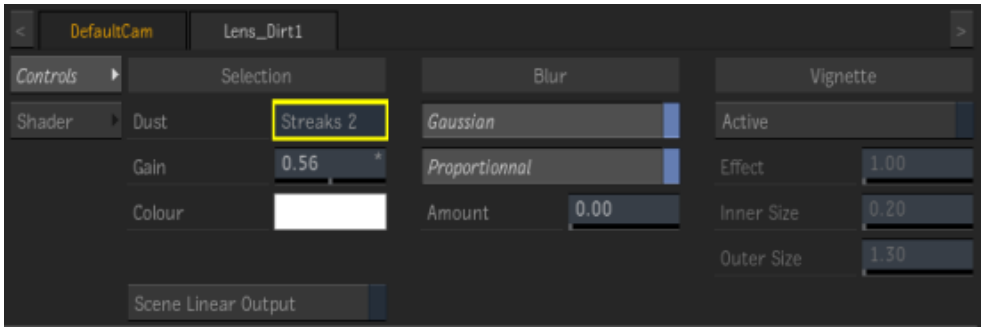


Lens Flare schematic with Matchbox replacements



Result

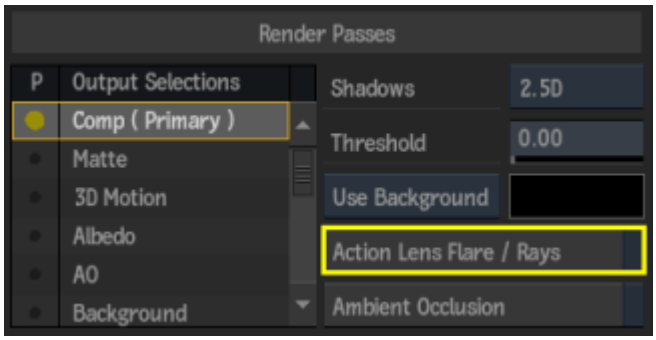
In each Matchbox Lens effect's menu, you can use the texture grid functionality to select different looks. In this example, You can select different Dust elements for the Lens Dirt effect.



Using a Matchbox Lens Flare in the Camera FX Pipeline

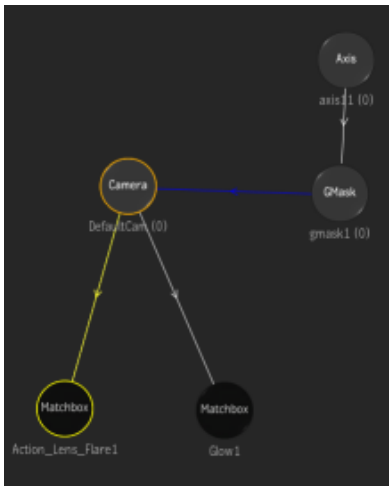
You can also use a Lens Flare Matchbox shader as part of the Camera FX pipeline. Since Lens Flares (and Rays) are rendered at the end of the Camera FX pipeline, you can use instead the ActionLensFlare Matchbox shader, which allows you to put the Lens Flare result anywhere in the Camera FX pipeline.

For this use case, make sure that you disable the Action Lens Flare / Rays button from the Comp output Render Passes menu:

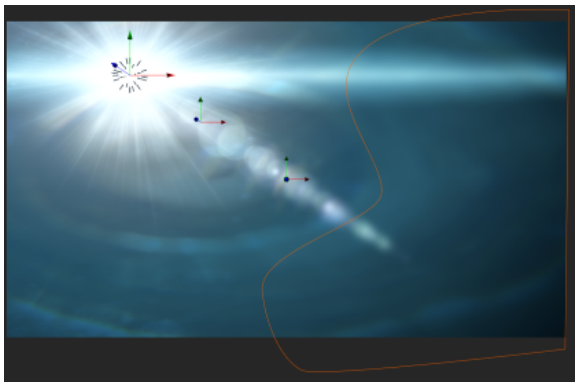


In this example, there is also a Matchbox Glow effect in the Camera FX pipeline, as well as a GMask that allows you to blur part of the Lens Flare result without affecting the Action result. The Glow can affect both

the Lens Flare and the Action result, since the ActionLensFlare Matchbox allows you to place it anywhere in the rendering pipeline (and in this case before the Glow effect).



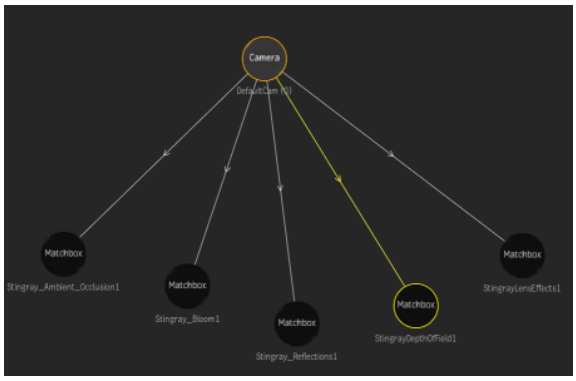
Lens Flare Matchbox as Camera FX



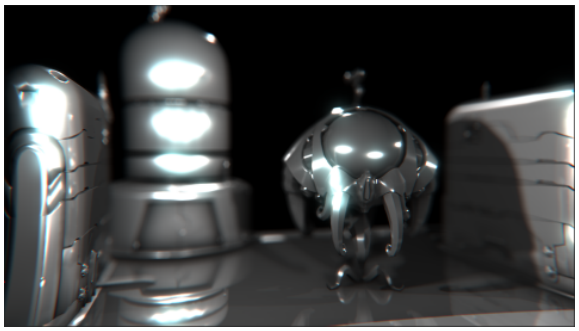
Result

Using Stingray Matchbox Shaders

Stingray Matchbox shaders are based on the Autodesk Stingray game-engine technology, allowing you to produce visually stunning results in Action. These shaders can be combined (as in the Camera FX example below). You can also use GMasks and 3D Selectives with the Stingray shaders to help you limit the effects to certain areas of your image.



Stingray Matchbox Shaders as Camera FX



Result

These are the available Stingray Matchbox shaders, as well as some usage tips:

Stingray Motion Blur This shader allows you to create motion blur effects, based on the motion vector provided (either by the Action Motion Vector output or the Batch Motion Analysis node). This shader can also use a Z-Depth input to help provide better results. When used as Camera FX in Action, it can automatically receive the current Camera Near/Far values, when Frustum is set to Auto, otherwise it can be set manually. The Exposure field is available to allow control on how much motion blur to create. While this tool can create quite nice small and subtle motion blur effects, it cannot create large motion blur effects due to the nature of the technique used; therefore the Exposure value is limited to 10.

Stingray Ambient Occlusion This shader allows you to create lighting occlusion, by computing how each point in a scene is exposed to ambient lighting. To do so, the shader requires the Z-Depth and Normals information of a 3D scene. You can control its impact in the scene by modifying the Intensity, Radius, and

Quality fields. The Use Z-Depth setting allows you to use a normal Z-Depth input, since, by default, the shader is expecting the Z-Depth HQ Action output for maximum quality (however a regular 16-bit Z-Depth input can be used). The Only AO setting allows you to output just the AO result without the scene result. When used as Camera FX, you can use both the Use GMask Input and the Selective options to tailor exactly where and how much ambient occlusion effect you want in your image. Multiple Stingray Ambient Occusion shaders can then be added, using different values and affecting different areas of the scene.

Stingray Bloom This shader allows you to reproduce an imaging artifact of real-world cameras, where bright areas create feathering (when overwhelming the camera capturing the scene). By default, the shader is set to use the Stingray Tone Map, which handles geometry content, so in the case of an image sequence, you may prefer using the Scene Linear Input option instead, if applicable. The first step of the Bloom shader is to create the bright pass which can be isolated using the Bright setting of the Pass option. To control how much of the image is contributing to the bright pass, you can adjust the Threshold and the Exposure settings. This bright pass becomes the source of the blooming effect to which the Blur Amount, Falloff, Offset, Bloom Colour, and Gain settings are then applied. When used as Camera FX, you can use both the Use GMask Input and the Selective options to tailor exactly where and how much Bloom effect you want in your image. Multiple Stingray Bloom shaders can then be added, using different values and affecting different areas of the scene.

Stingray Reflections This shader allows you to create single-bounce screenspace reflections, based on the Z-Depth and Normals information of a 3D scene. The shader uses the roughness information of the material through a Roughness map to control how sharp or diffuse the reflection is. To control this in Action, you can use the Shader node with either the Physically Based or the Cook-Torrance shading mode. The basic Reflection settings are the Intensity, Tint, Gain, and As Additive Ambient (which allows you to add the reflection as ambient light or blend it to the scene). The Use Z-Depth setting allows you to use a normal Z-Depth input, since, by default, the shader is expecting the Z-Depth HQ Action output for maximum quality (however a regular 16bit Z-Depth input can be used). The Only Reflections Output setting allows you to output just the Reflection result, without the scene result. You can use the Screen Edge and Surface Thickness Threshold settings to help control where the reflections are going to appear in the image. Since the Stingray Reflection shader is a single-bounce reflection, you may need to isolate which objects cast or which objects receive reflections, to get around some limitations of the technique. To do this, you can apply the shader as a Camera FX, and use the Casting and Receiving Selective settings. You can also use the GMask functionality to control the reflection further.

Stingray Depth of Field This shader allows you to create focus range blur effects. Use the Centre Z setting for focus position, and the Range setting to define the size of the focus area. Both of these settings are in Action camera-space coordinates. To control the Blur, you have independent control on the Near and Far side of the focus area. Max sets the blur size, while Distance sets how quickly you will reach the Max blur value, starting from the corresponding focus area position. You can further impact how the blur is going to behave in the defined Distance region, by adjusting the curves found in the Advanced tab section. By default it is set to linear, but it can be freely adjusted. Lastly, you can change the chromaticity of the effect by setting a colour to both the Far and Near blur area which affects the RGB channels independently.

Stingray Lens Effects This shader is a simple lens distort/correction tool with chromatic aberration capabilities. The Distortion setting distorts/corrects the image with a simple lens model, while the Fringe Intensity and Colour settings allow you to control the chromatic aberration of the lens. When used as a Camera FX, you can use the Use GMask Input option to only apply the overall effect to a portion of the image.

Using Matchbox as a Transition

You can use Matchbox shaders as transitions in the Timeline. You can create your own Matchbox transitions or use one of the many transition presets, supplied with the application.

To use Matchbox as a transition:

1. Add a Matchbox transition.

   The file browser opens at the location containing the shaders and transition presets.

2. Select a transition preset.

   The shader is loaded as a transition and you are taken back to the Timeline. However, the shader does not yet behave as a transition.

   Though the selected shader is marked "transition", it does not contain any of the animation or the parameter values that makes it behave as a transition. In order to get the transition effect, you can either manually create the transition animation or select one of the preset transition effects, from the Preset pop-up menu of the Quick menu, after having loaded the shader. This loads the animation data and parameter values necessary for a proper transition effect.

3. Select a preset transition from the Preset pop-up menu of the Quick menu.

   The shader now behaves as a transition.

4. Make any timing adjustments to your transition.

NOTE To create your own transition effect from a shader, you must manually keyframe your animation and set your shader parameter values and save them to a setup. An added benefit of working this way is that the setup contains the shader. This means that you can reload and use the effect even if the shader is not present on the workstation you're working on.

About Lightbox

Unlike Matchbox GLSL shaders, Lightbox shaders are available exclusively in Action, relying on the Light framework, to which a Lightbox node is parented in the Action scene. Lightbox shaders allow you to create and use a wide range of effects that affect one fragment at a time, allowing a casting of colour effects through the light cone, respecting every aspect of Light behaviour, including shading, decay, and feathering. Lightbox nodes can be useful in the lighting pipeline to develop a look at the scene level of Action, or to help with blue screen or green screen work.

Lightbox effects are based on the OpenGL Shading Language (glsl) programming language and can be created by users to use in Action, or to share with other Flame users (a number of preset and example shaders are also available). For this reason, a Lightbox API is available at *http://www.autodesk.com/shader_builder_api_guide*, that allows you to access Action scene information (such as distance, normals, camera position, and other 3D environment variables).

To add a Lightbox node in Action:

1. Do one of the following:
   - With a Light node selected in the schematic, double-click the Lightbox node in the All Nodes or Relighting bin.

     The file browser opens to allow you to select a Lightbox effect. When selecting a shader from the file browser, you can use the File Format box to display shaders in .glsl format or encrypted Lightbox .lx format.

   - With a Light selected in the schematic, double-click a Lightbox preset node from the dynamic Lightbox node bin (page 550).

     The Lightbox node is added to the scene as a child of the selected Light. If no light was selected prior to adding the Lightbox node, a new Directional Light is added to the scene as the parent of the Lightbox node.

TIP Lightbox nodes are displayed as white in the schematic to help you identify them. If you added the Lightbox from the dynamic node bin, the name of the effect also appears on the node in the schematic.

2 Double-click the Lightbox node in the schematic to display its menu.

The Lightbox node populates the menu dynamically, based on the .glsl (and .xml) code, but there are a few UI elements that are constant, located in the Shader tab.

3 If you added a Lightbox effect from the All Nodes or Relighting bin, you can change the effect by clicking Change Shader in the Shader tab of the Lightbox menu.

Lightbox Usage Tips

Here are a few highlights and tips to help you when working with Lightbox in Action:

- Multiple Lightbox nodes can be parented under the same Light, and the order of processing can be controlled in the Light Priority Editor (page 565).

  Since multiple Lightbox nodes parented under the same Light node respect a priority order, you can create a pipeline of Lightbox effects. For example, you can use a Selective 3D Lightbox shader to create a selective, then apply a second Lightbox shader, such as Colour Correct to affect only the selected areas.

- A GMask (or multiple GMasks) connected to the parent Light with a selective GMask Link (page 901) can help you create interesting effects, where the Lightbox is cast through the GMask shape. You can track the GMask, and combine GMask Links with Look At Links and Replica nodes to easily build a composite, such as simulating sunlight shining through windows.

- Lighting links (page 664) can be useful to connected Lightbox nodes in creating inclusive or exclusive constraints.

- Use the Action Object Solo (`F8`) view on the parent light to see the scene through the point of view of the light, as if it was a camera. The `F8` view also allows you to interact directly in the image window with icons, if they are present in the shader.

- If the loaded Lightbox shader has any preset starting points built into it (such as in the Duotone shader), you can find them in the Presets list in the Shader tab of the Lightbox menu.

- A folder is available in the channel editor for each Lightbox effect in your Action scene.

- Apart from the Lightbox menu with its dynamic settings generated from the .glsl code, there are a number of settings in the Light menu (page 668) that can help you set up your Lightbox effect (mostly in the Rendering section of the Basics tab, and in the Lightbox tab of the Light menu). By default, when a new Lightbox node is added to the scene with a new Light, the light is inactive, and global shading and Scene Ambient settings are set to active.

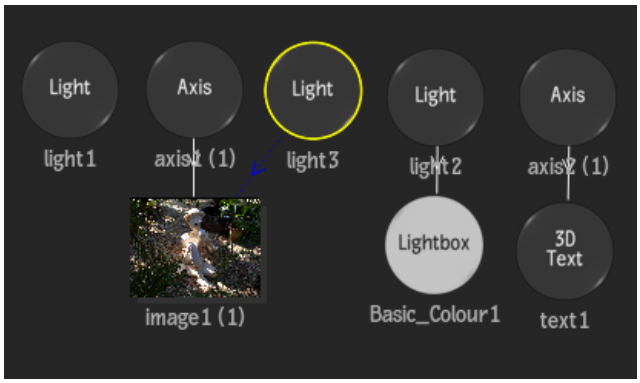
NOTE When the parent Light is inactive, certain Light settings are unavailable, such as colour and intensity.

Lightbox Examples

Use the following examples of Lightbox shaders to get a feel for the workflow of using different types of Lightbox effects in Action.

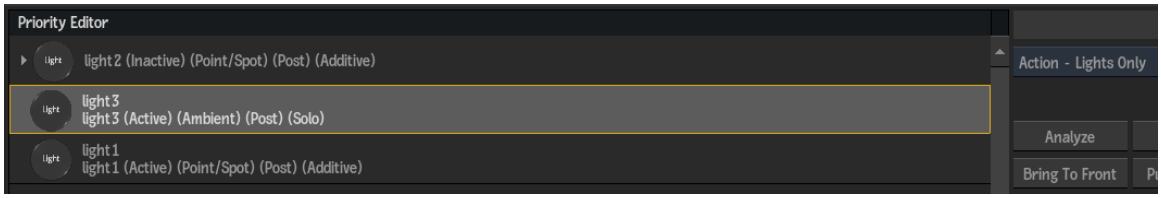
Combining Multiple Lights with Lightbox

A common scenario that you may encounter when working with Lightbox is how to combine images that you don't want to shade with a geometry that you do want to shade. You can use selective lighting links and the Priority Editor to solve this issue.



- Light 1 is a global light, set to Additive Light in the Light Source box.
- Light 2 is the parent of the Lightbox node, and is not set to Active.
- Light 3 is connected to the image node with a Light Inclusion link; the Light Type is Ambient, with the Shade set to 0%, and the Light source box set to Solo Light.

- These settings, when combined with Light 3 appearing in the Priority Editor after Light 1, produces the desired effect.

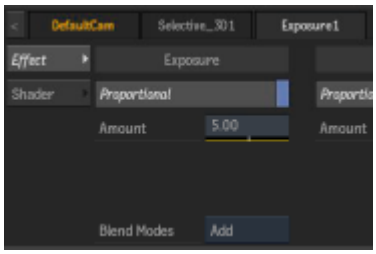


Lightbox Selectives

The goal when using the Selective 3D or Selective Noise 3D Lightbox shader is to create an alpha to use in the next Lightbox node in the pipeline.

Some tips for working with Lightbox Selective nodes:

- Add the Selective node as the first Lightbox to affect other Lightbox nodes that you can add later. It should appear at the bottom of the Priority list for the selected Light.
- Enable Selective Viewing in the Selective menu to display the selected area in magenta. Don't forget to disable Selective Viewing when you are done setting the selection.
- Choose a type of selective to create, such as 3D Sphere or 3D Cube for a Selective 3D node, or a noise type for a Selective Noise 3D node, and adjust other settings, as necessary. Enable Invert, if needed.
- Once the Selective is created, disable Selective Viewing, and add other Lightbox nodes under the same Light. The effect of this new Lightbox node only occurs in the Selective area that you created in the Selective node:

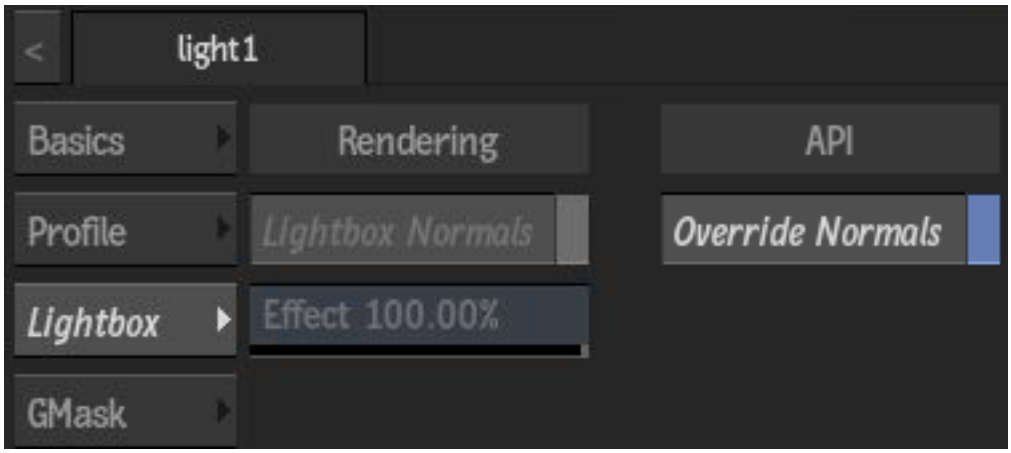


- You can have more than one Selective node parented under a Light. The Selective areas created by all the Selective nodes are combined to create one alpha selection.

Environmental Lightbox

You can use the Clouds and Fog Lightbox presets to simulate these environmental conditions in your Action scene.

For these effects, you do not want to use the normals attenuation from the parent Light, so make sure that the Override Normals button is enabled in the Lightbox tab of the Light menu. The state of this button can be coded directly in the shader (it is enabled by default for the Clouds and Fog presets).



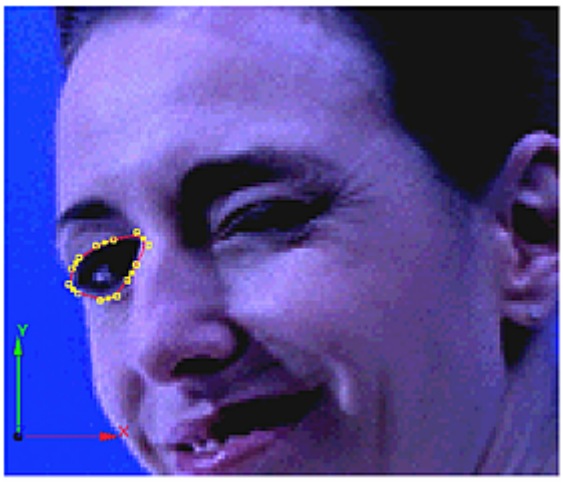
Warping and Morphing 25

About Distort

Use the Distort menu to warp a clip and create morphs between two clips. Both of these effects are created using splines, which are drawn and animated in a similar way to garbage masks.

Warping using the Distort Tool

Warps are created by drawing a source spline, and then offsetting a second corresponding destination spline. When you draw a source spline, an identical destination spline is automatically created. When you move the destination spline, the offset of the destination from its source spline is what warps the clip. For example, you can draw a source spline around the eye of an actor, and then edit the points of the corresponding destination spline to make the eye bulge.



Source spline drawn around eye

Image courtesy of Behavior Communications Inc.

Offsetting destination spline creates warp

Image courtesy of Behavior Communications Inc.

Distort Warping Workflow

The general warping workflow in the Distort tool includes the following steps.

Step:	Refer to:
1. Load clips into the Distort tool.	Accessing Distort (page 1372).
2. Draw a source spline.	Drawing Splines (page 1376). **TIP** Use Distort's schematic as you work with splines to help organize your work. See Using Distort's Schematic (page 1382).
3. Edit the source spline.	Editing Splines (page 1380).
4. Animate the source spline.	Animating and Tracking Splines (page 1390).
5. Unlink the destination spline from the source to enable warping.	Linking and Unlinking Splines and Axes (page 1379).
6. Edit the destination spline.	Editing Splines (page 1380).
7. Animate the destination spline.	Animating and Tracking Splines (page 1390).
8. Refine the warping effect.	Refining Distortions (page 1394).
9. Set what appears at edges of clip if the distortion effect pulls the clip away from the border, revealing empty space, or the background clip.	Filling in Gaps at Edges of Distorted Clips (page 1400).

NOTE Repeat this workflow for each warped element of the clip.

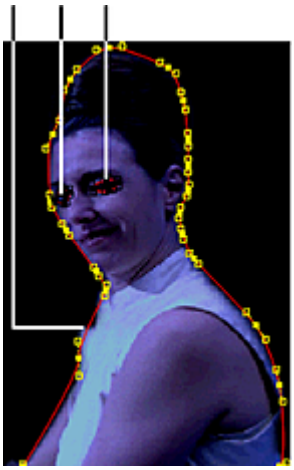
Morphing in Distort

Morphs are created by creating corresponding splines on two clips and linking them together. You then set interpolation values controlling how the features contained in the first set of splines morph into the features contained in the second set of splines. This method is called source interpolation. Finally, you blend the two clips over time to complete the morph. For example, you can draw splines around the outline of an actor and each of the actor's facial features. Then duplicate these splines onto a second clip containing the actor to whom the first actor will morph into. Edit the copied splines to more accurately contour the features of the second actor, link the corresponding splines, and then blend the two clips to complete the morph.

Both sets of splines, those on the first clip, and those on the second clip, can be animated. Additionally, you can edit and animate the interpolated splines.

Source splines drawn on actor

Image courtesy of Behavior Communications Inc.



Corresponding splines on second actor are linked

Image courtesy of Behavior Communications Inc.

Distort Morphing Workflow

The general morphing workflow in the Distort tool incudes the following steps.

Step:	Refer to:
1. Load clips into the Distort tool.	Accessing Distort (page 1372).
2. Draw a source spline on the Front1 clip.	Drawing Splines (page 1376). **TIP** Use Distort's schematic as you work with splines to help organize your work. See Using Distort's Schematic (page 1382).
3. Edit the Front1 source spline.	Editing Splines (page 1380).
4. Animate the Front1 source spline.	Animating and Tracking Splines (page 1390).
5. Display the Front2 clip.	Toggling Between Inputs (page 1374).
6. Duplicate the Front1 spline, then toggle the duplicated spline to become an Front2 spline.	Toggling Spline Nodes from One Input to Another (page 1388).
7. Edit the Front2 source spline.	Editing Splines (page 1380).
8. Animate the Front2 source spline.	Animating and Tracking Splines (page 1390).
9. In the Distort schematic, link the spline on the Front1 clip with the spline on the Front2 clip.	Linking Source Nodes (page 1389).

Step:	Refer to:
10. Set Blend and Interpolation values.	Blending Clips (page 1393) and Interpolating Features (page 1393).
11. Refine the morphing effect.	Refining Distortions (page 1394).

NOTE Repeat this workflow for each morphed feature you want to correlate from the Front1 clip to the Front2 clip.

Mesh-Based Warps and Morphs

While the Distort tool uses splines to define features of a clip you want to warp or morph, you can also use the Warper tool. Instead of splines, the Warper uses meshes to define affected features for similar warping and morphing effects.

Mesh-based warps and morphs created with the Warper can be more time-consuming since meshes have many more points to work with and usually cover the entire clip. The Warper can be useful, however, because you have more direct control over the parts of the clip being altered since you manipulate the mesh directly.

Accessing Distort

Use Distort to create warps and morphs of clips using spline-based shapes.

To access Distort, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

Distort accepts two front clips, two matte clips and one back clip as inputs, and outputs a result and an outmatte.

In Batch, you can create a more convincing morph, for example, by animating colour transitions between the first and second clips using Colour Corrector nodes.

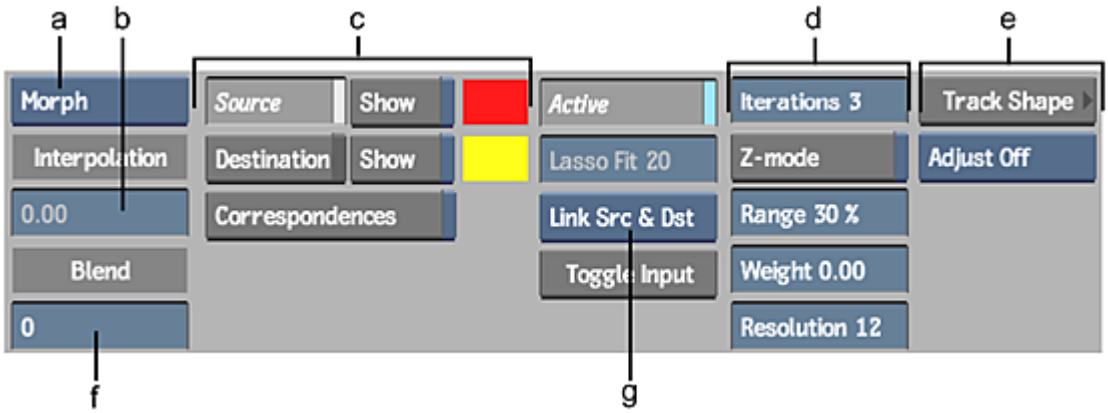
The Distort controls are described as follows.

Clip Display buttons Controls which clips, of those you load into Distort, are displayed. Use the Clip Display buttons to produce various effects, or to toggle on or off clips and mattes to facilitate your work.

Axis controls Edits the axis position, scale, and shear value of a spline.

Axis Stabilizer controls Gives access to the Stabilizer, for automatically tracking motion in the axis.

View box Sets the view in the image window. Two views available from this box, Input and Matte, each correlate with two distinct views, Front1 and Front2, or Matte1 and Matte2. You toggle between the input or matte views using the Input box.



(a) Warper button (b) Interpolation field (c) Spline Display controls (d) Refining controls (e) Shape Stabilizer controls (f) Blend field (g) Link box

TIP The distortion grid can be displayed by enabling Draw Grid in Distort's Setup menu. This can be useful when working with the Iterations and Range fields.

Using Distort's Views and Clip Display

You can change the view using the Distort menu as you create your effect.

Distort makes it easy to toggle between Front1 and Front2 clips or mattes independently in each viewport. For example, you can display Front1 in one viewport, and the matte of Front2 in another viewport.

Within a particular view, you can also toggle on or off clips to facilitate your work or create different effects. For example, if you loaded two inputs and their mattes, as well as a background clip, you can turn on or off any of the five clips that together make up the composite.

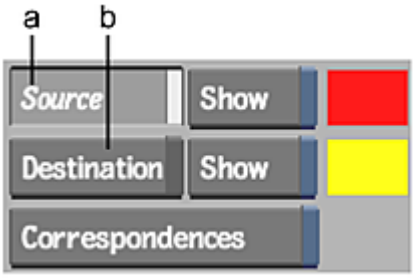
Changing Views

Use the appropriate view, available from the Distort menu's View box, for the task you are performing.

Select:	To view:
Input	The Front1 or Front2 clip. You toggle between them using the Input box.
Matte	Matte1 or Matte2. You toggle between them using the Input box.
Background	The background clip used in the warping or morphing effect.
Result	The result clip of the warping or morphing effect.
ResultMatte	The result matte that is created when you blend the IFront1 and Front2 clips when morphing.
DistortSchm	The Distort schematic.

Each of the Input and Matte views can correlate with two views if you loaded two inputs and their mattes into the Distort tool. See Toggling Between Inputs (page 1374).

While in the Input view, you can also toggle between viewing the unwarped input and the warped input by clicking between the Source and Destination buttons in the Warp and Morph menus.



(a) Source button (b) Destination button

Toggling Between Inputs

When you are morphing between clips, you can toggle between them so either Front1 or Front2 appears in the image window. The splines drawn on the clip are also displayed. For example, if you select Front1, the Front1 clip, along with its splines, is displayed in the image window. If you then select Front2, the Front2 clip appears with the splines drawn for Front2.

You can similarly toggle between Matte1 and Matte2.

To toggle between s or mattes:

1 From the Warp or Morph menu in the Distort tool, select an option from the Input box.

Select:	To:
Front1	Display the Front1 clip if Input is selected in the View box, or Matte1 if Matte is selected in the View box, along with the Front1 source and destination splines.
Front2	Display the Front2 clip if Input is selected in the View box, or Matte2 if Matte is selected in the View box, along with the Front2 source and destination splines.

The corresponding clip with its splines is displayed. As well, the splines that are not displayed, for example, splines added to the Front1 clip if you select Front2, are indicated as hidden in Distort's schematic.

Displaying Clips

Depending on the clips you load into Distort, you can use the Clip Display buttons to produce various effects, or to simply toggle on clips and mattes to facilitate your work. The following table shows how to set up the clips to produce various effects.

Front1	Front2	Matte1	Matte2	Back	Effect
On	Off	Off	Off	Off	Warps theFront1 clip only. This is the same as loading only one input clip.
On	On	Off	Off	Off	Morphs between two clips. This is the same as loading two input clips.

Front1	Front2	Matte1	Matte2	Back	Effect
On	Lock	Off	Off	Off	Wipes (page turns, rolls); regional warps (if the same clip is loaded as both the Front1 and Front2 clip).
On	Off	On	Off	Off	Warps the Front1 clip and Matte1 clip, composited on black.
On	On	On	Off	Off	Warps over a background with a matte clip.
On	Lock	On	Off	Off	Warps the Front1 and Matte1 clips composited on the Front2 clip.
On	On	On	On	On	Morphs between two clips and their mattes onto the selected background clip.

Creating Splines

You draw open or closed splines to match the features of a clip that you want to warp or morph. Splines are drawn in a similar way as garbage masks, by clicking to add points, or by dragging to draw freehand segments. You can also load a saved garbage mask setup to use as distort splines.

You draw a spline around a feature at the start of an effect. When you add a spline, both a source and destination spline is added to the clip. Since these splines are initially linked together and overlap, your manipulations to the source spline affect the destination spline in the same way. After you have drawn and animated the source spline, you can unlink the splines and edit the vertices and tangents of the destination spline independently.

When creating a morph between two clips, you draw splines on the Front1 clip, then optionally duplicate them to the Front2 clip. When you link the splines, those duplicated to the Front2 clip become the destination for splines drawn on the Front1 clip.

NOTE When morphing, you can also draw source spline on the Front2 clip to link to corresponding Front1 splines. Only splines with the same number of points, and drawn in the same orientation (clockwise or counter-clockwise), will create proper morphs.

You can show or hide source and destination splines, as well as change their colours.

Using Garbage Masks as Distort Splines

Garbage mask setups are created through the GMask node in Batch. You can import a saved garbage mask setup into Distort and use it as a spline. Note the following about how Distort interprets settings in the imported garbage mask setup:

- If the imported garbage mask has animation on its axes, the animation is recreated in axes above the main axis in the Distort schematic.
- If the imported garbage mask has shape animation or explicit animation on the gmask splines, the same animation on the source and destination splines is transformed into shape animations.
- If the imported garbage mask includes tracking data, the offset parameters on the geometry are duplicated on the offset parameters of both the source and destination splines.

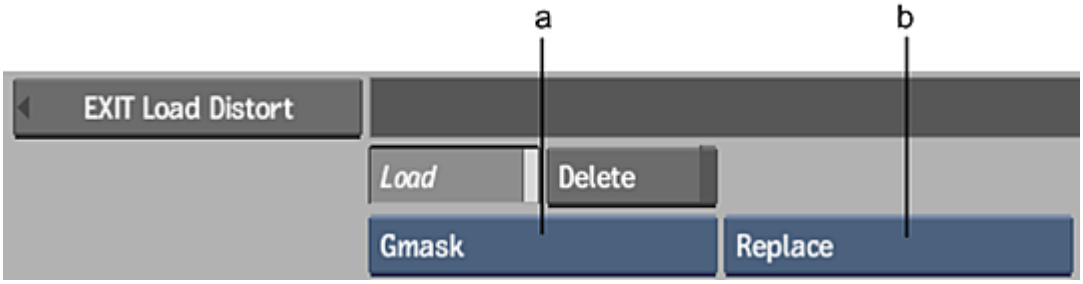
- Garbage mask setups that use multiples axes to manipulate instances of splines are translated to Distort instances. See Instancing Distort Splines (page 1381).

For help creating garbage masks, see Drawing a Mask (page 1106).

To import a garbage mask setup into Distort:

1. Do one of the following:
   - In the Distort menu, click Load to open the file browser.
   - From the Distort node in Batch, click NodeSetup to display Distort's Setup menu, then click Load Node to open the file browser.
2. From the Load Type box, select Gmask.

   The browser automatically points to the default Gmask folder.



(a) Load Type box (b) Load Mode box

3. From the Load Mode box, select whether you want the Gmask setup to append to or replace any existing Distort splines in your scene.
4. From the file browser, select the name of the setup to load.

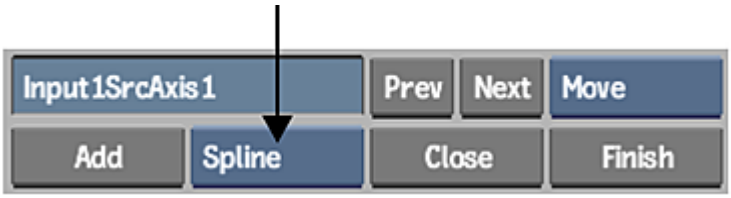
   TIP A library of preset garbage mask setups is available. To load a preset garbage mask, navigate to the */opt/Autodesk/project/<project_name>/gmask/default/* directory.

Drawing Splines

Draw single-point, open, or closed splines around the features you want to warp or morph at the beginning of the effect.

To draw a closed spline:

1. Go to the frame where you want the effect to begin.
2. Select Spline from the Node box.



3. If you are creating a morph, select the input where you want to draw the spline. To add a spline to Front1, for example, select Front1 in the Morph menu's Input box.

NOTE When creating a morph, a common strategy is to create the spline on just one input, and then duplicate it to the other. This has important advantages over creating splines on both inputs independently. See Toggling Spline Nodes from One Input to Another (page 1388).

4 Click Add.

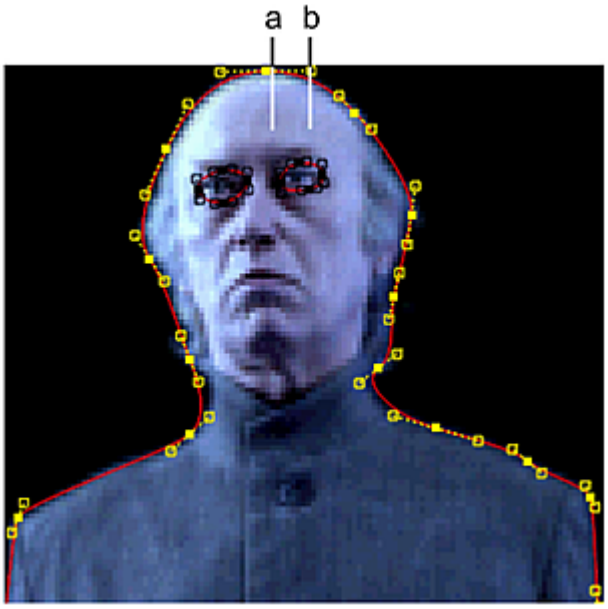
5 Do any of the following:

- Click to add vertices.
- `Shift`-drag to add freehand segments to the spline. Vertices are added where you drag, and appear after you release `Shift`. You can then use the Lasso Fit parameter to increase or decrease the number of vertices that define the freehand segments of the spline. If you add or delete vertices on the spline, or exit, the influence of the Lasso Fit parameter on the number of vertices is lost. See Adjusting the Number of Points in Freehand Segments (page 1109).

6 To close the spline, do one of the following:

- Click Close.
- Press `C`.
- Click the first vertex.
- While holding `Shift`, drag to the first vertex.

When the spline is closed, its vertices and tangents can then be edited.



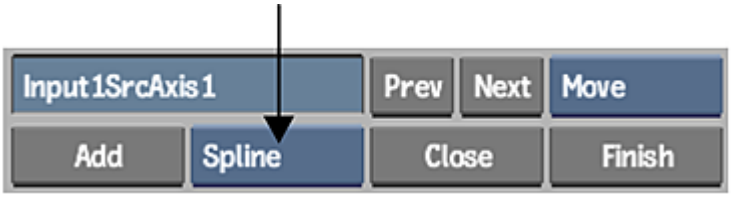
(a) Vertex (b) Tangent

Image courtesy of Behavior Communications Inc.

To draw a single-point or open spline:

1 Go to the frame where you want the effect to begin.

2 Select Spline from the Node box.



3 If you are creating a morph, select the input where you want to draw the spline. To add a spline to Front1, for example, select Front1 in the Morph menu's Input box.

NOTE When creating a morph, a common strategy is to create the spline on just one input, and then duplicate it to the other. This has important advantages over creating splines on both inputs independently. See Toggling Spline Nodes from One Input to Another (page 1388).

4 Click Add.

5 Do any of the following:

- Click to add vertices. If you are drawing a single-point spline, proceed to the next step.
- `Shift`-drag to add freehand segments to the spline. Vertices are added where you drag, and appear after you release `Shift`. You can then use the Lasso Fit parameter to increase or decrease the number of vertices that define the freehand segments of the spline. If you add or delete vertices on the spline, or exit, the influence of the Lasso Fit parameter on the number of vertices is lost. See Adjusting the Number of Points in Freehand Segments (page 1109).

6 To finish the spline, do one of the following:

- Click Finish.
- Press `F`.
- Click the last vertex.

When the spline is finished, its vertices and tangents can then be edited.

If you want to view the nodes that are added with each new spline, select DistortSchm from the View box. You can use Distort's schematic to access a menu, create parent-child relationships between splines and axes, delete splines, link Front1 and Front2 splines to create morphs, as well as perform other organizational tasks. See Using Distort's Schematic (page 1382).

To show and hide splines:

1 From the Warp or Morph menu in the Distort tool, do one of the following:

- To work with source splines, click Source.
- To work with destination splines, click Destination.
- To work with source splines while seeing the destination splines as a reference, click Destination Show.
- To work with destination splines while seeing the source splines as a reference, click Source Show.

NOTE To toggle between splines drawn on the Front1 and Front2 clip, select Front1 or Front2 from the Input box.

To colour splines:

1 Click the Source colour pot or the Destination colour pot, and use the colour picker that appears.
The colour you pick is applied to all source or destination splines accordingly.

Changing the Colour of Tangents

You can change the colour of tangents on the splines you draw. This is useful to better contrast the spline's tangents from the clip so that they are easier to work with.

To change the colour of tangents on splines:

1 In the Distort menu, click Setup.
2 In the Setup menu Display section, click the Tangents colour pot.

3 From the colour picker that appears, select a colour.
Tangents are displayed in the colour you set.

NOTE All spline tangents are initially displayed in magenta after they are drawn until a source spline and its corresponding destination spline are offset from one another.

Linking and Unlinking Splines and Axes

Warps are created by offsetting a destination spline from its source. By default, when you draw a spline, it is linked to its corresponding source or destination. After it is drawn, you might animate it, for example, so that it follows a subject in the clip. When the splines are linked, both the source and destination splines are animated in the same way.

When you are ready to create the warp, you unlink the destination spline from its source. Manipulations then affect each spline separately, so you can alter the shape of the destination. The differences between the two corresponding splines are what creates the warp.

You can also independently unlink the axis of a source spline and the axis of a destination spline. By default, the axis nodes are linked, even if the source and destination splines themselves are unlinked. You might unlink only the axes of corresponding splines, for example, if you want their offset to create the warping effect while the actual shapes of the splines remain the same.

You use a different linking method when using source interpolation to create a morph. See Linking Source Nodes (page 1389).

To link and unlink source and destination splines or axes:

1 Do one of the following:
- Select a spline by clicking it in a viewport or by clicking its spline node in the Distort schematic.

- Select an axis node in a spline node tree that is a parent to a spline node.

See Using Distort's Schematic (page 1382).

2 From the Warp or Morph menu in the Distort tool, click the Link box to toggle between Lnk Src & Dst and Distort.

Select:	To:
Lnk Src & Dst	Link source and corresponding destination splines, or their axes. With linked splines, if you move a spline or any of its vertices both the source and corresponding destination splines are affected in the same way. With linked axes, if you move the parent axes of a spline, the parent axes of the destination spline is affected in the same way.
Distort	Allow source and destination splines or axes to be offset from one another, which creates the warping effect.

Vertices on linked splines appear in magenta. Axes that are linked appear in magenta.

NOTE If you select Lnk Src & Dst and manipulate a vertex that has already been offset from its corresponding vertex, both the source and destination vertex snap to the same position. Similarly, if you manipulate the Axis values of a spline that has been offset from its corresponding spline with Lnk Src & Dst selected, all vertices and the splines' axes snap together.

Editing Splines

You edit splines in the same way as you edit garbage masks, working with their vertices and tangents. See Moving Vertices and Tangents (page 1112).

If Lnk Src & Dst is selected from the Link box, your edits affect both the source and destination spline regardless of where you make the edits. See Linking and Unlinking Splines and Axes (page 1379).

TIP The Link box has no effect if spline nodes are linked in the Distort schematic while morphing using source interpolation. See Linking Source Nodes (page 1389).

To edit a spline:

1 From the Tools box, select one of the following modes to edit a spline's vertices.

Select:	To:
Select	Select one or more vertices.

Select:	To:
Move	Move vertices to a new position.
Scale	Scale vertices in relation to the axis centre of the spline.
Delete	Remove vertices from the spline.
Break	Break tangents of vertices.

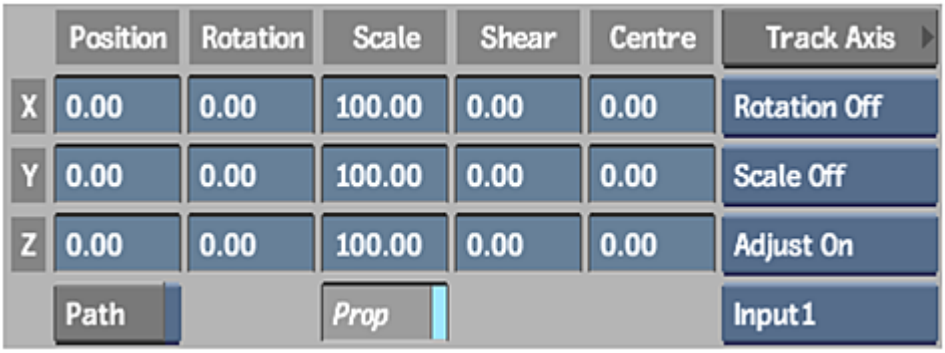
Transforming Splines

You can apply transformations such as translation, rotation, scaling, and shearing to a spline. The transformation data for the spline is stored in its Axis node. See Axis Nodes (page 1386).

All transformation parameters can be animated. You can also use a motion path to animate the position of a spline.

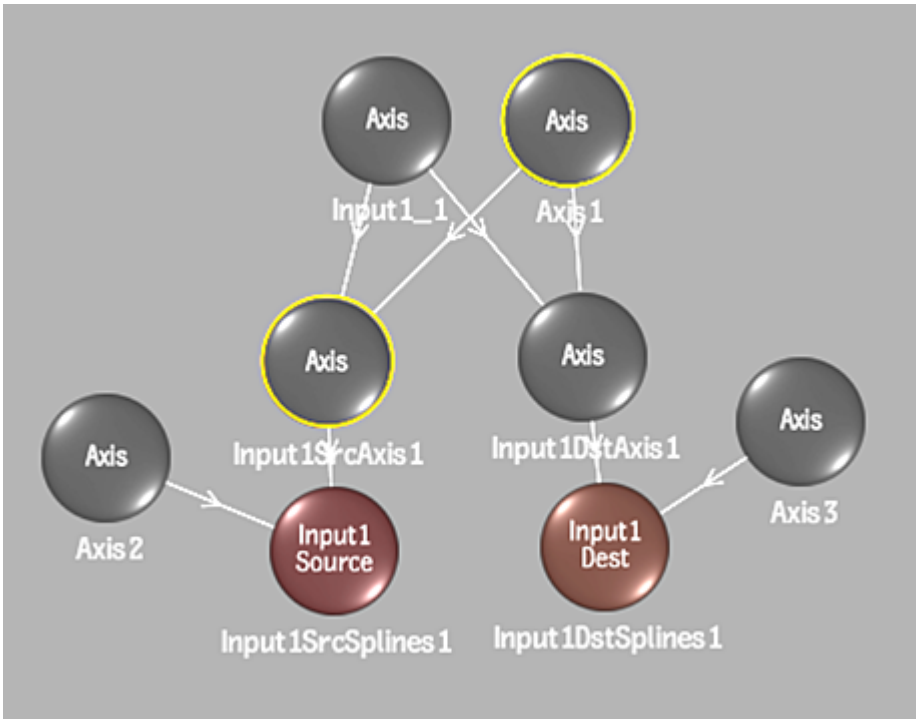
To transform a spline:

1 Use the Transformation fields to modify the position, shape, and size of a spline.



Instancing Distort Splines

When you add multiple axes above a pair of source and destination splines, each axis can be used to manipulate a separate instance of the pair of splines. You can then use the Distort settings for each axis to manipulate the instance of each spline. In the following example, the Input1_1 and Axis1 axes can be used to change instances of the source and destination splines. Axis2 and Axis3 cannot be used in this manner, since they are only attached to one input spline.



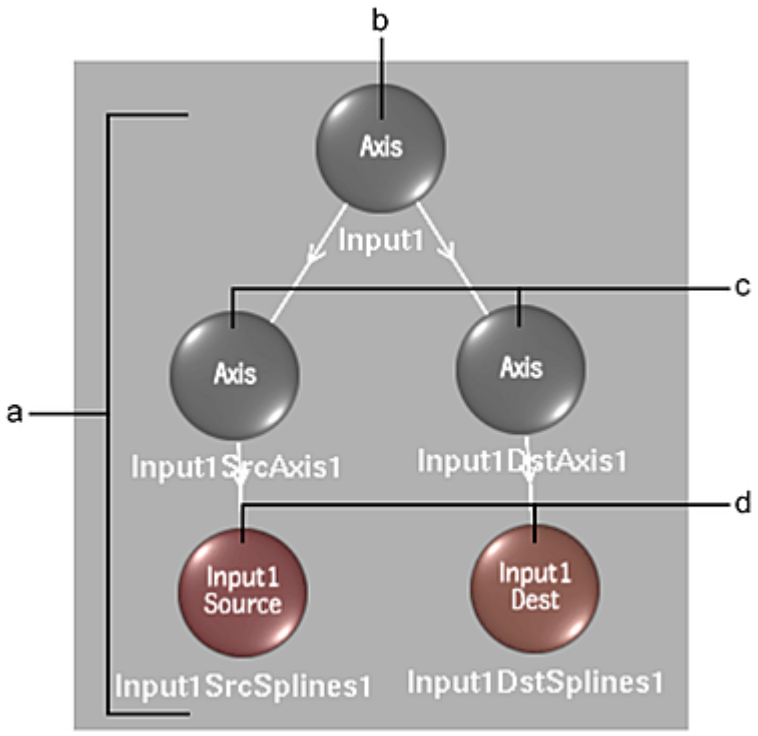
Using Distort's Schematic

The Distort schematic uses nodes to represent all the splines in the scene. The schematic shows relationships between source and destination splines, and between splines applied to the Front1 and Front2 clips.

Use the schematic to control all the splines in the scene, set parent-to-child relationships among nodes, as well as to select splines more easily.

You also use the schematic to link Front1 and Front2 source splines to create morphs using source interpolation. See Linking Source Nodes (page 1389).

When you add a spline to the Front1 clip, for example, the following spline node tree appears.



(a) Spline node tree representing corresponding splines added to Front1 (Input1) (b) Axis node for an Front1 (Input1) spline: the parent of corresponding source and destination splines (c) Axis nodes for each spline (d) Spline nodes for corresponding source and destination splines added to Front1 (Input1)

Each spline added to an input clip results in its own spline node tree in the schematic, with its own corresponding source and destination splines.

You can change an Front1 spline to an Front2 spline or an Front2 spline to an Front1 spline using the Toggle Input button. See Toggling Spline Nodes from One Input to Another (page 1388).

Visibility of the splines is controlled from the Warp or Morph menu using the Spline Display controls. When you toggle between the Front1 and Front2 clips with the Input box, splines are hidden for the inactive clip, which is also reflected in the schematic.

You can also set the transparency of the schematic. This is useful if you want to dim its display so as not to interfere with the main focus of your work.

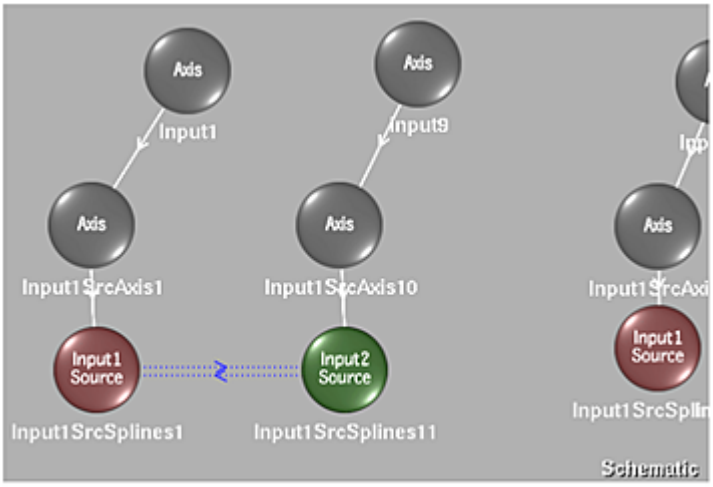
To view Distort's schematic:

1 Do one of the following:

- Select DistortSchm in the View box.

- Press the ~ key. Pressing the ~ key a second time returns to the previous view.

The Distort schematic appears.



TIP Display Distort's schematic and the input clips in a multiple viewport setup. This way, you can see the splines in the clips as you select and work with them in the schematic.

To set the transparency of the Distort schematic:

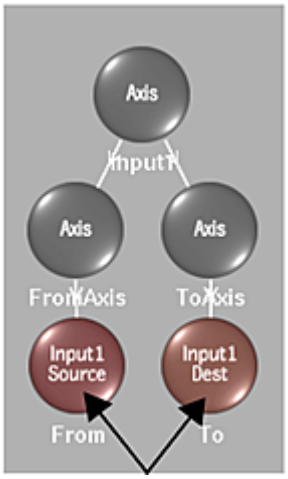
1. In the Distort menu, click Setup.
2. In the Setup menu Schematic section, adjust the value in the Transparency field.

NOTE The schematic transparency has a maximum value of 90%.

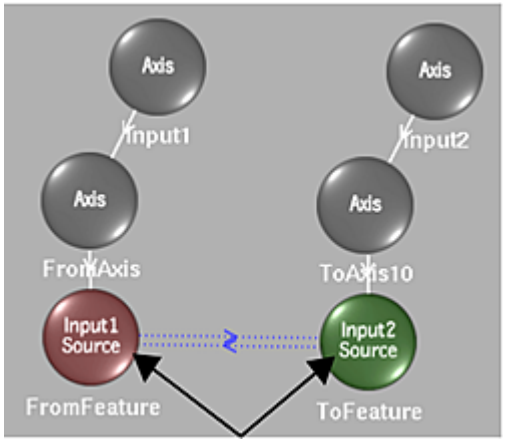
Node Types

In the Distort schematic, a node can be a spline or an axis.

Nodes help you organize warps that occur between corresponding source and destination splines in a spline node tree, and morphs that occur between linked Front1 and Front2 source nodes in different spline node trees.



Warps occur between corresponding source and destination splines that are offset from one another



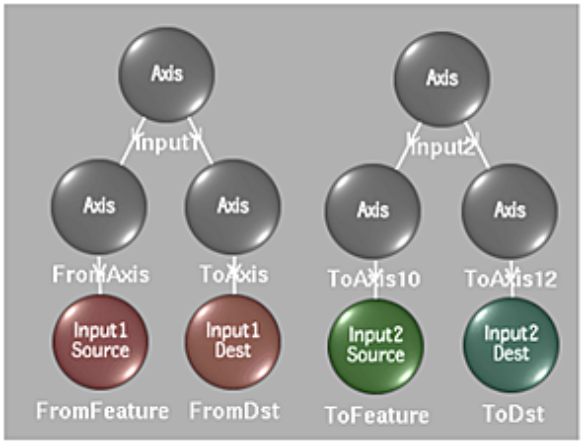
Morphs occur between corresponding Front1 (Input1) and Front2 (Input 2) source splines that are linked together

Spline Node Trees

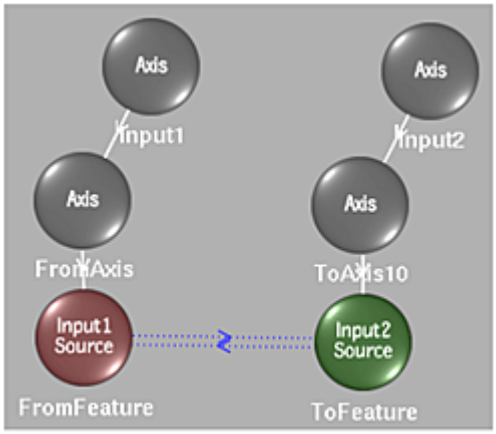
When you add a spline to the Front1 or Front2 clip, axis and spline nodes are added to the Distort schematic. These nodes represent the source spline and its corresponding destination spline, and are parented to axes in a spline node tree.

Warps are created when the source spline and destination spline in a spline node tree are offset from one another.

Morphs are created when you link a source spline from an Front1 spline node tree to a source spline from an Front2 spline node tree.



A spline node tree for a pair of corresponding splines on the Front1 (Input1) clip, and a spline node tree for a pair of corresponding splines on the Front2 (Input2) clip



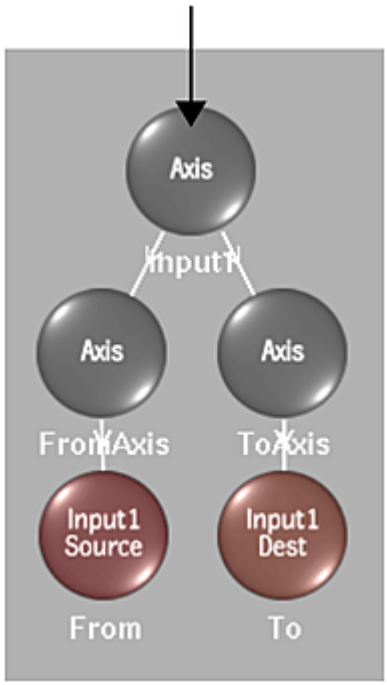
When you link source splines between Front1 (Input1) and Front2 (Input2) for source interpolation, the destination splines and their respective Axis nodes are removed from the spline node trees

Axis Nodes

Use Axis nodes to control the position, rotation, scale, and shear values of splines.

Initially when you add a spline, its source and destination node is parented to its own Axis node, and these Axis nodes are in turn parented to a common Axis node. So, for example, you can control the position of a particular Front1 source spline via its Axis node. Click the Axis node, the direct parent of the Spline node, then use the Axis menu in the Distort tool to apply tracking data to the node.

If you want to affect both a source spline and its corresponding destination spline, you can use the Axis node at the top of the spline node tree.



When you add splines, they are initially parented to their own spline node tree. You can, however, add more Axis nodes to the schematic, and parent splines from different spline node trees to the same Axis node. This can be useful, for example, if you create splines for the different features of an actor's face and want them to move in tandem with the actor.

Axis nodes for corresponding splines can be unlinked from each other by selecting a spline's axis node, then selecting Enable Warping from the Link box.

Spline Nodes

Use the Spline node to select the spline itself, including all its points. Each spline added to the schematic results in a source and destination Spline node. Each Spline node is representative of a particular spline, which consists of a closed shape defined by vertices. If Link Src and Dst is selected from the Link Box, transformations applied to a source or destination spline also affect the corresponding spline. This does not apply when Front1 and Front2 source splines are linked for source interpolation. See Linking Source Nodes (page 1389).

When you add a spline, it is applied to the Front1 or Front2 clip, depending on what is selected in the Input box. You can then select it and change it from an Front1 to an Front2 spline, or from an Front2 to an Front1 spline. This changes the input for both the source and corresponding destination spline. See Toggling Spline Nodes from One Input to Another (page 1388).

You can add more Axis nodes to the schematic and link the spline nodes to them, for example, if you want to affect the children splines in the same way.

You can, for example, track all the points for the selected spline using the Stabilizer. You can also select the Spline node for a spline if you are having trouble distinguishing between splines in a complex setup.

TIP When using source interpolation, you can select either one of the linked spline nodes when you want to set the interpolation value between them.

Adding Spline Node Trees

When you add a spline, a spline node tree is added for Front1 or Front2. The spline node tree includes both Spline and Axis nodes.

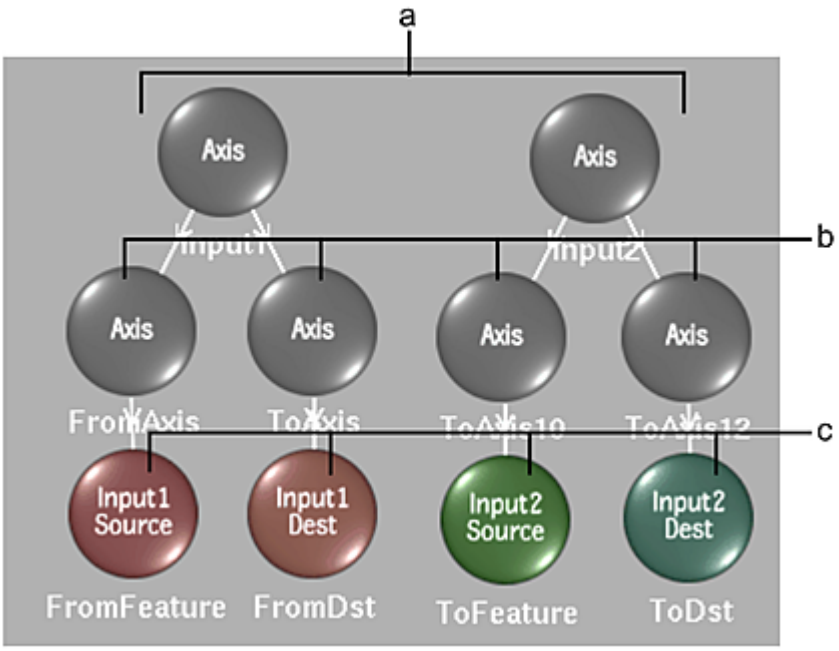
To add a spline node tree:

1. Select Front1 or Front2 from the Input box, depending on where you want to add splines.

   NOTE Splines can be changed from Front1 to Front2, or from Front2 to Front1. See Toggling Spline Nodes from One Input to Another (page 1388).

2. Select Spline from the Node box.
3. Click Add.
4. Draw a spline. See Drawing Splines (page 1376).

   A spline node tree is added for either Front1 or Front2 in the schematic. It includes source and destination spline nodes, each parented to their own Axis node, which are in turn parented to a common axis.



(a) Spline node trees for one Front1 (Input1), and one Front2 (Input2) spline (b) Axis nodes for individual spline wireframes (c) Spline nodes for corresponding source and destination splines

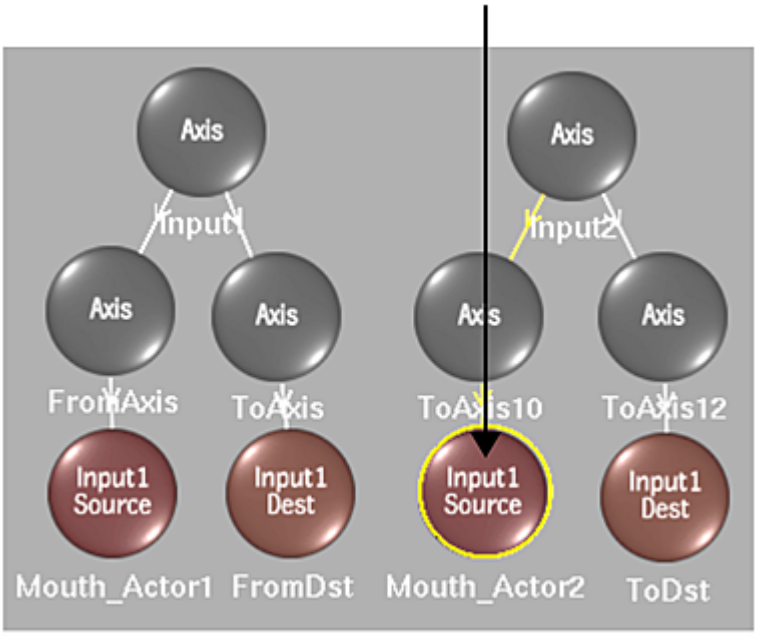
Toggling Spline Nodes from One Input to Another

Morphs work best when the source spline and destination spline have the same orientation—that is, were both drawn clockwise or counterclockwise—and the same number of vertices. You can ensure this is the case by drawing your spline on one input, then duplicating it to the other. To do this, first duplicate the spline, and then "move" the duplicate using the Morph menu's Toggle Input button. It toggles the selected spline from one input to the other.

TIP When morphing, you can duplicate splines and then toggle their input, which results in corresponding splines on the Front1 and Front2 clips that can then be linked for source interpolation.

To toggle a Spline node's input:

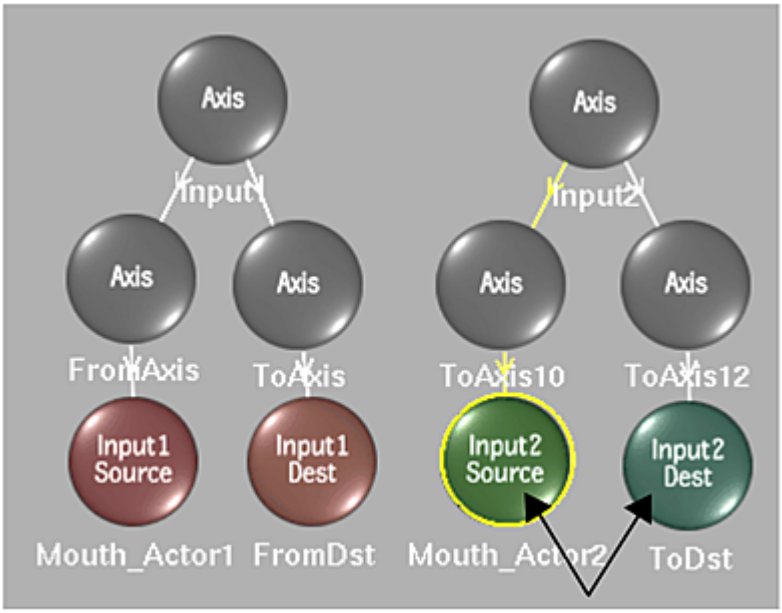
1 Select the source or corresponding destination Spline node you want to change from Front1 (Input1) to Front2 (Input2), or from Front2 (Input2) to Front1 (Input1).



NOTE You can also select the spline directly in the clip.

2 Click Toggle Input.

The source and destination spline nodes change either from Front1 (Input1) to Front2 (Input2), or from Front2 (Input2) to Front1 (Input1).



Linking Source Nodes

You link a spline source node from Front1 to a spline source node from Front2 to create interpolation between them for a morph. Destination splines are no longer used and cleared from the schematic. The source of the Front2 spline node tree becomes the destination of the Front1 spline node tree.

You can create as many relationships between Front1 and Front2 spline nodes by linking them in the schematic, as there are features to correlate between the clips.

To link source nodes:

1. Select Link from the Tools box.
2. Drag from an Front1 source spline node to an Front2 source spline node.

   NOTE You can only link a source spline from an Front1 to a source spline from Front2.

   The splines are linked to each other, as indicated by the blue dotted connector lines. The destination splines, and their respective Axis nodes, are removed from the two spline node trees. Once Spline nodes are linked, you can set an interpolation value to determine the percentage that the feature defined by the Front1 spline mixes with the feature defined by the Front2 spline. See Interpolating Features (page 1393).

To unlink nodes:

1. Select Link from the Tools box.
2. Drag the cursor across the line that joins the two linked source spline nodes.

   The spline nodes are unlinked, and the destination splines and their axes reappear for each of the affected spline node trees.

Animating and Tracking Splines

If the image you are warping or morphing is in motion, it will be necessary to animate each spline so it moves with the feature of interest. To animate a spline, set keyframes for its axes or vertices at various places in the clip. The motion between the keyframes is automatically interpolated. Alternately, use the Stabilizer to automate the process by tracking a reference point and connecting an axis or vertex to the resulting tracking data. Whichever method you choose, you can further control the shape of the spline by adding or removing vertices throughout the animation.

NOTE You can also animate correspondence points to refine the mapping of the source spline to its destination over time. See Working with Correspondence Points (page 1395).

For example, if you are creating a warp, you can load the Front1 clip into the Stabilizer and track all of the source spline's vertices. With Enable Warping selected from the Link box, you can then manually edit the points of the destination splines along the course of the clip to create the desired warp.

Similarly, if you are creating a morph, you can track the vertices of both source splines on the Front1 clip, as well as the source splines on the Front2 clip. When you then link the splines together in the Distort schematic for source interpolation, the morph animation is already set. You can refine the morph by further animating the destination spline.

At any frame, you can control the shape of the spline by adding and removing vertices. If the feature you are tracking gets larger, for example, you can add vertices to adjust the shape of the spline. Similarly, you can remove vertices that are no longer needed. Make them inactive from one keyframe to the next using the Active button, or permanently delete them. See Manipulating Vertices and Tangents (page 1111).

Animating Splines

Set keyframes for spline vertices and tangents over the course of the clip.

To animate a spline:

1. Move to the frame where you want to begin animating the spline.
2. If necessary, do one of the following to display the spline you want to animate:
   - From the Warp or Morph menu, click Source to display source splines, or click Destination to display destination splines. You can also use the Show buttons to display both source and destination splines.
   - If you are working with two clips, with either Input, Matte, or Result selected in the View box, toggle the Input box to Front1 or Front2, depending on which spline you want to animate.
3. Select the spline in the clip or its node in the Distort schematic.
4. Enable Auto Key in the Animation menu.
5. Select Move mode in the Edit box, then select and move one or more vertices to a new location. You can also move vertex tangents.
6. Advance the positioner to a different frame then move some more vertices.
7. Continue to move vertices and tangents as you progress through the clip, so the spline follows the movement of the feature you are warping or morphing.

Loading a Clip into the Stabilizer

Use the Stabilizer controls in the Warp menu or in the Morph menu to select the clip you want to track, the spline you want to apply the tracking data to, and the tracking mode (Axis or Shape). See How the Stabilizer Works (page 1064).

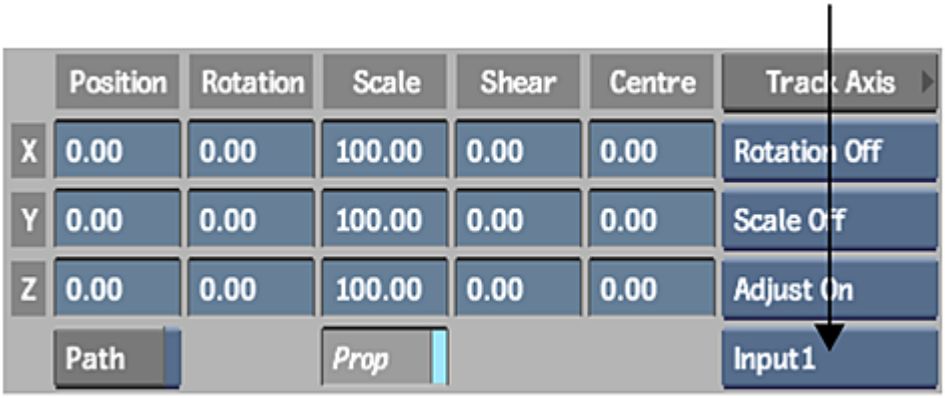
Select from the Track Clip box:	To track:
Front1	The Front1 clip.
Front2	The Front2 clip.
Matte1	The Matte1 clip.
Matte2	The Matte2 clip.
Background	The background clip.

Tracking by Axis

To make the spline follow the movement in the clip without changing shape, apply the tracking data to the spline's axis.

To track by axis:

1. Set the Stabilizer controls to apply the tracking data to either the source (Src) or destination (Dst) spline for either the Front1 or Front2 clip. Select the Axis node in the Distort schematic that is the parent of the spline you want to track.
2. Select the clip you want to track from the Track Clip box.



3. Click Track Axis.

   The Stabilizer menu appears.
4. Position the tracker on the image. Tracker 1 controls the position. Tracker 2 controls the rotation or scale. See Working with Trackers (page 1067).
5. To use two trackers, click Tracker 2, click Active, and position the second tracker on the image.
6. Click Analyse to generate the tracking data.
7. If necessary, fine-tune the analysis.

8 When you are satisfied with the tracking results, click Return to apply the results to the axis and return to Distort.

Tracking by Vertices

To make the spline follow the movement and shape change in the clip, apply the tracking data to the spline's vertices.

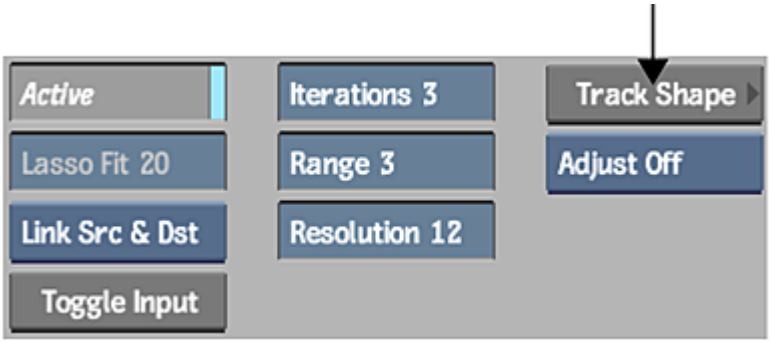
To track by vertices:

1 Select the spline's vertices that you want to affect. You can first select a spline by clicking its Spline node in the Distort schematic.

TIP Double-clicking a Spline node displays the Warp or Morph menu.

2 Set the Stabilizer controls to apply the tracking data to either the source (Src) or destination (Dst) spline for either the Front1 or Front2 clip.

3 Click Track Shape.



The Stabilizer menu appears. A tracker appears on each vertex that you selected. The reference frame is the frame from which you opened the Stabilizer.

4 If necessary, adjust the position of the trackers to ensure that each tracker has a good reference point.

5 Click Analyse to generate the tracking data, and if necessary, fine-tune the analysis.

For example, disable a problem tracker and re-analyse, as described in Tracking Difficult Shots and Correcting Errors (page 1091).

6 When you are satisfied with the tracking results, click Return to apply the results and return to Distort.

Duplicating and Pasting Tracking Data

You can duplicate and paste tracking data from one mesh to another using the Channel Editor. For example, if tracking was done using Axis, you can duplicate the translation of the Front1 source spline and apply it to the Front2 source spline.

Example: To duplicate translation data:

1 Click Animation to display the Channel Editor.

2 Select the X and Y channels in the Input1 _src translate folder.

3 Click Duplicate.

4 Select the X and Y channels in the Input1_dst translate folder.

5 Click Paste.

The translation channels containing the tracking data from the Front1 source spline are pasted into the Front1 destination spline.

Blending Clips

When you morph features from an Front1 clip to features from an Front2 clip or a background clip, you add a blend to the Front1. By blending the clip over time, the Front1 dissolves into the Front2 or the background. Matching features, for example, the mouth of an actor on the Front1 clip, and the mouth of an actor on the Front2 clip, as defined by the corresponding source and destination splines, can correspondingly blend from one to the other as the second clip becomes visible.

To blend the Front1 clip into the Front2 clip or background:

1 From the Morph menu, adjust the value in the Blend box. A value of zero results in the Front1 clip being totally opaque, while a value of 100 results in the clip being completely transparent.

TIP The blend value you set can be animated over time, and is displayed in the Channel Editor by a transparency channel in the Morph folder.

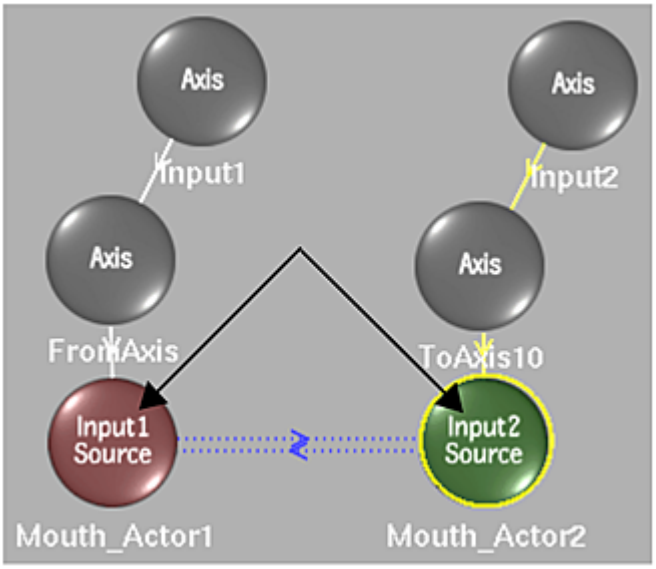
Interpolating Features

Set interpolation values for each set of Spline nodes you linked in the Distort schematic.

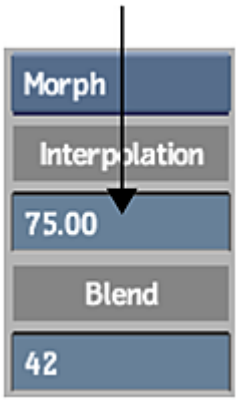
The interpolation value determines the percentage that the feature defined in the Front1 source spline mixes with the feature defined in the Front2 source spline.

To set the interpolation value between linked splines:

1 Select the Front1 (Input1) or Front2 (Input2) source Spline node of a linked pair in the Distort schematic.



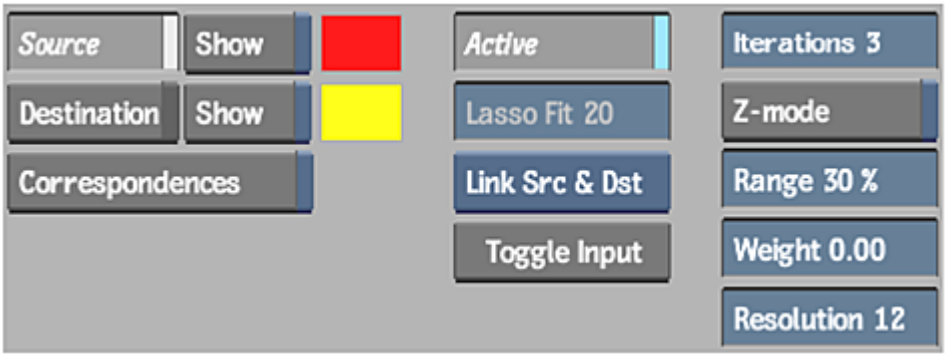
The current interpolation value for the linked splines appears in the Interpolation field.



2 Set the value for the interpolation. A value of 0 results in the Front1 feature remaining in its original form, while a value of 100 results in the Front1 feature distorting into the shape defined by the Front2 spline.

Refining Distortions

There are numerous methods and techniques to refine your distortions. You can adjust the correspondence between source and destination splines using correspondence points if a feature from Front1 is not mapped correctly to the corresponding Front2 feature. You can work with the number of iterations to increase the warping effect, or limit the affected region of the clip. As well, you can adjust clip resolution to affect how well clips distort.



Working with Correspondence Points

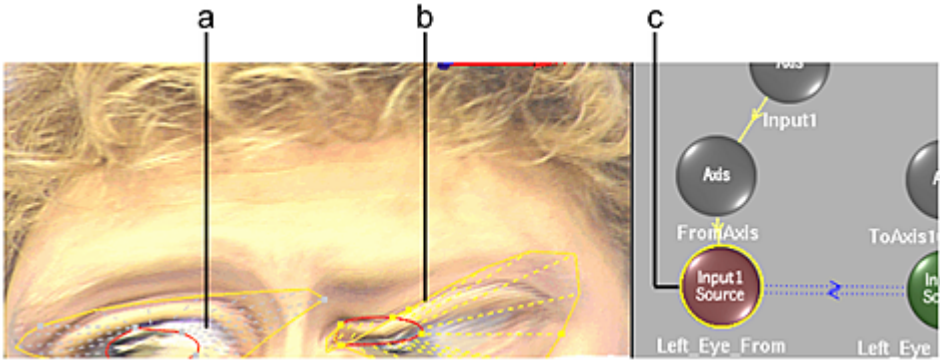
Correspondence points indicate how one frame is mapped to another in relation to source and destination splines, or Front1 and Front2 source splines when using source interpolation. If part of a clip is pulled or shifted too much during a distort, you can realign it by adjusting the correspondence points.

By default, correspondence points are added automatically. You can display the points, move them around the splines, animate them, as well as add and delete points to create a correspondence that better suits your needs. The more correspondence points a spline has, the more control you have over how the warp or morph is mapped.

To show correspondences:

1. Toggle on the visibility of the source splines and destination splines using the Show buttons.
2. In the Distort tool, click Correspondences.

   Correspondence points are shown on the source and destination splines, and are connected by dotted lines. Selected splines show the connections with yellow dotted lines; unselected splines shown them in grey.



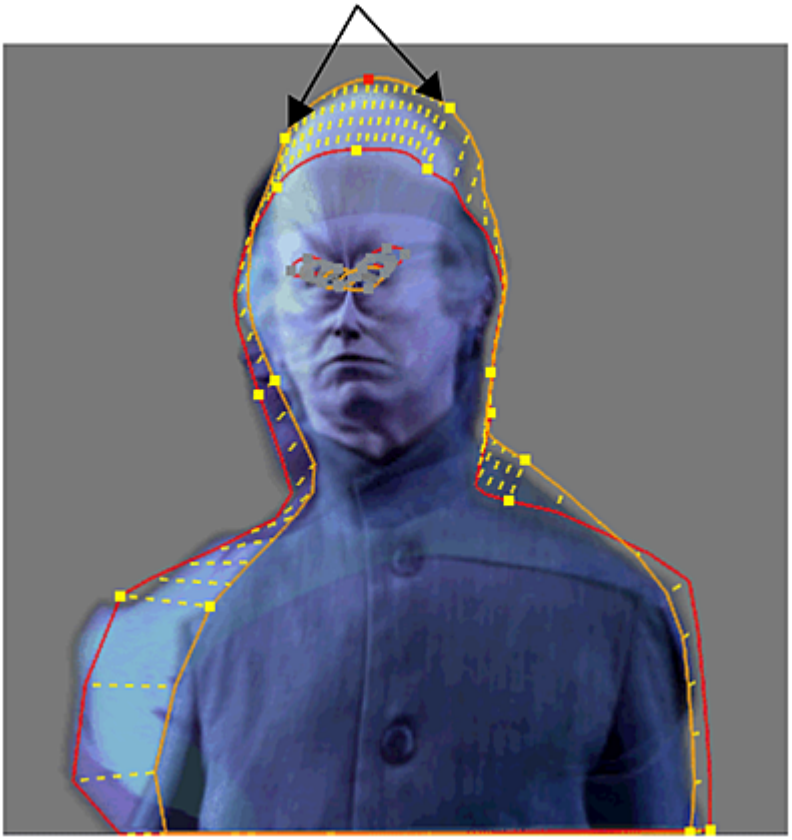
(a) Correspondences of unselected spline (grey) (b) Correspondences of selected spline (yellow) (c) Selected spline in Distort schematic

To move correspondence points:

1. Click Correspondences to display the correspondence points.
2. Select Move mode from the Edit box, then click and move a point on either the source or destination spline.

   You can also select multiple points by `Shift`-clicking them.

   Points can only be moved between the two points that surround it on the spline.



The correspondence point can only be moved between the two surrounding points

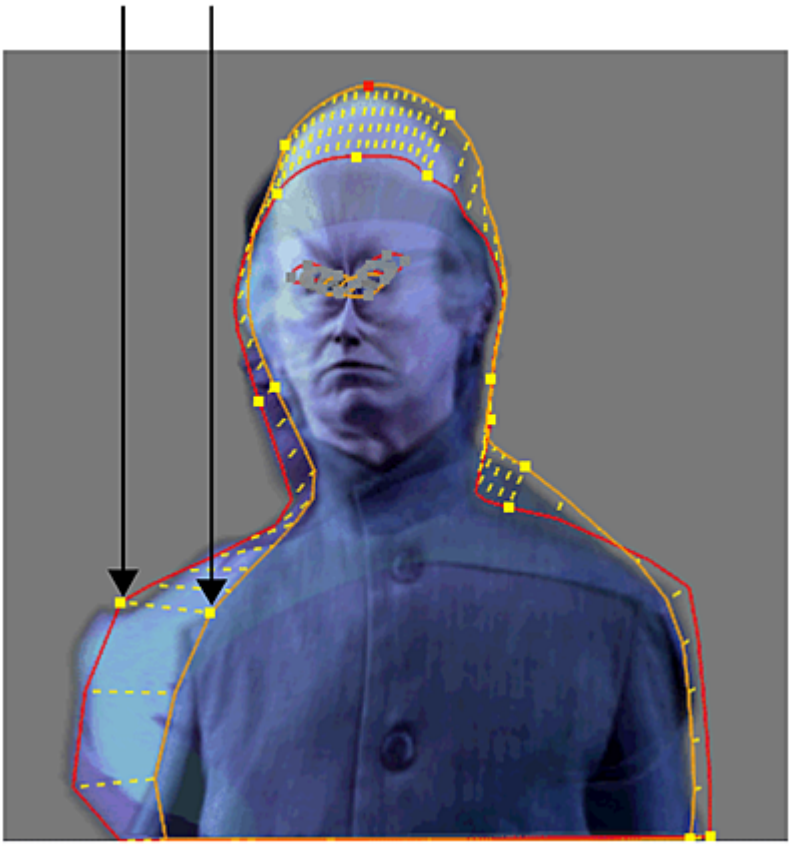
Image courtesy of Behavior Communications Inc.

The warped or morphed region is remapped according to the changes made to the correspondence points.

To add correspondence points:

1. Click Correspondences to display the correspondence points.
2. Select Add from the Tools box.
3. In the region of the clip where you want to improve the mapping, click the source or destination spline to add a new correspondence point.

   A point is added. As well, a corresponding point is added to the corresponding spline. You can then move either of these points to improve the mapping of the clip. The more correspondence points there are, the more refined the distortion.



Adding a correspondence point to a source spline adds one to the destination spline

Image courtesy of Behavior Communications Inc.

To animate correspondence points:

1. Move to the frame where you want to begin animating correspondence points.
2. Display a source and destination spline by clicking the Source Show and Destination Show buttons.
3. Click Correspondences to display correspondence points.

   TIP The source and destination splines should be offset from one another to best be able to animate their correspondence points.

4. Enable Auto Key in the Distort menu.
5. Select Move mode in the Edit box, and then move one, or more, correspondence points to a new location.
6. Advance the positioner to a different frame and then move a correspondence point again.
7. Continue to move correspondence points as you progress through the clip to refine the warp or morph over time.

   Correspondence point animation for each spline is kept in a corresPts channel in the Channel Editor. See Distort Animation Channels (page 1401).

To delete correspondence points:

1. Click Correspondences to display the correspondence points.
2. Select Delete from the Tools box.
3. Click a correspondence point on the source or destination spline.

The point you clicked and the corresponding point on the corresponding spline are deleted.

Adjusting Resolution

Resolution affects the sampling of the clips loaded into the Distort tool. The resolution value represents the number of pixel squares that the image is sampled at. Decrease the resolution value if you find your clips are not deforming smoothly.

NOTE Decreasing the resolution value slows down system performance.

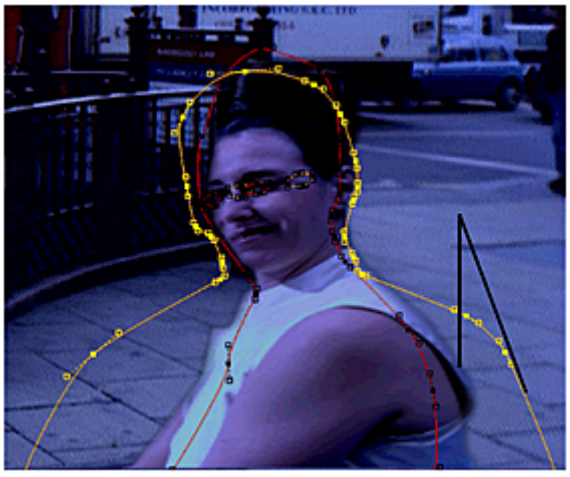
To adjust resolution:

1 Use the Resolution field to increase or decrease resolution. A smaller value increases the number of samples, producing a higher quality distortion.

Adjusting the Degree of Distortion

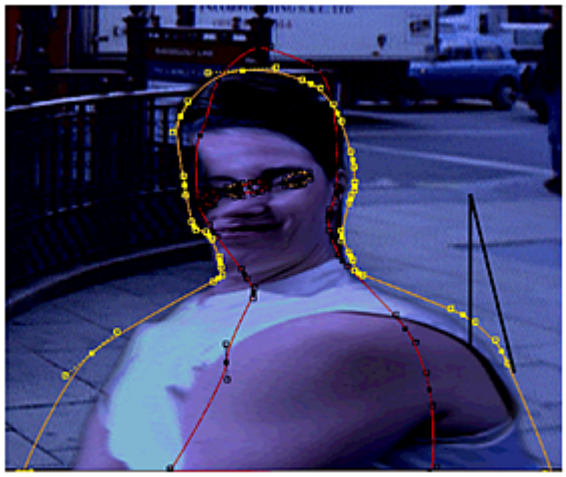
The Distort tool includes parameters that allow you to easily control the degree of distortion in a warp or morph. You can adjust the Iterations parameter so the distorted source pixels more closely approach the destination spline. Similarly, you can adjust the Range parameter to affect all the pixels in the image, or just those in the immediate region of the warp. Displaying the distortion grid can help you evaluate the effects of your changes.

To create a warp or morph, invisible iteration points are added between the source and destination splines. These iteration points determine how many timesthe pixels between the splines is examined when the calculations are performed. Adjusting the number of iteration points is relevant when the distance between the splines is great; that is, when you are trying to achieve a large warp. With a small iteration number, the warp will not “follow” the destination spline closely. Increasing the number of iterations will result in a more obvious distortion. For smaller warps or morphs, there is no advantage in using a greater number of iterations; it simply increases processing time unnecessarily.



With a low number of iterations, a distortion can sometimes be too subtle

Image courtesy of Behavior Communications Inc.



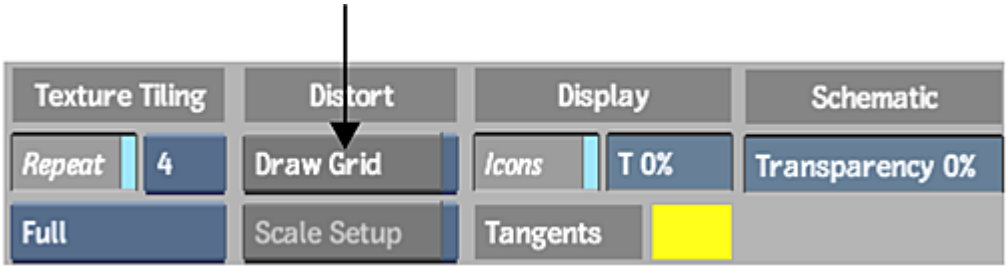
Increasing the number of iterations results in a more obvious distortion

Image courtesy of Behavior Communications Inc.

By adjusting the range, you can control the region affected by the distortion. A lower value decreases the region affected by the warp, while a higher one increases it. For example, a value of 100% affects all the pixels in the image, a value of 33% affects one-third of the image, a value of 25% affects one-quarter of it. To restrict the distortion to the area immediately surrounding the vertex, choose a higher region number. To broaden the effect, choose a lower number. This parameter has a smaller effect on processing time than the number of iteration points.

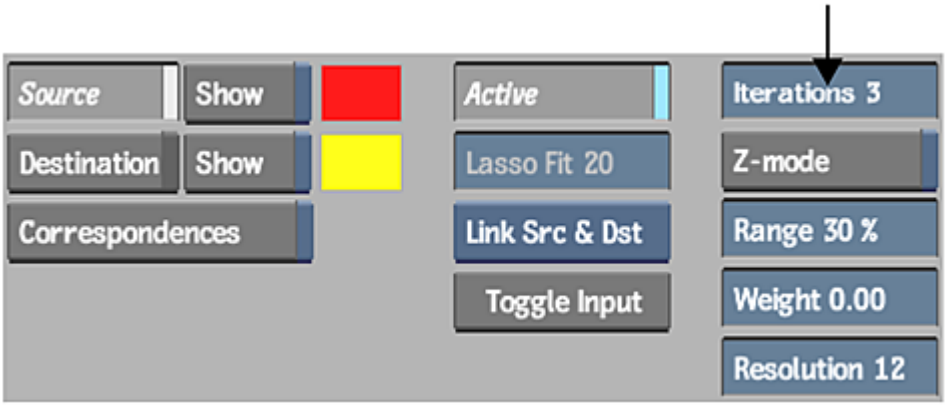
To display the distortion grid:

1 In Distort's Setup menu, enable Draw Grid.



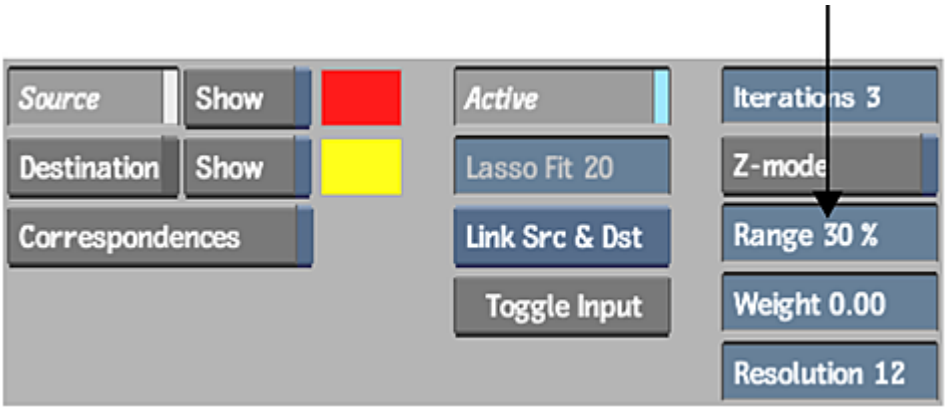
To adjust the amount the distortion "follows" the destination spline:

1 In the Warp or Morph menu, change the value in the Iterations field.



To change the size of the region affected:

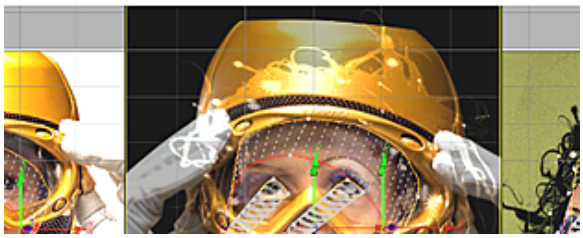
1 In the Warp or Morph menu, change the value in the Range field.



Filling in Gaps at Edges of Distorted Clips

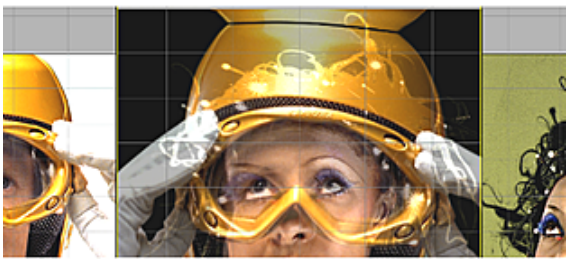
As you warp surfaces, sometimes their edges are pulled away from the absolute edge of the image, revealing either the background clip, or blank space.

You can fill this revealed area with the warped clip by repeating it.



Without Fill

Image courtesy of Das Werk and The House



With Fill, set to Part

Image courtesy of Das Werk and The House

To fill in gaps at edges of a distorted clip:

1 Click Setup.

(a) Repeat button (b) Repeat value (c) Amount box

2 In the Texture Tiling section, Enable Repeat.

3 Toggle the Repeat value to set the number of times the clip is repeated to fill the gaps:

- Select 4 to repeat the clip once on each side: top, bottom, left, and right.
- Select 8 to repeat the clip twice on each side.

4 Toggle the Amount button to control the portion of the clip to use to fill the gaps:

- Select Full to squeeze the whole clip into each of the gaps.
- Select Partial to use only a mirrored corresponding, and proportional portion of the clip to fill each of the gaps.

Warping Example

You can create many warps on a single input clip. Each destination spline offset from its source creates a warp on the clip. The following is an example of a simple warp.

To create a warp:

1. Once a spline shape has been made, click Warp to display the Warp menu.
2. From the View box, display the Result view to see the spline shape over the input clip.
3. Set the timebar positioner to the effect's first frame.
4. From the Warp menu, enable Source.

   All Front1 source splines are displayed.
5. If necessary, adjust the vertices of the shape by moving points or curve handles.

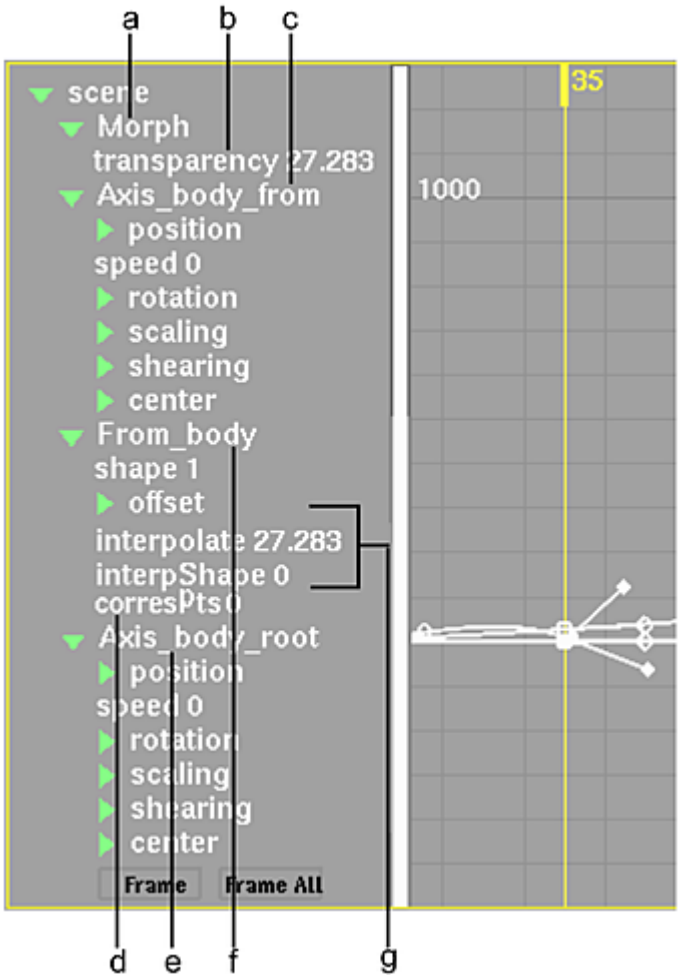
   TIP To prevent the warping of the clip as you edit a spline, make sure Link Src & Dst is selected in the Link box. Otherwise, as you edit either a source or destination spline, it will be offset from its corresponding source or destination—this is what will eventually create your warp. You can toggle between Link Src & Dst and Enable Warp by pressing the `W` keyboard shortcut.
6. Set the timebar positioner to the effect's final frame.
7. From the Link box, select Distort.

8. Enable Destination, and then adjust the vertices of the shape by moving points or curve handles. Edits to the destination spline-based shape will produce immediate warping.
9. Refine the warp as necessary. See Refining Distortions (page 1394).

Distort Animation Channels

The Distort tool includes channels for the Blend value as well as for all the Axis nodes in the schematic. Each spline in the effect contains a shape channel, with its vertex information, and an offset channel. As well, each spline contains a correspondence points channel containing point location information. All these channels are in the Morph folder in the Channel Editor.

When splines are linked, two additional channels appear under the Front1 spline's folder in the Channel Editor: an Interpolation channel corresponding to the value in the Interpolation field for the pair of linked spline nodes, and an interpShape channel. The interpShape channel contains the vertex information for the interpolated spline, which is the destination of the morph.

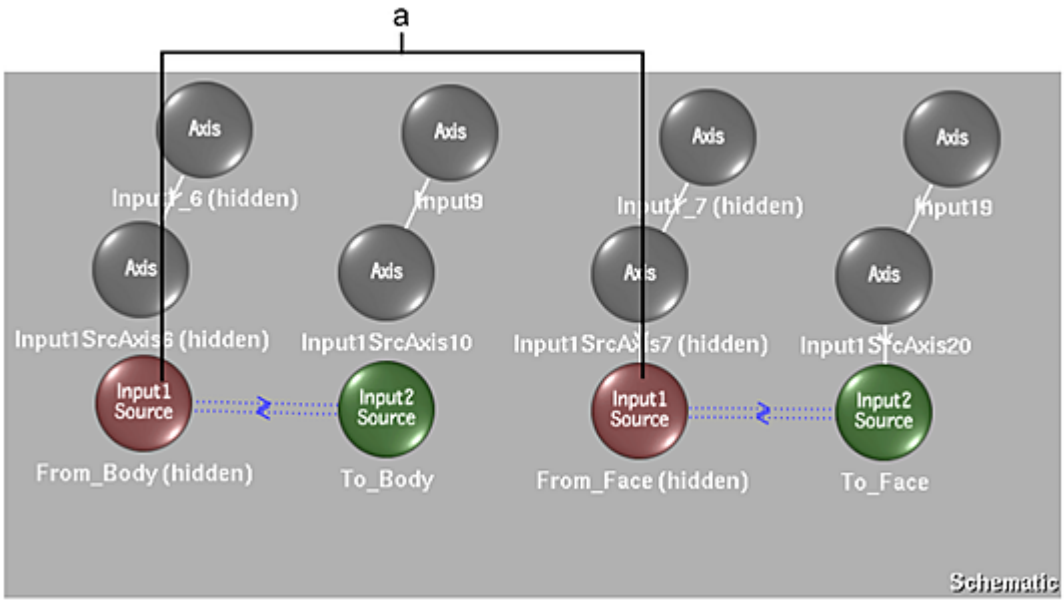


(a) Morph folder contains all folders and channels for both warps and morphs (b) Transparency channel corresponds to Blend parameter in Morph menu (c) Axis node folder for parent of Spline node (d) Correspondence points channel contains correspondence point information for the spline (e) Axis node folder for root of source and corresponding destination spline (f) Spline node folder for spline shape (g) Interpolation channels appear when spline nodes are linked in the schematic, under the Front1 (Input1) spline node's folder

Interpolation and InterpShape Channels

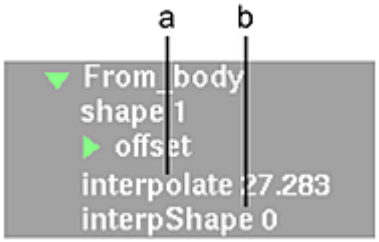
You can adjust the Interpolation and interpShape channels in the Channel Editor for each set of linked Front1 and Front2 splines to refine a morphing effect.

Each pair of linked Spline nodes produces an interpolated spline, which in turn, can be edited and animated independently.

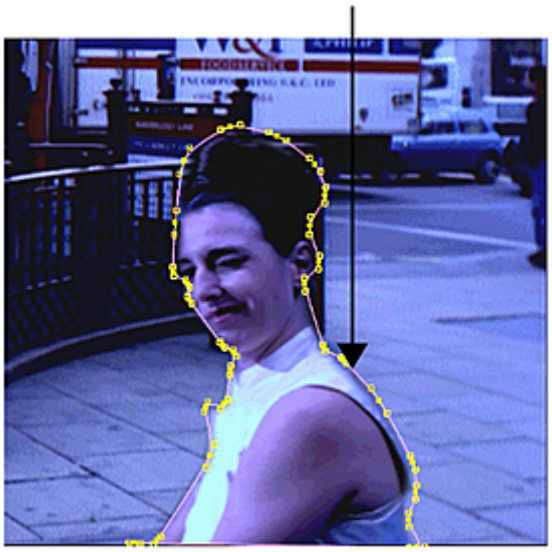


(a) Interpolation and interpShape channels appear for linked spline nodes

The two interpolation channels, corresponding to linked spline nodes, are always in the Front1 source Spline node's folder in the Channel Editor.



(a) Corresponds to value in Interpolation field of Warp or Morph menu (b) Contains vertex information for interpolated spline



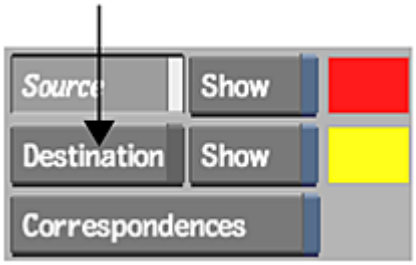
With Result selected in the View box, Front1 (Input1) selected in the Input box, and Source enabled, the source spline on Front1 (Input1) is displayed. It has a shape channel containing its vertex positions over time.

With Result selected in the View box, Front2 (Input2) selected in the Input box, and Source enabled, the source spline on Front2 (Input2) is displayed. It also has its own shape channel containing its vertex positions over time.

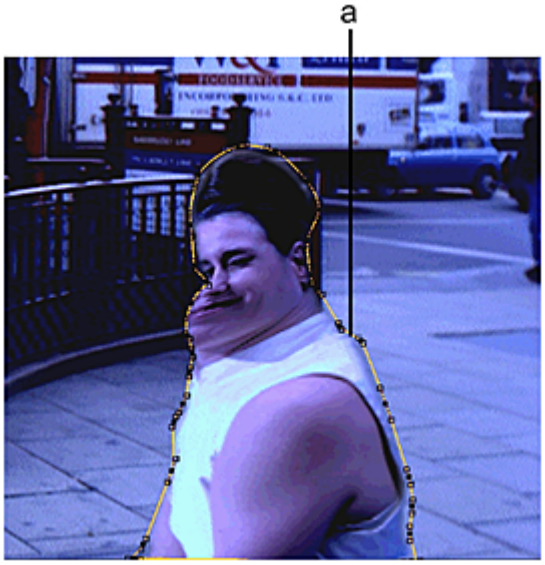
Image courtesy of Behavior Communications Inc.

Image courtesy of Behavior Communications Inc.

When an Front1 and Front2 spline are linked in the schematic, you select Result in the View box, Front2 in the Input box, and Destination to view the interpolated spline. The interpolated spline is the result of mixing the Front1 and Front2 source splines. Since it is the target of the morphing effect, you must click Destination to display it.



The interpolated spline's vertex information is contained in the interpShape channel in the corresponding Front1 spline's folder. Although the spline is a result of mixing the Front1 and Front2 source splines according to the Interpolation value, it can be controlled and animated independently.



(a) Morph follows contour of interpolated spline

Image courtesy of Behavior Communications Inc.

TIP Since each Interpolation channel, for every pair of linked Front1 and Front2 source splines, is independent, try using the Channel Editor's Link button on multiple Interpolation channels. This allows a common interpolation value for many sets of linked splines.

Using Distort in Batch

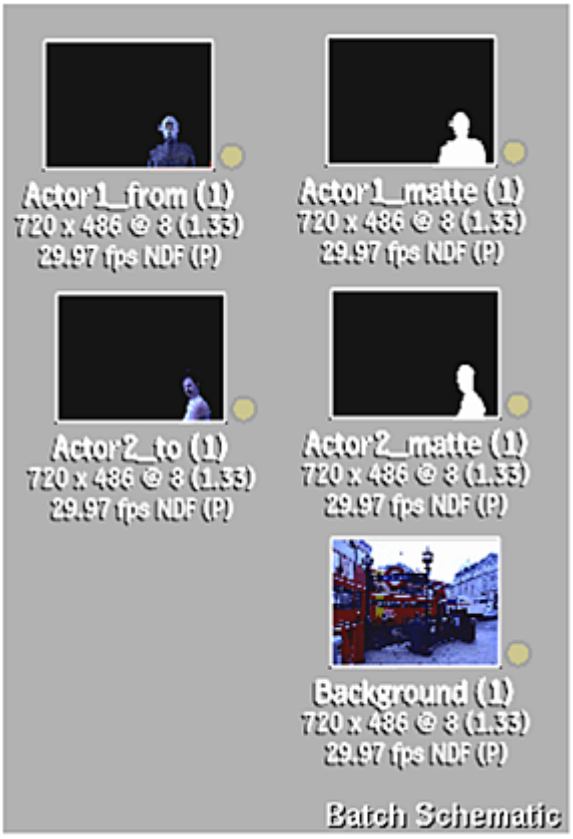
Use other nodes in combination with the Distort node in Batch to create more seamless warps and morphs that can be composited into media in Action.

Creating a Distortion to Feed into Action Media

You can create a morph effect in Distort, then feed the result, and the ResultMatte, into media in an Action node, to composite with additional media.

To create a distortion feeding into Action media:

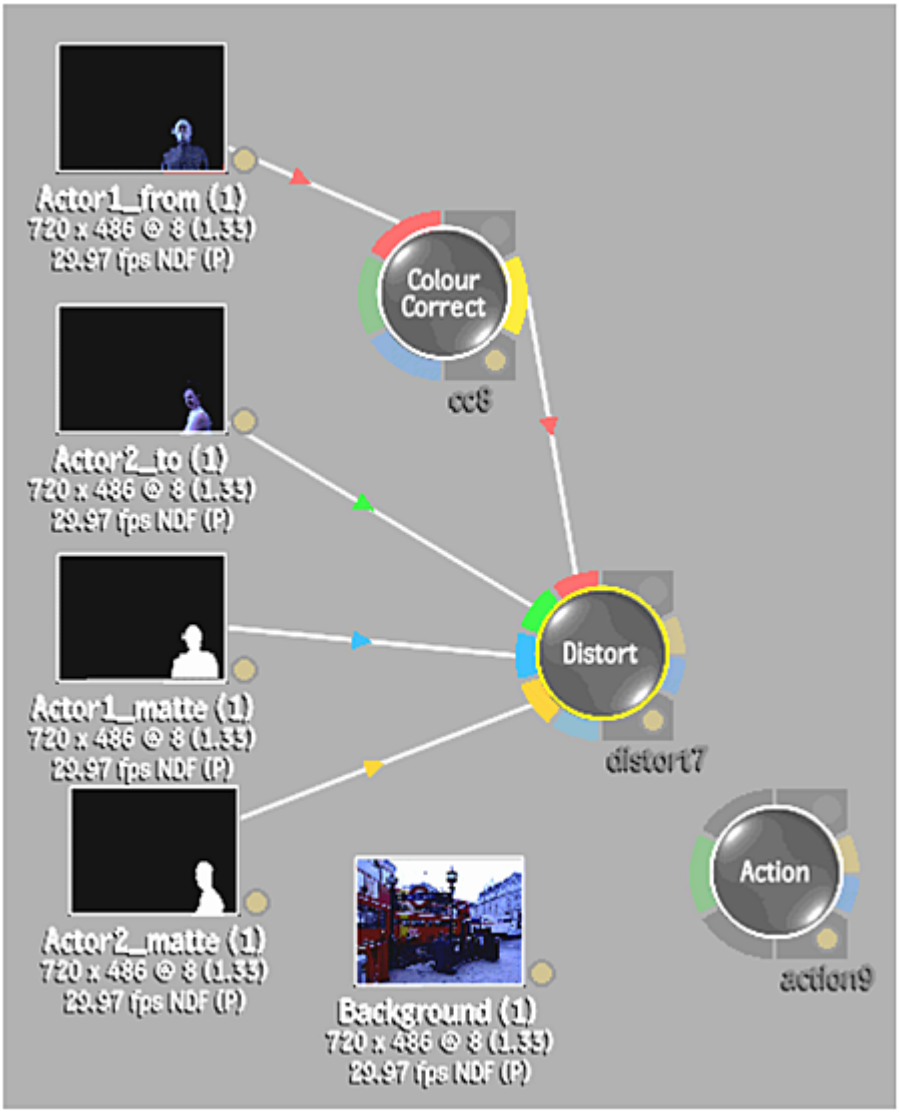
1. In Batch, add an Front1 clip and its corresponding matte, and an Front2 clip and its corresponding matte.
2. Add a background clip.



Images courtesy of Behavior Communications Inc.

NOTE The background clip will feed into the Action node, not the background socket of the Distort node.

3. Add a Distort node.
4. Add a Colour Corrector node.
5. Add an Action node.
6. Connect the Front1 clip to the Colour Corrector node.
7. Connect the Colour Corrector node to the Front1 socket of the Distort node.
8. Connect the other clips to their corresponding sockets on the Distort node: Front2, Matte1, and Matte2. There is no background connected to the background socket since Distort will feed into media of the Action node.

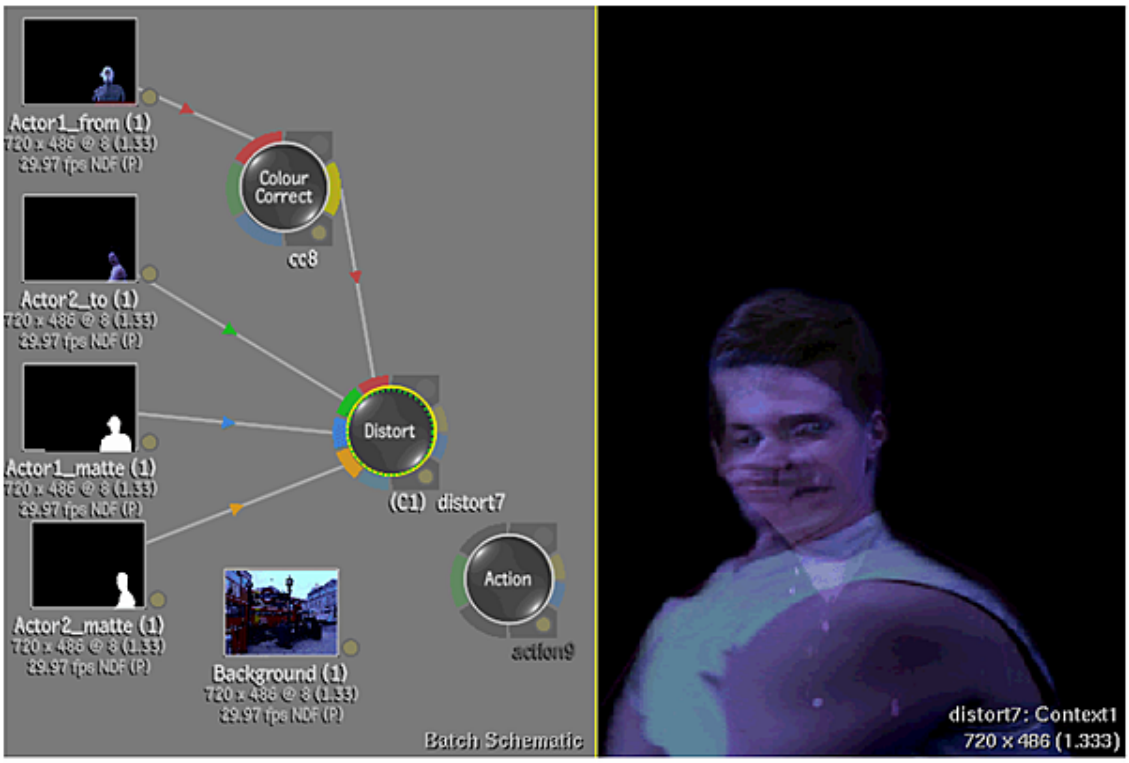


Images courtesy of Behavior Communications Inc.

9 Double-click the Distort node to load its controls into the work area.

10 Create the morph between the Front1 and Front2 clips.

11 Split the screen and create a context view for the Distort result and display it in a viewport.



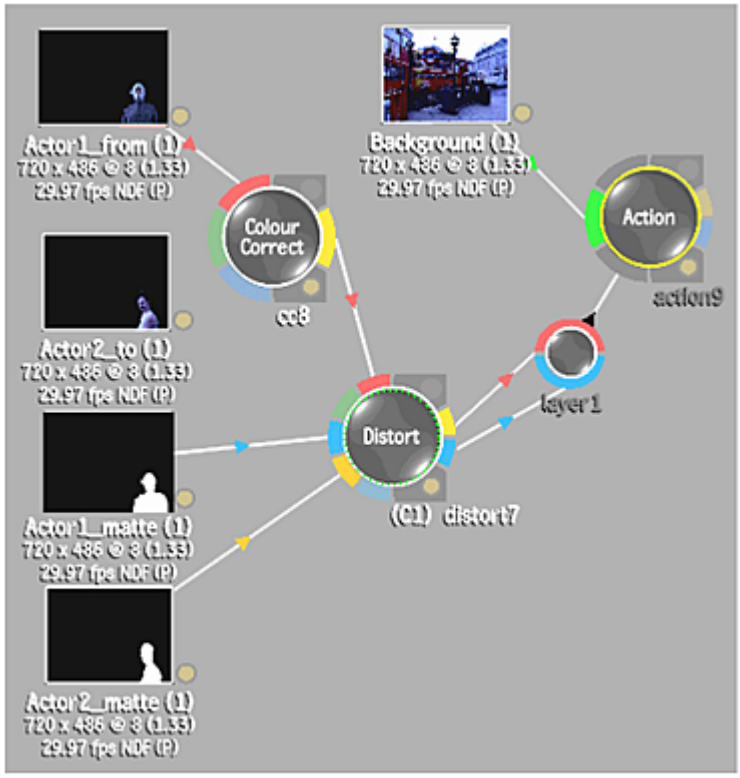
Images courtesy of Behavior Communications Inc.

12 In the other viewport, access the result of the Colour Corrector. Create an animated colour correction for the Front1 clip to gradually transition to the colours in the Front2 clip.

13 In the Action node, connect a background clip to it.

14 Within the Action node, `Ctrl`-click Add to add media that feeds into the Action node in the Batch schematic.

15 Feed the Result and the ResultMatte of Distort into the front and matte of the media, in turn feeding into Action.



Images courtesy of Behavior Communications Inc.

16 In Action, add a surface for the media.

17 Animate the media.

Saving Preferences

You can save preferences for the Distort tool and node.

The state of the following parameters are saved as preferences:

- Autokey
- Icons
- Icon Transparency
- Fill
- Source/Destination
- Show Destination
- Axis Motion Path
- Axis Scale Proportional

You can save preferences as a file you can reload, or as the tool's or node's defaults which are loaded when you reset all, or create new Distort nodes in Batch.

For more information on saving and loading preferences, and managing default preferences, see Setting Preferences (page 1933).

Saving and Loading Distort Setups

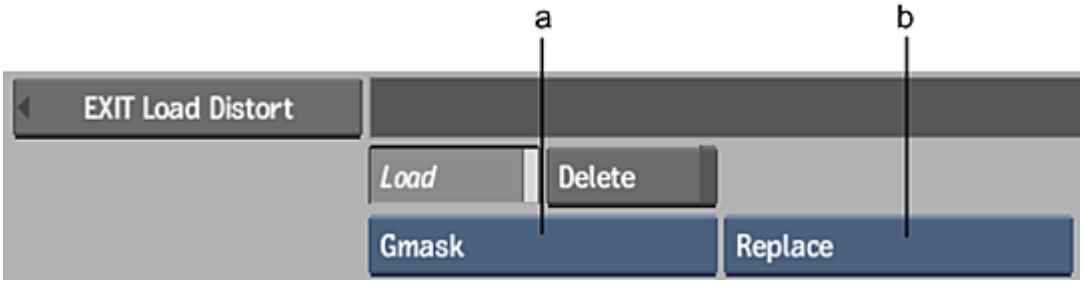
Distort setups can be saved and loaded as setup files, with spline and other information preserved. You can also exchange Distort setups between the Distort tool and the Distort node in Batch.

To save a Distort setup:

1. Do one of the following:
   - In the Distort menu, click Save to open the file browser.
   - From the Distort node in Batch, click NodeSetup to display Distort's Setup menu, then click Save Node to open the file browser.
2. Select Setup as the type of file to save.
3. Type the name of the setup, and click Save.

To load a Distort setup:

1. Do one of the following:
   - In the Distort menu, click Load to open the file browser.
   - From the Distort node in Batch, click NodeSetup to display Distort's Setup menu, then click Load Node to open the file browser.
2. From the Load Type box, select the type of files to load.



(a) Load Type box (b) Load Mode box

3. From the Load Mode box, select whether to want the setup to append to or replace any existing Distort splines in your scene.
4. From the file browser, select the name of the setup to load.

   NOTE If the Distort setup you are loading was created in a different resolution than the current project, click Scale Setup to scale the setup's splines.

About the Warper

Use the Warper to warp a clip or morph from a source clip to a result clip. Warping is free-form distortion of an image; it is a transition effect that matches the morphology of one clip gradually to that of another clip, such as when a human is morphing into an alien.

Accessing the Warper

You must select the required clips to access the Warper. You can enter the Warper with one to five clips. The number of clips that you load depends on the effect that you want to produce.

To access the Warper:

1. Select **Tools ➤ Filter ➤ Warper**.
   The cursor changes to Pick Front.
2. Select the types of clips you want to load into the Warper from the Input Mode box.

Select:	To load:	To produce:
Front1	Front clip only	A warp of a single image.
Front1/Front2	Front and back clips	A morph, a wipe, or a regional warp.
Front1/Matte1	Front and matte clips	A warp of a single image using a matte.
Front1/Front2/Matte1	Front, back, and matte clips	A warp over a background using a matte.
F1/F2/M1/M2/Back	Front, matte, back, back matte, and background clips	A morph using two clips and their mattes on a background clip instead of on black.

3. Select the front clip.
   The cursor changes to Render Here.
4. Click on any free (or a grey) area on the workspace.
   You are now in the Warper Tool.

   NOTE Regardless of which clips you load into the Warper, there are always two compositing layers available: a front layer and a back layer. If you do not load a back clip, black is used for the back layer.

To reset the Warper:

1. Click Reset All, and click Confirm when prompted.
   Warper options are restored to their default settings.

Setting Up the Clips

Use the Clip Setup boxes to specify how each clip loaded into the Warper will be used.

For Front and Back clips, the Clip Setup box contains On, Off, and Lock options. For Matte and Back Matte clips, the Clip Setup box contains On, Off, and Invert options. Background clips contain On and Off options.

Select:	To:
On	Warp the clip.
Off	Disable the clip. When a clip is disabled, it is not displayed and is not used in the processed result.
Lock	For Front and Back only, process the front or back clip in the final composite without warping it.
Invert	For Matte and Back Matte only, invert the front or back matte.

Activating Clips

The following table shows how to set up the clips to produce various effects.

Front1	Front2	Matte1	Matte2	Back	Effect
On	Off	Off	Off	Off	Warps the front clip only. This is the same as loading a front clip only.
On	On	Off	Off	Off	Morphs between two clips. This is the same as loading a front clip and a back clip.
On	Lock	Off	Off	Off	Wipes (page turns, rolls); regional warps (if the same clip is loaded as the front clip and the back clip).
On	Off	On	Off	Off	Warps the front clip and matte clip, composited on black.
On	On	On	Off	Off	Warps over a background with a matte clip.
On	Lock	On	Off	Off	Warps the front and matte clips composited on the back clip.
On	On	On	On	On	Morphs between two clips and their mattes onto the selected background clip.

Creating Meshes

After you load and set up your clips in the Warper, you create meshes to define the shape and position of the input image—the original image before the warp or morph—and the output image—the result of the warp or morph. The meshes use control points to determine the change, in shape and position, and create a convincing transition from the input image to the output image.

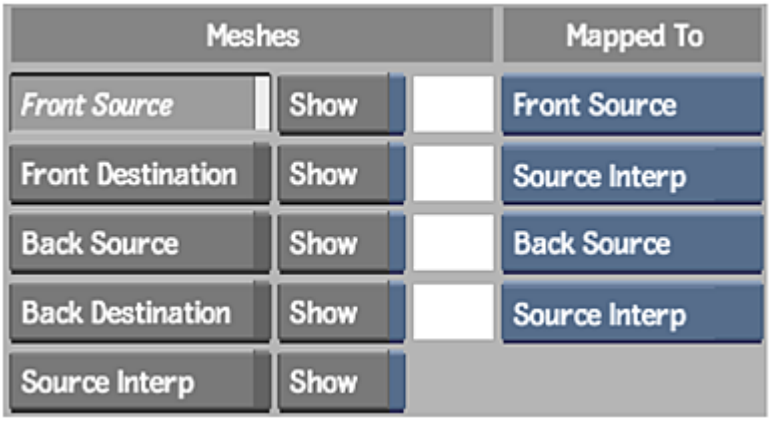
You use two meshes, a source mesh and a destination mesh, to generate the warp effect. For instance, if you want to morph a square into a circle, the source mesh defines the input image, which is a square, and the destination mesh defines the output image, which is a circle.

Defining a Mesh

When creating a mesh, you anchor the mesh to the surrounding area. To ensure this, create the mesh over a slightly larger area of the image than the part you want to affect. For example, to warp a person's mouth, extend the mesh to cover the lower half of the person's face. This “anchors” the area surrounding the warp so that it is not affected by the warp. You begin by defining the number of patches in the mesh and then placing it over the required area in the image.

To define a mesh:

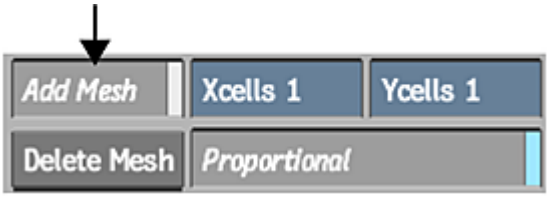
1. Go to the frame where you want the effect to begin.
2. Specify which mesh will be created.



Click:	To create:
Front Source	The front source mesh.
Front Destination	The front destination mesh.
Back Source	The back source mesh in a morph.
Back Destination	The back destination mesh in a morph.

For information on when to create each type of mesh, see Warping (page 1424) and Morphing (page 1426).

3. Click Add Mesh.



4 In the Xcells and Ycells fields, set the number of patches for the mesh. For example, to create a mesh that is 10 patches wide by 5 patches high, enter 10 in the Xcells field and 5 in the Ycells field.

Enable the Proportional button to automatically set the X and Y cells fields to the same value.

5 Click the image and drag the cursor to draw the mesh.

TIP If you cannot see the mesh properly against the colours in the image, you can customize the colours of the various meshes using the colour pots beside each mesh name.

Adding Patches

As you build the mesh, you may need to add patches to cover more of the image.

To add patches:

1 In the Warper menu, click Add Mesh.
2 To add only one patch, click the edge of the patch that is closest to where you want to add a patch.

The new patch is approximately the size of the adjacent patch.

3 To add several patches, drag from the edge of an existing patch over the area in which to add patches.

Each new patch added is approximately the size of its adjacent patch.

Deleting Patches

You can delete patches if you make a mistake while creating the mesh.

To delete patches:

1 In the Warper menu, click Delete.

2 To delete a patch that lies along the boundary of the mesh, click the middle of the patch.
3 To delete a line that connects two patches that are not along the boundary, click both sides of the line.

Modifying a Mesh

Use the modelling controls in the Warper menu to modify the meshes to define the original and the final shape of the image. You can modify the shape and size of the patches or the location and orientation of the mesh, divide the patches into smaller subpatches, and split the edges into smaller segments.

NOTE The more detail you add to the mesh, the more control you have when warping or morphing a clip. However, if you have too much detail, the meshes may be difficult to modify.

If you cannot see the mesh properly against the colours in the image, you can customize the colours of the various meshes, as well as those of the splines, tangents, and vertices. See Display Settings (page 1432).

Moving Control Points, Edges, and Tangent Handles

You can manipulate control points and tangent handles on the mesh to add precision to your warp effect. Click the control points to access their tangent handles, then click the tangent handles to change the shape of a patch.

Click the Move button to move a vertex, an edge, or a tangent handle.

To:	Click:
Move a control point	The control point and drag. Note that the X, Y, and Z coordinates of the selected vertex appear in the corresponding fields in the menu.
Move an edge	Near the centre of the edge and drag.
Move a tangent handle	The corresponding vertex to display its tangents, then click the tangent handle and drag.

Selecting Multiple Control Points

Select multiple control points to translate, scale, or rotate all or part of the mesh while maintaining the spatial relationship between the selected control points.

To select multiple control points:

1 In Warper menu, click Select.

2 Hold down the `Ctrl` key and drag a selection box over the control points that you want to select.

NOTE To deselect individual points, click the points. To deselect all, click outside the area of selected control points.

Translating, Scaling, and Rotating the Mesh

Use the Translate, Scale, and Rotate options to change the position, size, and orientation of part or all of the mesh.

To translate, scale, or rotate the mesh:

1 Select the appropriate area of the mesh. See Selecting Multiple Control Points (page 1414).

2 In the Warper menu, select Translate, Scale, or Rotate.

3 Click Move.

4 Click one of the selected control points and drag.

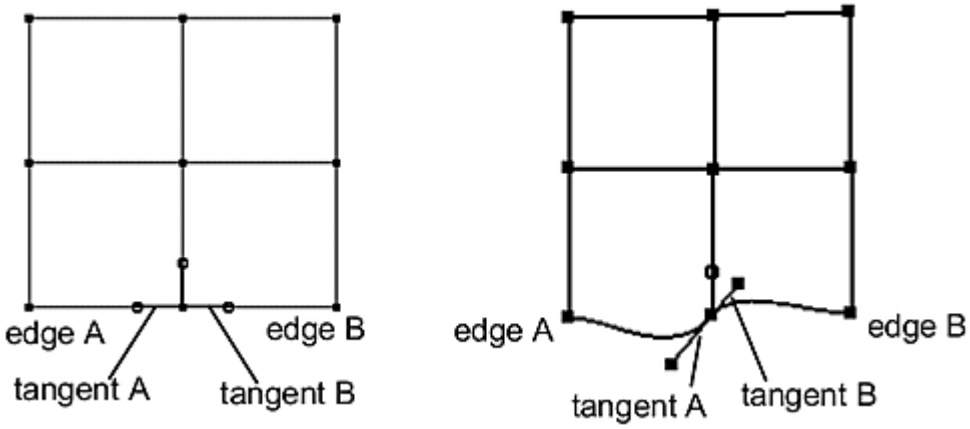
The selected area of the mesh is translated, scaled, or rotated.

NOTE You can also translate, scale, and rotate the entire mesh using the Translate, Scale, and Rotate channels in the Channel Editor. However, you cannot use the Channel Editor to affect only part of the mesh.

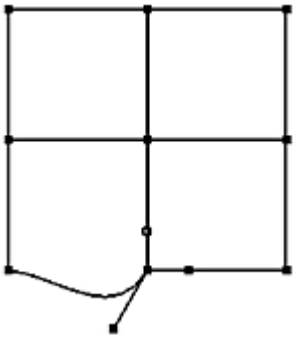
Breaking Tangents

By default, a vertex's tangents work together. Moving one tangent has the inverse effect on the opposite one to maintain a smooth curve at the vertex. In certain cases, you may want the tangents to move independently of each other.

Consider the mesh in the following example.



Suppose that you want to move tangent A to change the shape of edge A. Moving tangent handle A also causes tangent B to move in the opposite direction (as shown above); tangent A and tangent B are locked. You can use the Break command to break the tangents and move them independently (as shown here).



After using Break to break two tangents, you can use the Auto command to rejoin them. Note that resetting the tangents also resets the corresponding edges to their default positions.

To break tangent handles:

1. In the Warper menu, click Move.
2. Click a vertex to display its tangents.
3. Click Break, then click a tangent handle.

   The tangent and the opposite one can now be moved independently. The tangent handles are solid indicating they are broken.

   NOTE If the vertex has four tangents, click one handle of each pair to make all four tangents move independently.

To reset broken tangent handles:

1. In the Warper menu, click Move.
2. Click a vertex to display its tangents.
3. Click Auto, then click a tangent handle.
   The tangents and edges are reset.

Subdividing a Patch

You can divide patches in a mesh into smaller subpatches and split the edges into smaller segments using the Subdivide button.

To subdivide a patch:

1. In the Warper menu, click Subdivide.

2. Click the edge of a patch and drag the cursor to the opposite edge.

   NOTE To delete an edge created with Subdivide, enable Delete and click the edge.

Adding a Vertex

For more control when manipulating an edge, add a vertex to the edge.

To add a vertex:

1. In the Warper menu, click Split.

2 Click the edge of a patch where you want to add a vertex.

NOTE To delete a vertex created using the Split button, click Delete, then click the vertex.

Using the Freehand Mesh Tool

Use the Freehand Mesh tool to perform intricate mesh modelling.

To create a freehand mesh:

1 In the Warper menu, click Freehand.

2 Click the edge of a patch, and continue clicking until you complete the mesh.

3 `Ctrl`-click the final vertex.

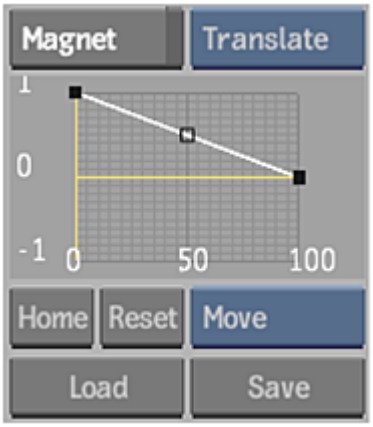
To delete a freehand mesh:

1 Click Delete.

2 Click the spline or vertex you want to delete, or click Undo.

Warping with the Magnet Tool

Use the Magnet tool to define the shape of the warped image. The Magnet tool warps the area of the mesh under the Magnet. You can set the size of the Magnet and define its effect on the selected area.



Note that the Magnet's effect is most noticeable when the selected area includes many control points (that is, when the grid has many patches, when you are using a large Magnet, or both).

To use the Magnet tool:

1. In the Warper menu, click Magnet.
2. Place the cursor over the mesh and adjust the Magnet size by pressing `Ctrl+S` and dragging.
3. Place the Magnet over the control points you want to warp and click.

   The affected control points turn red.
4. Drag the Magnet to produce the effect you want.

   NOTE By default, the Magnet translates the mesh. To scale or rotate the mesh, select the Scale or the Rotate option in the selection box beside the Magnet button.

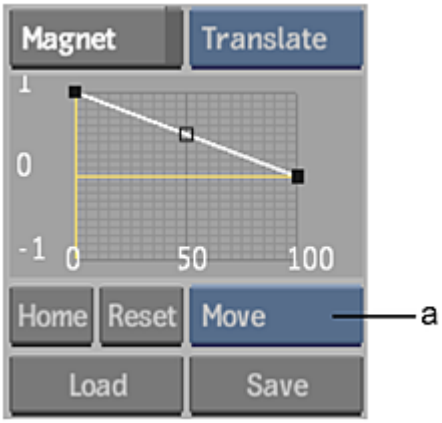
Customizing the Magnet's Effect

You can customize the Magnet's effect by changing the shape of the curve in the Magnet Curve Editor. Note that, by default, the Magnet has the greatest effect on the pixels at the centre of the Magnet and the least effect on the pixels at its perimeter.

To customize the Magnet's effect:

1. In the Warper menu, click MEditor.

   The Magnet Curve Editor appears.



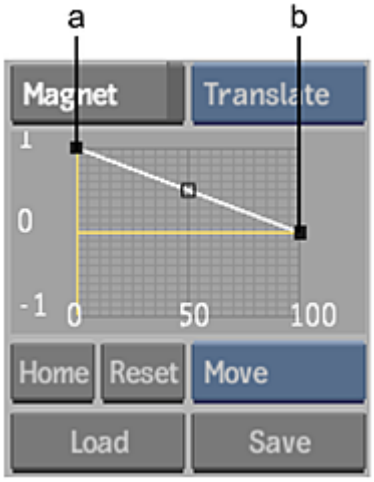
(a) Tools box

2 Adjust the shape of the Magnet curve.
You can also use the Edit Mode options to modify the curve.

NOTE You can save and load custom Magnet settings using the Save and Load buttons in the Magnet Curve Editor.

Tips and Tricks

You can experiment with the Magnet curve to achieve different effects.



(a) Point representing effect at the Magnet's centre (b) Point representing effect at the Magnet's perimeter

- Invert the curve to affect the pixels at the perimeter of the Magnet more than those at the centre.
- Flatten the curve along the horizontal axis to have the same effect on all pixels under the Magnet.
- Add extra control points and create a sine or wave curve to make the control points move in opposite directions.
- Move the curve into the negative horizontal axis to create a magnet with an opposing magnetic effect.

Modifying the Meshes for Live-Action Clips

If you are warping or morphing a live-action or moving clip, you must define the source mesh at several frames in the clip. The Warper interpolates the location and shape of the mesh between frames.

Creating a mesh over the length of a live-action or moving clip can be time-consuming. To simplify the mesh-building process, create the mesh for the first frame of the required clip, then use the Stabilizer to generate the successive meshes.

NOTE You can track the source or destination mesh for either the front or the back clip.

With the Stabilizer, you can track the movement in the clip, and then apply the tracking data to the mesh (tracking by Axis). This translates the mesh so that it follows the movement in the clip without changing shape. Alternatively, you can track the movement of selected points in the image, and apply the tracking data to the mesh (tracking by Vertices). This translates and changes the shape of the mesh so that it matches the position and shape of the image over the length of the clip.

Loading a Clip into the Stabilizer

Use the Stabilizer controls in the Warp menu (if you are warping) or in the Morph menu (if you are morphing) to select the clip you want to track, the mesh you want to apply the tracking data to, and the tracking mode.

Select:	To:
Front	Track the front clip.
Back	Track the back clip.
Axis	Apply the tracking data to the axis of the mesh. You can use one or two trackers in the Stabilizer.
Vertices	Apply the tracking data to the mesh control points (vertices). You must select the control points you want to track before entering the Stabilizer. You can track up to 1000 control points. Note that tracking by vertices sets a shape keyframe at every frame of the clip.
Source	Apply the tracking data to the source mesh of the selected clip.
Destination	Apply the tracking data to the destination mesh of the selected clip. This option can be useful if front and back destination meshes are different.

Tracking by Axis

To make the mesh follow the movement in the clip without changing shape, apply the tracking data to the mesh axis.

To track by axis:

1. Set the Stabilizer controls to apply the tracking data to either the Source or Destination mesh for either the Front or Back clip.
2. Select Axis.
3. Click the Stabilizer button.

   The Stabilizer menu appears.
4. Position the tracker on the image. See Working with Trackers (page 1067).
5. To use two trackers, click Tracker 2, click Active, and position the second tracker on the image.
6. Click Analyse to generate the tracking data.
7. If necessary, fine-tune the analysis.
8. When you are satisfied with the tracking results, click Return to apply the results and return to the Warper.

Tracking by Vertices

To make the mesh follow the movement and shape change in the clip, apply the tracking data to the mesh vertices.

To track by vertices:

1. Select the mesh control points that you want to affect. See Selecting Multiple Control Points (page 1414).
2. Set the Stabilizer controls to apply the tracking data to either the Source or Destination mesh for either the Front or Back clip.
3. Select Vertices.
4. Click the Stabilizer button.

   The Stabilizer menu appears. A tracker appears on each vertex that you selected. The reference frame is the frame from which you opened the Stabilizer.
5. If necessary, adjust the position of the trackers to ensure that each tracker has a good reference point.
6. Click Analyse to generate the tracking data, and if necessary, fine-tune the analysis.

   For example, disable a problem tracker and re-analyse, as described in Tracking Difficult Shots and Correcting Errors (page 1091).
7. When you are satisfied with the tracking results, click Return to apply the results and return to the Warper.

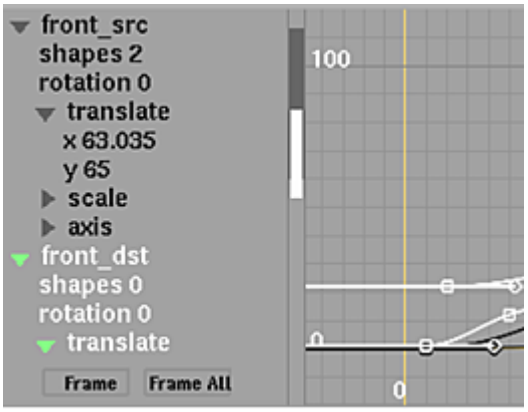
Copying and Pasting Tracking Data

You can copy and paste tracking data from one mesh to another using the Channel Editor. For example, if tracking was done using Axis, you can copy the translation of the front source mesh and apply it to the back source mesh.

To copy translation data:

1. Click Animation to view the Channel Editor.
2. Select the X and Y channels in the front_src translate folder.
3. Click Copy.
4. Select the X and Y channels in the front_dst translate folder.
5. Click Paste.

   The translation channels containing the tracking data from the front source mesh are pasted into the front destination mesh.



If tracking was done using Vertices, you can copy the shape animation of the front source mesh and apply it to the front destination mesh.

Example: To copy shape animation data:

1. Select the front_src shapes channel.
2. Click Copy.
3. Select the front_dst shapes channel.
4. Click Paste.

   The shapes channel containing the tracking data from the front source mesh is pasted into the front destination mesh.

Copying and Pasting Meshes for a Live-Action Clip

For a live-action clip, use the Channel Editor to copy the shapes curve of one mesh and paste it into another mesh. The shape and translation of the mesh at each frame are copied.

To copy and paste meshes for a live-action clip:

1. Create the mesh at the first frame of a clip and modify it over several frames to define the movement in the clip.
2. Click Animation to view the Channel Editor.
3. Select the shapes channel for the mesh you are copying (front_src, front_dst, back_src, or back_dst).
4. Click Copy.
5. Select the shapes channel for the mesh that you want to be the same as the copied mesh (front_src, front_dst, back_src, or back_dst).
6. Click Paste.

   NOTE If Auto Key in the Setup menu is enabled, a keyframe is created in the associated animation channel at the current frame every time the mesh is copied to a new frame and modified. When you copy the shapes curve to a new mesh, you copy the mesh and all its keyframes.

Copying and Pasting Meshes for a Still Clip

For a still clip, use the Copy and Paste buttons in the Warper menu to copy a mesh and paste it into another mesh.

To copy and paste meshes for a still clip:

1. Create the mesh at the first frame of the clip.
2. Click Copy in the Warper menu to copy the mesh—including its translation, scale, and rotation values.
3. Go to the required frame and display the mesh (source or destination) where you want to paste the copy.
4. Click Paste.

Warping

Warping changes the shape of the image in a clip. When you warp an image, you start by drawing a mesh over the region of the image you want to warp. You then modify the mesh over the image at specific keyframes to distort the image.

To create the warp, define a source mesh and a destination mesh for the front clip. The source mesh defines the area of the image that you want to warp; it outlines the shape of the non-distorted image in each frame of the clip. The destination mesh defines what the warped image looks like; it outlines the shape of the distorted image in each frame of the clip. Typically, you set the destination mesh at several frames of the clip. The Warper generates a smooth transition between frames.

The general steps for warping:

1. Load the appropriate clips into the Warper. If you are warping just a region of the clip, load the same clip as the Front and the Back clip. See Warping a Region of a Clip (page 1426).
2. In the Warper menu, click Warp.
3. Set up the clips. See Setting Up the Clips (page 1410). In general:
   - Set Front to On.
   - If you are warping just a region of the clip, set Back to Lock.
4. Click Front to view the front clip.
5. Define the source mesh for the original image.
6. Define the destination mesh for the warped image.
7. Process the clip.

Defining the Source Mesh

Use the modelling tools to define the source mesh so that it outlines the original shape of the image in the front clip.

To define the source mesh for the original image:

1. Go to the first frame of the clip.
2. Click Front Source to define the source mesh for the front clip.
3. Add a mesh over the area to warp and modify its shape to match the original image.

4 If you are using a live-action or moving clip, the source mesh must follow the movement of the image. You can either modify the mesh manually throughout the clip, or use the Stabilizer to track the movement in the clip. See Modifying the Meshes for Live-Action Clips (page 1420).

Defining the Destination Mesh

Set the destination mesh to be the same as the source mesh at the first frame, then change the shape of the destination mesh at different frames of the front clip to outline the required shape of the distorted image.

To define the destination mesh for the warped image:

1 Click Front Source.
2 Go to the first frame in the clip.
3 Click Copy to copy the source mesh.
4 Click Front Destination to select the destination mesh for the front clip.
5 Click Paste to paste the copied mesh into the destination mesh.
6 At several frames in the clip, modify the destination mesh to outline the required shape of the warped image.

TIP In the view box, select Result to see the pixels move as you move the mesh points.

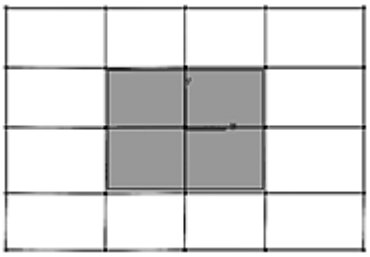
7 When you finish modifying the meshes, go to frame 1 and click Render.

Warping a Still Clip

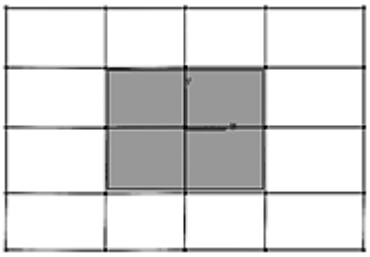
To warp a still clip, you define the source mesh only for the first frame of the clip since the object to warp has the same shape and location in each frame of the clip. Typically, you have to define the destination mesh only for the first and the last frames of the clip. The Warper generates a smooth transition between the two frames.

The following illustration shows the source and destination meshes used to warp a square into an oval.

Source Mesh:

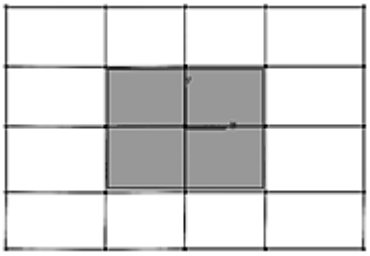


First Frame

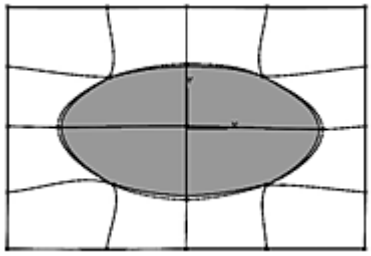


Last Frame

Destination Mesh:



First Frame



Last Frame

Warping a Region of a Clip

When a clip is processed, only the area defined by the mesh is processed. If you want to warp only a region in an image (known as regional warping), but you want to see the entire image in the result, you must load the same clip as the front and the back clip, and then lock the back clip.

Morphing

Morphing gradually transforms an image in a front clip into an image in a back clip. The effect is achieved by warping the two images and dissolving between the front and back clips. Unlike warping, which requires only a front source mesh and a front destination mesh, morphing requires a source mesh and a destination mesh for both the front clip and the back clip.

The front source mesh defines the original shape of the image in each frame of the front clip. Similarly, the back source mesh defines the original shape of the image in each frame of the back clip.

NOTE You should create the mesh over a slightly larger area of the image than just the part you want to affect. See Defining a Mesh (page 1412).

Both the front and the back destination meshes correspond to the warped image. Since the front clip transforms into the back clip:

- At the first frame, the front and back destination meshes correspond to the shape of the front clip.
- At the last frame, the front and back destination meshes correspond to the shape of the back clip.

The general steps for morphing:

1. Load the appropriate clips into the Warper. You need to load at least a front clip and a back clip.
2. In the Warper menu, click Morph.
3. Set up the clips (see Setting Up the Clips (page 1410)). In general:
   - Set Front to On.
   - Set Back to On.
4. Click Front to view the front clip.
5. Define the front source mesh to match the shape of the front clip at each frame.
6. Define the back source mesh to match the shape of the back clip at each frame.
7. Define the front and back destination meshes. You can either use Source interpolation to automatically modify the destination meshes or copy and paste the meshes and set the keyframes manually.

8 When you finish modifying the meshes, go to frame 1 and click Render.

Defining the Front Source Mesh

The front source mesh defines the original shape of the image in each frame of the front clip.

To define the source mesh for the front clip:

1 Go to the first frame of the clip.
2 In the Morph menu, click Front Source.
3 Add a mesh over the area to be morphed and modify its shape to match the image.
4 Make the front source mesh match the image at each frame of the front clip. You can either modify the mesh manually or use the Stabilizer to track the movement in the front clip.

Defining the Back Source Mesh

The back source mesh always contains the same number of control points and patches as the front source mesh. Copy the front source mesh to use as a starting point, then adapt the mesh shape to the image at each frame of the back clip.

If you are using the Stabilizer and you need more detail for the back source mesh, add subdivisions to the mesh. Be sure to add the same subdivisions to the front source mesh.

NOTE Enable the Fixed button in the Stabilizer menu.

To define the source mesh for the back clip:

1 Go to the first frame of the clip.
2 Click Front Source, then click Copy to copy the front source mesh.
3 In the Morph menu, click Back Source to select the back source mesh.
4 Click Paste to paste the copied mesh into the back source mesh.
5 Click Back to view the back clip.
6 Modify the back source mesh to match the image in the back clip at each frame. You can either modify the mesh manually or use the Stabilizer to track the movement in the back clip.

Using Source Interpolation

Use source interpolation to automatically define the destination meshes. Source interpolation automatically creates the transition from the front source mesh at the first frame to the back source mesh at the last frame of the clip.

NOTE If you want to add another effect, such as scaling, during the morph, do not use source interpolation. You must define the front and back destination meshes manually at several frames. See Setting Keyframes Manually (page 1428).

Dissolve and Interpolation Curves

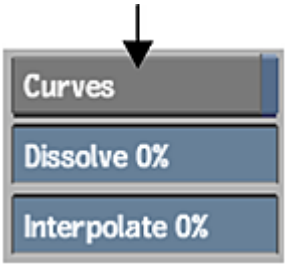
Source interpolation uses two predefined animation curves—dissolve and interpolation—to calculate the percentage change from the front source mesh to the back source mesh over the length of the clip. These values are set in the Dissolve and Interpolate fields:

- The Dissolve field defines the percentage change in the colour of the pixels at the current frame.
- The Interpolate field defines the percentage change in the position of the pixels at the current frame.

The default values in these fields are 0% at the first frame and 100% at the last frame. Using the default values results in a smooth dissolve from the pixels in the front image to those in the back image, as well as a smooth transition from the shape of the front image to that of the back image. You can change the rate of the morph by changing the interpolation and dissolve values.

To use source interpolation:

1. Click Morph to display the Morph menu.
2. Under Mapped To, select Source Interp to use source interpolation for the Front Destination mesh and Back Destination mesh.
3. Enable Curves to activate the interpolation and dissolve curves.



4. In the View box, select Result to view the result, and play the clip.

To change the rate of the morph:

1. In the Warper menu, click Animation to display the Channel Editor.
2. Select the dissolve or the interpolate channel.
3. Modify the curve.

Setting Keyframes Manually

If you are not satisfied with the source interpolation result, you can set the keyframes manually. However, you will have to set the keyframes throughout the clip—you cannot use source interpolation in conjunction with manual keyframes.

To set a keyframe manually:

1. Go to the frame corresponding to the keyframe you want to set.
2. Click Front Destination to display the front destination mesh.
3. Modify the mesh.
4. Copy the front destination mesh and paste it into the back destination mesh for that frame.

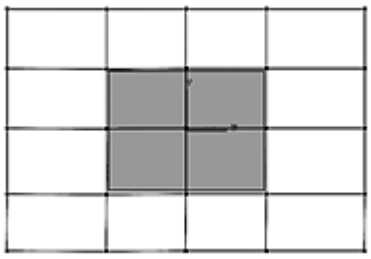
   Alternatively, if you want the back destination mesh to always match the front destination mesh, select the Front Destination under Mapped To adjacent to the Back Destination Meshes button.

Morphing Between Two Still Clips

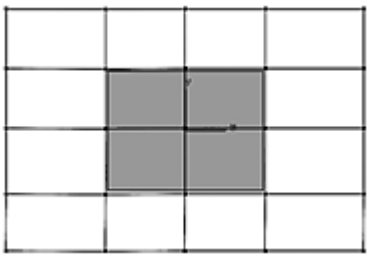
To morph between two still clips, you define the source meshes only for the first frame of the front and back clips (since the objects have the same shape and location in each frame of the clips). You can use source interpolation to create a smooth morph.

The following illustration shows the source and destination meshes needed to morph a square into a circle.

Front Source Mesh:

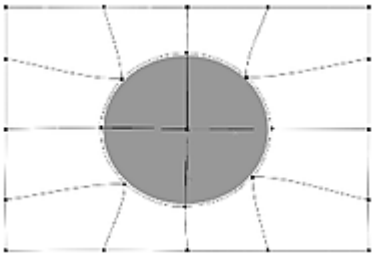


First Frame

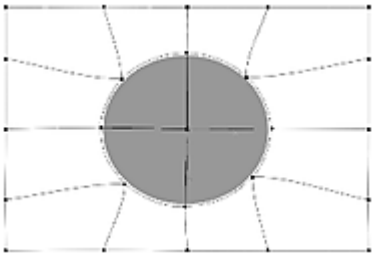


Last Frame

Back Source Mesh:

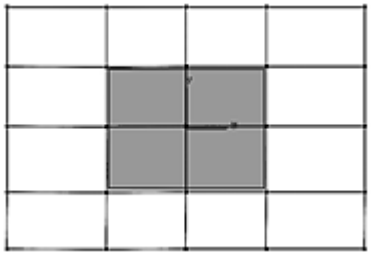


First Frame

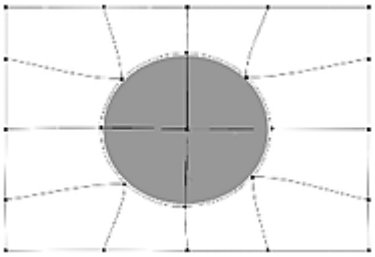


Last Frame

Front and Back Destination Meshes:



First Frame



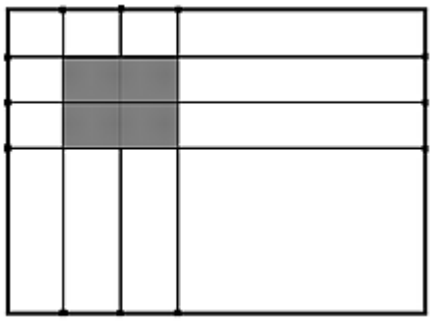
Last Frame

Morphing Between Two Live-Action Clips

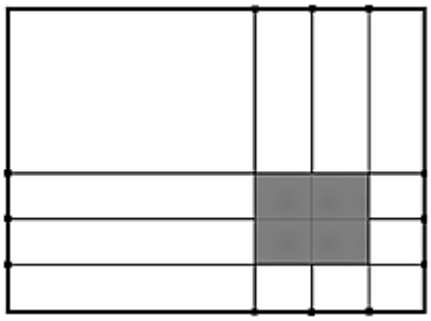
To morph between two live-action clips, you have to define the source and destination meshes for the front and back clips for the duration of the clips—since the objects change shape and location in each frame of the clips. You can use the Stabilizer to track the movement of the objects and source interpolation to create a smooth morph.

The following illustration shows the source and destination meshes needed to morph a square into a circle, both moving from the upper-left to the lower-right corner of the image window.

Front Source Mesh:

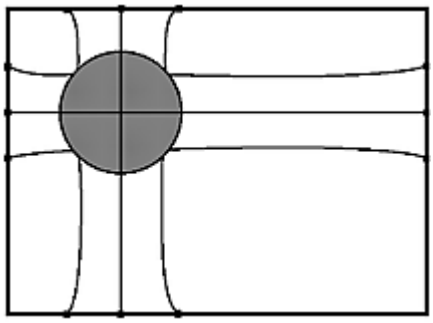


First Frame

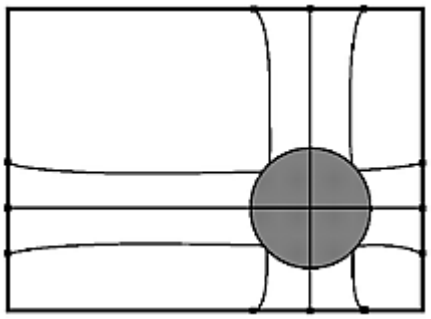


Last Frame

Back Source Mesh:

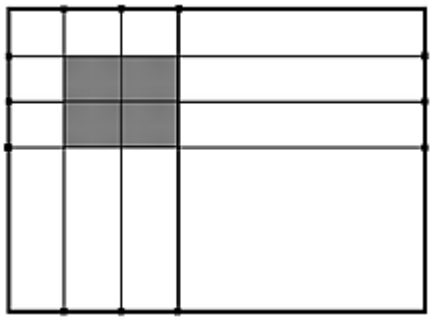


First Frame

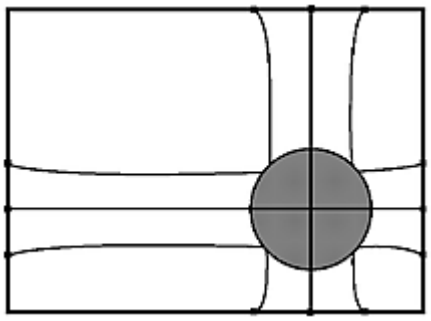


Last Frame

Front Destination Mesh:

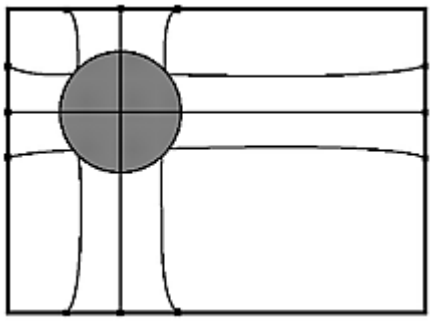


First Frame

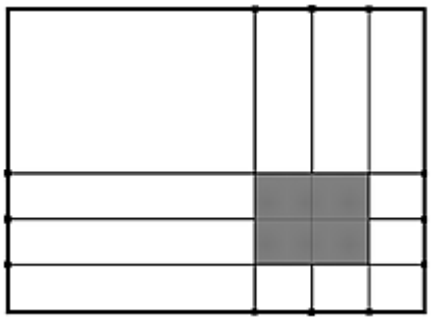


Last Frame

Back Destination Mesh:



First Frame



Last Frame

Setup Options

From the Setup menu, you can customize display colours and set processing options.

Rendering Settings

Use the Rendering settings to improve the final output quality of your image.

Texture Settings

Use the following controls to choose between hardware texture-mapping or polygons for processing.

Repeat Specifies how pixels are handled when a source mesh goes outside of the borders of an image. When this button is disabled, anything outside of the image is considered black. When the button is enabled, the image is repeated. TexRepeat is disabled by default.

Warper Settings

Use these settings to define general warper properties.

Slip Settings

Slips a clip by a specified number of frames. Click in the field for the clip you want to slip and enter the number of frames. The first frame of the clip will be repeated by the number specified to result in a delayed start.

Select:	**To slip:**
Front	The front clip.
Back	The back clip.
Matte	The front matte.
BMatte	The back matte.

Display Settings

Enable the Icons button to display splines, tangents, and vertices in the image. You can also customize the colours of splines, tangents, and vertices using the colour pot next to each button.

Effects and Tools Reference

26

This section provides information about the effects and tools that you can use on your media.

2D Histogram

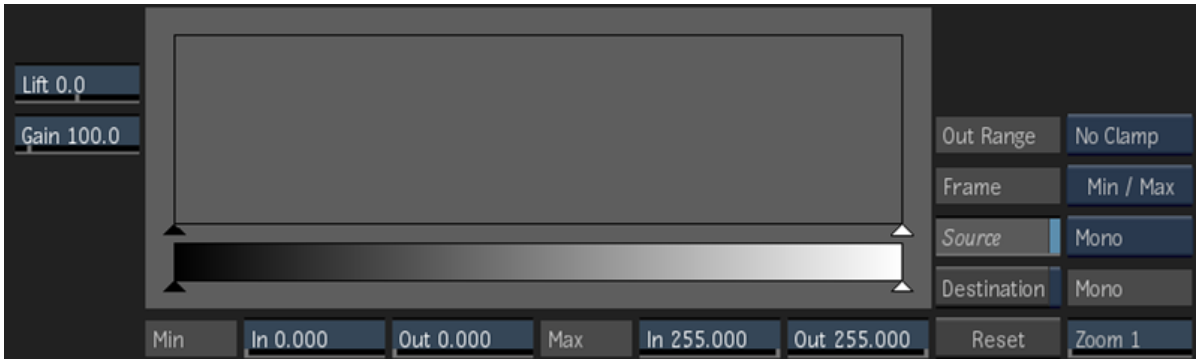
Use the 2D Histogram to display the luminance distribution of the matte.

To access the 2D Histogram menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

2D Histogram Menu Settings



Lift field Displays the value added to the resulting pixels to create the final matte. Editable.

Gain field Displays the value that the resulting pixel values are multiplied by to create the final matte. Editable.

Histogram Displays a bar graph that is used to adjust the luminance values of the image. In the Ranges menu, the curves for the three luma ranges are also displayed.

Minimum Input field Displays the lower limit of the luminance values. Pixels with lower values are mapped to black.

Minimum Output field Displays the lower limit of the luminance values for black pixels.

Maximum Input field Displays the upper limit of the luminance values. Pixels with higher values are mapped to white.

Maximum Output field Displays the upper limit of the luminance values for white pixels.

Out Range box Select a curve that is constant (Clamp) or linear (No Clamp) before the first point of the curve and after the last point of the curve.

When using 16-bit floating point images, you can select Clamp to clamp colour and luminance values, or No Clamp to allow pixel floating point values to be less the 0 or more than 1.

Frame Selection box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Destination button Enable to show a histogram of the colour values in the result or destination clip.

The destination colour values are obtained from the current frame. If you input both front and matte clips, the colour values in the front clip that are defined by the white part of the matte are displayed.

Reset button Resets to default histogram settings.

Zoom field Displays the vertical zoom value of the histogram.

You can also zoom horizontally by pressing `Ctrl+spacebar` and dragging left or right in the histogram. To pan horizontally, click `spacebar` and drag left or right in the histogram.

2D Transform

Use 2D Transform to apply basic axis transformations and camera shake effects to clips.

To access the 2D Transform menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front and a matte clip, and outputs a result and an outmatte.

2D Transform Menu Settings

Timeline FX Quick Menu Settings

To see the full 2D Transform menu, click the Editor button.

2D Transform Tab selector Select the tab settings you want to modify.

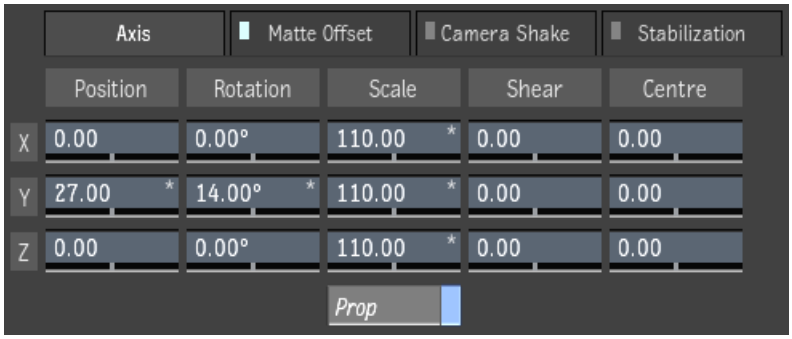
General Settings

Transform Type box Select Perspective to modify all available settings, or Pan & Scan to modify only Position and Scale. In Pan & Scan mode, some Axis and Camera Shake settings are unavailable.

Rendering Mode box Select whether to render in Automatic, Progressive or Interlaced mode.

Show Icons button Enable to display the vertex editing tool in the image window.

Axis Settings



Position X field Displays the X-axis offset applied to the clip. Editable.

Position Y field Displays the Y-axis offset applied to the clip. Editable.

Position Z field Displays the depth offset of the clip. Available when Transform Type is set to Perspective. Editable.

Rotation X field Displays the degree of rotation on the X axis. Available when Transform Type is set to Perspective. Editable.

Rotation Y field Displays the degree of rotation on the Y axis. Available when Transform Type is set to Perspective. Editable.

Rotation Z field Displays the degree of rotation along the Z axis. Available when Transform Type is set to Perspective. Editable.

Scale X field Displays the X-axis scaling offsets applied to the clip. Editable.

Scale Y field Displays the Y-axis scaling offsets applied to the clip. Editable.

Scale Z field Displays the Z-axis scaling offset. Available when Transform Type is set to Perspective. Editable.

Proportional button Enable to affect the Scale fields proportionally.

Shear X field Displays the offset value for the horizontal edges of the clip. Available when Transform Type is set to Perspective. Editable.

Shear Y field Displays the offset value for the vertical edges of the clip. Available when Transform Type is set to Perspective. Editable.

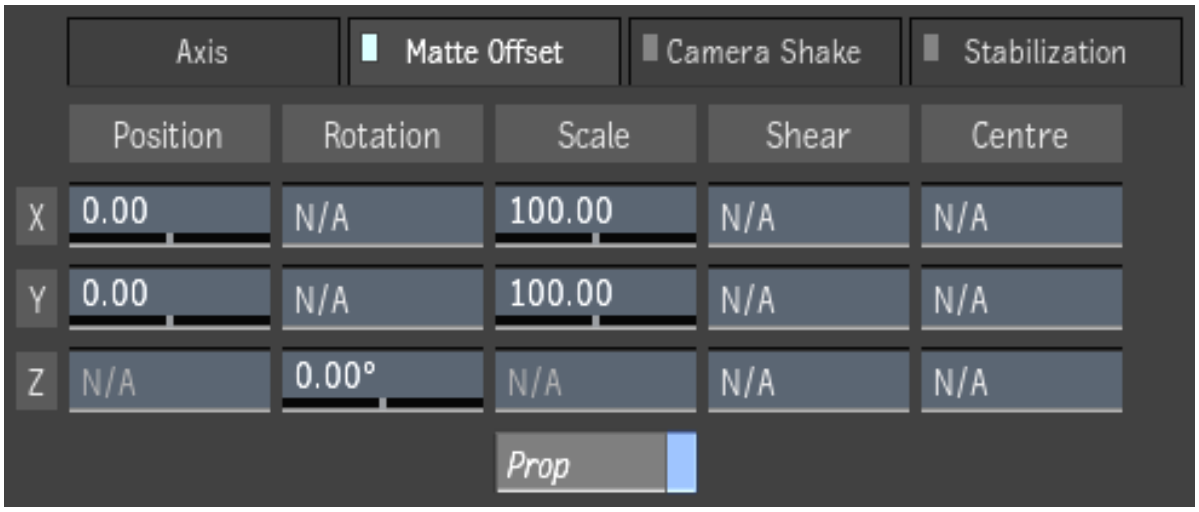
Shear Z field Displays the depth value of the horizontal edges of the clip. Available when Transform Type is set to Perspective. Editable.

Centre X field Displays the centre point value of the clip along the horizontal axis. Available when Transform Type is set to Perspective. Editable.

Centre Y field Displays the centre point value of the clip along the vertical axis. Available when Transform Type is set to Perspective. Editable.

Centre Z field Displays the centre point value of the clip along the Z axis. Available when Transform Type is set to Perspective. Editable.

Matte Offset Settings



Position X field Displays the X-axis offset applied to the matte. Editable.

Position Y field Displays the Y-axis offset applied to the matte. Editable.

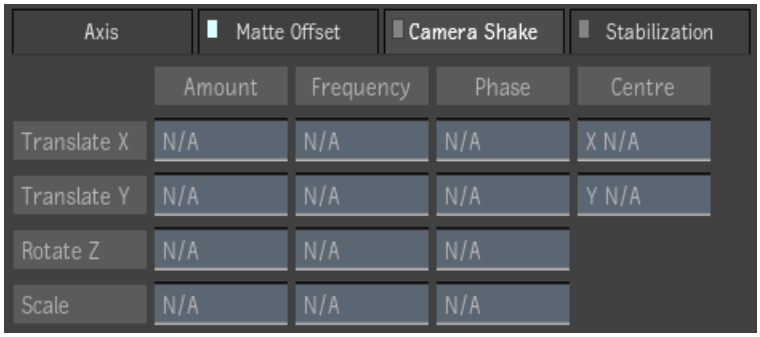
Rotation Z field Displays the degree of rotation along the Z axis. Editable.

Scale X field Displays the X-axis scaling offset applied to the matte. Editable.

Scale Y field Displays the Y-axis scaling offset applied to the matte. Editable.

Proportional button Enable to affect the Scale fields proportionally.

Camera Shake Settings



Camera Shake button Enable to simulate camera shake in the image.

Translate X Amount field Displays the amount of horizontal movement applied to the clip. Editable.

Translate Y Amount field Displays the amount of vertical movement applied to the clip. Editable.

Rotation Z Amount field Displays the degree of rotation on the Z axis. Available when Transform Type is set to Perspective.Editable.

Scale Amount field Displays the amount of scaling applied to the clip. Editable.

Translate X Frequency field Displays the frequency of horizontal movement applied to the clip. Editable.

Translate Y Frequency field Displays the frequency of vertical movement applied to the clip. Editable.

Scale Frequency field Displays the frequency of scaling applied to the clip. Editable.

Rotation Z Frequency field Displays the frequency of rotation on the Z axis. Available when Transform Type is set to Perspective.Editable.

Translate X Phase field Displays the phase of horizontal movement applied to the clip. Editable.

Translate Y Phase field Displays the phase of vertical movement applied to the clip. Editable.

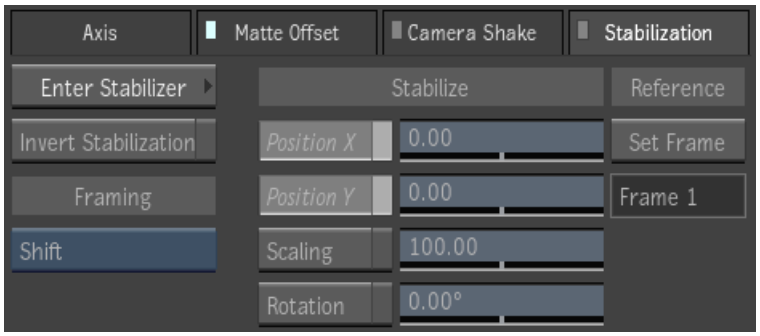
Rotation Z Phase field Displays the phase of rotation on the Z axis. Editable.

Scale Phase field Displays the phase of scaling applied to the clip. Editable.

X Centre field Displays the centre point value of the clip along the horizontal axis. Available when Transform Type is set to Perspective. Editable.

Y Centre field Displays the centre point value of the clip along the vertical axis. Available when Transform Type is set to Perspective.Editable.

Stabilization Settings



Enter Stabilizer button Enable to enter the Stabilizer editor.

Invert Stabilization button Enable to apply an inverse of the stabilization data to recreate the original camera shake.

Framing Mode box Select a method to fill or remove the area where the image has been shifted after stabilization.

Position X button Enable to include the X-axis position in the stabilization.

Position X field Displays the amount of stabilization data found in the animation channel for the X position.

Position Y button Enable to include the Y-axis position in the stabilization.

Position Y field Displays the amount of stabilization data found in the animation channel for the Y position.

Scaling button Enable to include scaling in the stabilization.

Scaling field Displays the amount of stabilization data found in the animation channel for scaling.

Rotation button Enable to include the rotation in the stabilization.

Rotation field Displays the amount of stabilization data found in the animation channel for rotation.

Set Frame button Enable to set the current frame as a reference frame that has no transformation data.

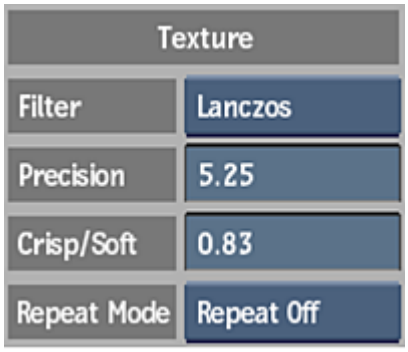
Reference Frame field Displays the frame number of the reference frame that has no transformation data. Non-editable.

Perspective and Random Seed Settings

Perspective field Displays the field of view. Available when Transform Type is set to Perspective. Editable.

Random Seed field Displays the random seed value of the camera shake. Editable.

Texture Settings



Filter box Displays available filters to control the quality of resize operations. The list of filters changes when you switch between Perspective and Pan & Scan Transform Types.

Precision field Displays the frequency cut-off point used during resize. Editable only when doing a pan and scan with Shannon or Lanczos filters.

Crisp/Soft field Displays the amount of blurring used during resize. Editable only when doing a pan and scan with Mitchell, Shannon or Lanczos filters.

Repeat Mode box Select an option to fill the empty portions of the frame.

Select:	To:
Repeat Last	Repeats last pixel color.

Select:	To:
Tile Repeat	Wrap the image around to fill the area.
Mirror Repeat	Repeats the source pixels both vertically and horizontally.
Repeat Off	Use a colour to fill the area. Use the adjacent colour pot to pick the colour. You will lose texture and will fill the rest of the image with a black border.

Anti-Aliasing Settings

Anti-Aliasing button Enable to activate sampling and softness.

Sampling box Select the number of samples to use in the anti-aliasing process.

Softness field Displays the level of softness of the samples. Editable.

Motion Blur Settings

Motion Blur button Enable to apply a motion blur to a selected clip.

Samples field Displays the number of frames to sample when creating the blur. The samples include the current frame, and an equal distribution of past and future frames. Editable.

Phase field Displays the level of softness of the samples. Editable.

Shutter field Displays the number of frames for which the shutter stays open. For example, when the shutter value is set to 3, every third frame is a sample. Editable.

Lock Transform button Enable to lock the transformation of the motion blur effect.

Result Output Settings

Result Output box Select an output option. Select Premultiplied to output alpha values in the colour channels of the result.

Canvas Resolution Settings

Resize Fit Mode box Select whether to resize the clip with width and height values.

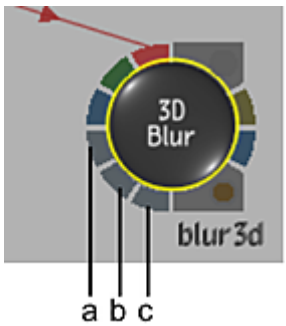
Same As Input	The output resolution remains the same as the input resolution.
User Defined	You define the output resolution.
Auto Scale	The output resolution changes dynamically, which insures that any transformation applied will not crop the image across the whole length of the clip.

Width field Displays the width of the image. Editable.

Height field Displays the height of the image. Editable.

3D Blur

Use the 3D Blur to create a plausible blur effect using Z-depth map information.



(a) Z-depth input tab (b) Forward Flow input tab (c) Kernel input tab

to access the 3D Blur menu, use:

- Batch, then select a node from the Node bin.

- Timeline, then use Batch FX (see Creating Batch FX (page 496)).

This node accepts a front, back, and matte as input. Additional inputs are available for the Z-depth map, forward vector, and kernal media. It outputs a result and an outmatte.

The 3D Blur can be used in the following ways:

- As a defocus node that can be modulated by a Z-depth map.
- As an artistic blur node, by modifying the pattern of the highlights. This can be done by editing the pattern profile curve or attaching an external clip to the appropriate node input.
- As a motion blur node, by including forward motion data.
- As a node using a combination of these scenarios.

3D Blur Menu Settings

Creating a Lens Blur

Lens blur simulates the blur created by a camera lens, such as a rack defocus.

The primary input of the 3D blur node is the front clip and the matte clip. A blur effect can be created with a front clip only. However a matte clip allows you to determine how the matte is used. The Blending settings also determine how the input is composited in the Result view.

Divide Front button Enable to unpremultiply the front clip using the matte clip. This button is active when a matte is in use.

Output box Select an option to determine how to composite the output in the Result view. The output can be a blurred premultiplied or unpremultiplied result using the front clip and the matte clips, or a composite of the blurred front clip over a background clip. Premultiplied output is equivalent to compositing the resulting front on a black background using the resulting matte.This button is active when a matte is in use.

Use the 3D Blur type settings to refine and constrain the amount of blur to apply to the image and to constrain the amount of blur. Higher blur values increase rendering time. You can increase rendering performance by decreasing the blur quality. When you are ready to create a result clip and an output matte, you can select the highest quality blur.

Blur Type box Select the type of 3D Blur to apply. Some of the settings differ based on the blur type you choose.

Blur Width field Displays the horizontal blur amount in pixels. Editable.

Blur Height field Displays the vertical blur amount in pixels. Editable.

Blur Proportional button Enable to constrain blur amount proportions.

Sides field Displays the number of sides in a Gaussian blur shape.Available when Gaussian is chosen as the blur type. Editable.

Rotation field Displays the angle of rotation of a Gaussian blur shape. Available when Gaussian is chosen as the blur type.Editable.

Kernel Precision box Select the quality and speed of the defocus blur. For example, a 1x1 Kernel results in a fast, but coarse defocus blur.Available when Defocus is chosen as the blur type.

Z Slicing field Displays the amount of slices to blend to determine the Z-Depth interpolation of a defocus blur. Available when Defocus is chosen as the blur type.Editable.

You can offset the light and colour values in the source image. Highlights are applied to the image before the blur is applied.

Adjust button Enable to activate highlight controls.

Gain field Displays the light values of the image. Editable.

Offset field Displays each colour value adjusted by this increment. Editable.

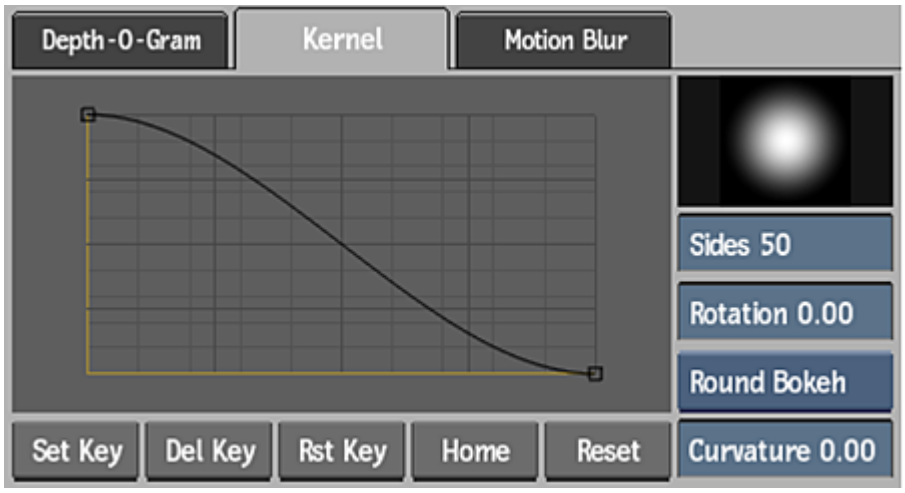
Threshold field Displays the minimum colour value included in highlights. Editable.

Ramp Range field Displays the difference between the Threshold value and the value at which the highlights take full effect. Editable.

Max Effect At field Displays the value at which the highlights take full effect. Non-editable.

Changing the Blur Pattern

Use the Kernal tab to edit the blur kernel pattern.



The kernel is the basic blur shape. The shape of the kernel is determined by its number of sides, its rotation, and the shape of its S-curve. This curve represents the shape of the pattern, from its centre to the outside. The default S-curve defines the softness of the blur. You can change the curve by manipulating the two points that define the curve, or you can add points to the curve. Use the Tools box to add and delete points on the curve.

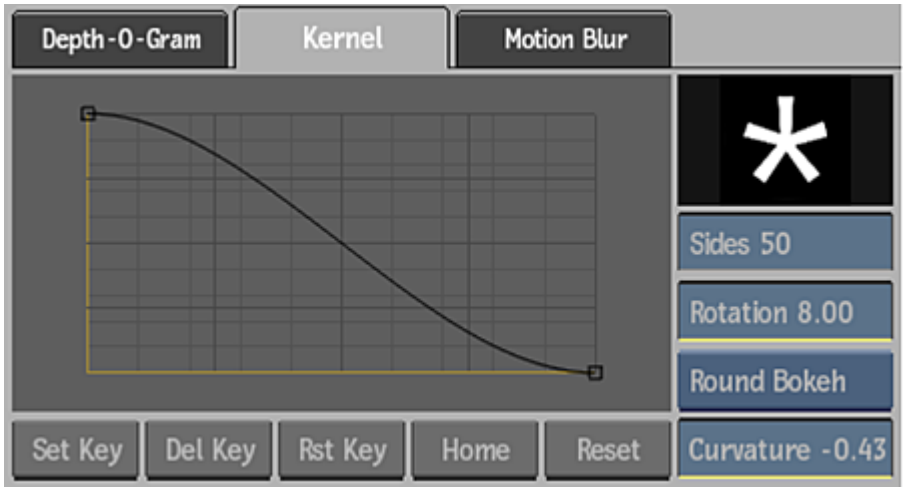
Sides field Displays the number of sides in the kernel shape. Editable, if kernel information is not attached to the node.

Rotation field Displays the angle of rotation of the kernel shape. Editable, if kernel information is not attached to the node.

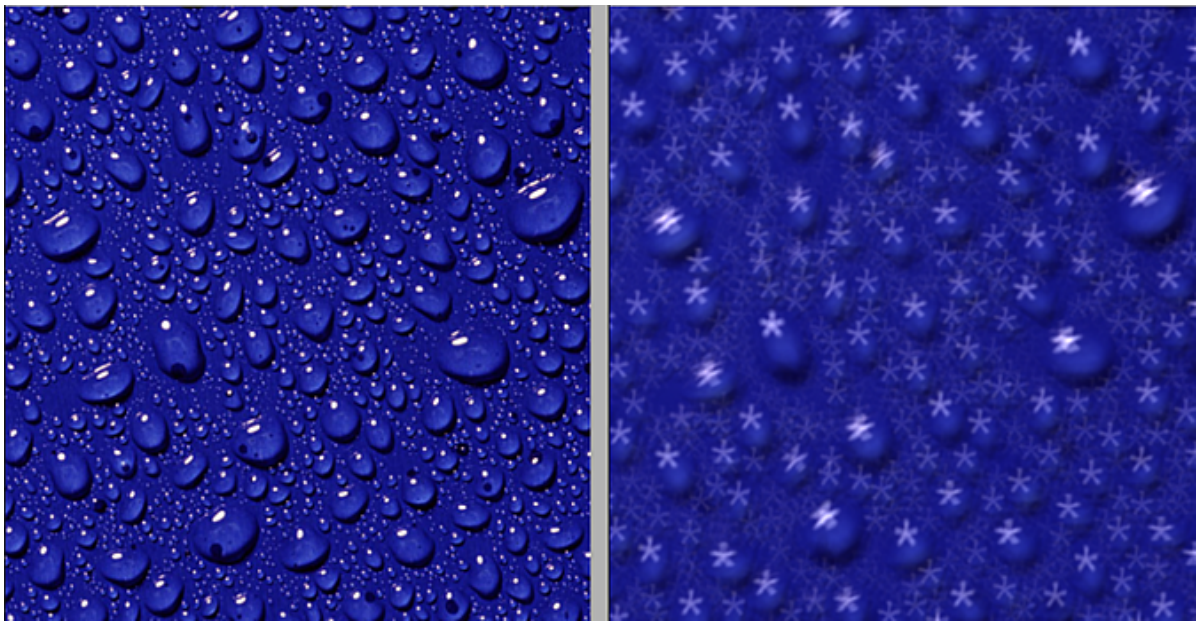
Bokeh Type box Select whether to use a round or angle bokeh curve to define the kernel shape.

Curvature field Displays the amount of curvature applied to a round bokeh kernel shape. Editable.

You can define the blur shape using external kernel information by attaching an input to the Kernel tab. The input can be a different resolution than the other node inputs.

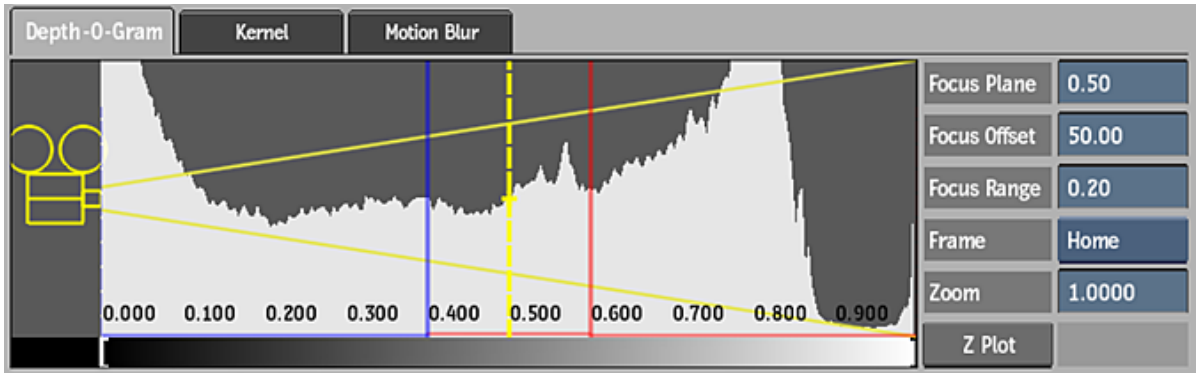


External blur information is not editable. Like the default blur shape selections, the shape determines the influence of the kernel's pixels on each pixel in the front input.

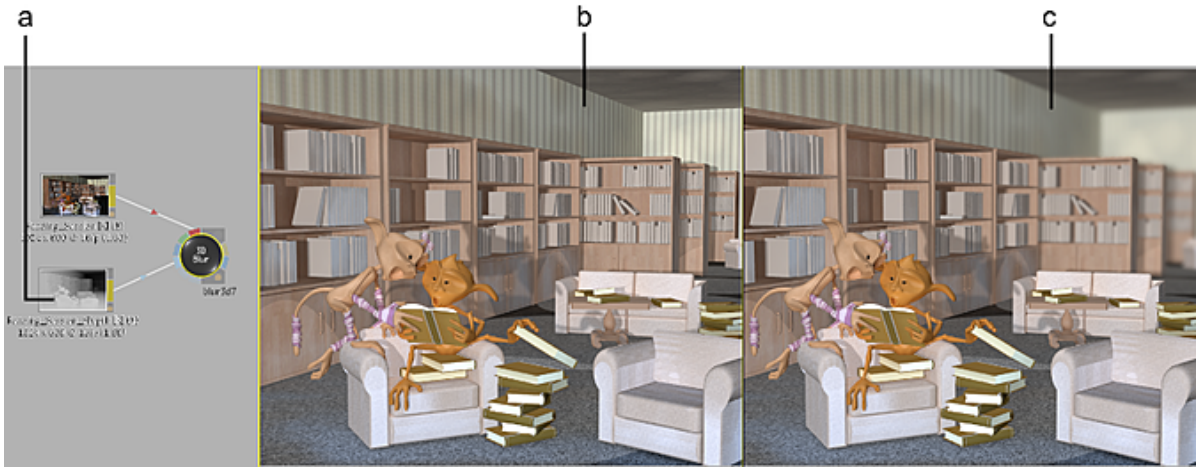


Simulating a Depth of Field Effect

Depth blur effects control the amount of blur based on a depth of field matte. Depth blur settings are displayed in the Depth-O-Gram tab.



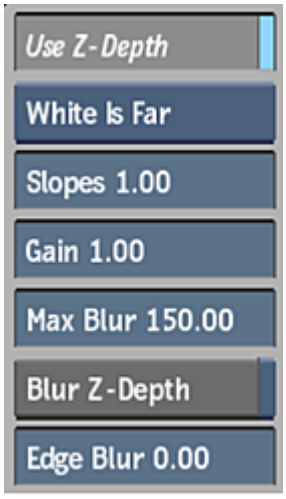
A depth of field map (Z-depth map) can be connected to the node. Black portions of the map are in focus. White portions display the highest level of blur. Note that a Z-depth map imported from another application may use the opposite convention and may need to be inverted.



(a) Z-depth map (b) Front View (c) Blurred portion in Result View

Image courtesy of Jean-Marc Belloncik

Depth Map settings can be used to invert colour values in the Z-depth map for reuse with the 3D Blur node.



Z-Depth button Enable to activate depth-of-field controls and apply the effect to the image.

White Value box Select whether white pixels represent the furthest point or nearest point on the Z-axis.

Slopes field Displays the gamma value. Applies a gamma curve to the Z-depth map before it is used. Editable.

Gain field Displays the white balance value. Applies a gain curve to the Z-depth map before it is used. Editable.

Max Blur Field Displays the maximum total horizontal and vertical blur amount to clamp the value and avoid unnecessary rendering. The cumulative effect of the high blur amount (Basic tab), and gamma and gain correction applied to the Depth map (Depth tab) may result in very large values, which greatly increases the rendering time. Editable.

Blur Z-Depth button Enable to allow pre-blurring of the Z-Buffer. Disable to increase performance time.

Edge Blur field Displays the minimum blur value of "in focus" pixels. In some instances, an out-of-focus object that appears closer to the camera (that is, in front of the focus plane) can have hard edges. In this case, increase the value in the field to add a blur to these edges. Editable.

Use the Depth-O-Gram settings to refine focus values.

Focus Plane field Displays the distance of your focus point, which is the point at which there is no blur on the image. Editable.

Focus Offset field Displays the distance between the focus plane and the near offset represented as a percentage of the total offset range. Select 50% to make the near and far offsets equidistant from the focus point. Editable.

Focus Range field Displays the distance the near and far offset. Editble.

Frame option box Select how you want to frame the histogram.

Zoom field Displays a vertical zoom value for the histogram to display. You can zoom horizontally by pressing Ctrl+spacebar and dragging left or right in the histogram. To pan horizontally, click spacebar and drag left or right in the histogram.Editable.

Z Plot button Click to activate the pick cursor. Use to select a pixel in the image to display its depth.

Z Plot Colour pot Displays the colour that indicates the plane on which the plotted value is located.

Use On-Screen Widgets section in the NodeSetup menu to define display settings.

Near/Far button Enable to display the focus offset plane in the image window.

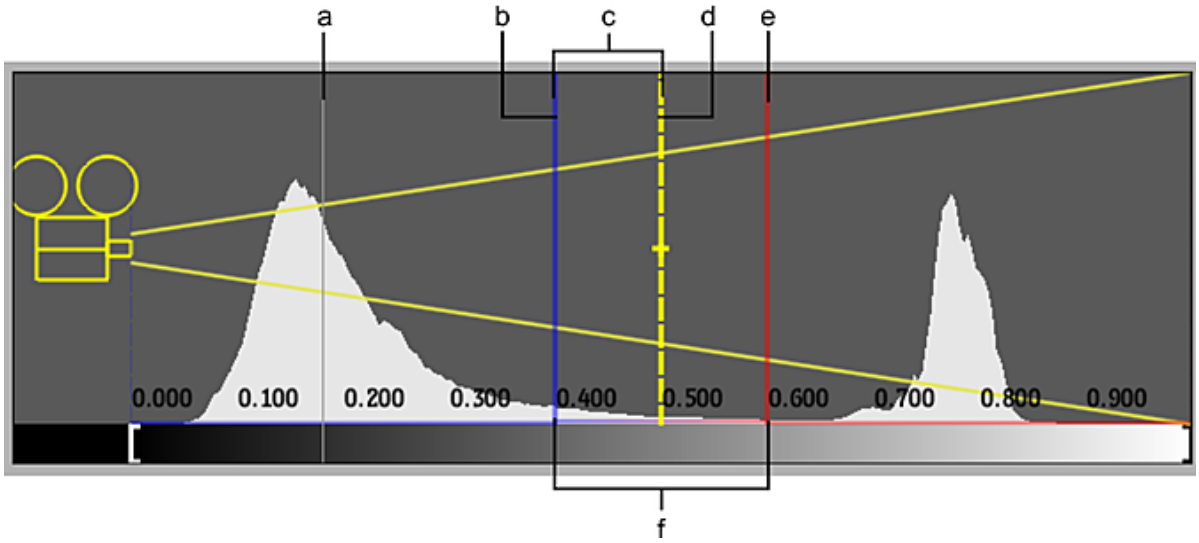
Near and Far colour pots Select the colours that indicate the nearest and furthest points of focus.

Focus Plane button Enable to display the plane on which the focus point lies in the image window.

Focus Plane colour pot Select the colour that indicates the plane on which the focus point is located.

Modifying Depth of Field Gesturally

Depth blur effects can be modified gesturally by dragging the focus, near offset, and far offset planes in the graphic representation of the depth of field. As you drag these elements, the planes are also displayed in the Result view as a preview of the areas that will be in focus. Use the depth control fields to change the gamma and gain, and to change the focus range while keeping the focus plane constant. These parameters are updated in the depth of field display automatically.

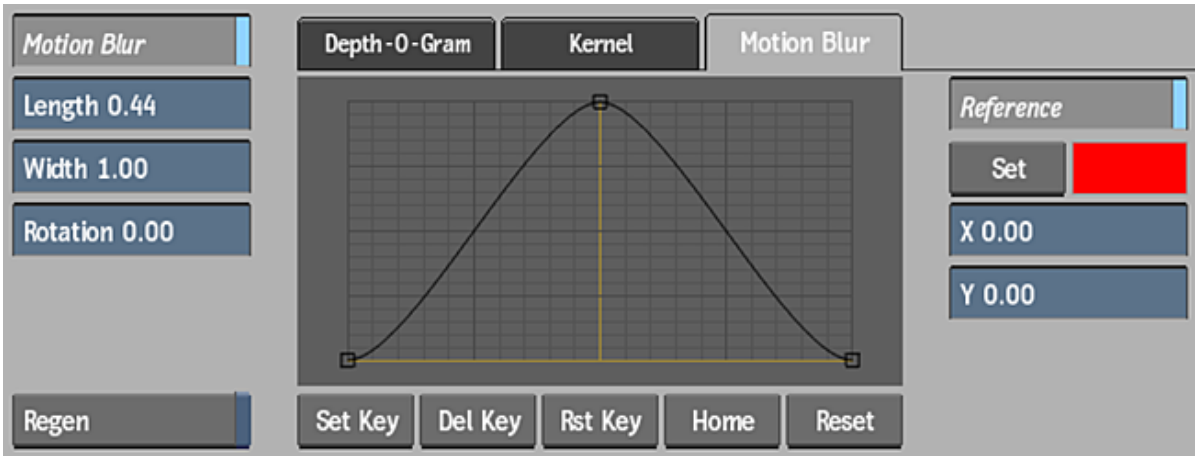


(a) Plot Value Plane (b) Near Focus Offset Plane (c) Focus Offset (d) Focus Plane (e) Far Focus Offset Plane (f) Focus Range

Simulating a Motion Blur Effect

The motion blur simulates the blur created by a fast-moving object or camera. A blur effect can be created with front clip input and a clip with forward motion data input into the Forward Flow tab.

Use the Motion Blur Profile to define the opacity of the blur. The opacity increases at the top of the curve. By default, the blur is more transparent further away from the pixel. The midpoint represents the opacity of the blur at the position of the pixel.



Motion Blur button Enable to activate the motion blur controls and apply the effect to the image.

Length field Displays the amplitude of the motion blur. Editable.

Width field Displays the width of the blur. Editable.

Rotation field Displays the angle of rotation to apply to motion vectors. Motion is rotated counterclockwise. Editable.

Reference button Enable to activate the reference controls and apply the effect to motion blur.

Set button Enable to apply the motion blur to objects that do not have the same relative motion as the selected pixel. Enabled if the X or Y field is edited.

Set Reference colour pot Displays the colour of the crosshair that marks the reference point in the image window.

X field Displays the horizontal position of the pixel to use as a reference point at the selected frame. A reference point can be selected directly in the frame using the cursor. Editable.

X field Displays the vertical position of the pixel to use as a reference point at the selected frame. A reference point can be selected directly in the frame using the cursor. Editable.

Auto Matte

Use the Auto Matte to generate a high-contrast matte from a clip.

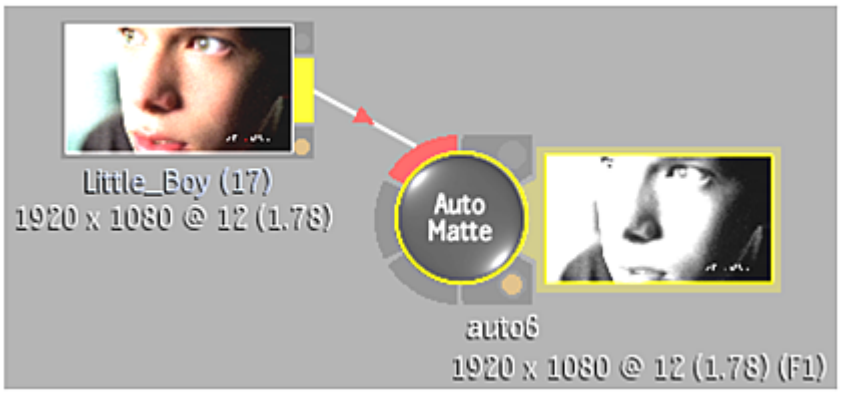


Image courtesy of Optimus

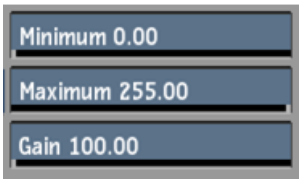
To access the Auto Matte menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip as input, and outputs a result.

Auto Matte Menu Settings

General Settings



Minimum field Displays the minimum luma value considered to be part of the opaque area of the matte. Editable.

Maximum field Displays the maximum luma value considered to be part of the transparent area of the matte. Editable.

Gain field Displays a value, in percentage, that is multiplied to the pixel values in the resulting matte to reduce or increase the grey levels.

Auto Stabilize

Use the Auto Stabilize to analyse the movement in a clip and remove camera instability. For example, you can remove camera jitter or lock an object's position over a sequence of frames to make it appear motionless.

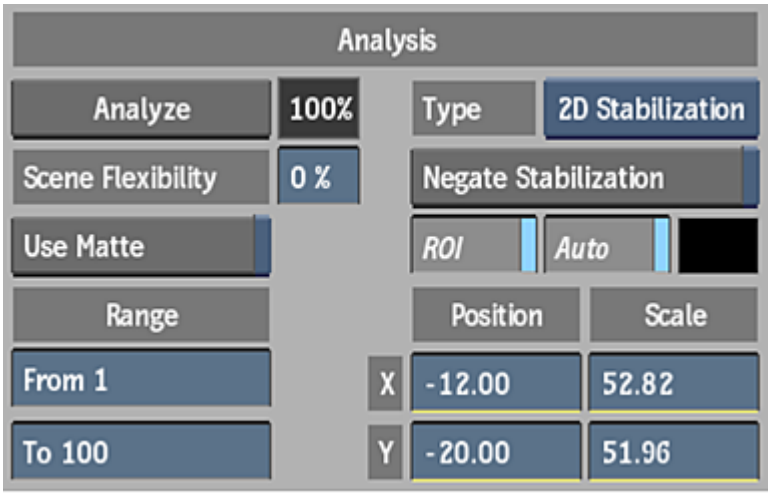
To access the Auto Stabilize menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts front and matte clips, and outputs a result and an outmatte.

Auto Stabilize Menu Settings

Analysis Settings



Analyze button Click to run the stabilization analysis. During analysis, the viewport defaults to the Front view, and the frame advances as the stabilization is calculated. Progress can be monitored by the keyframes that have been created during analysis and the percentage of completion displayed next to the Analyze button. Track points are displayed on the clip to indicate how the stabilization was tracked.

Progress field Displays the percentage of the analysis that is complete. Non-editable.

Scene Flexibility field Displays the level of focus on rigid objects that move in the clip. Enter 0% to focus on a single rigid object as it moves or 100% to analyse all visible motion paths. If the analysis has no scene flexibility, the stabilization will focus on a single rigid object as it moves in the clip. Use the following values as a general guideline:

Select:	To:
0%-20%	Stabilize a pan, tilt, zoom, or the motion of a single rigid object that is visible during the entire analysis.
30%-70%	Stabilize objects that change their shape or depth, and exclude objects that are smaller or moving rapidly.
80%-100%	Stabilize all visible movements.

Use Matte button Enable to exclude black areas on the matte from analysis on the front clip, and constrain the Analysis region.

Range From field Displays the value of the first frame to include in the analysis. Editable.

Range To field Displays the value of the last frame to include in the analysis. Editable.

Stabilization Method box Select whether to perform a two-dimensional or perspective analysis.

Negate Stabilization button Enable to apply the inverted parameter values.

Region of Interest button Enable to display a rectangle that can be resized to indicate the region you want to analyse on the front clip. Select Front from the View box.

Auto button Enable to automatically track motion within the region of interest. At each frame, the position of the region of interest is updated based on the motion within the area. If this button is disabled, and the region of interest was not previously tracked, the region of interest remains static.

Stabilization Settings

Position X button Enable to include the X-axis position in the stabilization.

Position X Option box Select whether to lock the X-axis position at the reference frame to all frames in the analysis (Fixed), or to apply smoothness to the stabilization curve on the X-axis.

Position Y button Enable to include the Y-axis position in the stabilization.

Position Y Option box Select whether to lock the Y-axis position at the reference frame to all frames in the analysis (Fixed), or to apply smoothness to the stabilization curve on the Y-axis.

Rotation button Enable to include the rotation in the stabilization.

Rotation Option box Select whether to lock the rotation at the reference frame to all frames in the analysis (Fixed), or to apply smoothness to the stabilization curve on the rotation.

Scaling button Enable to include scaling in the stabilization.

Scaling Lock Option box Select whether to lock the scaling at the reference frame to all frames in the analysis (Fixed), or to apply smoothness to the stabilization curve on the scaling.

NOTE You can change the centre of rotation and scaling for the image. By default, the centre of rotation and scaling is the centre of the image, represented by a yellow crosshair. Switch to Front view, then hold down C and click the new centre point.

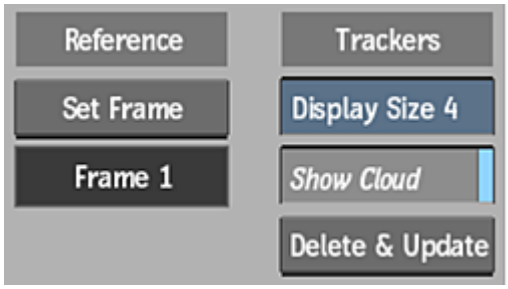
Perspective button Enable to include perspective in the stabilization.

Perspective Option box Select whether to lock the perspective at the reference frame to all frames in the analysis (Fixed), or to apply smoothness to the stabilization curve on the perspective.

Smoothness field Displays the level of smoothness of the stabilization curves generated by the analysis and determines the mix between the original motion and the completely stabilized transformation. Editable.

Mix field Displays the percentage of smoothness used in the final stabilization output and determines how smooth the stabilization curves are. Editable.

Reference and Tracker Settings



Set Reference Frame button Click to set the current frame as the reference frame for the stabilization.

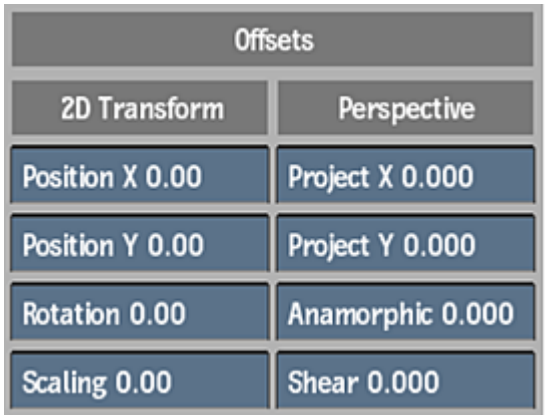
Reference Frame field Displays the current reference frame. Non-editable.

Display Size field Displays the pixel width and height of track points. Editable.

Show Cloud button Enable to display the track point cloud.

Delete and Update button Click to delete selected track points and update stabilization curves. Selected points are displayed in red in the image.

Offset Settings



Position X field Displays the X-axis offset applied to the clip. Editable.

Position Y field Displays the Y-axis offset applied to the clip. Editable.

Rotation field Displays the rotation offset applied to the clip. Editable.

Scaling field Displays the scaling offsets applied to the clip. Editable.

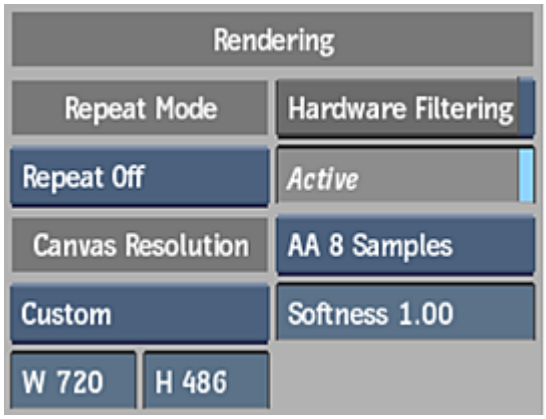
Project X field Displays the X-axis projection offsets applied to the clip. Editable.

Project Y field Displays the Y-axis projection offsets applied to the clip. Editable.

Anamorphic field Displays the anamorphic offsets applied to the clip. Editable.

Shear field Displays the shearing offsets applied to the clip. Editable.

Rendering Settings



Repeat Mode box Select an option to fill the empty portions of the frame.

Select:	To:
Repeat Last	Repeats last pixel color.

Select:	To:
Tile Repeat	Wrap the image around to fill the area.
Mirror Repeat	Repeats the source pixels both vertically and horizontally.
Repeat Off	Use a colour to fill the area. Use the adjacent colour pot to pick the colour. You will lose texture and will fill the rest of the image with a black border.

Resize Fit Mode box Select whether to resize the clip with width and height values.

Same As Input	The output resolution remains the same as the input resolution.
User Defined	You define the output resolution.
Auto Scale	The output resolution changes dynamically, which insures that any transformation applied will not crop the image across the whole length of the clip.

Select:	To:
Same as Input	The output resolution remains the same as the input resolution.
Custom	You define the output resolution.

Width field Displays the width of the image. Editable.

Height field Displays the height of the image. Editable.

Hardware Filtering button Enable to filter subpixel information.

Anti-Aliasing button Enable to activate sampling and softness.

Sampling box Select the number of samples to use in the anti-aliasing process.

Softness field Displays the level of softness of the samples. Editable.

Average

Use Average to simulate the motion blur of a moving object in a clip. An image averaging function is applied to the frames to create the motion blur.

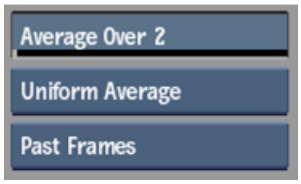
To access the Average menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip, and outputs a result.

Average Menu Settings

The Average Node processes frames based on data from preceding and subsequent sample frames. A frame set to No Media with missing media may include a sample.



General Settings

Average Over field Displays the number of frames used to calculate the average and create the transparency for the motion blur effect. Editable.

Weighted/Uniform box Select whether to uniformly average surrounding frames or give frames closest to the current frame a greater weight when calculating the transparency for the motion blur effect.

Frames box Select whether the blur is calculated from previous frames only or future frames as well.

Blend and Comp

Use Blend & Comp to apply basic blending and compositing between two inputs, and their mattes, over a background.

To access the Blend & Comp menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts up to two front and matte clips, as well as a back clip, and outputs a result and outmatte.

Blend and Comp Menu Settings

General Settings

Blend Type box Select the set of blend mode operations available in the Blend Mode box.

Blend Mode box Select a blend operation. Options determined by Blend Type box.

Swap Inputs button Enable to switch the rendering order of the inputs.

Input Settings

NOTE Input 1 and 2 settings are identical. Activate and use based on inputs selected for Blend & Comp.

Active button Enable to activate the input.

Clamping box Select a clamping option.

Matte box Select how the matte is used when blending.

Pre-multiplication box Select whether the colour values in the input are pre-multiplied or divided from the alpha, or left as-is when blending.

Transparency field Displays the transparency value. Editable.

Gain Trackball Adjusts the gain of the input.

Saturation field Displays level of colour purity in the image. Editable.

Gamma field Displays the gamma level. Editable.

Offset field Displays a value that modifies all of the colour parameters. Editable.

Red Gain field Set the percentage of colour values in the red channel. Editable.

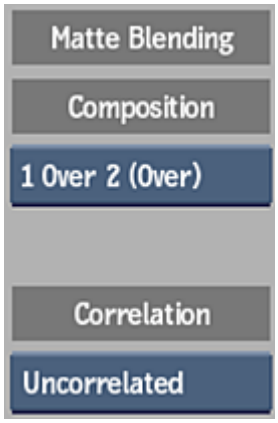
Green Gain field Set the percentage of colour values in the green channel. Editable.

Blue Gain field Set the percentage of colour values in the blue channel. Editable.

Luma Gain field Set the percentage of luma gain value to display. Editable.

Proportional button Enable to adjust the gain of the colour values proportionally.

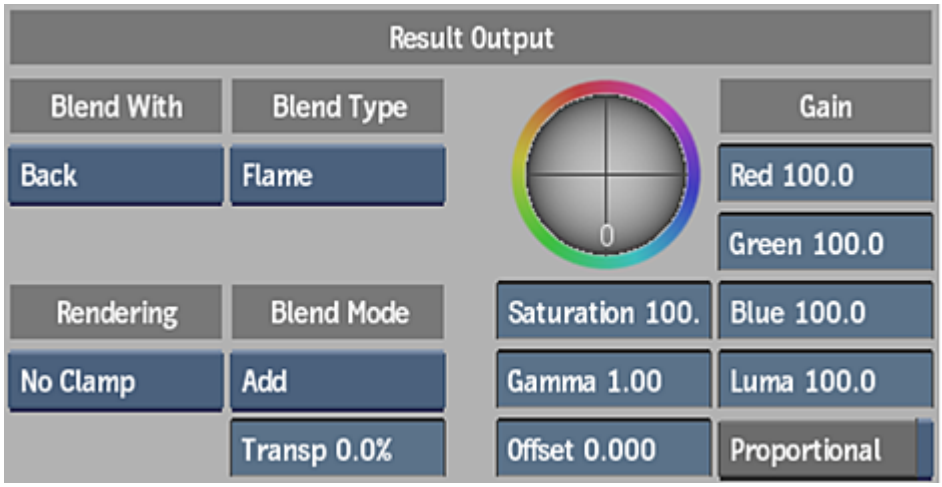
Matte Blending Settings



Composition box Select how the input mattes are combined.

Correlation box Select whether the mattes are related to each other.For example, select Correlated if the mattes are from different, but continuous parts on the same object.

Result Output Settings



Blend With box Select whether you want to blend the inputs with the second input clip, the background clip or a colour.

Blend Type box Select the set of blend mode operations available in the Blend Mode box.

Clamping box Select a clamping option.

Blend Mode box Select a blend operation. Options determined by Blend Type box.

Transparency field Displays the transparency value. Editable.

Gain Trackball Adjusts the gain of the input.

Saturation field Displays level of colour purity in the image. Editable.

Gamma field Displays the gamma level. Editable.

Offset field Displays a value that modifies all of the colour parameters. Editable.

Red Gain field Set the percentage of colour values in the red channel. Editable.

Green Gain field Set the percentage of colour values in the green channel. Editable.

Blue Gain field Set the percentage of colour values in the blue channel. Editable.

Luma Gain field Set the percentage of luma gain value to display. Editable.

Proportional button Enable to adjust the gain of the colour values proportionally.

Blur

Use Blur to apply and customize a blur filter to a clip.

To access the Blur menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.
- The Action Media list (see Adding Effects to Media (page 607)).

This node accepts a front and a matte clip, and outputs a result and outmatte. The outmatte clip can have a different level of blur than the result clip.

Blur Menu Settings

Quick Menu Settings

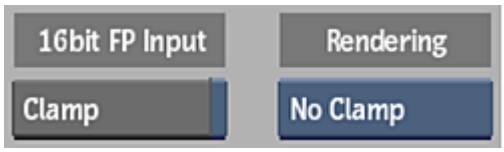
The following selector is unique to the quick menu when Defocus is selected in the Blur Type box. To see the full Blur menu, click the Editor button.

Blooming and Bokeh Menu selector Select whether to display Blooming settings or Bokeh blur settings.

General Settings

Regen button Enable to get dynamic updating of the image as you make changes.

Setup Settings



Clamp Input button Enable to clamp colour and luminance values on input in the 16-bit floating point rendering pipeline.

Clamp Render box Select a clamping option for colour and luminance values on output in the 16-bit floating point rendering pipeline.

Blur Type Settings

Blur Type for Gaussian

Blur Type for Radial

Blur Type box Select the type or shape of blur filter to apply to the clip. Depending on the blur type, some of the other blur settings may vary.

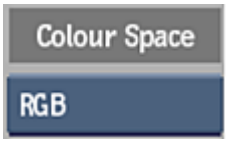
Bokeh Blur field Displays the amount of smoothness applied to sharp bokeh edges. This creates the blur that is applied to out-of-focus points of light to simulate a shallow focus. Editable.

Bias field Displays the direction of a blur. Enter a positive value for forward, a negative value for backward, or 0 for a blur that moves in both directions. Available when Directional or Radial Stamp is chosen as the blur type.

Radial Mode box Select whether a radial blur or glow moves in one circular direction (Spin) or two rotating directions (Twist). Available when Radial is chosen as the blur type.

Samples field Displays the quality of a Radial Stamp blur or glow. Editable. Available when Radial Stamp is chosen as the blur type.

Color Space Settings



Colour Space box Select whether to blur in RGB or YUV space. The Channels settings change to reflect this setting.

Centre Settings (Radial Only)

Stabilizer button Opens the Stabilizer menu to track the centre of the blur from the source clip.

Centre X field Displays the X position of the centre of the radial blur (or gesturally move the red circle in the image). Editable.

Centre Y field Displays the Y position of the centre of the radial blur (or gesturally move the red circle in the image). Editable.

NOTE You can also move the red circle on the image to set the position of the centre of the blur. The Centre X and Y fields update accordingly.

Absolute/Relative box Select whether to position and offset the centre of the radial blur in a relative mode (expressed as a percentage) or absolute mode (expressed in pixels).

Offset X field Offsets the centre along the X axis (or press Ctrl and gesturally move the red circle in the image).This field is useful to apply changes to the centre after tracking has been performed. Editable.

Offset Y field Offsets the centre along the Y axis (or press Ctrl and gesturally move the red circle in the image). This field is useful to apply changes to the centre after tracking has been performed. Editable.

NOTE You can also press `Ctrl`and move the red circle on the image to set the offset of the centre blur.

Front Settings

Width field Displays the width of the blur.Increasing the blur increases the render time. Editable.

Height field Displays the height of the blur.Increasing the blur increases the render time. Editable.

Proportional button Enable to affect the width and height proportionally.

Pixel Ratio button Enable to blur the image using the same proportion as its aspect ratio.

Gain field Displays a value by which pixel colour values are multiplied. The offset value is added to this value to determine the final colour. Editable.

Offset field Displays the value to add to current pixel colour values. The resulting colour value is clipped at 0. Editable.

Length field (Directional) Displays the radius amount of a directional blur. Editable.

Angle field (Directional) Displays the angle of a directional blur. Editable.

Amount field (Radial, Radial Stamp) Displays the amount of radial blur. Editable.

Rotation field (Radial, Radial Stamp) Displays the angle of rotation for a radial blur. Editable.

Blooming Settings (Defocus Only)

Basic/Additive Blooming button Switch between basic blooming and additive blooming bokeh effects. The basic blooming mode displays the gain applied to highlights in non-HDR images, to allow the creation of bokeh patterns without affecting colour integrity. The additive blooming mode allows you to create higher intensity bokeh patterns from any source image, using minimum and maximum thresholds for highlight segregation.

Basic Blooming field Displays the gain applied to highlights in non-HDR images, to allow the creation of bokeh patterns without affecting colour integrity. This creates the glow effect that is applied to the bright spots of the image to simulate light bleeding, or blooming, over the edges. Editable.

Additive Blooming field Displays the amount of high intensity bokeh patterns that can be created from any source image generating extreme highlight content . Editable.

Additive Blooming minimum field Displays the minimum threshold for highlight segregation. Editable.

Additive Blooming maximum field Displays the maximum threshold for highlight segregation. Editable.

Matte Settings

Matte Settings for Gaussian, Defocus, or Box

Matte Settings for Radial or Radial Stamp

Matte Settings for Directional

Matte Width field Displays the width of the blur for the matte. Editable.

Matte Height field Displays the height of the blur for the matte. Editable.

Matte Length field Displays the radius amount of a directional blur for the matte. Editable.

Matte Angle field Displays the angle of a direction blur of the matte. Editable.

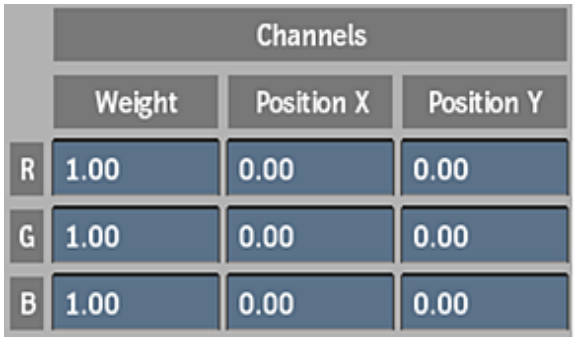
Matte Amount field Displays the amount of radial blur for the matte. Editable.

Matte Rotation field Displays the angle of rotation for a radial blur for the matte. Editable.

Lock To Front button Enable to keep the matte values the same as their corresponding values for the front clip.

Invert Matte button Enable to blur the region outside the area defined by the matte.

Channel Settings



RGB Channels

YUV Channels

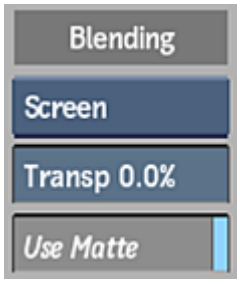
Weight field Displays the weighted value of the channel.

Position X field Displays the horizontal offset of the channel.

Position Y field Displays the vertical offset of the channel.

YUV Colour box Select whether to affect Luma or Chroma only, or include both when blurring in YUV colour space.

Blending Settings



Screen option box Select a logical operation that can be used to blend the front clip and the result clip.

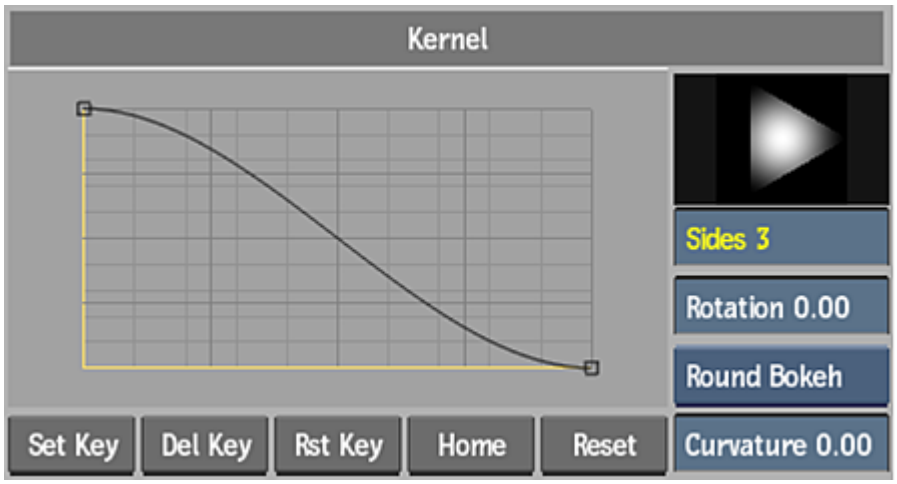
Transparency field Displays the percentage of blending when the result is composed on the front clip. Editable.

Use Matte button Enable to apply the blur with the areas defined by the matte.

Rendering Settings

Rendering Mode box Select whether to render in Automatic, Progressive or Interlaced mode.

Kernel Settings (Defocus Only)



The kernel is the defocus blur shape. The shape of the kernel is determined by its number of sides, its rotation, and the shape of its S-curve. This curve represents the shape of the pattern, from its centre to the outside. The default S-curve defines the softness of the blur. You can change the curve by manipulating the two points that define the curve, or you can add points to the curve. Use the Tools box to add and delete points on the curve.

Sides field Displays the number of sides in the kernel shape. Editable, if kernel information is not attached to the node.

Rotation field Displays the angle of rotation of the kernel shape. Editable, if kernel information is not attached to the node.

Bokeh Type box Select whether to use a round or angle bokeh curve to define the kernel shape.

Curvature field Displays the amount of curvature applied to a round bokeh kernel shape. Editable.

Bump Displace

Use the Bump Displace effect to create displacement-like emboss effects using light and surface bumps. You can select Front or Front/ZDepth or Front/ZDepth/Matte from the Input Mode box.

To access the Bump Displace menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip and a displacement clip (usually a matte), and outputs a result.

Bump Displace Menu Settings

Bump Settings

Minimum Bumps field Displays the lower limit of the bump depth. Editable.

Maximum Bumps field Displays the upper limit of the bump depth. Editable.

Bump Height field Displays the height value for the effect. Positive numbers create an embossed look. Negative numbers create an engraved look. Editable.

Bump Softness field Displays a blurring value for the effect. Editable.

Light Settings

Light X Position field Displays the horizontal position of the light. Editable. You can also use the light icon in the image window to change the position of the light.

Light Y Position field Displays vertical position of the light. Editable. You can also use the light icon in the image window to change the position of the light.

Light Y Position field Displays the depth of the light. Editable. You can also use the light icon in the image window to change the position of the light.

TIP You can also use the light icon in the image window to gesturally position the light.

Light Scale field Displays a value that you can modify to see the results of moving your light when using extreme X, Y or Z values. Editable.

Shaded field Displays the level of intensity for the image. Editable.

Ambient field Displays the percentage of overall lighting for the image. Editable.

Threshold field Displays the percentage of bumps filtered out by using light exposure. Editable.

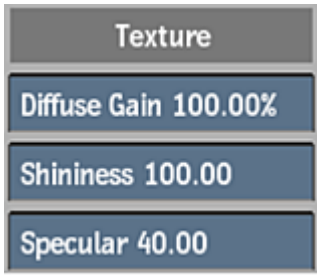
Red Light field Displays the red value of the light. Editable.

Green Light field Displays the green value of the light. Editable.

Blue Light field Displays the blue value of the light. Editable.

Light Colour Pot Displays the colour of the light. Click to open the Colour Picker and specify a value.

Texture Settings



Diffuse Gain field Displays a value for the contrast in the image. Use to increase or decrease the intensity of highlights and shadows in surface features. Editable.

Shininess field Displays a value for the amount of shine in the image. Editable.

Specular field Displays a value for the reflectiveness of the image. Editable.

RGB Distortion Settings

Red Distortion field Displays the amount of distortion in the red channel. Editable.

Green Distortion field Displays the amount of distortion in the green channel. Editable.

Blue Distortion field Displays the amount of distortion in the blue channel. Editable.

Repeat Mode Settings

Repeat Mode Settings Select an option to fill the empty portions of the frame.

Anti-Aliasing Settings

Anti-aliasing button Enable to activate sampling and softness.

Sampling box Select the number of samples to use in the anit-aliasing process.

Softness field Displays the level of softness of the samples. Editable.

Burn-in Letterbox

Use Burn-in Letterbox to burn a letterbox into a clip.

To access the Burn-in Letterbox menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip, and outputs a result.

Burn-in Letterbox Menu Settings

General Settings

Aspect Ratio box Select a preset or custom aspect ratio.

Offset field Displays the vertical offset. Drag left or right to apply the letterbox to a lower or higher portion of the image. Editable.

Border colour pot View the current border colour. Editable.

Ratio field Displays the custom aspect ratio. Editable.

NOTE The Ratio field is only available when Custom is selected from the Aspect Ratio box.

Burn-in Metadata

Use Burn-in Metadata to burn-in clip information.

To access the Burn-in Metadata menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

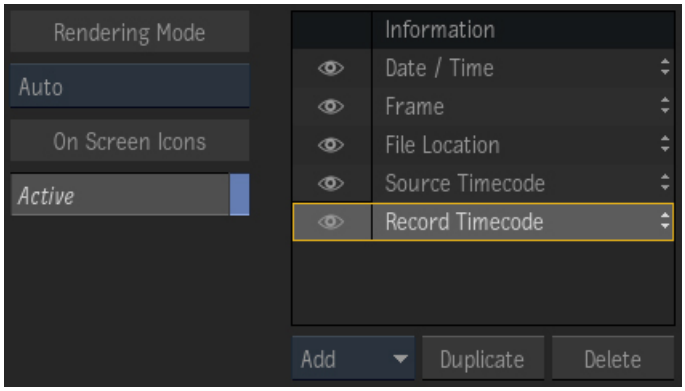
This node accepts a front clip, and outputs a result. It renders a black frame whenever its input is unlinked media or a No Media slate.

Note that some tokens and layers are either not available for display, or display different information, depending where Burn-in Metadata is being used.

NOTE The Burn-In Timecode tool is no longer available in Batch or Batch FX and the Tools tab. It can however be restored from an Archive or from a Wired project; Burn does not translate in a Burn-in Metadata.

Burn-in Metadata Menu Settings

General Settings

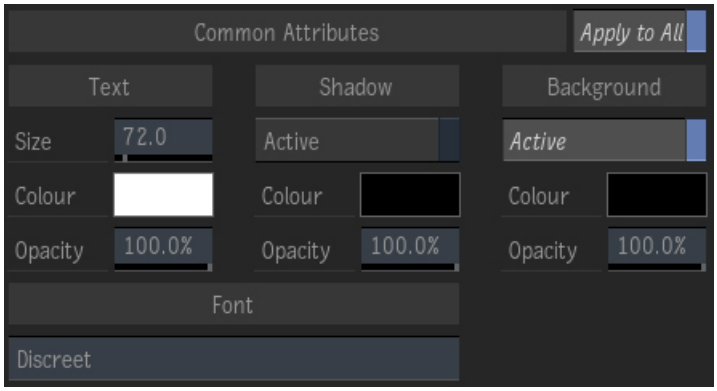


Rendering Mode box Select Automatic to render using the scan format of the input, or render in Progressive or Interlaced mode.

On Screen Icons button Enable to display layer positioners in the viewport. Use the icons to move the layer in X and Y, or rotate in Z. Alignment options allow you to affect only the selected layer or every layers.

Displayed Information list Lists all of the added information layers. Use the buttons below to populate the list with the information layers you want. Use the eye icon to mute a layer. Click a layer to display its Layer Specific options.

Common Attributes



Apply to All button Enable to have the Common Attributes applied to every layer already in the Displayed Information list. Disable to apply the attributes only to the currently selected layer.

Text Size field Displays the font size of the burned-in text. Editable.

Text colour pot Select the current colour of the burned-in text. Select to open the colour picker.

Text Opacity field Displays the opacity of the burned-in text. Editable.

Enable Shadow button Enable to add a drop shadow to the burned-in text.

Shadow colour pot Displays the current shadow colour. Select to open the colour picker.

Shadow Opacity field Displays the opacity of the drop shadow. Editable.

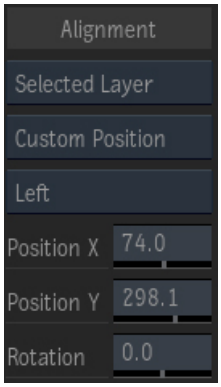
Enable Background button Enable to add a rectangular background to the burned-in text.

Background colour pot Displays the current background colour. Select to open the colour picker.

Background Opacity field Displays the opacity of the background. Editable.

Font field Displays the font used for the burned-in text. Editable.

Alignment



Layer Selection box Use Selected Layer to modify only the selected layer. All Layers modifies all the layers at once, but the modifications are treated as offsets for each individual layer. Global Axis modifies the layers in one block keeping their relative positions.

Alignment Selection box Selects how the layer is positioned: custom coordinates, or relative to Safe Action or Title margins.

Text Alignment box Sets the text justification relative to its layer's axis. Disabled when the Layer Selection box is set to Global Axis.

X Position field Sets the X position of the selected layers. Editable.

Y Position field Sets the Y position of the selected layers. Editable.

Rotation field Sets the Z rotation of the selected layers. Editable.

Layer Specific Options

This section lists all the possible layers, some of which are available from the Tools or Timeline FX Burn-in Metadata, but not from Batch or Batch FX.

Resolution

Displays the resolution of the current frame. If you use Burn-in Metadata as a Timeline FX or from the Tools tab, you can display the resolution of either the timeline or the source media file.

In Batch or Batch FX, you can propagate Frame information to the Burn-in Metadata node by `T-clicking` a Clip, a Render, or a Write File node.

Resolution Text field Displays the layer's text to burn-in. Use Add Token to insert predefined information. Editable.

Add Token box Adds a token to the Text field at the insertion point.

Displayed Resolution box Sets which resolution to display. Not available in Batch or Batch FX.

Date/Time

In Batch or Batch FX, you can propagate Frame information to the Burn-in Metadata node by `T-clicking` a Clip.

Date/Time Text field Displays the layer's text to burn-in. Use Add Token to insert predefined information. Values of the tokens only update when Update is clicked. Editable.

Add Token box Adds a token to the Text field at the insertion point.

Update Field button Updates the tokens in the text field to current values.

Frame Rate

Displays the frame rate. If you use Burn-in Metadata as a Timeline FX or from the Tools tab, you can display the frame rate of either the timeline or the source media file.

In Batch or Batch FX, you can propagate Frame information to the Burn-in Metadata node by `T-clicking` a Clip, a Render, or a Write File node.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Numeric Only button Enable to remove the 'fps' units from the frame rate.

Frame Rate Mode button Sets the frame rate to display, either the one of the timeline/clip, or the one from the source media file. Not available in Batch or Batch FX.

Source Timecode

In Batch or Batch FX, you can propagate Frame information to the Burn-in Metadata node by `T-clicking` a Clip, a Render, or a Write File node.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Custom Start button Enable to define a customized start value for the Source timecode.

Custom Start Value field Sets a customized start value for the Source timecode. Editable.

Frame Code Mode box Sets the frame code mode (FCM) for the customized Source timecode.

Record Timecode

In Batch or Batch FX, you can propagate Frame information to the Burn-in Metadata node by `T-clicking` a Clip, a Render, or a Write File node.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Custom Start button Enable to define a customized start value for the Record timecode.

Custom Start Value field Sets a customized start value for the Record timecode. Editable.

Frame Code Mode box Sets the frame code mode (FCM) for the customized Record timecode.

Keycode

This layer displays *No Keycode* unless a valid keycode is manually specified or automatically derived from the input clip.

In Batch or Batch FX, you can propagate Frame information to the Burn-in Metadata node by `T-clicking` a Clip.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Custom Start Keycode button Enable to define a customized start keycode.

Custom Keycode field Sets the customized keycode start. Editable.

Frame

Displays the current frame number. How that number is determined in the Timeline and in the Tools tabs is determined by the First Frame box setting. In Batch or Batch FX, it matches that of the Current Frame field.

In Batch or Batch FX, you can propagate Frame information to the Burn-in Metadata node by `T-clicking` a Clip, a Render, or a Write File node. And if you `T-click` a Render or a Write File node, the Max Frame Value field is automatically set to the Render Range's *To* value.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

First Frame box Sets how frames are counted. Segment restarts the count at each segment, with Max Frame as the segment duration. Timeline/Clip count starts with the timeline or clip, with Max Frame as the duration. Source Media counts from the source absolute start frame, with Max Frame as the source duration.

Not available in Batch or Batch FX.

Padding field Sets the padding for the frame number. Editable.

Custom Start Frame button Enable to define the value of the start frame for the clip. Disables First Frame box option if available. Editable.

Custom Start field Sets the value of the start frame. Editable.

Count Direction button Sets the direction of the count.

Max Frame button Enable to display of the Max Frame Value field.

Max Frame Value field Sets a maximum frame value. Editable.

User

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

User Nickname

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Project

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Project Nickname

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Workstation

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Custom Text

Text field Displays customized text. Editable.

File Location

Displays the location of the source media file.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Include Filename button Enable to display the name of the media file

Custom Level button Enable to truncate the displayed file path.

Truncate box Sets the directory levels to keep in the displayed path. Keep Last preserves the end of the file path. Remove First removes the beginning of the file path.

Directory field Sets the number of directory levels to preserve or remove, according to the Truncate box. Editable.

Source Version ID

Displays the source version ID of the clip or segment currently displayed. Not available in Batch or Batch FX.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Source Version Name

Displays the source version name of the clip or segment currently displayed.

Not available in Batch or Batch FX.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Comment

Displays the contents of the Timeline segment's comment field.

Not available in Batch or Batch FX.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Tape/Reel/Source

In Batch or Batch FX, you can propagate Frame information to the Burn-in Metadata node by `T-clicking` a Clip, a Render, or a Write File node. And if you `T-click` a Render or a Write File node, the Max Frame Value field is automatically set to the Render Range's *To* value.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Source Name

Displays the source's name.

Not available in Batch or Batch FX.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Shot Name

Displays the contents of the Timeline segment's Shot Name field.

Not available in Batch or Batch FX.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Segment Name

Displays the contents of the Timeline segment's Segment Name field.

Not available in Batch or Batch FX.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Name

Displays the clip Name if you use Burn Metadata from the Timeline or the Tools tab. Displays the Node Name if you use Burn Metadata as a node in Batch or Batch FX.

In Batch or Batch FX, you can propagate Frame information to the Burn-in Metadata node by `T-clicking` a Clip, a Render, or a Write File node.

Label button Enable to display as a label the contents of the Label field below.

Label field Displays the layer's label to burn-in. Enabled by Label. Editable.

Clamp

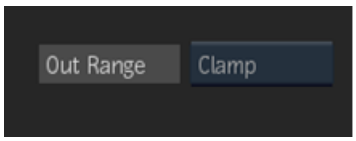
Use the Clamp node to clamp 16-bit floating point OpenExr clips.

To access the Clamp menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

Clamp Menu Settings

General Settings



Out of Range box Select a curve that is constant (clamp) or clamps negative values below 0.

Colour Corrector

About Colour Corrector

The Colour Corrector includes tools that provide precise control over colour values. You can modify luma ranges in a clip (shadows, midtones, and highlights), sample colours, and adjust the colour balance. You can rewire colour channels and suppress colours, as well as animate a colour correction by manipulating the animation curve in the Channel Editor.

When working with large images, you can free up additional screen space with the Overlay user interface. This feature is exclusive to the Colour Corrector and the Colour Warper. The Colour Warper's Overlay user interface is not available when accessing the Colour Warper as a Batch or Batch FX node.

For details on using the Overlay user interface, see Overlay User Interface (page 408).

Accessing the Colour Corrector

To access the Colour Corrector, you must load clips of the same resolution. If the clips you want to load have different resolutions, resize them so that they have the same resolution.

You can load a front clip, a front and back clip, or a front, back, and matte clip for colour correction. Changes in colour are applied to the front clip.

To access the Colour Corrector menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- Modular Keyer, Action or Paint tool.

Colour Correct Menu Settings

Timeline FX Quick Menu Settings

To see the full Colour Correct menu, click the Editor button.

Luma Range Quick Menu selector Select the quick menu for the luma range you want to modify.

Colour Channel Quick Menu selector Select the quick menu for the colour channel you want to modify.

Use Matte button Enable to use the matte input as part of the effect processing.

Main Menu

Setup button Opens the Setup menu where you setup either Colour Correct or Colour Warper options.

Animation button Opens the Animation menu where you display the animation channel.

Master button Modifies the luma range for the entire image after individual range modifications.

Shadows button Modifies the luma range for the dark areas in the image.

Midtones button Modifies the luma range for the midlevel areas in the image.

Highlights button Modifies the luma range for the light areas in the image.

Regen button Enable to get dynamic updating of your colour corrections.

Rewire box Select an option to rewire the RGB channels of an image, or create a monochrome or negative image.

Red Rewiring box Select a color or luma channel to replace the red values with the selected channel values. Select 1-R to replace red values with its inverse values. Active when Master range is selected.

Green Rewiring box Select a color or luma channel to replace the green values of an image with its inverse (1-R), red, blue, or luma values. Active when Master range is selected.

Blue Rewiring box Select a color or luma channel to replace the blue values of an image with its inverse (1-R), red, green, or luma values. Active when Master range is selected.

Reset Basics Resets colour correction controls.

Hue field Displays the colour range. Editable.

Saturation field Displays level of colour purity in the image. Editable.

Pivot field Displays the percentage value of the colour range around which the contrast pivots. The default value is 50% for integer input and 18% for floating point input. Editable.

Hue field Displays the colour range. Editable.

Gain field Drag left or right to increase or decrease light greys from the white area of the image.

RGB Gamma field Adjust the grey values. Editable.

Red Gamma field Adjust the grey values in the red channel. Editable.

Green Gamma field Adjust the grey values in the green channel. Editable.

Blue Gamma field Adjust the grey values in the blue channel. Editable.

RGB Gain field Set the percentage of original colour values to use in the image. Editable.

Red Gain field Set the percentage of colour values in the red channel to display in the image. Editable.

Green Gain field Set the percentage of colour values in the green channel to display in the image. Editable.

Blue Gain field Set the percentage of colour values in the blue channel to display in the image. Editable.

RGB Offset field Adjust the colour values by adding an offset value. Editable.

Red Offset field Adjust the colour values in the red channel by adding an offset value. Editable.

Green Offset field Adjust the colour values in the green channel by adding an offset value. Editable.

Blue Offset field Adjust the colour values in the blue channel by adding an offset value. Editable.

Contrast field Adjust the gradations between the light and dark areas in the image. Editable.

Red Contrast field Adjust the gradations between the light and dark areas in the red channel of image. Editable.

Green Contrast field Adjust the gradations between the light and dark areas in the green channel of image. Editable.

Blue Contrast field Adjust the gradations between the light and dark areas in the blue channel of image. Editable.

Red Suppression button Enable to suppress red colour values in the image.

Green Suppression button Enable to suppress green colour values in the image.

Blue Suppression button Enable to suppress blue colour values in the image.

Cyan Suppression button Enable to suppress cyan colour values in the image.

Magenta Suppression button Enable to suppress magenta colour values in the image.

Yellow Suppression button Enable to suppress yellow colour values in the image.

Front colour patch Displays colour sampled from the front clip.

Result colour patch Displays the result of front clip colour correction.

Back colour patch Displays colour sampled from the back clip.

Sampling box Select the type of colour values to display for sampled colours.

Sample Data Type box Select the type of measurement to use for RGB values of sampled colours.

Histogram Menu

Histogram tab Displays the Histogram menu where you adjust the red, green, blue, and luminance channels of the image.

Histogram Displays a bar graph that is used to adjust the luminance values of the image. In the Ranges menu, the curves for the three luma ranges are also displayed.

Minimum Input field Displays the lower limit of the luminance values. Pixels with lower values are mapped to black.

Minimum Output field Displays the lower limit of the luminance values for black pixels.

Maximum Input field Displays the upper limit of the luminance values. Pixels with higher values are mapped to white.

Maximum Output field Displays the upper limit of the luminance values for white pixels.

Channel Selection box Select the luminance or an RGB channel to extract its values for the key.

Out Range box Select a curve that is constant (Clamp) or linear (No Clamp) before the first point of the curve and after the last point of the curve.

When using 16-bit floating point images, you can select Clamp to clamp colour and luminance values, or No Clamp to allow pixel floating point values to be less the 0 or more than 1.

Frame Selection box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Source button Enable to show a histogram of the source colour values.

The source colour values are obtained from the current frame. If you input both front and matte clips, the colour values in the front clip that are defined by the white part of the matte are displayed.

Source View box Select whether to display all RGB Source histograms or only the histogram you are working on (Mono).

Destination button Enable to show a histogram of the colour values in the result or destination clip.

The destination colour values are obtained from the current frame. If you input both front and matte clips, the colour values in the front clip that are defined by the white part of the matte are displayed.

Destination View box Select whether to display all RGB Destination histograms or only the histogram you are working on (Mono).

When viewing the Source and Destination histograms at the same time in Mono mode, the Destination histograms are lighter; in RGB mode, the Destination histograms are displayed with a white outline.

Reset button Resets to default histogram settings.

Zoom field Displays the vertical zoom value of the histogram.

You can also zoom horizontally by pressing `Ctrl+spacebar` and dragging left or right in the histogram. To pan horizontally, click `spacebar` and drag left or right in the histogram.

Curves Menu

Curves tab Displays the Curves menu to display luminance and RGB curves.

Curves Display the curves for the luminance and each RGB channel.

Home button Restores the position of panned or zoomed curves to the default setting.

Match button Matches colours in the front and back colour patches, and assigns a number to the current match.

Match field Displays which match is currently displayed.

ReMatch button Overrides the selected match operation with current settings.

Out Range box Select a curve that is constant (Clamp) or linear (No Clamp) before the first point of the curve and after the last point of the curve.

Reset button Resets to default curve settings.

Ranges Menu

Ranges tab Displays the HLS Ranges menu where you can adjust highlight, midtone, and shadow ranges.

Reset button Resets to default histogram and curve settings.

Setup

Invert Matte button Enable to colour correct the region outside the area defined by the matte.

Background field Displays the background brightness value. Editable

Colour Curves

Use Colour Curves to access a clip's colour menu, remove colour spill from a front clip, and perform a hue shift.

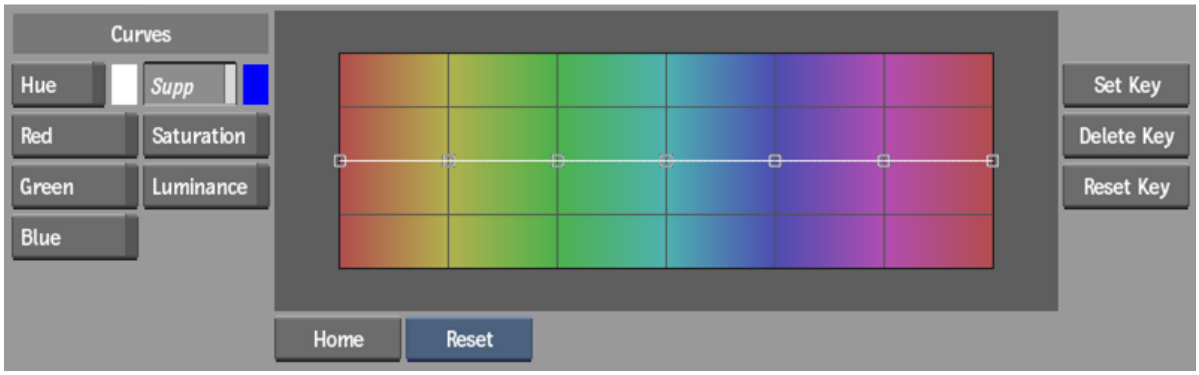
To access the Colour Curves menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Colour Curves Menu Settings

General Settings



Curve Channel button Enable this button to select the colour curve for a specific channel. Only one Curves button can be enabled at a time. Each Curves button is active when you modify its curve in the graph.

Hue colour pot Displays the colour to shift toward when the curve is modified. Editable.

Suppression colour pot Displays the colour to suppress in the clip. Editable.

Home button Reverts to the original view.

Reset box Resets the curve view.

Set Key button Sets a keyframe at the selected frame.

Delete Key button Deletes the selected keyframe.

Reset Key button Resets the curve at the selected keyframe.

Colour Management

This node accepts a front clip, and outputs a result.

General settings

Colour Management Mode box Select the type of colour management tool to use.

Bit-Depth box Select the output bit-depth. Note: Select "16 float" when converting to a scene-linear colour space.

Tagged Colour Space box Select the colour space to tag your media with. This does not alter pixel values, but it does affect how the viewports show the media.

Invert button Invert the colour transform to convert from the original destination back to the original source colour space. In Custom mode, you may also invert individual component transforms by clicking in the Invert column to toggle it.

Export button Export this transformation as a single CTF file.

Pixel Sampler Click on this box and then drag the cursor through the image to sample values.

Colour Transform details Shows descriptive information about the transform including the mathematical operations applied. (The bit-depths show the original transform scaling but internal processing is done at 32-bit float and then scaled to the selected output depth).

Input Transform settings

Input Colour Space menu Select the source colour space of your media. "From File or Rules" uses the File Input Rules set in Colour Management Preferences. For a description of the available colour spaces, click Input Colour Spaces, on the right.

Working Space menu Select the destination colour space of your media. Set the Working Space first in order to filter the list of Input Colour Spaces to show only the compatible options. "From Project" takes the value from the Colour Management Preferences.

View Transform settings

Tagged Colour Space menu Select "From Source" to use the tagged colour space of the input or select another value to override it.

View Transform menu Select "From Rules" to use the highest priority Viewing Rule for the Tagged Colour Space and Display. Select from the Viewing Rules category to choose one of the Viewing Rules compatible with this Tagged Colour Space and Display. Or select any other viewing transform. Auto-conversion will be applied to connect the Tagged Colour Space to the colour space expected by the view transform and to connect the output colour space of the view transform to the selected Display.

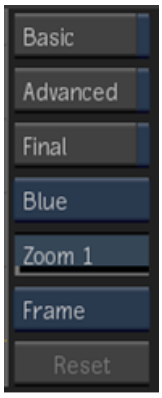
Display menu Select the Display colour space that you want to produce media for. The "Current Broadcast/Graphics Monitor" options are taken from the Colour Management Preferences for this workstation.

Colour Management Menu Settings (LUT Editing)

General Settings

Export button Opens the Export LUT menu where you select the location to export the LUT.

Imported LUT field Displays the name of the imported LUT.



Basic button Enable to display the basic conversion LUT curve.

Advanced Editing button Enable to display advanced editing curves for each colour channel. RGB curves use blending to distinguish overlapping curves.

Final button Enable to display the final conversion LUT curve.

Edit Curve box Select to display the editing curve for the selected colour channel.

Zoom field Displays the zoom percentage of the histogram. Editable.

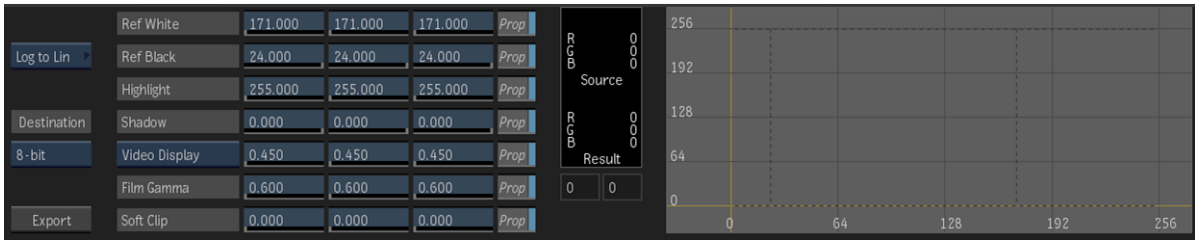
Home box Select whether to recenter the graph or center the frame.

Reset button Applies default settings.

NOTE When accessed as a Timeline FX, you can use the quick menu to change some settings for each LUT type. To access the full LUT Editor menu, click the Enter Editor button.

Quick Menu Selector Select the quick menu that you want to modify.

Log to Lin, Lin to Log, and Gamma Settings



Red Reference White field Displays the lower limit at which red channel values are considered white. Editable.

Green Reference White field Displays the lower limit at which green channel values are considered white. Editable.

Blue Reference White field Displays the lower limit at which blue channel values are considered white. Editable.

Red Reference Black field Displays the upper limit at which red channel values are considered black. Editable.

Green Reference Black field Displays the upper limit at which green channel values are considered black. Editable.

Blue Reference Black field Displays the upper limit at which blue channel values are considered black. Editable.

Red Highlight field Displays the value at which greater red channel values are clamped. Editable.

Green Highlight field Displays the value at which green channel values are clamped. Editable.

Blue Highlight field Displays the value at which blue channel values are clamped. Editable.

Red Shadow field Displays the value at which lower red channel values are clamped. Editable.

Green Shadow field Displays the value at which lower green channel values are clamped. Editable.

Blue Shadow field Displays the value at which blue channel values are clamped. Editable.

Gamma Correction box Select preset gamma correction values (Video Display) or custom values. This setting is available for Log to Lin and Lin to Log.

Red Gamma Correction field Displays the gamma correction value for the red channel. Editable.

Green Gamma Correction field Displays the gamma correction value for the green channel. Editable.

Blue Gamma Correction field Displays the gamma correction value for the blue channel. Editable.

NOTE The following settings are available for Log to Lin and Lin to Log.

Red Film Gamma field Displays the gamma correction value for the red channel. Editable.

Green Film Gamma field Displays the gamma correction value for the green channel. Editable.

Blue Film Gamma field Displays the gamma correction value for the blue channel. Editable.

Red Soft Clip field Displays the level of adjustment to the shoulder of the conversion curve for the red channel. The upper limit is dependent on the channel's current reference white value. To create softer highlight, drag right when using a linear curve. Drag left when using a logarithmic curve.

Green Soft Clip field Displays the level of adjustment to the shoulder of the conversion curve for the green channel. The upper limit is dependent on the channel's current reference white value. To create softer highlight, drag right when using a linear curve. Drag left when using a logarithmic curve.

Blue Soft Clip field Displays the level of adjustment to the shoulder of the conversion curve for the blue channel. The upper limit is dependent on the channel's current reference white value. To create softer highlight, drag right when using a linear curve. Drag left when using a logarithmic curve.

Proportional button Enable to constrain channel value proportions.

EXR Display Settings

Red Exposure field Displays the luminance level of the red channel for the display image. Editable.

Green Exposure field Displays the luminance level of the green channel for the display image. Editable.

Blue Exposure field Displays the luminance level of the blue channel for the display image. Editable.

Red Defog field Displays the value to be subtracted from red color values to reduce fogging of the image. Editable.

Green Defog field Displays the value to be subtracted from green color values to reduce fogging of the image. Editable.

Blue Defog field Displays the value to be subtracted from blue color values to reduce fogging of the image. Editable.

Red Knee Low field Displays the lower limit of the compressed pixel range for the red channel. Editable.

Green Knee Low field Displays the lower limit of the compressed pixel range for the green channel. Editable.

Blue Knee Low field Displays the lower limit of the compressed pixel range for the blue channel. Editable.

Red Knee High field Displays the upper limit of the compressed pixel range for the red channel. Editable.

Green Knee High field Displays the upper limit of the compressed pixel range for the green channel. Editable.

Blue Knee High field Displays the upper limit of the compressed pixel range for the blue channel. Editable.

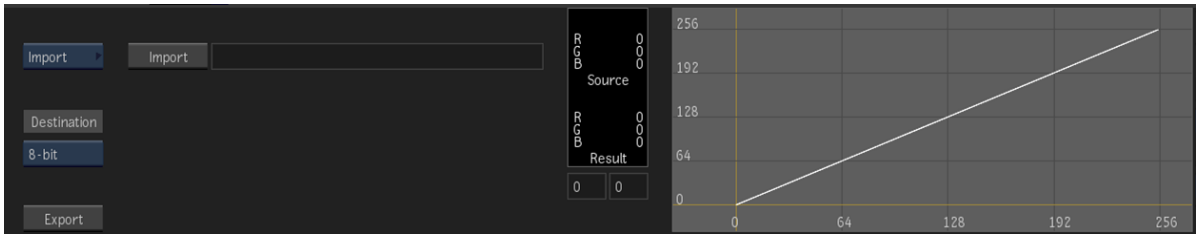
Red Gamma Correction field Displays the gamma correction value for the red channel. Editable.

Green Gamma Correction field Displays the gamma correction value for the green channel. Editable.

Blue Gamma Correction field Displays the gamma correction value for the blue channel. Editable.

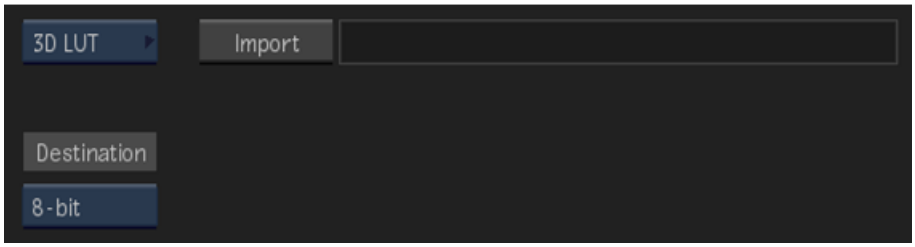
Proportional button Enable to constrain channel value proportions.

Import Settings



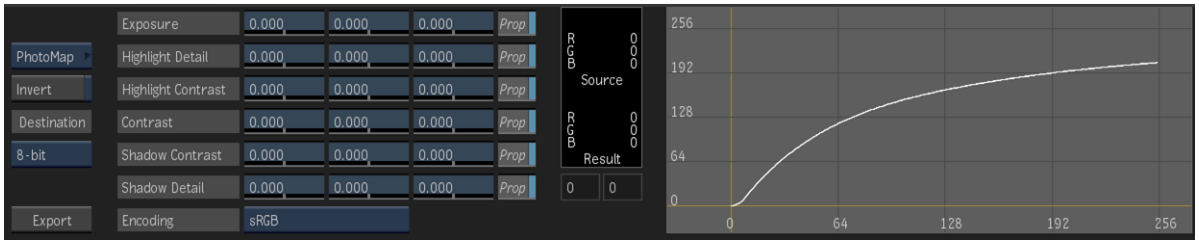
Import button Open the file browser to select a LUT for import.

3D LUT Settings



Import button Open the file browser to select a LUT for import.

PhotoMap Settings



Invert button Enable to apply the inverse of the tone-mapping parameters function to convert video media to scene-linear media.

Red Exposure field Displays the luminance level of the red channel for the display image. Editable.

Green Exposure field Displays the luminance level of the green channel for the display image. Editable.

Blue Exposure field Displays the luminance level of the blue channel for the display image. Editable.

Red Highlight Detail field Displays the value for the detail in the light areas of the red channel. Editable.

Green Highlight Detail field Displays the value for the detail in the light areas of the green channel. Editable.

Blue Highlight Detail field Displays the value for the detail in the light areas of the blue channel. Editable.

Red Highlight Contrast field Displays the value for the contrast in the light areas of the red channel. Editable.

Green Highlight Contrast field Displays the value for the contrast in the light areas of the green channel. Editable.

Blue Highlight Contrast field Displays the value for the contrast in the light areas of the blue channel. Editable.

Red Contrast field Displays the value for the midtone contrast in the red channel. Editable.

Green Contrast field Displays the value for the midtone contrast in the green channel. Editable.

Blue Contrast field Displays the value for the midtone contrast in the blue channel. Editable.

Red Shadow Contrast field Displays the value for the contrast in the dark areas of the red channel. Editable.

Green Shadow Contrast field Displays the value for the contrast in the dark areas of the green channel. Editable.

Blue Shadow Contrast field Displays the value for the contrast in the dark areas of the blue channel. Editable.

Red Shadow Detail field Displays the value for the detail in the dark areas of the red channel. Editable.

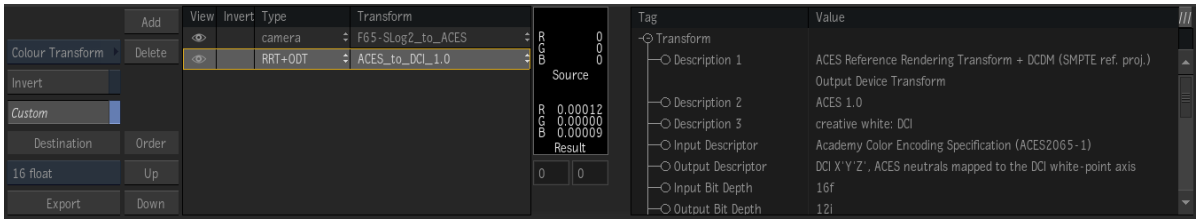
Green Shadow Detail field Displays the value for the detail in the dark areas of the green channel. Editable.

Blue Shadow Detail field Displays the value for the detail in the dark areas of the blue channel. Editable.

Proportional button Enable to constrain channel value proportions.

Encoding option box Select a standard format for the colour space.

Colour Transform Settings



Import button Use to browse and select a colour transform.

See LINK for a list of supported colour transforms.

Custom button Apply a custom chain of color transforms.

Add button Add a new row to the end of the chain. Click in the Type column to select or change the transform type, and then click in the Transform column to select a transform.

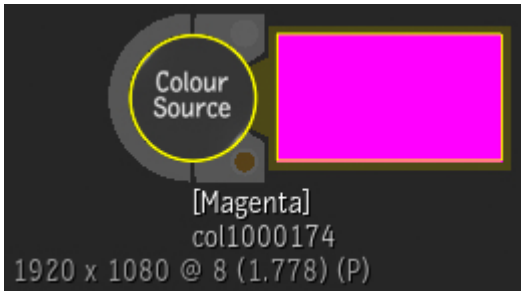
Delete button Remove the selected transform from the chain.

Up button Move the selected transform earlier in the chain.

Down button Move the selected transform later in the chain.

Colour Source

Use Colour Source to generate a colour bar, noise, gradient, or colour clip that can be used as the clip for other nodes.



To access the Colour Source menu, use:

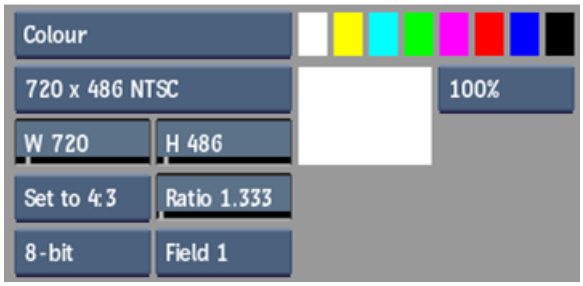
- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

This node outputs a result.

The Colour Source node includes eight colour pots for storing and selecting customized colours for creating frames. You can specify a frame's resolution and bit depth.

Colour Source Menu Settings

General Settings



Source Type box Select whether to generate frames of solid colour, noise, or colour bars.

Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

Width field Displays the custom width resolution of the clip. Editable.

Height field Displays the custom width resolution of the clip. Editable.

Aspect Ratio Presets box Select a standard frame aspect ratio. Select the Set to w:h option to set the clip to use square pixels. Select Custom to define a custom frame aspect ratio in the Aspect Ratio field.

Aspect Ratio field Displays the custom render/output aspect ratio. Editable

Bit Depth box Select the render/output bit depth of clips.

Scan Mode box Select the scan mode of clips.

Current Colour bar Displays the pattern applied to the colour source.

Luminance box Select whether to apply 75% or 100% luminance to the colour frames.

Tagged Colour Space box Select a colour space to tag the result with.

Color Warper

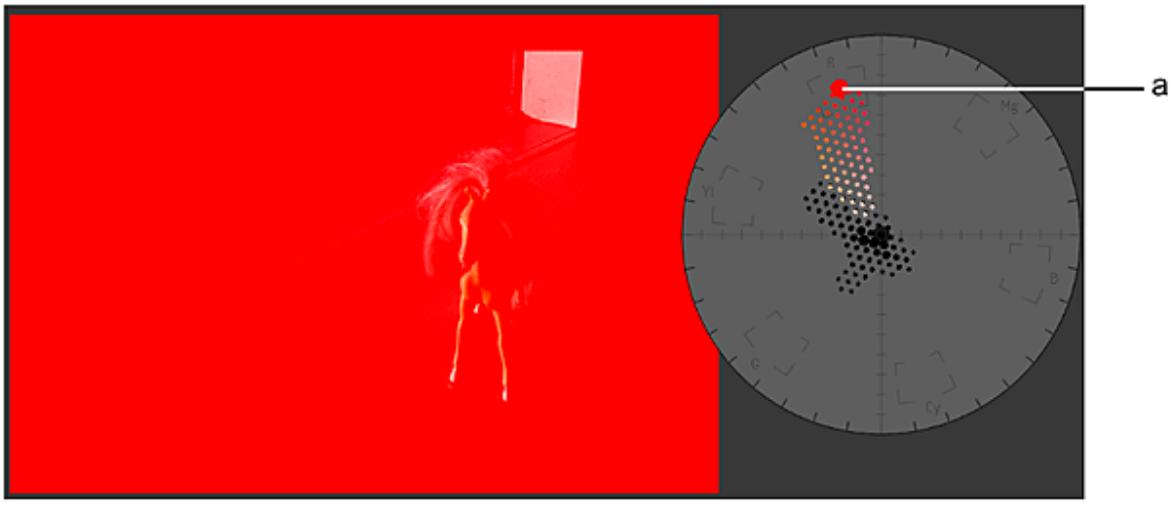
About the Colour Warper

When you colour correct an image or clip, use the Colour Warper to perform advanced colour corrections and create artistic colour effects. The way in which you approach these tasks depends on your goal, the number of clips you are using, and the type of clips being used.

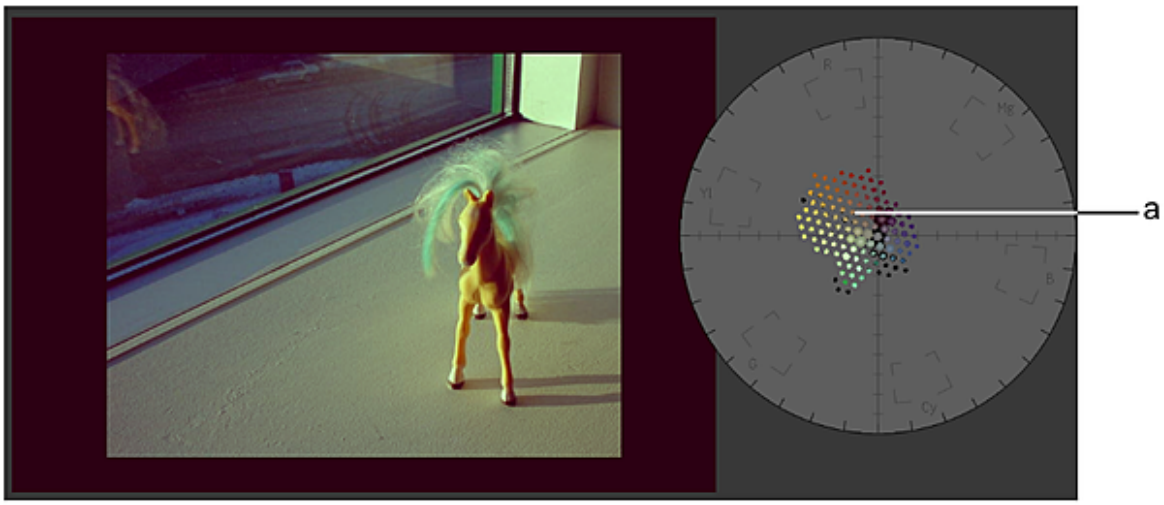
Clips created from source material shot with the same camera equipment under the same lighting conditions may be colour corrected quickly and easily to correct lighting and colour imbalances. Clips created from source material shot at different times of the day, in different seasons, at different locations, or using different equipment require more work. With the Colour Warper, you can manipulate colours with precision and ease, working on the entire clip as you would with traditional tools or working with a matte to adjust a range of colour in the clip.

Use the Colour Warper to gesturally set black and white levels, adjust specific colours and colour ranges, and accurately match colours in one clip to another. You can also perform hue shifts and suppress colour to remove colour spill or create visual effects such as a colour cast. While you manipulate the colour content of a clip, you can monitor reference clips as well as changes in the colour distribution to ensure that you achieve the result you want. Original data is always preserved, so you can adjust colours without the risk of permanently losing colour information.

When working with 16-bit floating point images in the Colour Warper, you can plot colours outside of the 0 to 1 range. Even when working with integer images, clamped colour information (colours that go beyond the RGB range) can be retrieved using the Colour Warper controls.



(a) Clamped colour



(a) Colour information restored

When working with large images, you can free up additional screen space with the Overlay user interface. This feature is exclusive to the Colour Corrector and the Colour Warper. The Colour Warper's Overlay user interface is not available when accessing the Colour Warper as a Batch or Batch FX node.

For details on using the Overlay user interface, see Overlay User Interface (page 408).

Accessing the Colour Warper

Use the Colour Warper to modify the colour content in your clips. You can manipulate colour content with intuitive controls that provide precise colour correction, and view histograms that help you visualize your image's colours. You can also output a matte corresponding to selected colours.

You can access the Colour Warper as a Batch or Batch FX node that you drag and drop into the process tree or the Modular Keyer's processing pipeline. Access the Colour Warper from the Modular Keyer to remove colour spill, or access it from Batch or Batch FX to modify the colour content of your clips.

To access the Colour Warper menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- Modular Keyer, Action or Paint tool.

Color Warper Menu Settings

Timeline FX Quick Menu Settings

To see the full Colour Warper menu, click the Editor button.

Use Mask button Enable to use the matte to delimit the Colour Correct or Colour Warper effect.

Colour Warper Menu Settings

Basics button Opens the Basics menu where you perform basic colour correction, warping, suppression and saturation of your clips.

Subsetups button Opens the menu where you store, compare, and share Colour Warper subsetups.

Basics Menu

Colour correct your clips. You can adjust shadows, midtones, and highlights, and control black and white levels, hue, and saturation. You can also perform colour warping, colour suppression, and colour saturation on a limited range of colour as well as adjust gamma and luminance curves.

Home button Positions a vectorscope at its default location and size.

Scope button Enable to display a 2D or 3D vectorscope in the Result view.

Work On box Select an option to apply colour corrections to all or part of an image. Select Master to apply colour corrections to the entire image or select a selective to generate a matte and perform selective colour correction.

Clear /Reset box Select whether to clear the value at the current frame or reset the value for the entire animation curve.

Selective View box Select whether to view the matte, selective, or result image.

Clear /Reset box Select whether to clear the value at the current frame or reset the value for the entire animation curve.

Black field Displays the luminance value in the image shadows without affecting the chrominance values. Editable.

White field Displays the luminance value in the image highlights without affecting the chrominance values. Editable.

Hue field Displays the luminance value in the image shadows without affecting the chrominance values. Editable.

Saturation field Displays the global saturation value. Editable.

Red Gamma field Displays the gamma level in the red channel.

Clear /Reset box Select whether to clear the value at the current frame or reset the value for the entire animation curve.

Green Gamma field Displays the gamma level in the green channel.

Clear /Reset box Select whether to clear the value at the current frame or reset the value for the entire animation curve.

Blue Gamma field Displays the gamma level in the blue channel.

Clear /Reset box Select whether to clear the value at the current frame or reset the value for the entire animation curve.

Select button Activates the pick cursor. Use to sample an area to adjust. Activate the cursor again and select the area to match.

Plot button Activates the pick cursor. Use to sample an area to display its result clip colour value.

Ref button Activates the pick cursor. Use to sample an area to display its reference clip colour value.

Trackball option box Select a method of adjustment for a range of colour.

Luma field Displays the luminance of the destination colour. Editable.

Tools box Select a tool to work with.

Home button Restore the histogram to its default zoom and pan settings.

Clear /Reset box Select whether to clear the value at the current frame or reset the value for the entire animation curve.

Source button Enable to show a histogram of the colour values in the front, or source clip.

Destination button Enable to show a histogram of the colour values in the result clip.

Zoom field Displays the zoom ratio of the histogram. Editable.

Selective menu

Define colour ranges using softness and tolerance to create a matte and apply colour correction to a selected region.

1 Enable to apply the matte for Selective 1 to the result.

2 Enable to apply the matte for Selective 2 to the result.

3 Enable to apply the matte for Selective 3 to the result.

Invert button Enable to invert the matte or selective.

Gaussian button Enable to apply a Gaussian blur to the matte. Disable to apply a box blur.

X field Displays the width of the blur. Editable.

Y field Displays the height of the blur. Editable.

Pick Custom button Define the tolerance range based on a sample from the image. Drag the cursor over the image to define initial tolerance.

Red button Define the tolerance range based on the red channel.

Green button Define the tolerance range based on the green channel.

Blue button Define the tolerance range based on the blue channel.

Cyan button Define the tolerance range based on the cyan channel.

Magenta button Define the tolerance range based on the magenta channel.

Yellow button Define the tolerance range based on the yellow channel.

Shadows button Define the tolerance range based on the dark areas in the image.

Midtones Define the tolerance range based on the midlevel areas in the image.

Highlights button Define the tolerance range based on the light areas in the image.

Adjusting box Select Tolerance to add tolerance to the matte, +Softness to add softness to the matte, or -Softness to remove softness from the matte.

Sharpness field Displays the percentage of sharpness of the matte. Editable.

Tolerance button Enable to apply the grey tolerance indicator on the hue cube.

Softness button Enable to apply the black softness indicator on the hue cube.

Move/Zoom box Select Move to adjust the softness and tolerance by moving the handles on the hue cube or Zoom to zoom in by dragging on the cube.

Low Softness field Displays the minimum value for the softness range. Editable.

Low Tolerance field Displays the minimum value for the tolerance range. Editable.

High Tolerance field Displays the maximum value for the tolerance range. Editable.

High Softness field Displays the maximum value for the softness range. Editable.

Frame Options box Select Home to return the hue cube to its default position, Plot Colour to enlarge the gradient to include the plot and reference colours, or Autoframe to view the complete gradient range.

Auto Key button Set keyframes automatically when changes are made to a frame.

Gamma Options

RGB button Enable to display the RGB gamma curve.

Red button Enable to display the red gamma curve.

Green button Enable to display the green gamma curve.

Blue button Enable to display the blue gamma curve.

Luminance button Enable to display the luminance curve.

Warp Options

Source colour pot Displays the colour you want to modify. Editable.

Destination colour pot Displays the corrected colour.

Subsetups Menu

Store, compare, and share Colour Warper subsetups.

Current button Access the most recently committed setup.

Load button Open the file browser to load a group of subsetups within a directory.

Save button Open the file browser to save active subsetups in a specific directory.

Subsetup A LED button Indicates the subsetup is stored.

Subsetup A button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup B LED button Indicates the subsetup is stored.

Subsetup B button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup C LED button Indicates the subsetup is stored.

Subsetup C button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup D LED button Indicates the subsetup is stored.

Subsetup D button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup E LED button Indicates the subsetup is stored.

Subsetup E button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup F LED button Indicates the subsetup is stored.

Subsetup F button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup G LED button Indicates the subsetup is stored.

Subsetup Gbutton Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup H LED button Indicates the subsetup is stored.

Subsetup H button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup I LED button Indicates the subsetup is stored.

Subsetup I button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup J LED button Indicates the subsetup is stored.

Subsetup J button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Setup Menu

Scope box Displays the vectorscope you want to use to view the clip.

Canvas button Enable to show the vectorscope canvas. When the canvas is off, the vectorscope is transparent, but can be outlined.

Lines button Enable to show the vectorscope outline.

Bars box Select the SMPTE bars setting for the 2D vectorscope.

Size box Select the size of the coloured squares or cubes that make up the histogram.

Clear Buffer button Clears the Undo buffer of all previous undo operations.

Levels field An upper limit for the number of undo or redo operations. Editable.

Combine

Use Combine to combine the individual color space channels of three different source clips (RGB, YUV, or HSL).

To access the Combine menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front, back and matte clip, and outputs a result.

Combine Menu Settings

General Settings

Colour Space Type box Select which type of colour space to combine.

Comp

Use Comp to blend two inputs and their mattes. The type of composition is determined by the selected blending mode.

To access the Comp menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

The Timeline FX pipeline uses:

- The upper track as the primary clip and matte (optional) that comprise the foreground of the composition.
- The lower tracks as the secondary clip and matte (optional) that comprises the background.

This node accepts:

- The primary clip (Front) and matte that comprise the foreground of the composition.
- The secondary clip (Back) and back matte (optional) that comprise the background.

TIP The default inputs for the foreground and background can be switched using the Swap Inputs button in the Comp menu.

This node outputs a result and an output matte.

Comp Menu Settings

Timeline FX Quick Menu Settings

To see the full Comp menu, click the Editor button.

Input Settings

NOTE Input Front and Input Back settings are identical.

Matte box Select how the matte is used when blending.

Clamping box Select a clamping option for colour and luminance values on input in the 16-bit floating point rendering pipeline.

Premultiplication box Select whether the colour values in the input are premultiplied by the matte or not.

Transparency field Displays the transparency value. Editable.

Blending Settings

Blend Type box Select whether to use Flame or Photoshop blend modes.

Blend Mode box Select how the front and back inputs are composited. The available list of modes depends on the selection in the Blend Type box.

Swap Inputs button Enable to switch the rendering order of the inputs.

Blend Matte Settings

Composition box Select how the input mattes are combined.

Correlation box Select whether the mattes are related to each other. For example, select Correlated if the mattes are from different, but continuous parts on the same object.

Output Controls Settings

Clamp Render box Select a clamping option for colour and luminance values on output in the 16-bit floating point rendering pipeline.

Pre-multiplication box Select whether the colour values in the output are pre-multiplied or not.

Compound

Use Compound to combine the contents of several frames into a single frame.

To access the Compound menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip, and outputs a result.

Compound Menu Settings

General Settings

First Rendered Frame field Displays the first frame at which output is rendered from the node. Unrendered output does not display any media.Editable.

Compound Over field Displays the number of frames that are combined into one frame. Editable.

Damage

Use Damage to apply a large variety of film or video degradation effects to a clip.

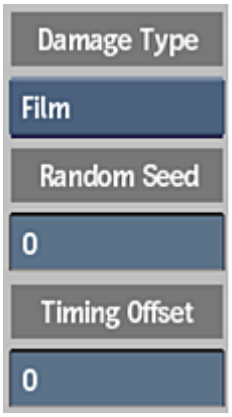
To access the Damage menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip, and outputs a result.

Damage Menu Settings

General Settings



Damage Type box Select an option for the media type of damage effects to use. Selecting one of the effects groups from the Damage Type box will display a menu of options for this effect.

Random Seed field Displays the number used to generate random variations in the damage effects. Editable.

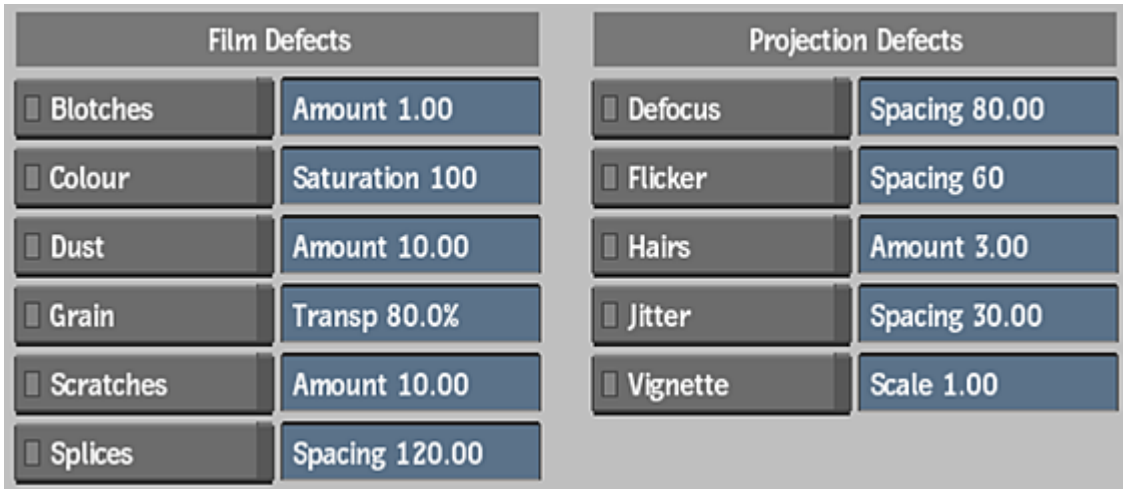
Timing Offset field Displays the number used for timing offset in the damage effects. Editable.

Damage Effect buttons Enable to display options for an effect. Click LED to switch between On (blue) and Off (grey).

Film Settings

Use the Film Damage effects to apply a large variety of film degradation effects to a clip.

When you select Film from the Damage type box, the Film Damage Effects are displayed, along with the most useful adjustment box to the right of each.



Blotches Settings

Blotches button Use this effect to simulate blotches on old film.

Blotch Amount field Displays the amount of blotches applied to the image. Editable.

Blotch Transparency field Displays the percentage of transparency applied to the blotches. Editable.

Blotch Transparency Variation field Displays the percentage of variation for the transparency applied to the blotches. Editable.

Blotch Size field Displays the average size of the blotches. Editable.

Blotch Size Variation field Displays the percentage of variation in the size of the blotches. Editable.

Blotch Softness field Displays the percentage by which the blotches are out of focus. Editable.

Blotch Negative Mix field Displays the amount of negative blotches in the mix. Editable.

Blotch colour pot Displays the tint applied to the blotches in the image. Editable.

Colour Settings

Colour button Use this effect to simulate colour fading and variation in old film.

Colour Gain trackball Drag to adjust the colour gain. This is an alternate control combining the Red Gain, Green Gain, Blue Gain, and Luma fields.

Colour Saturation field Displays the level of colour purity in the image. Editable.

Colour Gamma field Displays the amount of gamma correction applied to the image. Editable.

Colour Offset field Displays the amount by which all the colour parameters are modified. Editable.

Red Gain field Displays the amount of gain applied to the red channel. Editable.

Green Gain field Displays the amount of gain applied to the green channel. Editable.

Blue Gain field Displays the amount of gain applied to the blue channel. Editable.

Luma field Displays the amount of gain applied to the luminance channel. Editable.

Proportional button Enable to synchronize changes in red, green, blue, and luma channels.

Hue field Displays the colour range. Editable.

Shadows colour pot Displays the tint applied to shadows in the image. Editable.

Highlights colour pot Displays the tint applied to highlights in the image. Editable.

Minimum Colour Value field Displays the luminance value in the image shadows. Editable.

Maximum Colour Value field Displays the luminance value in the image highlights. Editable.

Dust Settings

Dust button Use this effect to simulate dust particles on old film.

Dust Amount field Displays the amount of dust particles applied to the image. Editable.

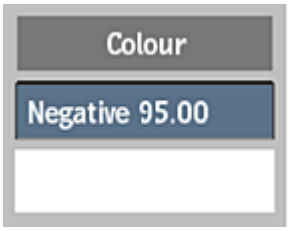
Dust Transparency field Displays the percentage of transparency applied to the dust particles. Editable.

Dust Transparency Variation field Displays the percentage of variation in transparency applied to the dust particles. Editable.

Dust Size field Displays the average size of the dust particles. Editable.

Dust Size Variation field Displays the percentage of variation applied to the size of the dust particles. Editable.

Dust Softness field Displays the percentage by which the dust is out of focus. Editable.

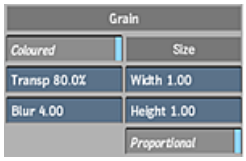


Dust Negative Mix field Displays the amount of negative dust particles in the mix. Editable.

Dust colour pot Displays the tint applied to the dust particles in the image. Editable.

Grain Settings

Grain button Use this effect to simulate grain on old film.



Grain Coloured button Enable for the grain to be coloured. Disable for the grain to be monochrome.

Grain Transparency field Displays the percentage of transparency applied to the grain. Editable.

Grain Blur field Displays the amount of blur applied th the grain. Editable.

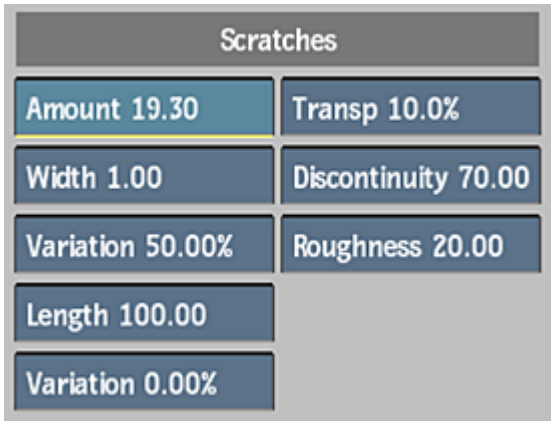
Grain Width field Displays the width of the grain in pixels. Editable.

Grain Height field Displays the height of the grain in pixels. Editable.

Grain Proportional button Enable to effect the width and height proportionally. Editable.

Scratches Settings

Scratches button Use this effect to simulate scratches in old film.



Scratch Amount field Displays the amount of scratches applied to the image. Editable.

Scratch Width field Displays the average width of the scratches. Editable.

Scratch Width Variation field Displays the percentage of variation in the width of the scratches. Editable.

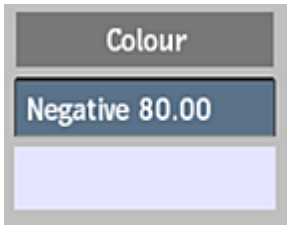
Scratch Length field Displays the average length of the scratches. Editable.

Scratch Length Variation field Displays the percentage of variation in the length of the scratches. Editable.

Scratch Transparency field Displays the transparency of the scratches. Editable.

Scratch Discontinuity field Displays the percentage of gaps that occur in the scratches. Editable.

Scratch Roughness field Displays the amount of irregularities and changes in alignment of the scratches. Editable.



Scratch Negative Mix field Displays the amount of negative scratches in the mix. Editable.

Scratch colour pot Displays the tint applied to the scratches. Editable.

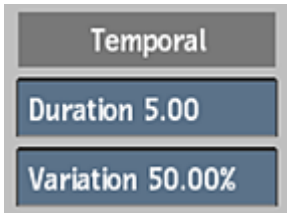
Scratch Travel field Displays the amount in pixels that the scratches move along. Editable.

Scratch Travel Variation field Displays a percentage of variation that the scratches travel. Editable.

Scratch Angle Variation field Displays the angle at which the scratches occur. Editable.

Scratch Spread field Displays the area of the frame over which the scratches are spread. Editable.

Scratch Horizontal Offset field Displays the horizontal offset of the scratches. Editable.



Scratch Duration field Displays the duration in frames of the scratch effect. Editable.

Scratch Duration Variation field Displays the variation in duration for the scratch effects. Editable.

Splices Settings

Splices button Use this effect to simulate splices on old film.

Splice Auto box Select whether the splice occurs automatically or is manually inserted. Select Manual to display the Create Splice and Delete Splice buttons.

Splice Type box Select what kind of splice to be applied to the image.

Splice Border Width field Displays the width of the splice border. Editable.

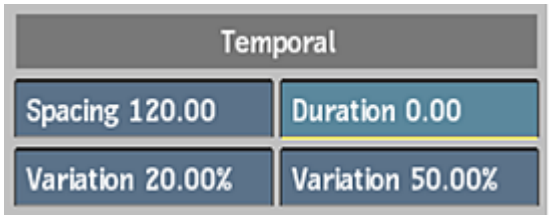
Splice colour pot Displays the colour used for the splice effects. Editable.

Splice Projector Roll field Displays the amount of space that the frame will travel as a percentage. Editable.

Splice Motion Blur field Displays the amount of motion blur applied to the splice effects. Editable.

Splice Jitter field Displays the amount of jitter applied to the splice effects. Editable.

Splice Defocus field Displays the amount of defocus applied to the splice effects. Editable.

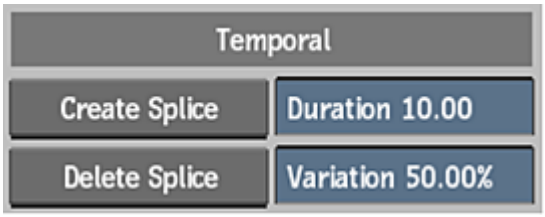


Splice Spacing field Displays the spacing in frames between splice effects. Editable.

Splice Spacing Variation field Displays the percentage of variation in spacing between splice effects. Editable.

Splice Duration field Displays the duration in frames of the splice effects. Editable.

Splice Duration Variation field Displays the percentage of variation in duration for the splice effects. Editable.

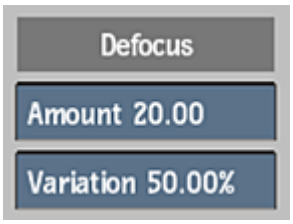


Create Splice button Creates a splice at the current frame. Available when Splice Auto is set to Manual.

Delete Splice button Deletes the splice at the current frame. Available when Splice Auto is set to Manual.

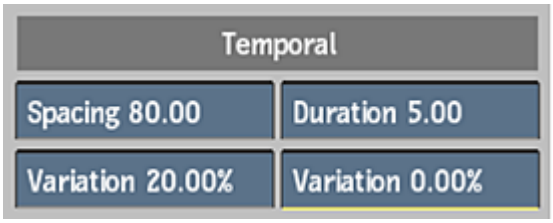
Defocus Settings

Defocus button Use this effect to simulate projection defocus on old film.



Defocus Amount field Displays the amount of defocus applied to the image. Editable.

Defocus Amount Variation field Displays the percentage of variation in the amount of defocus applied to the image. Editable.



Defocus Spacing field Displays the spacing in frames between defocus effects. Editable.

Defocus Spacing Variation field Displays the percentage of variation in the spacing between defocus effects. Editable.

Defocus Duration field Displays the duration in frames of the defocus effects. Editable.

Defocus Duration Variation field Displays the percentage of variation in duration for the defocus effects. Editable.

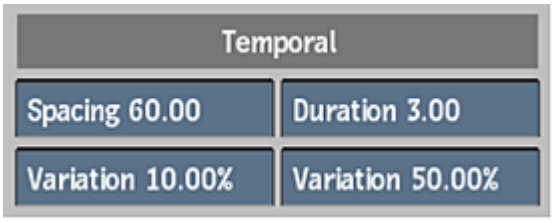
Flicker Settings

Flicker button Use this effect to simulate frame flicker on old film.

Flicker Amount field Displays the amount of flicker added to the image. Editable.

Flicker Amount Variation field Displays the percentage of variation in the amount of flicker added to the image. Editable.

Flicker Balance field Displays the ratio of dark and bright flickers used for the effect. Editable.



Flicker Spacing field Displays the spacing in frames between flicker effects. Editable.

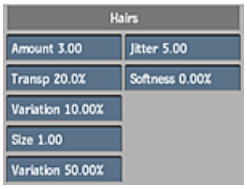
Flicker Spacing Variation field Displays the percentage of variation in the spacing between flicker effects. Editable.

Flicker Duration field Displays the duration in frames of the flicker effects. Editable.

Flicker Duration Variation field Displays the percentage of variation in duration for the flicker effects. Editable.

Hairs Settings

Hairs button Use this effect to simulate hairs on old film.



Hair Amount field Displays the amount of hairs applied to the image. Editable.

Hair Transparency field Displays the transparency applied to the hairs. Editable.

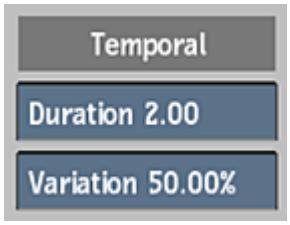
Hair Transparency Variation field Displays the percentage of variation in transparency applied to the hairs. Editable.

Hair Size field Displays the average size of the hairs. Editable.

Hair Size Variation field Displays the percentage of variation in size of the hairs. Editable.

Hair Jitter field Displays the amount of jitter applied to the hairs. Editable.

Hair Softness field Displays the percentage by which the hair is out of focus. Editable.



Hair Duration field Displays the duration in frames of the hair effects. Editable.

Hair Duration Variation field Displays the percentage of variation in duration for the hair effects. Editable.

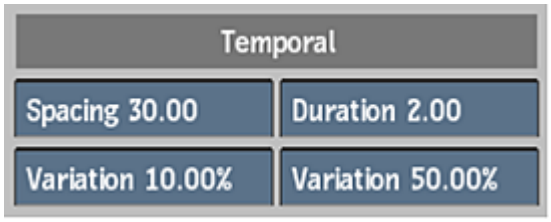
Jitter Settings

Jitter button Use this effect to simulate projection jitter on old film.

Jitter X Amount field Displays the amount of horizontal movement applied to the jitter effect. Editable.

Jitter Y Amount field Displays the amount of vertical movement applied to the jitter effect. Editable.

Jitter Amount Variation field Displays the percentage of variation applied to the movement of the jitter effect. Editable.



Jitter Spacing field Displays the spacing in frames between jitter effects. Editable.

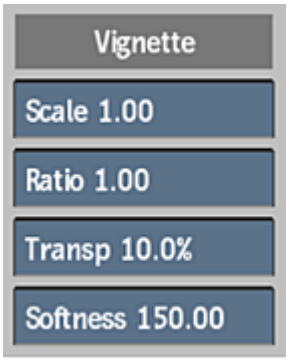
Jitter Spacing Variation field Displays the percentage of variation in the spacing between jitter effects. Editable.

Jitter Duration field Displays the duration in frames of the jitter effects. Editable.

Jitter Duration Variation field Displays the percentage of variation in duration for the jitter effects. Editable.

Vignette Settings

Vignette button Use this effect to simulate vignetting on old film.



Vignette Scale field Displays the size of the vignette effect compared to the size of the frame. Editable.

Vignette Ratio field Displays the ratio of width to height of the vignette effect. Editable.

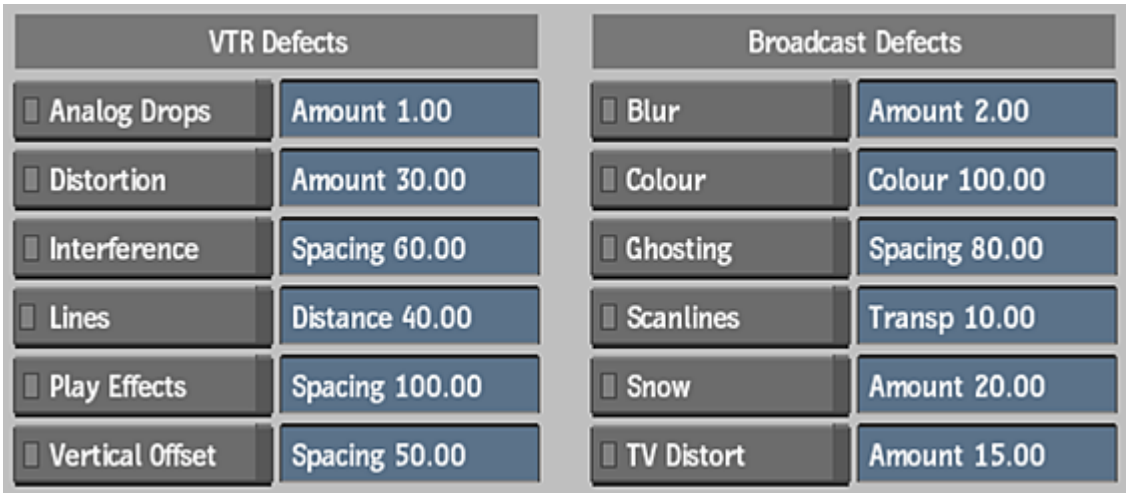
Vignette Transparency field Displays the amount of transparency applied to the vignette effect. Editable.

Vignette Softness field Displays the amount of softness added to the edge of the vignette effect. Editable.

Analog Video Settings

Use the Analog Video Damage effects to apply a large variety of analog video degradation effects to a clip.

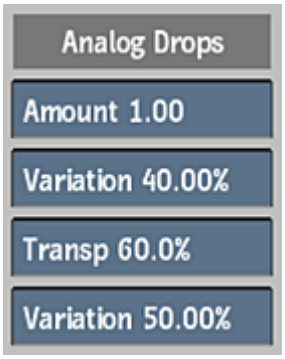
When you select Analog Video from the Damage type box, the Analog Video Damage Effects are displayed, along with the most useful adjustment box to the right of each.



Analog Drop Settings

Analog Drops button Use this effect to simulate analog drops causing noise or shash when playing back old tapes on VTRs. .

Drop Type box Select an option for the type of noise.



Drop Amount field Displays the amount of shash applied to the image. Editable.

Drop Amount Variation field Displays the percentage of variation in the amount of shash applied to the image. Editable.

Drop Transparency field Displays the percentage of transparency applied to the shash. Editable.

Drop Transparency Variation field Displays the percentage of variation in the transparency applied to the shash. Editable.

Drop Type Variation field Displays the percentage of variation applied to the drop type. Editable.

Distortion Settings

Distortion button Use this effect to simulate edge distortion when playing back old tapes on VTRs.

Edge Distortion Amount field Displays the amount of edge distortion applied to the image. Editable.

Edge Distortion Amount Variation field Displays the percentage of variation in the amount of edge distortion. Editable.

Edge Distortion Transparency field Displays the percentage of transparency applied to the edge distortion effect. Editable.

Edge Distortion Transparency Variation field Displays the percentage of variation in transparency applied to the edge distortion effect. Editable.

Edge Distortion Coverage field Displays the percentage of vertical spread for the edge distortion effect. Editable.

Edge Distortion Scale field Displays the size scaling applied to the edge distortion effect. Editable.

Edge Distortion Vertical Offset field Displays the vertical offset applied to the edge distortion effect. Editable.

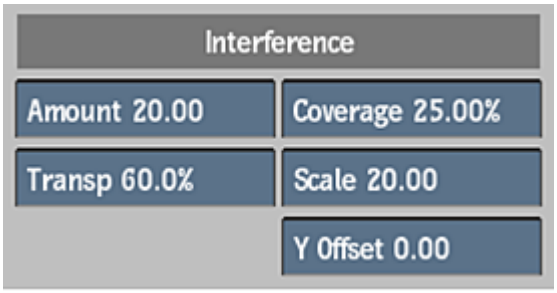
Edge Distortion Speed field Displays the speed at which the edge distortion effect moves across the image. Editable.

Edge Distortion Speed Variation field Displays the percentage of variation in the speed at which the edge distortion effect moves. Editable.

Edge Distortion Speed Variation Period field Displays the time period in frames within which the speed variation occurs. Editable.

Interference Settings

Interference button Use this effect to simulate electromagnetic Interference when playing back old tapes on VTRs.



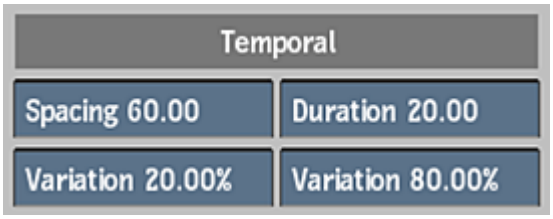
Interference Amount field Displays the amount of electromagnetic interference applied to the image. Editable.

Interference Transparency field Displays the percentage of variation in the amount of electromagnetic interference applied to the image. Editable.

Interference Coverage field Displays the percentage of vertical spread applied to the electromagnetic interference effect. Editable.

Interference Scale field Displays the scaling applied to the electromagnetic interference effect. Editable.

Interference Vertical Offset field Displays the vertical offset applied to the electromagnetic interference effect. Editable.



Interference Spacing field Displays the spacing in frames between electromagnetic interference effects. Editable.

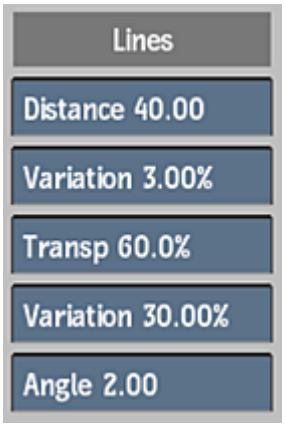
Interference Spacing Variation field Displays the percentage of variation in spacing between electromagnetic interference effects. Editable.

Interference Duration field Displays the duration in frames of the electromagnetic interference effects. Editable.

Interference Duration Variation field Displays the percentage of variation in duration for the electromagnetic interference effects. Editable.

Lines Settings

Lines button Use this effect to simulate ones caused by noise artifacts when playing back old tapes on VTRs.



Lines Distance Amount field Displays the amount of distance between the noise artifact lines applied to the image. Editable.

Lines Jittering Amount Variation field Displays the percentage of variation in the jitter applied to the lines. Editable.

Lines Transparency field Displays the percentage of transparency applied to the noise artifact lines. Editable.

Lines Transparency Variation field Displays the percentage of variation in the transparency of the lines. Editable.

Lines Angle field Displays the angle of the noise artifact lines. Editable.

Lines Speed field Displays the speed at which the noise artifact lines move around. Editable.

Lines Speed Variation field Displays the percentage of variation in the speed of the noise artifact lines. Editable.

Lines Speed Variation Period field Displays the time period in frames within which the noise artifact lines occur. Editable.

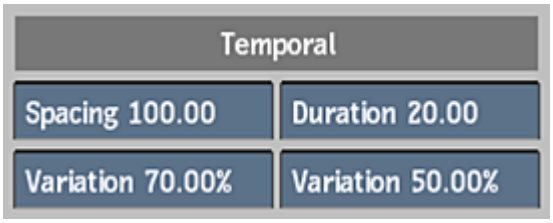
Play Effects Settings

Play Effects button Use this effect to simulate play effects caused by misaligned or dirty tape heads when playing back old tapes on VTRs.

Play Effects Amount field Displays the amount of play effects added to the image. Editable.

Play Effects Speed field Displays the speed at which the play effects occur. Editable.

Play Effects Freeze field Displays the number of repeat frames added to create a jerky play effect. Editable.



Play Effects Spacing field Displays the spacing in frames between play effect errors. Editable.

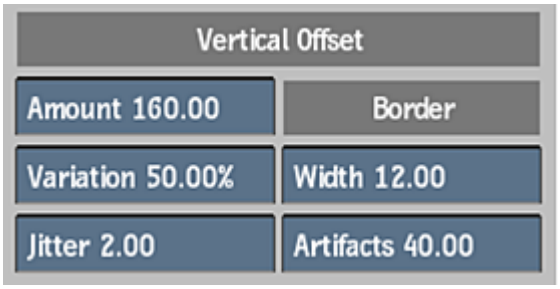
Play Effects Spacing Variation field Displays the percentage of variation in spacing between play effect errors. Editable.

Play Effects Duration field Displays the duration in frames of the play effect errors. Editable.

Play Effects Duration Variation field Displays the percentage of variation in duration of the play effects. Editable.

Vertical Offset Settings

Vertical Offset button Use this offset to simulate vertical rolls when playing back old tapes on VTRs.



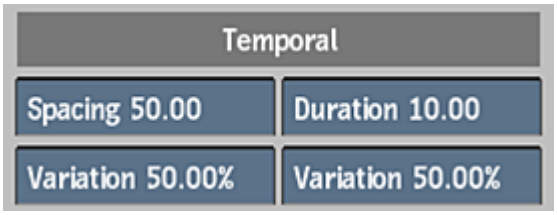
Vertical Offset Roll Amount field Displays the amount of vertical roll applied to the image. Editable.

Vertical Offset Roll Amount Variation field Displays the percentage of variation in the amount of the vertical roll. Editable.

Vertical Offset Jitter field Displays the amount of jitter applied to the vertical roll. Editable.

Vertical Offset Border Width field Displays the width of the vertical roll border. Editable.

Vertical Offset Border Artifacts field Displays the amount of artifacts within the vertical roll border. Editable.



Vertical Offset Spacing field Displays the spacing in frames between vertical roll errors. Editable.

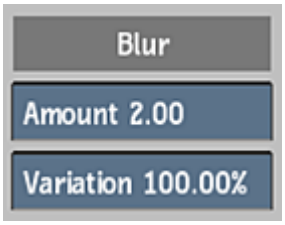
Vertical Offset Spacing Variation field Displays the percentage of variation in spacing between vertical roll errors. Editable.

Vertical Offset Duration field Displays the duration in frames of the vertical roll errors. Editable.

Vertical Offset Duration Variation field Displays the percentage of variation in duration for the vertical roll errors. Editable.

Blur Settings

Blur button Use this effect to simulate blur from poor signal reception.

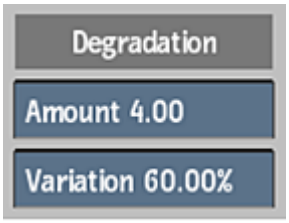


Blur Amount field Displays the amount of blurring added to the image. Editable.

Blur Amount Variation field Displays the percentage of variation in the amount of the blur effect. Editable.

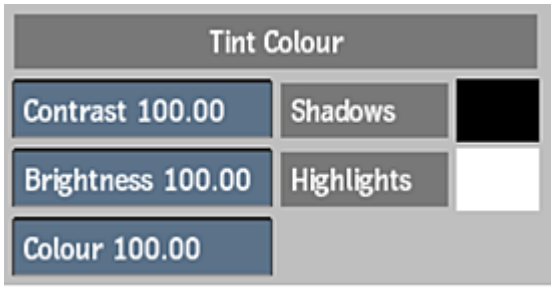
Colour Settings

Colour button Use this effect to simulate analog colour degradation, shift and variations from poor signal reception.



Analog Colour Degrade Amount field Displays the amount of colour degradation applied to the image. Editable.

Analog Colour Degrade Amount Variation field Displays the percentage of variation in the amount of the colour degradation. Editable.



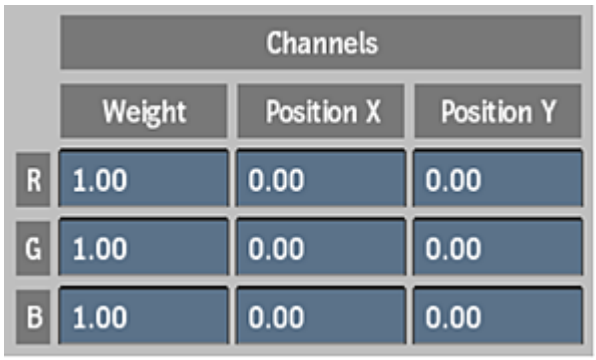
Analog Colour Contrast Transparency field Displays the amount of contrast applied to the image. Editable.

Analog Colour Brightness Transparency Variation field Displays the amount of brightness applied to the image. Editable.

Analog Colour Saturation field Displays the amount of colour saturation applied to the image. Editable.

Analog Colour Shadows colour pot Displays the tint applied to shadows in the image. Editable.

Analog Colour Highlights colour pot Displays the tint applied to highlights in the image. Editable.



Analog Colour Red Channel Gain field Displays the amount of gain applied to the red channel. Editable.

Analog Colour Green Channel Gain field Displays the amount of gain applied to the green channel. Editable.

Analog Colour Blue Channel Gain field Displays the amount of gain applied to the blue channel. Editable.

Analog Colour Red Channel X Shift field Displays the amount of horizontal offset applied to the red channel. Editable.

Analog Colour Green Channel X Shift field Displays the amount of horizontal offset applied to the green channel. Editable.

Analog Colour Blue Channel X Shift field Displays the amount of horizontal offset applied to the blue channel. Editable.

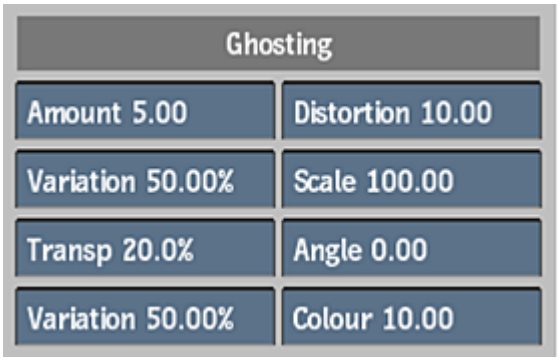
Analog Colour Red Channel Y Shift field Displays the amount of vertical offset applied to the red channel. Editable.

Analog Colour Green Channel Y Shift field Displays the amount of vertical offset applied to the green channel. Editable.

Analog Colour Blue Channel Y Shift field Displays the amount of vertical offset applied to the blue channel. Editable.

Ghosting Settings

Ghosting button Enable to simulate ghosting from poor signal reception.



Ghosting Amount field Displays the number of repeated ghost images applied to the original image. Editable.

Ghosting Amount Variation field Displays the percentage of variation in the number of ghost images. Editable.

Ghosting Transparency field Displays the percentage of the transparency applied to the ghosted images. Editable.

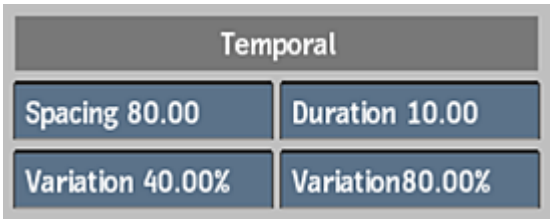
Ghosting Transparency Variation field Displays the percentage of variation in the transparency of the ghosted images. Editable.

Ghosting Distortion field Displays the amount of distortion applied to the ghosted images. Editable.

Ghosting Scale field Displays the amount of scaling applied to the ghosted images. Editable.

Ghosting Angle field Displays the angle applied to the ghosted images. Editable.

Ghosting Colour Degradation field Displays the amount of colour degradation applied to the ghosted images. Editable.



Ghosting Spacing field Displays the spacing in frames between ghosting errors. Editable.

Ghosting Spacing Variation field Displays the percentage of variation in spacing between ghosting errors. Editable.

Ghosting Duration field Displays the duration in frames of the ghosting errors. Editable.

Ghosting Duration Variaton field Displays the percentage of variation in the duration of ghosting errors. Editable.

Scanlines Settings

Scanlines button Use this effect to simulate scanlines from poor signal reception.

Scanlines Scale field Displays the size of the scanlines applied to the image. Editable.

Scanlines Variation field Displays the percentage of variation in the size of the scanlines. Editable.

Scanlines Variation Speed field Displays the speed at which the scanline variation occurs. Editable.

Scanlines Transparency field Displays the percentage of transparency applied to the scanlines.

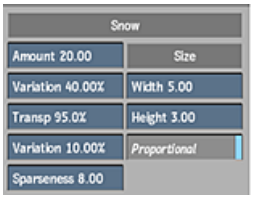
Scanlines Vertical Offset field Displays the vertical offset for the scanline effect. Editable.

Scanlines Softness field Displays the amount of softness applied to the scanlines. Editable.

Scanlines Angle field Displays the angle at which the scanlines occur. Editable.

Snow Settings

Snow button Enable to simulate snow from poor signal reception.



Snow Amount field Displays the amount of snow applied to the image. Editable.

Snow Amount Variation field Displays the percentage of variation in the amount of snow applied to the image.

Snow Transparency field Displays the percentage of transparency or softness, applied to the snow. Editable.

Snow Transparency Variation field Displays the percentage of variation in the transparency or softness, applied to the snow. Editable.

Snow Sparseness field Displays the density of the snow. Editable.

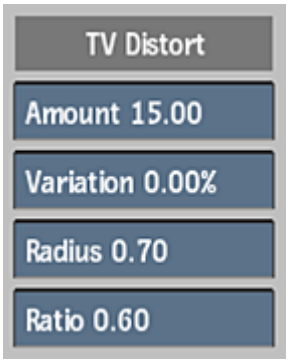
Snow Width field Displays the width in pixels of the snow particles. Editable.

Snow Height field Displays the height in pixels of the snow particles. Editable.

Snow Proportional button Enable to effect the width and height proportionally. Editable.

TV Distort Settings

TV Distort button Use this effect to simulate TV lens distortion from a poor video camera.



TV Distort Amount field Displays the amount of TV lens distortion applied to the image. Editable.

TV Distort Amount Variation field Displays the percentage of variation in the amount of TV lens distortion applied to the image.

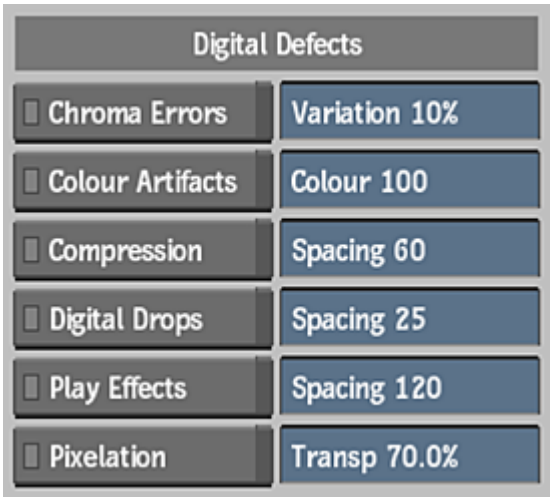
TV Distort Radius field Displays the size of the image to which the TV lens distortion is applied. A value of 1 includes the entire image. Editable.

TV Distort Ratio field Displays the ratio of height to width that is affected by the TV lens distortion effect. Editable.

Digital Video Settings

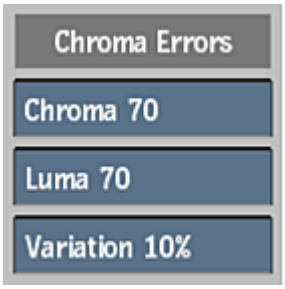
Use the Digital Video Damage effects to apply a large variety of digital video degradation effects to a clip.

When you select Digital Video from the Damage type box, the Digital Video Damage Effects are displayed, along with the most useful adjustment box to the right of each.



Chroma Errors Settings

Chroma Errors button Use this effect to simulate chroma errors in badly encoded digital video transmissions.



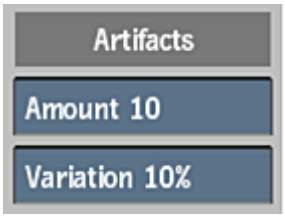
Chroma Amount box Displays the value representing the change to the chroma channel. Editable.

Luma Amount box Displays the value representing the change to the luminance channel. Editable.

Variation Amount box Displays the percentage of variation applied to the Chroma and Luma channels.

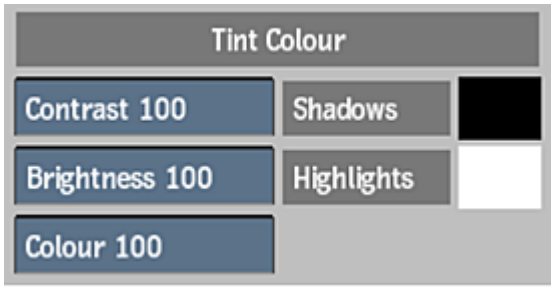
Colour Artifacts Settings

Colour Artifacts button Use this effect to simulate digital colour artifacts and degradation appearing in badly encoded digital video transmissions.



Colour Degradation Amount box Displays the amount of digital colour degradation applied to the image. Editable.

Colour Degradation Variation box Displays the percentage of variation in the amount of colour degradation applied to the image. Editable.



Contrast box Displays the amount of contrast applied to the image. Editable.

Brightness box Displays the amount of brightness applied to the image. Editable.

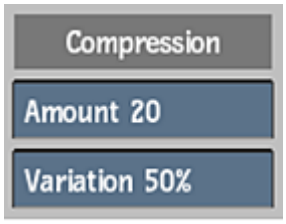
Tint Colour box Displays the tint value applied to the colour in the image. Editable.

Shadows colour pot Displays the tint applied to shadows in the image. Editable.

Highlights colour pot Displays the tint applied to highlights in the image. Editable.

Compression Settings

Compression button Use this effect to simulate digital compression errors in badly encoded digital video transmissions.



Compression Amount field Displays the amount of the digital compression effect applied to the image. Editable.

Compression Amount Variation field Displays the percentage of variation in the amount of digital compression applied to the image. Editable.

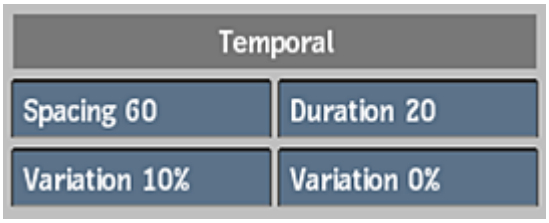
Compression Edges Active button Enable to activate edge detection errors in compression.

Compression Edges Minimum field Displays the minimum threshold value used to detect edges. Available when Active button is enabled. Editable.

Compression Edges Maximum field Displays the maximum threshold value used to detect edges. Available when Active button is enabled. Editable.

Compression Edges Width field Displays the width of the edge in pixels. Available when Active button is enabled. Editable.

Compression Edges Softness field Displays the softness of the edge. Available when Active button is enabled. Editable.



Compression Spacing field Displays the spacing in frames between digital compression errors. Editable.

Compression Spacing Variation field Displays the percentage of variation in spacing between compression errors. Editable.

Compression Duration field Displays the duration in frames of the compression errors. Editable.

Compression Duration Variation field Displays the percentage of variation in duration of the compression errors. Editable.

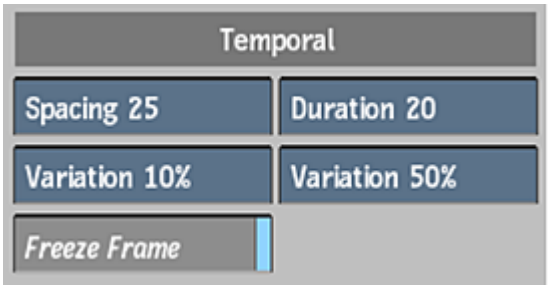
Digital Drops Settings

Digital Drops button Use this effect to simulate digital drop errors in badly encoded digital video transmissions.

Digital Drops Amount field Displays the amount of digital drop errors applied to the image. Editable.

Digital Drops Coherence field Displays the amount of coherence to digital drop errors in the image. Editable.

Digital Drops Amount Variation field Displays the percentage of variation in the amount of digital drop errors applied to the image. Editable.



Digital Drops Spacing field Displays the spacing in frames between digital drop errors. Editable.

Digital Drops Spacing Variation field Displays the percentage of variation in spacing between digital drop errors. Editable.

Digital Drops Freeze Frame button Displays the number of repeat frames added to create a jerky digital drop effect. Editable.

Digital Drops Duration field Displays the duration in frames of the digital drop errors. Editable.

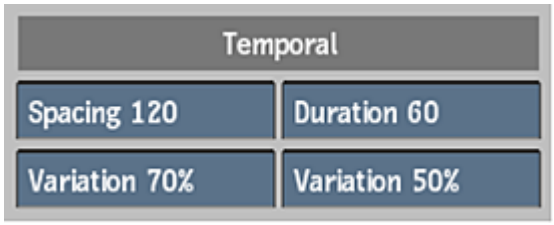
Digital Drops Duration Variation field Displays the percentage of variation in duration for the digital drop errors. Editable.

Play Effect Settings

Play Effects button Use this effect to simulate digital play effects in badly encoded digital video transmissions.

Play Effects Speed field Displays the amount of change to the speed of playback applied to the image. Editable.

Play Effects Shuttering field Displays the amount of shuttering during playback that is applied to the image. Editable.



Play Effects Spacing field Displays the spacing in frames between play effects errors. Editable.

Play Effects Spacing Variation field Displays the percentage of variation in spacing between play effects errors. Editable.

Play Effects Duration field Displays the duration in frames of the play effects errors. Editable.

Play Effects Duration Variation field Displays the percentage of variation in the duration for play effects errors. Editable.

Pixelation Settings

Pixelation button Use this effect to simulate reduced resolution and digital pixelation errors in badly encoded digital video transmissions.

Pixelation Width Amount field Displays the width in pixels of the new pixelation block. Editable.

Pixelation Height Amount field Displays the height in pixels of the new pixelation block. Editable.

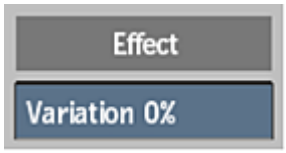
Pixelation Proportional button Enable to effect the width and height proportionally. Editable.

Pixelation Average button Displays the amount of averaging applied between the pixel block and its surroundings. Editable.

Pixelation Transparency field Displays the percentage of transparency, or softness, applied to the pixel blocks. Editable.

Pixelation X Offset field Displays the horizontal offset for the pixelation effect. Editable.

Pixelation Y Offset field Displays the vertical offset for the pixelation effect. Editable.



Pixelation Size Variation field Displays the percentage of variation in the size of pixels during playback. Editable.

Deal

Use Deal to deal out the frames of a single clip evenly to any number of destination clips.

To access the Deal menu, use:

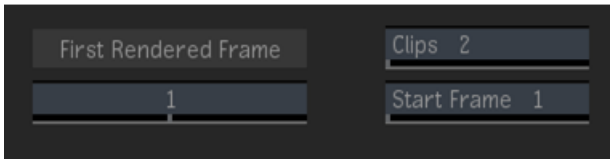
- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip, and outputs a result.

The Clip field defines the number of times the clip is split; the Frame determines which of the dealt clips is output. The First Processed Frame field set the value at which output is processed from the node. Unprocessed output does not display any media.

Deal Menu Settings

General Settings



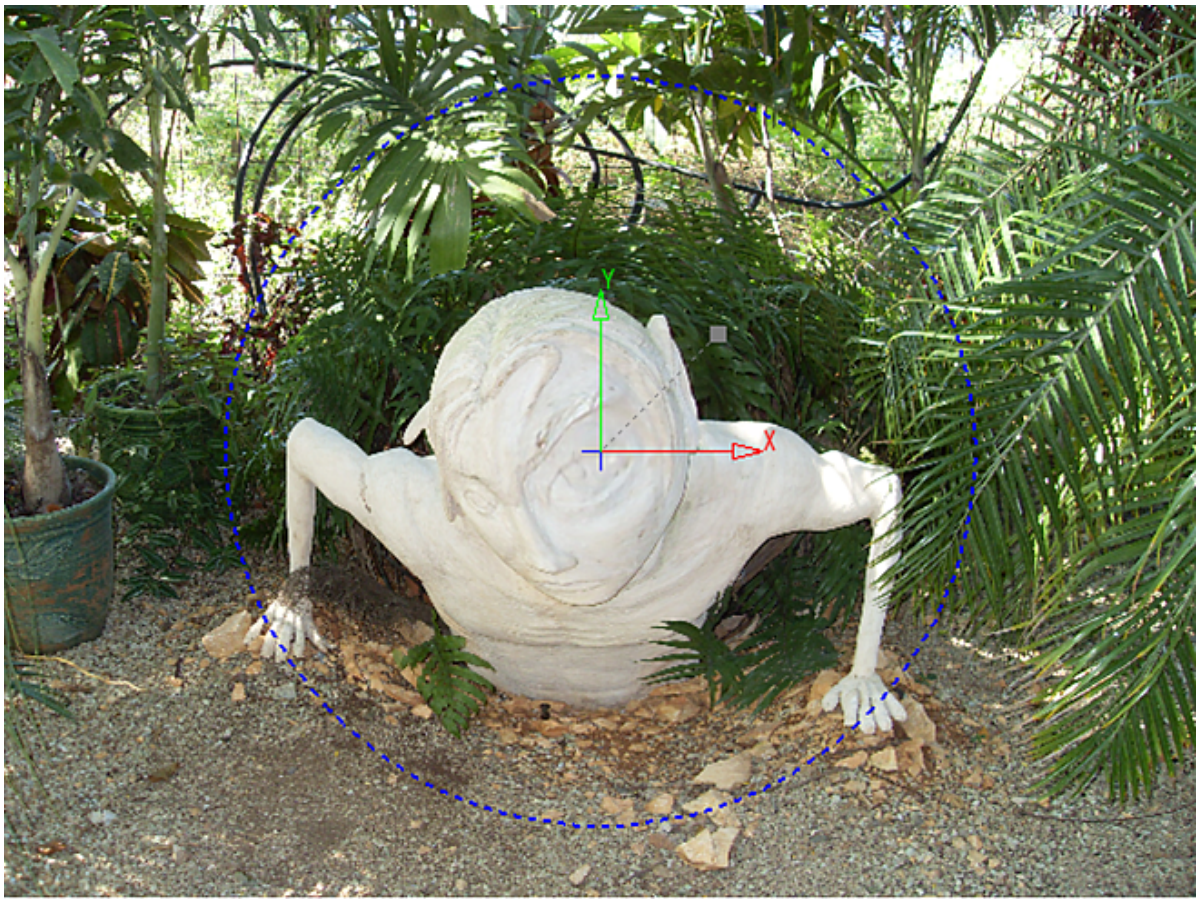
First Rendered Frame field Displays the first frame at which output is rendered from the node. Unrendered output does not display any media.Editable.

Clips field Displays the number of times the clip is split. Editable.

Start Frame field Displays which frame of the input clip used to determine which dealt clip is output. Editable.

Deform

Use Deform to apply various types of deformation effects to clips.



To access the Deform menu, use:

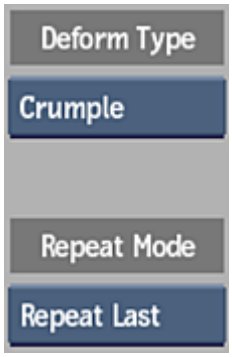
- Batch, then select a node from the Node bin.

- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front and a matte clip, and outputs a result and an outmatte.

Deform Menu Settings

General Settings



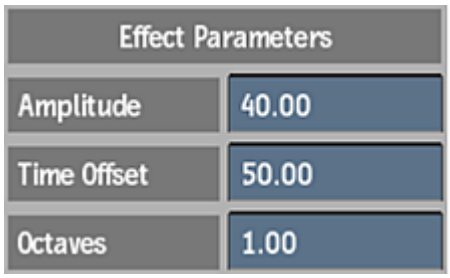
Deform Type button Select the type of deformation you want to create. The available effect parameters change based on the selection.

Deform Ripple box (not shown) Select how you want the ripples created. Pond Ripples create ripples with a 45 degree offset. Out from Center pushes the ripples away from the center, extending the first half, and compressing the second half of each ripple. Around Center rotates the crest of each ripple in a clockwise direction around the center.

Repeat Mode box Select an option to fill the empty portions of the frame.

Effect Parameters

NOTE The following settings are available when Crumple is selected from the Deform Type button.

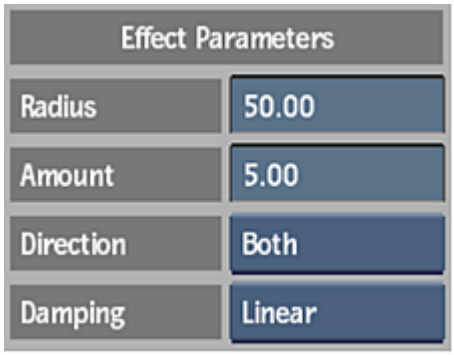


Amplitude field Displays the amount of deformation. Increase the value to increase the effect. Editable.

Time Offset field Displays the time offset interval of the crumpling. Editable.

Octaves field Displays a value for the number of layers summed in the operation, from zero to 10. Increase the value to increase the fractal effect. Editable.

NOTE The following settings are available when Magnify is selected from the Deform Type button.



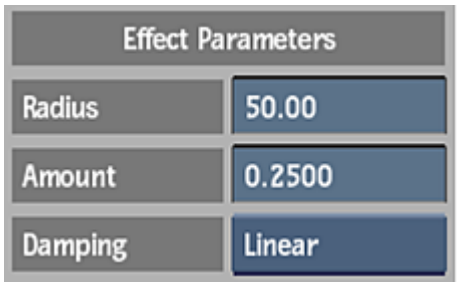
Radius field Displays the size of the affected area. You can also drag the circle in the image window to change the size of the radius. Editable.

Amount field Displays the amount of the deformation. Editable.

Direction box Select Horizontal, Vertical, or Both to indicate the direction of the magnification.

Damping box Select how the effect is applied within the radius. Select None to apply no damping, Linear to decrease the effect linearly to 0, or Quadratic to decrease the effect quadratically to 0.

NOTE The following settings are available when Pinch is selected from the Deform Type button.

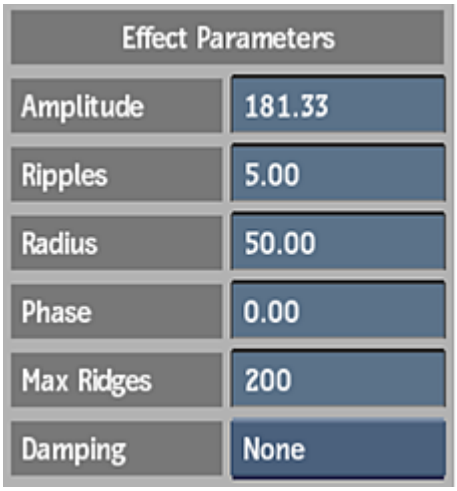


Radius field Displays the size of the affected area. You can also drag the circle in the image window to change the size of the radius. Editable.

Amount field Displays the amount of the deformation. Editable.

Damping box Select how the effect is applied within the radius. Select None to apply no damping, Linear to decrease the effect linearly to 0, or Quadratic to decrease the effect quadratically to 0.

NOTE The following settings are available when Ripple is selected from the Deform Type button.



Amplitude field Displays the amount of deformation. Increase the value to increase the effect. Editable.

Ripples field Displays the frequency of the ripples (from zero to 60). Editable.

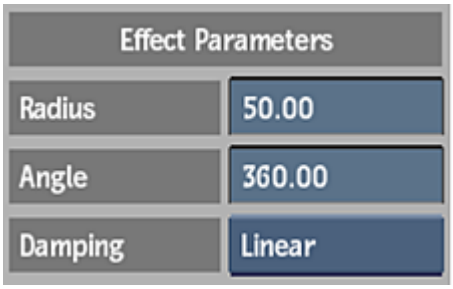
Radius field Displays the size of the affected area. You can also drag the circle in the image window to change the size of the radius. Editable.

Phase field Displays the spread of the ripples to the centre. Use this value to animate the ripple effect. Editable.

Max Ridges field Displays the total number of ripples that can be generated. For example, set to 5 to create 5 rings of distortion. Editable.

Damping box Select how the effect is applied within the radius. Select None to apply no damping, Linear to decrease the effect linearly to 0, or Quadratic to decrease the effect quadratically to 0.

NOTE The following settings are available when Twirl is selected from the Deform Type button.

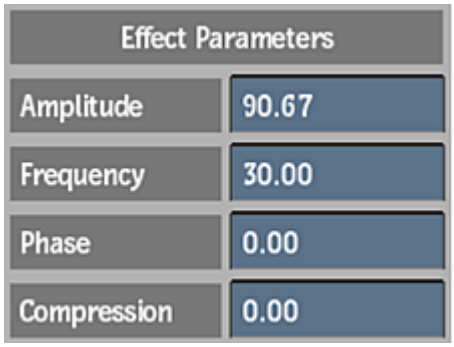


Radius field Displays the size of the affected area. You can also drag the circle in the image window to change the size of the radius. Editable.

Twirl Angle field Displays the direction of the twirl. Editable.

Damping box Select how the effect is applied within the radius. Select None to apply no damping, Linear to decrease the effect linearly to 0, or Quadratic to decrease the effect quadratically to 0.

NOTE The following settings are available when Wave is selected from the Deform Type button.



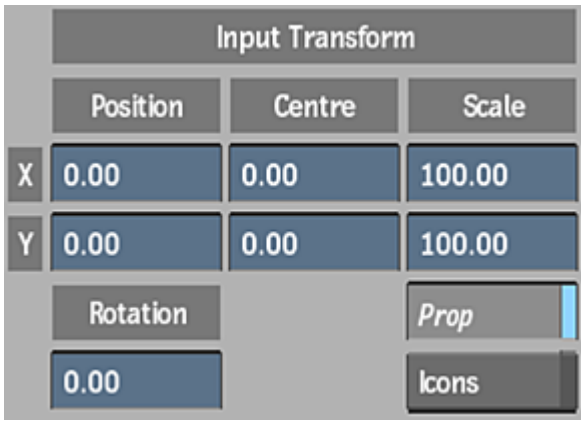
Amplitude field Displays the amount of deformation. Increase the value to increase the effect. Editable.

Wave Frequency field Displays the number of waves. Editable.

Phase field Displays the spread of the ripples to the centre. Use this value to animate the ripple effect. Editable.

Compression field Displays a pixel flattening value (negative values flatten to the left, and positive values flatten to the right). Editable.

Transform Settings



NOTE Transform settings for Input, Deform and Output are identical.

Transform X Position field Displays the horizontal position of the transformation. Enable the Icons button to change the position by dragging the vertex tool in the image window. Editable.

Transform Y Position field Displays the vertical position of the transformation. Editable. Enable the Icons button to change the position by dragging the vertex tool in the image window. Editable.

Transform Rotation field Displays the rotation of the transformation. Editable. Enable the Icons button to change the rotation by dragging the vertex tool in the image window. Editable.

Transform Centre X field Displays the centre point value of the transform along the horizontal axis. Editable.

Transform Centre Y field Displays the centre point value of the transform along the vertical axis. Editable.

Transform X Scale field Displays the horizontal scale factor. Editable.

Transform Y Scale field Displays the vertical scale factor. Editable.

Proportional button Enable to scale X and Y values proportionally.

Icons button Enable to display the vertex editing tool in the image window.

Anti-Aliasing Settings

Active button Enable to activate software anti-aliasing.

Anti-Aliasing Sampling box Select the number of samples to use in the anti-aliasing process.

Anti-Aliasing Softness field Displays the softness value for software anti-aliasing. Editable.

Degrain

Use Degrain to remove grain from the RGB channels of a selected colour in an image.

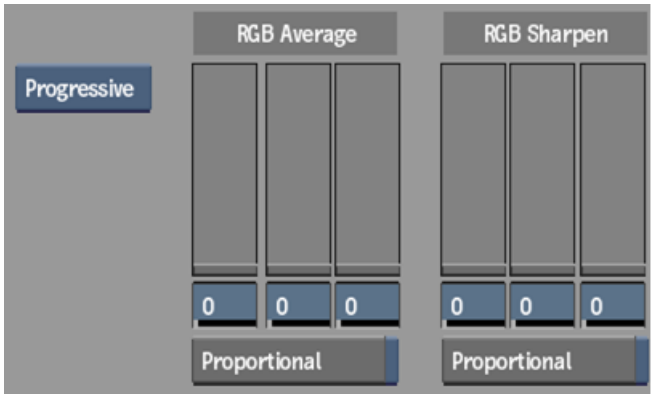
To access the Degrain menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Degrain Menu Settings

General Settings



Render Method box Select whether you are removing grain from progressive or interlaced media.

Red Average slider Displays the amount of blur applied to the red channel.

Red Average field Displays the amount of blur applied to the red channel. Editable.

Green Average slider Displays the amount of blur applied to the green channel.

Green Average field Displays the amount of blur applied to the green channel. Editable.

Blue Average slider Displays the amount of blur applied to the blue channel.

Blue Average field Displays the amount of blur applied to the blue channel. Editable.

Proportional RGB Average button Enable to blur RGB channels proportionally.

Red Sharpen slider Displays the amount of sharpness applied to the red channel.

Red Sharpen field Displays the amount of sharpness applied to the red channel. Editable.

Green Sharpen slider Displays the amount of sharpness applied to the green channel.

Green Sharpen field Displays the amount of sharpness applied to the green channel. Editable.

Blue Sharpen slider Displays the amount of sharpness applied to the blue channel.

Blue Sharpen field Displays the amount of sharpness applied to the blue channel. Editable.

Proportional RGB Sharpen button Enable to sharpen RGB channels proportionally.

Deinterlace

Use Deinterlace to separate the odd and even scanlines of a clip.

To access the Deinterlace menu, use:

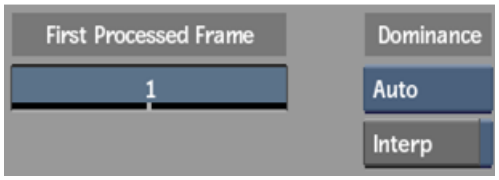
- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip as input, and outputs a result.

For each frame of the clip, the result clip contains one frame with odd scanlines (Field 1) and one frame with even scanlines (Field 2).

Deinterlace Menu Settings

General Settings



First Rendered Frame field Displays the first frame at which output is rendered from the node. Unrendered output does not display any media.Editable.

Field Dominance box Select Field 1 or Field 2 dominance, or Auto to have the application detect automatically the field dominance.

Interpolation box Enable to interpolate a blend between adjacent lines and fill in the isolated scan lines. This also reduces interlacing artifacts.

Denoise

Use the Denoise effect to reduce or remove noise and grain from your source media.

To access the Denoise menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result clip.

Denoise Menu Settings

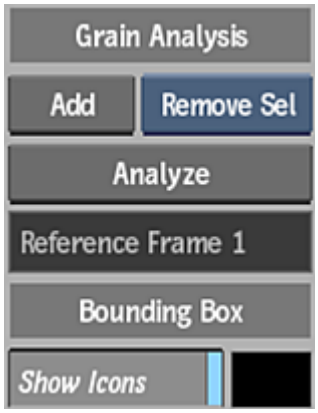
Setup Settings

Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Export button Click to export a Denoise setup to be used as a Lustre Degrain preset.

Import button Click to open the browser to select a Lustre Degrain preset to import.

Grain Analysis Settings



Add Bounding Box button Adds a new sub-region. Click and drag the bounding box in the image window to set the location.

Remove Bounding Box box Select an option to remove the currently selected bounding box or all the bounding boxes.

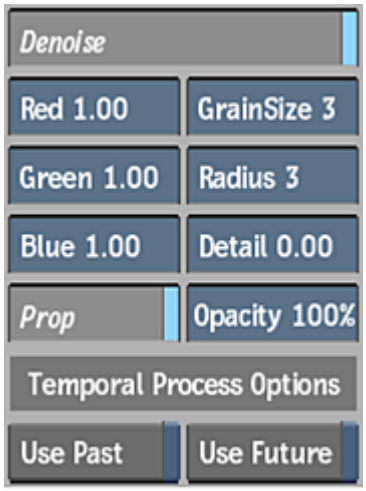
Analyze button Click to determine the grain structure.

Reference frame field Displays the frame number that is analysed. You can use this frame analysis as a reference that can be edited and applied to the clip.

Show Icons button Enable to display all the defined bounding boxes.

Bounding Box colour pot Displays the colour used for the border of the bounding boxes. Click to change the colour.Editable.

Denoise Settings



Denoise button Enable to use noise removal on the clip.

Red Gain field Displays the gain for red channel colour values. Editable.

Green Gain field Displays the gain for green channel colour values. Editable.

Blue Gain field Displays the gain for blue channel colour values. For film scans, the grain is often greater in this channel. Editable.

Proportional button Enable to change gain values proportionally for all three colour channels.

Grain Size field Displays a value in pixels that is proportional to the size of the grain. The default value is 3, but may be higher for 4K images. Editable.

Smoothing Radius field Displays the blur radius. For smoother results, a higher value will add more pixels to the blur, but increase rendering time. Editable.

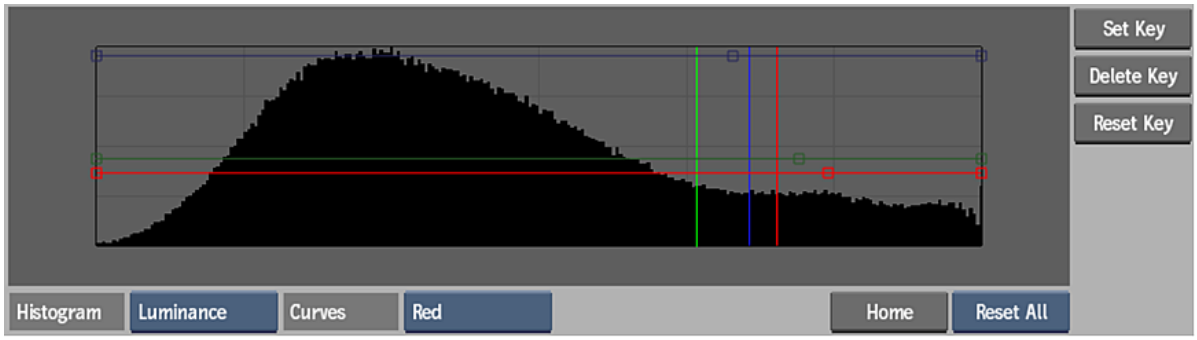
Detail field Displays the amount of detail to preserve when the Smoothing Radius is set to a high value. It is recommended you enter 0.05 to 0.15 as an initial value. Editable.

Opacity field Displays a percentage of the level of opacity between the source image and the output with the applied grain filter. Editable.

Use Past button Enable to compare with pixel data from previous frames.

Use Future button Enable to compare with pixel data from subsequent frames.

Histogram Settings



Histogram box Select to display the red, green, blue, or luminance histogram in the graph. Select Current Curve to display the histogram for the currently selected Curves Channel.

Curves Channel box Select to highlight the red, green, or blue channel curve in the graph.

Home button Reverts to the original view.

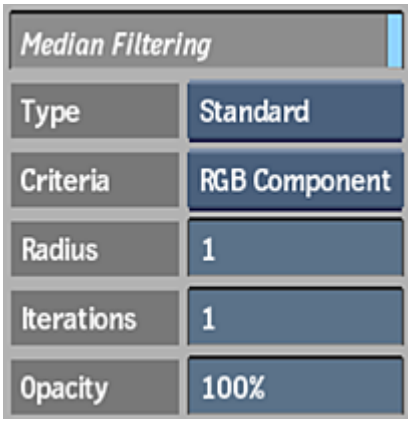
Reset box Resets the curve view.

Set Key button Sets the current values for the selected channels in the current frame.

Delete Key button Deletes the selected keyframe or curve.

Reset Key button Select to reset the current curve or all the curves to default.

Median Filtering Settings



The Median Filter removes noise by calculating the median value for each pixel (the most probable pixel value) and applies an edge-preserving smoothing filter.

Median Filtering button Enable to use the median filter.

Type box Select the filtering mode. Options are:

- Standard: Applies the standard fitering algorithm.
- Advanced: Applies a more advanced filtering algorithm, but is more resource intensive. Use the Advanced Filter Mode on particularily noisy shots.

Criteria box Select the ranking criteria by which the median value is chosen. Options are:

- Luminance :Uses the pixel luminance value to calculate the median value.
- RGB Vector: Uses the RGB coordinate values to calculate the median value.
- RGB Component: Uses the R, G and B channels independently to calculate the median value.

Radius field Displays the size of the filtering region in pixels from the centre. A higher value results in more pixels being taken into account when calculating the median value. Editable.

Iterations field Displays the number of times that the filter is applied recursively. Editable.

Opacity field Displays how much the result of one median iteration is combined with the original input. Editable.

Depth of Field

Use Depth of Field to create a blur that is applied to out-of-focus points of light to simulate a shallow focus.

To access the Depth of Field menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front, Z-depth and matte clip, and outputs a result, out Z-depth and outmatte clip. The output matte clip can have a different level of depth of field than the result clip.

Depth of Field Menu Settings

Blur and Blooming Settings

Blur Width field Displays the horizontal blur amount in pixels. Editable.

Blur Height field Displays the vertical blur amount in pixels. Editable.

Blur Proportional button Enable to constrain blur amount proportions.

Bokeh Blur field Displays the amount of smoothness applied to sharp bokeh edges. This creates the blur that is applied to out-of-focus points of light to simulate a shallow focus. Editable.

Basic/Additive Blooming button Switch between basic blooming and additive blooming bokeh effects. The basic blooming mode displays the gain applied to highlights in non-HDR images, to allow the creation of bokeh patterns without affecting colour integrity. The additive blooming mode allows you to create higher intensity bokeh patterns from any source image, using minimum and maximum thresholds for highlight segregation.

Basic Blooming field Displays the gain applied to highlights in non-HDR images, to allow the creation of bokeh patterns without affecting colour integrity. This creates the glow effect that is applied to the bright spots of the image to simulate light bleeding, or blooming, over the edges. Editable.

Additive Blooming field Displays the amount of high intensity bokeh patterns that can be created from any source image generating extreme highlight content . Editable.

Additive Blooming minimum field Displays the minimum threshold for highlight segregation. Editable.

Additive Blooming maximum field Displays the maximum threshold for highlight segregation. Editable.

Bokeh Settings

Sides field Displays the number of sides in the kernel shape. Editable, if kernel information is not attached to the node.

Rotation field Displays the angle of rotation of the kernel shape. Editable, if kernel information is not attached to the node.

Bokeh Type box Select whether to use a round or angle bokeh curve to define the kernel shape.

Curvature field Displays the amount of curvature applied to a round bokeh kernel shape. Editable.

Slices and Edge Artifacts Settings

Slices Number field Displays the number of slices to blend to determine the Z-Depth interpolation of a depth-of-field focus blur. You can see the slices in the depth-o-gram to allow a better understanding of the focus plane behaviour, and to help define the number of slices required, depending on your Z-depth colour information. Editable.

Slices Overlap field Displays the amount of blending overlap between adjacent slices. Editable.

Foreground Expand field Displays the amount of mixing between the edges of selected objects and their surroundings in the foreground. Editable.

Background Blend field Displays the amount of mixing between the edges of selected objects and the background image. Editable.

Z-Depth Settings

Z-Depth Minimum field Displays the lower limit of the Z-Depth values. Pixels with lower values are mapped to black. Editable.

Z-Depth Z Maximum field Displays the upper limit of the Z-Depth values. Pixels with higher values are mapped to white. Editable.

White Value box Select whether white pixels represent the furthest point or nearest point on the Z-axis.

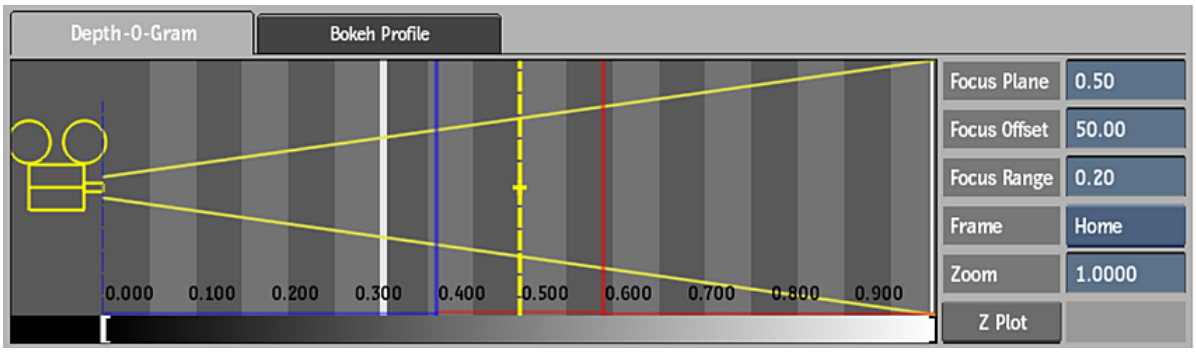
Premultiplied Depth button Enable to have the alpha channel information premultiplied with the colour depth data. Use to prevent a black halo from forming around blurred areas.

Max Blur Field Displays the maximum total horizontal and vertical blur amount to clamp the value and avoid unnecessary rendering. The cumulative effect of the high blur amount (Basic tab), and gamma and gain correction applied to the Depth map (Depth tab) may result in very large values, which greatly increases the rendering time. Editable.

Slopes field Displays the gamma value. Applies a gamma curve to the Z-depth map before it is used. Editable.

Depth-O-Gram Settings

Use the Depth-O-Gram settings to refine focus values.



Depth blur effects control the amount of blur based on a depth of field matte. Depth blur settings are displayed in the Depth-O-Gram tab.

A depth of field map (Z-depth map) can be connected to the node. Black portions of the map are in focus. White portions display the highest level of blur. Note that a Z-depth map imported from another application may use the opposite convention and may need to be inverted.

Focus Plane field Displays the distance of your focus point, which is the point at which there is no blur on the image. Editable.

Focus Offset field Displays the distance between the focus plane and the near offset represented as a percentage of the total offset range. Select 50% to make the near and far offsets equidistant from the focus point. Editable.

Focus Range field Displays the distance the near and far offset. Editble.

Frame option box Select how you want to frame the histogram.

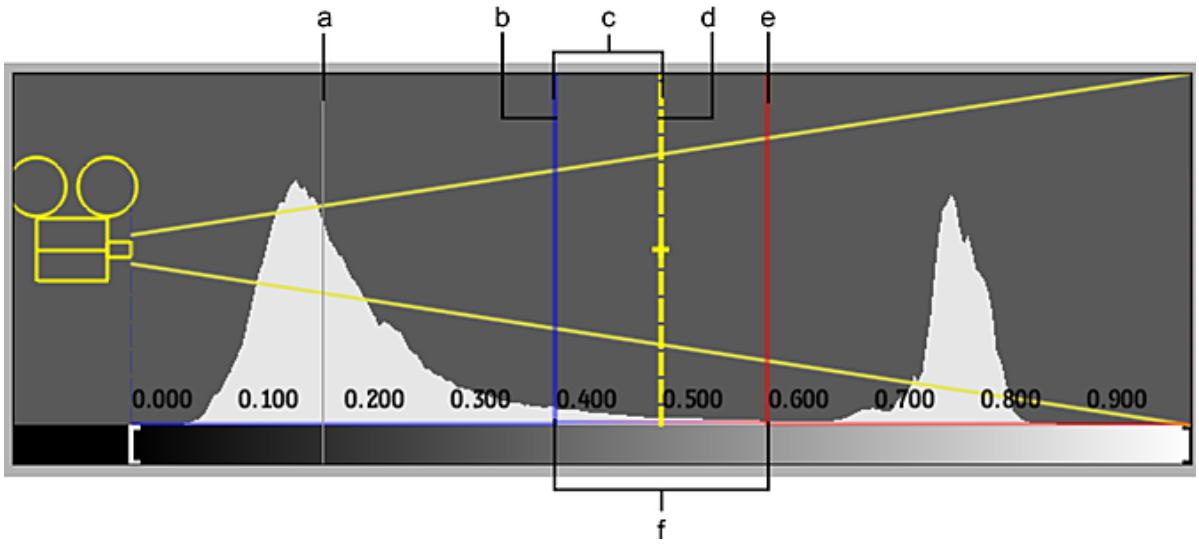
Zoom field Displays a vertical zoom value for the histogram to display. You can zoom horizontally by pressing Ctrl+spacebar and dragging left or right in the histogram. To pan horizontally, click spacebar and drag left or right in the histogram.Editable.

Z Plot button Click to activate the pick cursor. Use to select a pixel in the image to display its depth.

Z Plot Colour pot Displays the colour that indicates the plane on which the plotted value is located.

Modifying Depth of Field Gesturally

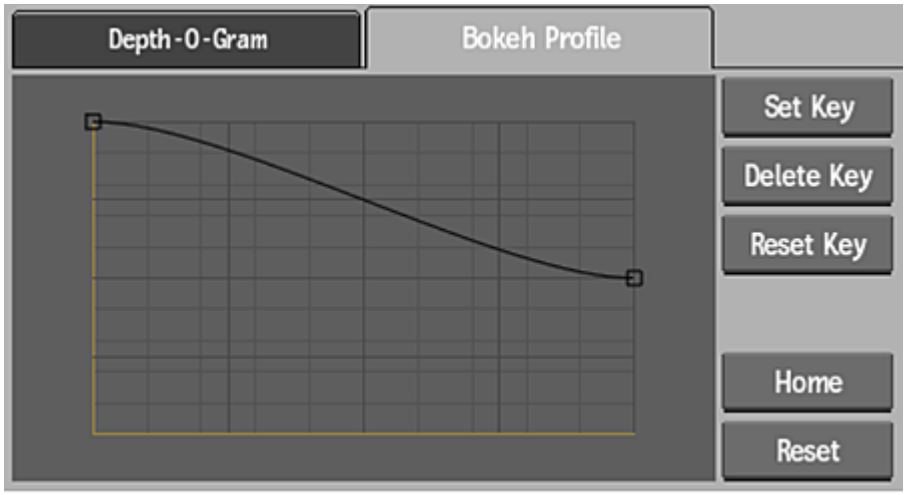
Depth blur effects can be modified gesturally by dragging the focus, near offset, and far offset planes in the graphic representation of the depth of field. As you drag these elements, the planes are also displayed in the Result view as a preview of the areas that will be in focus. Use the depth control fields to change the gamma and gain, and to change the focus range while keeping the focus plane constant. These parameters are updated in the depth of field display automatically.



(a) Plot Value Plane (b) Near Focus Offset Plane (c) Focus Offset (d) Focus Plane (e) Far Focus Offset Plane (f) Focus Range

Bokeh Profile Settings

Use the Bokeh Profile tab to edit the blur kernal pattern.



The kernel is the basic blur shape. The shape of the kernel is determined by its number of sides, its rotation, and the shape of its S-curve. This curve represents the shape of the pattern, from its centre to the outside. The default S-curve defines the softness of the blur. You can change the curve by manipulating the two points that define the curve, or you can add points to the curve. Use the Tools box to add and delete points on the curve.

Set Key button Sets a keyframe at the selected frame.

Delete Key button Deletes the selected keyframe.

Reset Key button Resets the curve at the selected keyframe.

Home button Resets the profile view.

Reset button Resets the blur pattern profile curve to its default values.

Difference Matte

Use the Difference Matte to generate a matte clip from two source clips with the same background but different foreground elements.

To access the Difference Matte menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front and back clip as input, and outputs a result.

The Difference Matte allows you to remove an image from one context and add it to another. The matte is created using the Tolerance and Softness values that you specify.

Difference Matte Menu Settings

General Settings

You can enable the RGB or the YUV settings to generate a matte clip.

RGB Settings

RGB button Enable to use values in the RGB colour space to generate a matte.

R button Enable to use the red colour channel to create the matte.

G button Enable to use the green colour channel to create the matte.

B button Enable to use the blue colour channel to create the matte.

Red Tolerance colour pot Displays the value of the red channel used to create the matte. Editable.

Red Tolerance field Displays the value of the red channel used to create the matte. Editable.

Green Tolerance colour pot Displays the value of the green channel used to create the matte. Editable.

Green Tolerance field Displays the value of the green channel used to create the matte. Editable.

Blue Tolerance colour pot Displays the value of the blue channel used to create the matte. Editable.

Blue Tolerance field Displays the value of the blue channel used to create the matte. Editable.

Red Softness colour pot Displays the softness value applied to the red channel to create the matte. Editable.

Red Softness field Displays the softness value applied to the red channel to create the matte. Editable.

Green Softness colour pot Displays the softness value applied to the green channel to create the matte. Editable.

Green Softness field Displays the softness value applied to the green channel to create the matte. Editable.

Blue Softness colour pot Displays the softness value applied to the blue channel to create the matte. Editable.

Blue Softness field Displays the softness value applied to the blue channel to create the matte. Editable.

Gain field Displays the value that the resulting pixel values are multiplied by to create the final matte. Editable.

Lift field Displays the value added to the resulting pixels to create the final matte. Editable.

YUV Settings

YUV button Enable to use values in the YUV colour space to generate a matte.

Y button Enable to use the Y channel (luminance) to create the matte.

U button Enable to use the U channel (chrominance) to create the matte.

V button Use the V channel (chrominance) to create the matte.

Y Tolerance colour pot Displays the value of the Y channel used to create the matte. Editable.

Y Tolerance field Displays the value of the Y channel used to create the matte. Editable.

U Tolerance colour pot Displays the value of the green channel used to create the matte. Editable.

U Tolerance field Displays the value of the U channel used to create the matte. Editable.

V Tolerance colour pot Displays the value of the V channel used to create the matte. Editable.

V Tolerance field Displays the value of the V channel used to create the matte. Editable.

Y Softness colour pot Displays the softness value applied to the Y channel to create the matte. Editable.

Y Softness field Displays the softness value applied to the Y channel to create the matte. Editable.

U Softness colour pot Displays the softness value applied to the U channel to create the matte. Editable.

U Softness field Displays the softness value applied to the U channel to create the matte. Editable.

V Softness colour pot Displays the softness value applied to the V channel to create the matte. Editable.

V Softness field Displays the softness value applied to the V channel to create the matte. Editable.

Gain field Displays the value that the resulting pixel values are multiplied by to create the final matte. Editable.

Lift field Displays the value added to the resulting pixels to create the final matte. Editable.

Distort

Use Distort to create warps and morphs of clips using spline-based shapes.

To access Distort, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).

■ Tools, then select from the menu (See Accessing Tools (page 493)).

Distort accepts two front clips, two matte clips and one back clip as inputs, and outputs a result and an outmatte.

Distort Menu Settings

Distort Settings

Input1 Display box Select a display mode for Input1.

Input2 Display box Select a display mode for Input2.

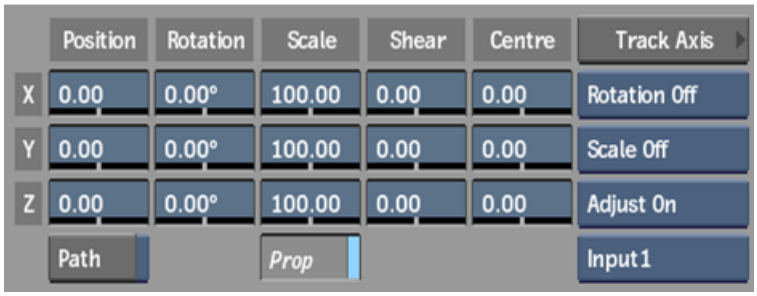
Matte1 Display box Select a display mode for Matte1.

Matte2 Display box Select a display mode for Matte2.

Back Display box Select a display mode for the background clip.

Input box Toggles between Input1 and Input2 clips based on the selection in the View box.

If Input is selected in the View box, this box toggles between Front1 and Front2. If Matte is selected in the View box, this box toggles between Matte1 and Matte2. If Result is selected in the View box, this box toggles between showing the splines for Front1 and Front2 drawn on top of the result.



X Position field Displays the position of the X axis. Editable.

Y Position field Displays the position of the Y axis. Editable.

Z Position field Displays the position of the Z axis. Editable.

Motion Path button Enable to animate the position of the axis using a spline drawn in the scene.

X Rotation field Displays the rotation value of the X axis. Editable.

Y Rotation field Displays the rotation value of the Y axis. Editable.

Z Rotation field Displays the rotation value of the Z axis.

X Scale field Displays the scale value of the X axis. Editable.

Y Scale field Displays the scale value of the Y axis. Editable.

Z Scale field Displays the scale value of the Z axis. Editable.

Prop Scale button Enable to scale the X, Y, and Z axes proportionally.

X Shear field Displays the shear value of the X axis. Editable.

Y Shear field Displays the shear value of the Y axis. Editable.

Z Shear field Displays the shear value of the Z axis. Editable.

X Centre field Displays the centre value of the X axis. Editable.

Y Centre field Displays the centre value of the Y axis. Editable.

Z Centre field Displays the centre value of the Z axis. Editable.

Track Axis button Opens the Stabilizer menu to apply tracking data to an axis.

Tracking Rotation box Select whether tracking rotation is On, Off, or Inverted.

Tracking Scale box Select whether tracking scaling is On, Off, or Inverted.

Tracking Adjust box Select whether an offset is applied relative to the spline's axis.

Track Clip box Select the clip to track.

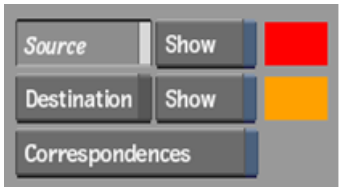
Warper button Select whether to display the Warp or Morph menu.

Interpolation field Displays the percentage of mixing between the Input1 spline and the Input2 spline when source splines are linked for source interpolation. Editable.

You independently set an interpolation value for each set of linked Front1 and Front2 source splines.

Blend field Displays the level of blend between the Input1 and Input2 clips when morphing. Editable.

NOTE The Blend field is only available from the Morph menu.



Source button Enable to display source splines for manipulation. Also use to display the unwarped input clip.

Source Show button Enable to display source splines for reference when working with destination splines.

Source colour pot Displays the colour of all source splines. Editable.

Destination button Enable to display destination splines for manipulation. Also use to display the warped input clip.

Destination Show button Enable to display destination splines for reference when working with source splines.

Destination colour pot Displays the colour of all destination splines. Editable.

Correspondences button Enable to display correspondence points and connector lines, which indicate how the source spline maps to the destination spline.

You can add, edit, animate, and delete correspondence points. The more correspondence points a spline has, the greater effect it has on the overall warp.

Active button Enable to allow greater control over an animated spline by toggling selected vertices on and off.

Lasso Fit field Displays the number of vertices in freehand segments of a spline. Editable.

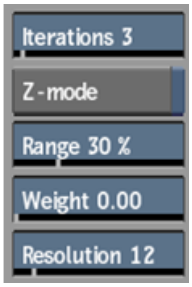
Increasing the value decreases the number of vertices, while decreasing the value increases the number of vertices. Only segments of the spline drawn using freehand mode are affected, while segments created by simple clicks remain unaffected. The Lasso Fit parameter loses its influence over freehand segments of a spline if you edit vertices.

Link box Controls whether source and destination splines are linked together or manipulated independently.

Set this to Lnk Src & Dst to keep the source and destination splines linked as you draw and animate the source spline. Set this to Enable Warping to control the destination spline independently to create a warp. You can also independently link and unlink the axis nodes that are the parents of a source spline and its corresponding destination spline.

Toggle Input button Switch between an Input1 spline and an Input2 spline.

This is useful when you have animated a spline on one input and wish to apply it to the other.



Iterations field Displays a value for the distance that the distortion follows the destination spline. Editable.

If Z-mode is disabled, the maximum distance of the distortion is limited to avoid overlapping. When there is a large distance between a source and destination spline, you will achieve better results by increasing the number of iterations. For smaller warps or morphs, there is no advantage in using a greater number of iterations—it will increase processing time unnecessarily.

Z-mode button Enable to allow overlapping in the image to create 3D-like effects where parts of the image are pulled over other parts.

NOTE When Z-mode is enabled, the Iterations field is disabled.

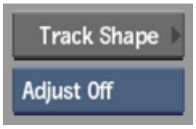
Range field Displays the size of the region affected by the warping effect. Editable.

The greater the value, the larger the portion of the image affected. A value of 100% affects the entire image, a value of 33% affects one-third of the image, for example.

Weight field Displays the relative weight on the selected spline, to control its influence. Editable.

For example, you can set a higher weight on a spline in an area of the image that you do not want to move.

Resolution field Displays the pixel resolution of the clip being warped or morphed. A lower pixel resolution creates warps and morphs that better follow the contours of the distortion splines, although it can slow down system performance.



Track Shape button Opens the Stabilizer menu to apply tracking data to a spline's vertices.

Adjust box Select whether an offset is applied relative to the spline's tangents.

Distort Setup Settings

Repeat button Enable to fill in gaps by repeating the clip.

Repeat Value box Select 4 to fill gaps by repeating the clip once on each side, or 8 to repeat the clip twice on each side.

Amount box Select the amount of clip to repeat.

Draw Grid button Enable to display the distortion grid.

Scale Setup button Enable to scale a loaded setup's splines to the resolution of the current project.

Icons button Enable to display splines, tangents, and vertices in the image.

Icons Transparency field Displays the transparency level of the icons in the image. Editable.

Tangent colour pot Displays the colour of tangents on the splines. Editable.

Schematic Transparency field Displays the transparency level of the schematic. Editable.

Undo Levels field Displays the levels of Undo available in the Undo/Redo lists. Editable.

Miscellaneous Settings

Exit button Exits the Distort module.

Load button Loads a setup.

Save button Saves a setup.

Setup Name field Displays the name of the last saved setup.

Revert button Reverts to the last saved setup.

Player button Opens the Player to view the last rendered clip.

Step Process field Displays the frame number interval to be processed (to see intermediate results, for example). Editable.

Preview button Displays the distort effect without processing.

Setup button Opens the Setup menu, where you can set various preference and display settings.

Animation button Opens the channel editor, where you can animate various distort settings.

Set Key button Sets a keyframe at the selected frame.

Delete Key button Deletes the selected keyframe.

Copy button Copies an object, a branch (object and its children), or all, depending on what is selected in the Selection Mode box.

Delete button Deletes an object, a branch (object and its children), or all, depending on what is selected in the Selection Mode box.

Selection Mode box Select whether to copy, delete, or reset.

Reset button Resets an object, a branch (object and its children), or all, depending on what is selected in the Selection Mode box.

Node Name field Displays the name of the selected node in the schematic. Editable.

Previous button Selects the previous node in the schematic.

Next button Selects the next node in the schematic.

Add button Adds an axis or spline, depending of the selection of the Node box.

Node box Select whether to add an axis or spline, then click Add.

Close button Closes a spline at the first vertex.

Finish button Finishes a single-point or open spline at the last vertex.

Edge Detect

Use Edge Detect to trace the edges in a clip based on colour. This data can then be used to create a range of artistic effects in various colour spaces.

To access the Edge Detect menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front and matte clip, and outputs a result and an outmatte.

Edge Detect Settings

General Settings

Colour Type box Select the colour space that you want to use to isolate the edge.

Result Output box Select Result to output the composite of the input image, input matte, and the matte generated by the edge detection. Or, select Edge Only to output only the result of compositing the input image with the matte generated by the edge detection.

Matte Output button Select Input to output the input matte or Edge Matte to output the matte generated by the edge detection.

Edge Detection Settings

Red Weight field Displays a value for the relative intensity of red used to detect edges. This value is scaling factor, not the color value of the channel itself. Editable.

Green Weight field Displays a value for the relative intensity of green used to detect edges. This value is scaling factor, not the color value of the channel itself. Editable.

Blue Weight field Displays a value for the relative intensity of blue used to detect edges. This value is scaling factor, not the color value of the channel itself. Editable.

Proportional Weight button Enable to adjust the red, green, and blue weights proportionally.

Minimum Edge Detection Threshold field Displays the lower limit for edge detection. Editable.

Maximum Edge Detection Threshold field Displays the upper limit for edge detection. Editable.

Softness Width field Displays a value for the width of the edge. Editable.

Softness Gain field Displays a value for the softness of the edge. Editable.

Edge Effects Settings

Double Edge button Enable to do a second edge detection pass and create twice as many edges.

Minimum Edge Detection Threshold field Displays the lower limit for edge detection. Editable.

Maximum Edge Detection Threshold field Displays the upper limit for edge detection. Editable.

Softness Width field Displays a value for the width of the edge. Editable.

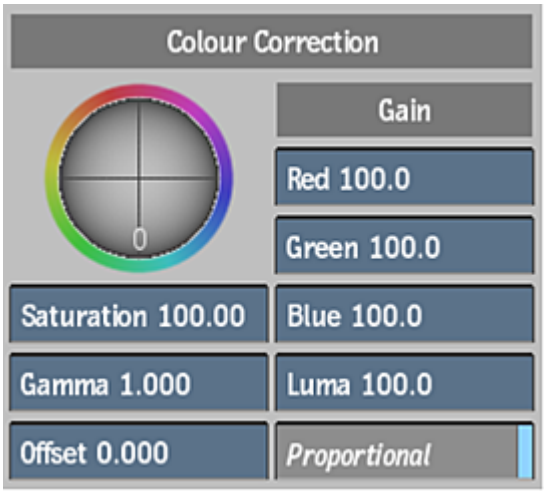
Softness Gain field Displays a value for the softness of the edge. Editable.

Directional button Enable to create the edge in the direction indicated in the Angle field.

Edge Angle field Displays a value for the direction of the edge. Editable.

Colour Offset pot Displays a colour offset value for the image outside of the detected edges.

Colour Correction Settings



Saturation field Displays level of colour purity in the image. Editable.

Gamma field Displays the gamma level. Editable.

Offset field Displays a value that modifies all of the colour parameters. Editable.

Red Gain field Set the percentage of colour values in the red channel. Editable.

Green Gain field Set the percentage of colour values in the green channel. Editable.

Blue Gain field Set the percentage of colour values in the blue channel. Editable.

Luma Gain field Set the percentage of luma gain value to display. Editable.

Proportional button Enable to adjust the gain of the colour values proportionally.

Blending Settings

Blend Mode option box Select an operation to blend the input image with the matte created by the edge detection, and the input matte (if the Use Matte button is enabled).

Edge Transparency field Displays a value for the transparency applied to the edges. Editable.

Use Matte button Enable to use the input matte to constrain the blending of the input image and the matte created by the edge detection.

Exposure

Use Exposure to apply plausible exposure and contrast settings suitable to the image data type of an input clip. The values can be adjusted separately for each colour channel or controlled simultaneously.

Front view of a 16-bit floating point image

Result view of the image after the exposure and contrast have been modified

To access the Exposure menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

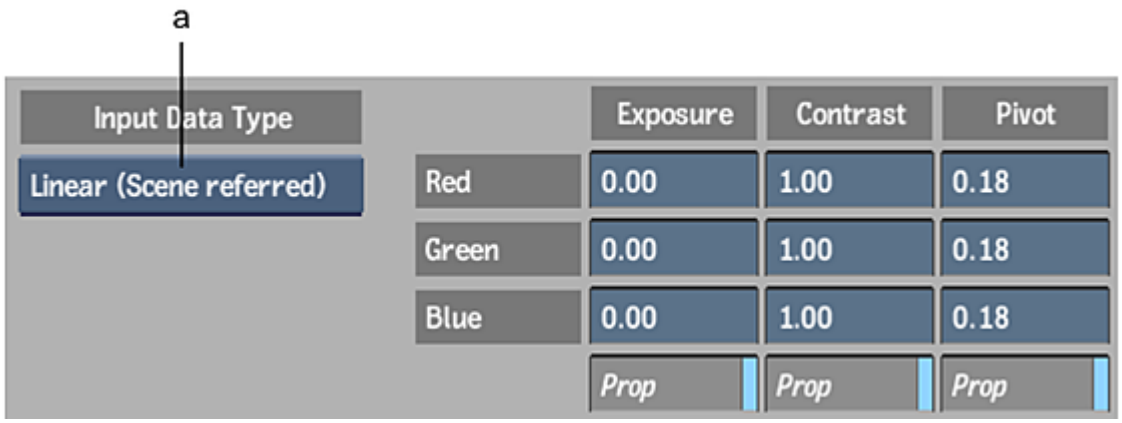
This node accepts a front clip and a matte clip, and outputs a result. The matte input is used to limit the areas of the front clip that are affected by exposure and contrast changes.

NOTE To adjust the exposure, contrast, and image data type of the image for display purposes only, set the image display viewer options in the View menu or edit them gesturally in the current viewport. See Displaying Multiple Views (page 406).

Exposure Menu Settings

Exposure Node

Calculations for exposure and contrast adjustments are based on the type of image that is selected in the Exposure menu.



(a) Input Data Type box

Input Data Type box Select the type of image data being input to the node.Your selection determines the most suitable type of transformation to apply to the clip. The algorithm used to calculate the result is dependent on this option.

Select:	To:
Logarithmic	Apply a transformation to a logarithmic film scan.
Video	Apply a transformation to a video clip.
Linear	Apply a transformation to a 16-bit floating-point image, with a high dynamic range.

Red Exposure field Displays the exposure offset of the red channel.

Green Exposure field Displays the exposure offset of the green channel.

Blue Exposure field Displays the exposure offset of the blue channel.

Proportional button Enable to change the exposure offset value for a channel and update the values for the other channels proportionally.

Red Contrast field Displays the contrast level of the red channel.

Green Contrast field Displays the contrast level of the green channel.

Blue Contrast field Displays the contrast level of the blue channel.

Proportional button Enable to change the contrast value for a channel and update the values for the other channels proportionally.

Red Pivot field Displays the red value used as a pivot when generating contrast. The pivot value affects the way the contrast is calculated. The default value is mid-gray on a logarithmic scale.

Green Pivot field Displays the green value used as a pivot when generating contrast. The pivot value affects the way the contrast is calculated. The default value is mid-gray on a logarithmic scale.

Blue Pivot field Displays the blue value used as a pivot when generating contrast. The pivot value affects the way the contrast is calculated. The default value is mid-gray on a logarithmic scale.

Proportional button Enable to change the pivot value for a channel and update the values for the other channels proportionally.

Field Merge

Use Field Merge to remove field jitter by merging the fields of a clip.

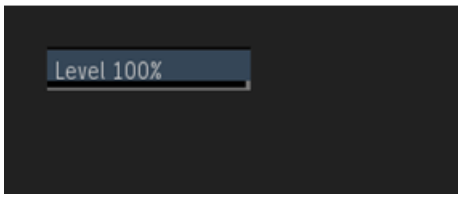
To access the Field Merge menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip, and outputs a result.

Field Merge Menu Settings

General Settings



Level field Enter the percentage of blending between fields, or drag on the field to modify its value.

Filter

Use Filter to apply different effects to a clip, including textures, blurring, edge detection, embossing, and sharpening.

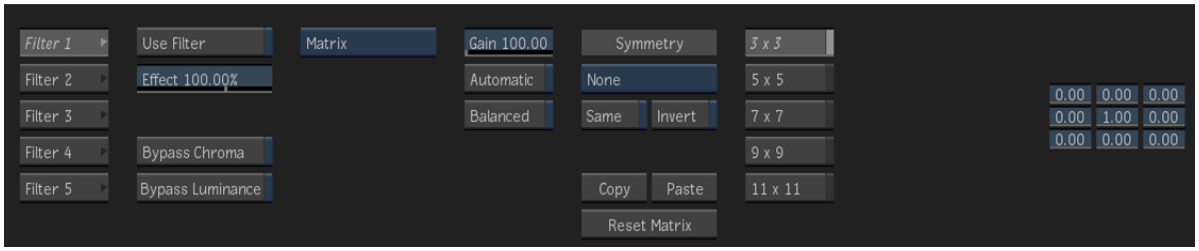
To access the Filter menu:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip, and outputs a result.

Filter Menu Settings

General Settings



Filter 1 button Modifies the set of image operations in the first filter element that is applied in the final composite filter.

Filter 2 button Modifies the set of image operations in the second filter element that is applied in the final composite filter.

Filter 3 button Modifies the set of image operations in the third filter element that is applied in the final composite filter.

Filter 4 button Modifies the set of image operations in the fourth filter element that is applied in the final composite filter.

Filter 5 button Modifies the set of image operations in the fifth filter element that is applied in the final composite filter.

Use Filter button Enable to apply the filter element to the final composite filter.

Effect field Displays the level of filtering required. Editable.

Bypass Chroma button Enable to ignore hue and saturation channels of an image.

Bypass Luminance button Enable to ignore luminance channels of an image.

Filter Type box Select whether to use matrix calculations or a predefined procedure to alter the image.

Matrix Settings

Gain field Displays the light values of the image. Editable.

Automatic button Enable to preserve the average luminance of the clip.

Balanced button Enable to activate a compensating algorithm when a value is entered in the matrix, where the value is divided by the remaining fields, and then the result is subtracted from each value in the matrix.

Symmetry Type box Select an arrangement used to change symmetrical elements.

Same button Enable to change a symmetrical field to the same value as a field that is being edited.

Invert button Enable to change a symmetrical field to the equal and opposite value as the field value that is being edited.

Copy button Click to copy the current matrix setup.

Paste button Click to paste a copied matrix setup.

Reset Matrix button Reset all values in the matrix.

3x3 button Enable to apply a matrix of three rows and three columns. The field in the centre represents the pixel that is currently being evaluated; the others represent surrounding pixels. The matrix displays the relative influence of each pixel surrounding the current one.

5x5 button Enable to apply a matrix of five rows and five columns. The field in the centre represents the pixel that is currently being evaluated; the others represent surrounding pixels. The matrix displays the relative influence of each pixel surrounding the current one.

7x7 button Enable to apply a matrix of seven rows and seven columns. The field in the centre represents the pixel that is currently being evaluated; the others represent surrounding pixels. The matrix displays the relative influence of each pixel surrounding the current one.

9x9 button Enable to apply a matrix of seven rows and seven columns. The field in the centre represents the pixel that is currently being evaluated; the others represent surrounding pixels. The matrix displays the relative influence of each pixel surrounding the current one.

11x11 button Enable to apply a matrix of eleven rows and eleven columns. The field in the centre represents the pixel that is currently being evaluated; the others represent surrounding pixels. The matrix displays the relative influence of each pixel surrounding the current one.

Procedural Settings

Invert button Enable to invert the image.

Rect button Enable to blur the image.

Width field Displays the height of the blur. Editable.

Height field Displays the width of the blur. Editable.

Sobel button Enable to apply an edge-detection filter that uses the Sobel operator.

Sobel Direction box Select the direction in which the filter is applied.

Prewitt button Enable to apply an edge-detection filter that uses the Prewitt operator.

Prewitt Direction box Select the direction in which the field is applied.

Flip

Use Flip to generate a mirror image of a clip. You can flip frames in a clip horizontally, vertically or both.

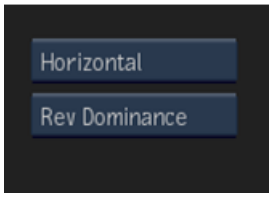
To access the Field Merge menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip, and outputs a result.

Flip Menu Settings

General Settings



Flip Direction box Select to flip the clip horizontally or vertically.

Reverse Dominance box Select an option for correcting the field dominance when flipping clips vertically.

Garbage Mask

Use the Garbage Mask to isolate particular areas of an image to include with, or exclude from, the opaque area of the matte.

To access the Garbage Mask menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).

  When you access the GMask as a Timeline FX, you can enable Use Matte to use the matte input as part of the effect processing.
- Tools, then select from the menu (See Accessing Tools (page 493)).

■ The Modular Keyer, then select a node from the Node bin.

NOTE A different GMask node is available from within Action (page 900) and the Mask Tracer (page 1099).

This node accepts a front and a matte clip as input, and outputs a result.

GMask usage notes:

■ The Garbage Mask menu includes the Tracer and the region of Interest (ROI) functions, which can also be found in the Modular Keyer.

■ The node processes gaps in clips set to No Media as black frames. An unconnected front clip will return an error, while and unconnected back clip will process black frames.

■ You can save and load GMask setups directly in Batch or Batch FX.

Garbage Mask Menu Settings

Timeline Quick Menu Settings

Garbage Mask Quick Menu selector Select which menu to display.

Mask Selection box Apply changes to the selected mask or all masks.

Garbage Mask General Settings

Regenerate button Enable to get dynamic updating of your mask as you make changes.

X Position field Position the X axis.

Y Position field Position the Y axis.

Z Position field Position the Z axis.

Path button Enable to animate the position of the axis using a spline drawn in the scene.

X Rotation field Rotate the X axis.

Y Rotation field Rotate the Y axis.

Z Rotation field Rotate the Z axis.

X Scale field Scale the X axis.

Y Scale field Scale the Y axis.

Z Scale field Scale the Z axis.

Proportional button Enable to constrain proportions while scaling.

X Shear field Shear the X axis.

Y Shear field Shear the Y axis.

Z Shear field Shear the Z axis.

X Centre field Centre the X axis.

Y Centre field Centre the Y axis.

Z Centre field Centre the Z axis.

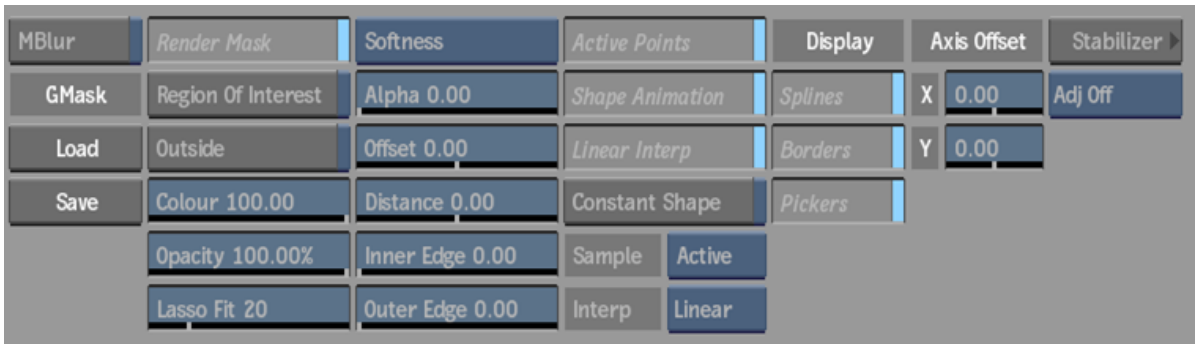
Stabilizer button Opens the Stabilizer menu where you track reference points in a clip.

Tracking Rotation option box Select Rotation Off for one-point tracking, Rotation On for two-point tracking, Rotation Inv to invert the rotation data.

Tracking Scale option box Select Scaling Off for one-point tracking, Scaling On for two-point tracking, Scale Inv to invert the scaling data.

Adjust box Select Adj Offset if the mask is parented to one axis. Select Adj Axis if the mask is parented to a hierarchy of objects. Select Adj Tangents to have the tangents for the selected points adjusted while the points are tracked.

Motion Blur button Enable the blur effect for the selected axis (Motion Blur must be enabled in the Setup menu).



Motion Blur button Toggles the blur effect for the selected geometry (Motion Blur must be enabled in the Setup menu).

Load button Loads a garbage mask setup.

Save button Saves a garbage mask setup.

Render Mask button Enable to see the mask and its effect on the image.

Region of Interest button Enable to reveal the matte of multiple garbage masks.

Outside button Enable to apply the effect to the part of the image outside the mask.

Colour field Set the blend between the outgoing and incoming image inside the mask.

Opacity field Set the transparency of the mask.

Lasso Fit field Set the number of points used in the freehand part of the mask.

Edge Softness box Select an edge softness method. Select Softness to create a uniform gradient around the mask edge, or select Tracer to key out details around the mask edge. Different settings appear based on the selection.

Alpha field Set the transparency of the gradient from the mask edge.

Offset field Set the position of the gradient's border from the mask edge.

Clean button Enable to apply the clean algorithm to reduce random noise and artefacts.

Clean Effect field Displays the level of cleaning to apply. Editable.

Pickers button Enable to display two pickers to perform luma and chroma analysis of the area inside and outside of the mask and derive localized edge keys from this information.

Picker Size field Displays the size of the pickers. Editable.

Distance field Specify the area over which the Inner and Outer Edge adjustments have an effect.

Inner Edge field Smoothen the softness gradient towards the inside of the mask edge.

Outer Edge field Smoothen the softness gradient towards the outside of the mask edge.

Active Points button Enable to use all points as a reference for the mask. Disable at a specific frame to mute picker values previously sampled at another frame.

Shape Animation button Enable to animate a mask using the Shape channel.

Linear Interpolation button Enable to use linear interpolation of the mask border between keyframes. Disable to use rounded interpolation. This button is active when the Shape Animation button is enabled.

Constant Shape button Enable to modify the mask's shape without setting keyframes.

Sample Status box Select how picker values are sampled. Active resamples picker values at every frame (the default). Passive disables resampling for one or more frames. This box is active when the Shape Animation button is disabled.

Interpolation Mode box Select the type of interpolation between keyframes when the Sample Status box is set to Active.

Splines button Enable to display inner and outer splines of the mask.

Borders button Enable to display inner and outer borders for advanced gradients.

Pickers button Enable to display elements that allow for detailed chroma and luma analysis when refining a detailed key.

X Axis Offset field Offset the mask from its X axis.

Y Axis Offset field Offset the mask from its Y axis.

Stabilizer button Opens the Stabilizer menu where you apply tracking data to selected vertices.

Adjust box Select Adj Tangents to have the tangents for the selected vertices adjusted while the points are tracked.

Add button Click to add a gmask axis or geometry node.

Node Type box Select whether to add an axis or geometry node.

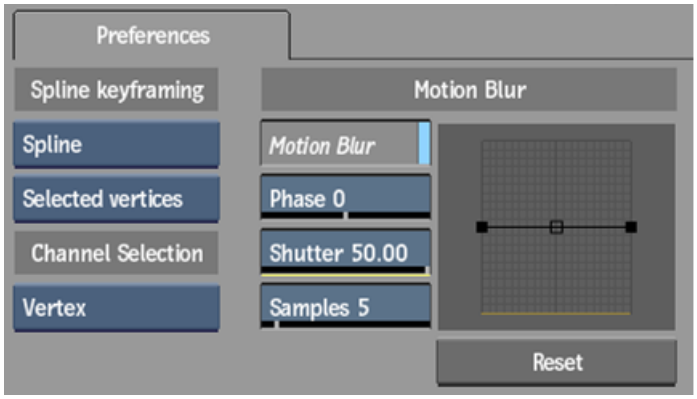
Close button Click to close the mask shape automatically.

Object Node Name field Displays the name of the selected object node.

Previous Node button Click to scroll to the previous similar node.

Next Node button Click to scroll to the next similar node.

Garbage Mask Node Setup Settings



Spline Keyframing box Select the parameters to use when animating a Garbage Mask spline. When Shape Animation is disabled in the Garbage Mask general settings, then the Spline & Tracer, Spline and Item options become available.

Vertex Keyframing box Select the conditions under which parameters are applied to specified vertices in the mask.

Channel Selection box Select which channels are selected in the Channel Editor when you select one or more vertices.

Motion Blur button Enable to apply motion blur to garbage masks that can be used to match the movement of objects in a clip.

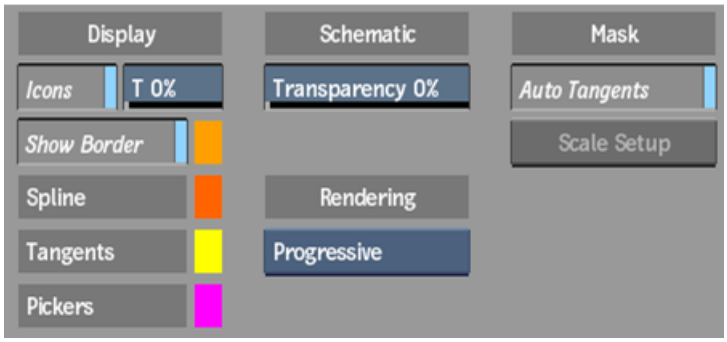
Phase field Displays whether the motion blur is based on the movement before or after the current frame. -100 places the motion blur before the frame; a value of 100 places it after the frame.

Shutter field Displays the duration of the motion blur at each frame. Editable.

Samples field Displays the number of samples taken at each frame to create the motion blur. Editable.

Motion Blur curve Displays the transparency of the samples that create the blur effect. Drag the curve or its handles to edit it gesturally.

Reset button Resets all motion blur controls and disables the Motion Blur button.



Icons button Enable to display garbage mask splines and axes.

Transparency field Displays the transparency of garbage mask splines and axes. Editable.

Show Border button Enable to display the Softness Offset wireframe border, defined in the Shape menu.

Show Border colour pot Select the colour for the Softness Offset wireframe.

Wireframe colour pot Select the colour of garbage mask splines.

Control Points colour pot Select the colour of garbage mask control points.

Pickers Display colour pot Select the colour of the pickers.

Transparency field Displays the transparency of nodes in the schematic. Editable.

Auto Insert button Enable to automatically insert a node when dragged between two connected nodes. When disabled, press Shift to auto insert.

Rendering box Select the rendering method.

Auto Tangents Enable to create tangents for new points.

Scale Setup button Enable to scale the mask. Use when you have a resolution mismatch between the clip and its applied mask.

Glow

Use Glow to create and customize a glow effect on a clip.

To access the Glow menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

■ The Modular Keyer, then select a node from the Node bin.

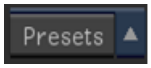
This node accepts a front and matte clip, and outputs a result an outmatte. You can add a Glow node to any part of the pipeline except to the Front and Matte pipes of MBLend nodes and the Matte pipes of CBlend nodes.

Glow Menu Settings

Using Glow Presets

A number of Glow presets are included, such as a basic glow effect or edge boost effect. These presets can help you learn how the glow effect works, or provide a good starting point to change settings to create better glow effects.

To use the Glow preset, select a preset from the Presets drop-down list in the Glow Setup menu. The Glow menu settings are changed to reflect the chosen preset.



Presets button Opens the Presets browser where you can select a preset.

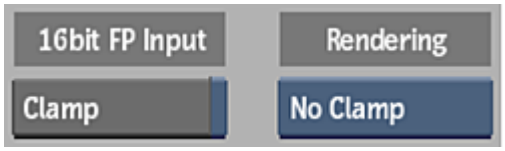
Presets dropdown list Select a preset from the dropdown list. Menu settings are changed to reflect the chosen preset.

General Settings

Rendering Mode box Select whether to render in Automatic, Progressive or Interlaced mode.

Regen button Enable to get dynamic updating of the image as you make changes.

Setup Settings



Clamp Input button Enable to clamp colour and luminance values on input in the 16-bit floating point rendering pipeline.

Clamp Render box Select a clamping option for colour and luminance values on output in the 16-bit floating point rendering pipeline.

Glow Type Settings

Glow Type for Gaussian

Glow Type for Directional

Glow Type for Radial

Glow Type for Radial Stamp

Glow Type box Select the type or shape of glow filter to apply to the clip. For example, a Box blur has rectangular edges.Depending on the glow type, some of the other glow settings vary.

Radial Mode box Select whether a radial blur or glow moves in one circular direction (Spin) or two rotating directions (Twist). Available when Radial is chosen as the blur type.

Bias field Displays the direction of a blur. Enter a positive value for forward, a negative value for backward, or 0 for a blur that moves in both directions. Available when Directional or Radial Stamp is chosen as the blur type.

Samples field Displays the quality of a Radial Stamp blur or glow. Editable. Available when Radial Stamp is chosen as the blur type.

Centre Settings (Radial only)

Stabilizer button Opens the Stabilizer menu to track the centre of the blur from the source clip.

Centre X field Displays the X position of the centre of the radial blur (or gesturally move the red circle in the image). Editable.

Centre Y field Displays the Y position of the centre of the radial blur (or gesturally move the red circle in the image). Editable.

NOTE You can also move the red circle on the image to set the position of the centre of the blur. The Centre X and Y fields update accordingly.

Absolute/Relative box Select whether to position and offset the centre of the radial blur in a relative mode (expressed as a percentage) or absolute mode (expressed in pixels).

Offset X field Offsets the centre along the X axis (or press Ctrl and gesturally move the red circle in the image).This field is useful to apply changes to the centre after tracking has been performed. Editable.

Offset Y field Offsets the centre along the Y axis (or press Ctrl and gesturally move the red circle in the image). This field is useful to apply changes to the centre after tracking has been performed. Editable.

NOTE You can also press Ctrl and move the red circle on the image to set the offset of the centre of the glow. The Offset X and Y fields update accordingly.

Input Settings

Hue field Displays the colour range in the image before the glow effect is applied. Editable.

Saturation field Displays the colour purity level in the image before the glow effect is applied. Editable.

Contrast field Displays the level of gradations between light and dark areas before the glow effect is applied. Editable.

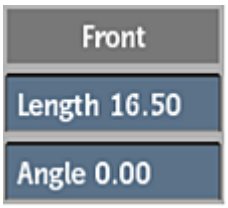
Gamma field Displays the level of grey in the image before the glow effect is applied. Editable.

Gain field Displays a value by which pixel colour values are multiplied. The offset value is added to this value to determine the final colour. Editable.

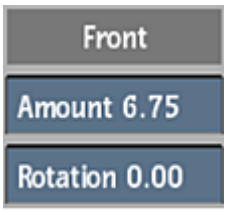
Offset field Displays the value to add to current pixel colour values. The resulting colour value is clipped at 0. Editable.

Front Settings

Front Settings for Gaussian or Box



Front Settings for Directional



Front Settings for Radial or Radial Stamp

Width field Displays the width of the blur.Increasing the blur increases the render time. Editable.

Height field Displays the height of the blur.Increasing the blur increases the render time. Editable.

Proportional button Enable to affect the width and height proportionally.

Pixel Ratio button Enable to blur the image using the same proportion as its aspect ratio.

Amount field (Radial, Radial Stamp) Displays the amount of radial blur. Editable.

Rotation field (Radial, Radial Stamp) Displays the angle of rotation for a radial blur. Editable.

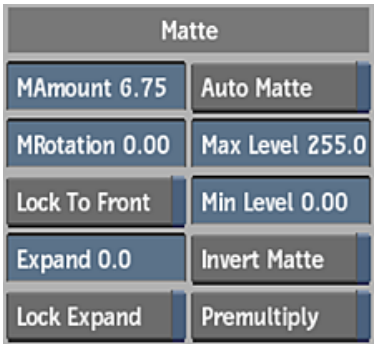
Length field (Directional) Displays the radius amount of a directional blur. Editable.

Angle field (Directional) Displays the angle of a directional blur. Editable.

Matte Settings

Matte Settings for Gaussian or Box

Matte Settings for Directional



Matte Settings for Radial or Radial Stamp

Matte Width field Displays the width of the blur for the matte. Editable.

Matte Height field Displays the height of the blur for the matte. Editable.

Matte Amount field Displays the amount of radial blur for the matte. Editable.

Matte Rotation field Displays the angle of rotation for a radial blur for the matte. Editable.

Matte Length field Displays the radius amount of a directional blur for the matte. Editable.

Matte Angle field Displays the angle of a direction blur of the matte. Editable.

Lock To Front button Enable to keep the matte values the same as their corresponding values for the front clip.

Expand field Displays the percentage of additional blur to the matte. Editable when the Lock Expand button is disabled.

Lock Expand button Enable to make the Expand value directly proportional to the value in the Width field.

Auto Matte button Enable to generate matte values from the front clip.

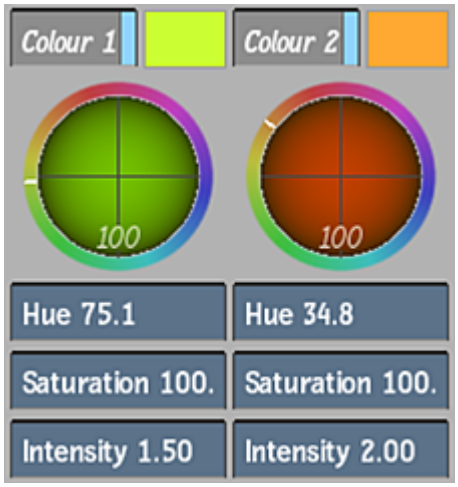
Max Level field Displays the upper limit of the luminance values included in the glow effect. Editable.

Min Level field Displays the lower limit of the luminance values included in the glow effect. Editable.

Invert Matte button Enable to apply the glow to the region outside the area defined by the matte.

Premultiply button Enable to multiply the matte clip to the front clip.

Colour Settings



Colour 1 box Enable to define the first colour used for the glow. If two colours are used, they are combined in additive mode to create the glow effect.

Colour 1 Colour pot Displays the hue and saturation colour of the first glow colour. Click to open the colour picker to select a different colour.

Colour 1 Trackball Adjusts the hue and saturation of colour 1.

Colour 1 Hue field Displays the colour range of colour 1. Editable.

Colour 1 Saturation field Displays the level of colour purity of colour 1. Editable.

Colour 1 Intensity field Displays the level of brightness of colour 1. Editable.

Colour 2 box Enable to define the second colour used for the glow. If two colours are used, they are combined in additive mode to create the glow effect.

NOTE If you are using both Colour 1 and Colour 2, they are combined in additive mode to create the glow effect.

Colour 2 Colour pot Displays the hue and saturation colour of the second glow colour. Click to open the colour picker to select a different colour.

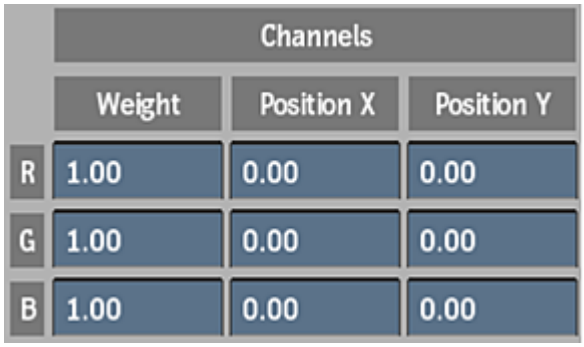
Colour 2 Trackball Adjusts the hue and saturation of colour 2.

Colour 2 Hue field Displays the colour range of colour 2. Editable.

Colour 2 Saturation field Displays the level of colour purity of colour 2. Editable.

Colour 2 Intensity field Displays the level of brightness of colour 2. Editable.

Channels Settings

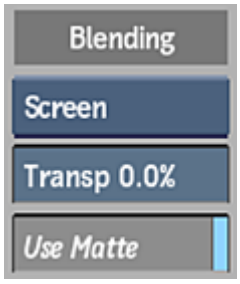


Weight field Displays the weighted value of the channel.

Position X field Displays the horizontal offset of the channel.

Position Y field Displays the vertical offset of the channel.

Blending Settings



Screen option box Select a logical operation that can be used to blend the front clip and the result clip.

Transparency field Displays the percentage of blending when the result is composited on the front clip. Editable.

Use Matte button Enable to include the matte in the blending.

Result Settings

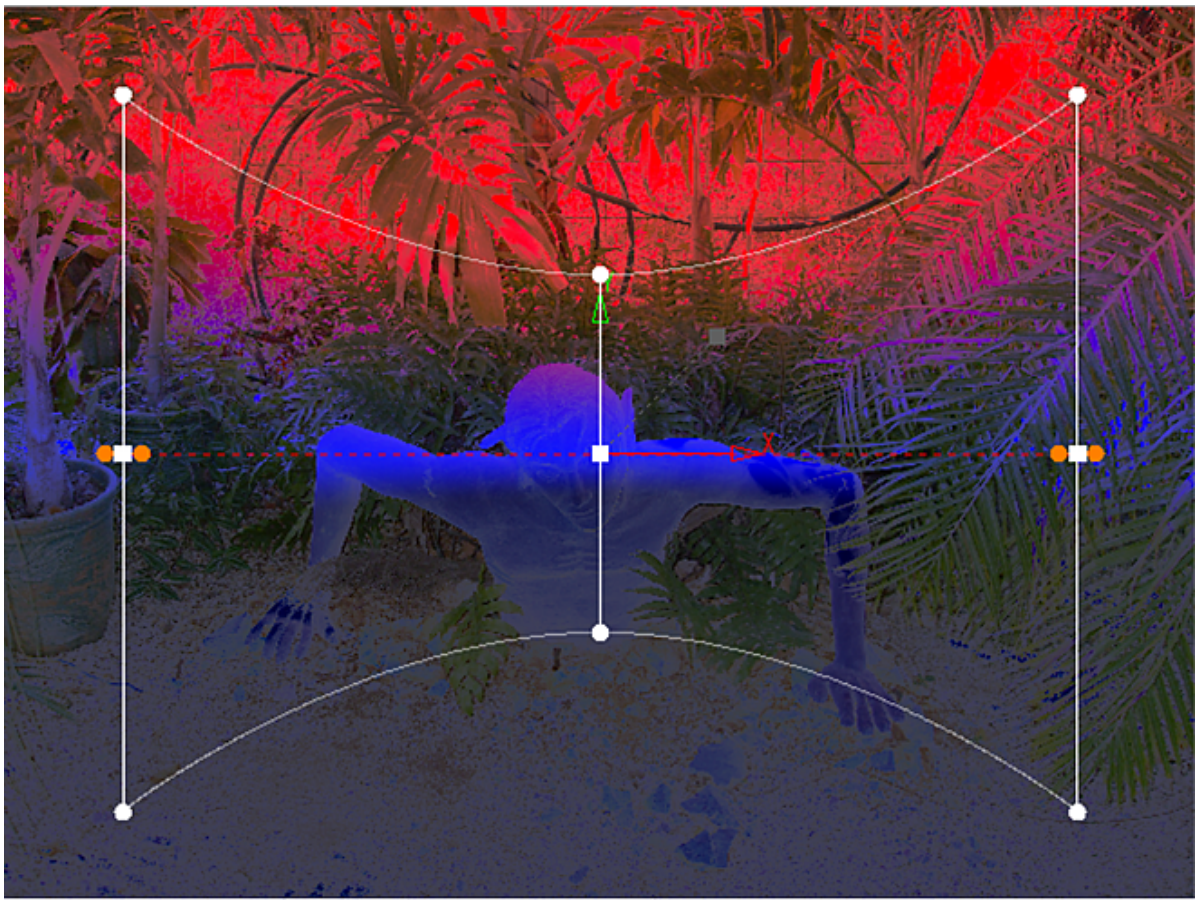
Result Output button Select whether to output the combined result (with blending), or only the glow effect itself.

Matte Output Settings

Matte Output button Switch between blend matte and glow luma. Blend matte allows you to output the matte used internally to perform the blending of the glow effect over the input image. Glow luma allows you to generate a straight-forward luminance conversion of the actual glow effect.

Gradient

Use the Gradient to create an effect based on colours and patterns of your choosing.



To access the Gradient menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front and matte clip, and outputs a result and outmatte.

Gradient Menu Settings

Setup Settings

Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

Width field Displays the custom width resolution of the clip. Editable.

Height field Displays the custom width resolution of the clip. Editable.

Aspect Ratio Presets box Select a standard frame aspect ratio. Select the Set to w:h option to set the clip to use square pixels. Select Custom to define a custom frame aspect ratio in the Aspect Ratio field.

Aspect Ratio field Displays the custom render/output aspect ratio. Editable

Bit Depth box Select the render/output bit depth of clips.

Scan Mode box Select the scan mode of clips.

Icons button Enable to display vertices, splines, and other gradient selection tools in the image window.

Transparency field Displays the transparency level of the gradient icons in the image window. Editable.

Vertex Colour Pot Displays the colour of vertices in the image window. Click to open the colour picker to select a different colour.

Spline Colour Pot Displays the colour of splines in the image window. Click to open the colour picker to select a different colour.

Amplitude Colour Pot Displays the colour of amplitude in the image window. Click to open the colour picker to select a different colour.

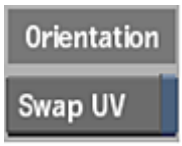
Type Settings

Gradient Type box Select the type of gradient to apply. Some of the gradient settings differ based on the gradient type you choose.

Select:	To create a gradient based on:
Directional	A single straight line spline.
Radial	Two circular geometries.
Spline	A spline that can be manipulated with vertices and tangents.
Point	Multiple coloured points.

For each gradient type, you can gesturally create your gradients in the image window with various widgets, such as splines and vertices. Make sure that Show Widgets is enabled in the Gradient Setup menu.

Orientation Settings



Swap UV button Enable to change the direction of the gradient.

Gradient Controls Settings

Position X field Displays the position of the gradient along the X axis. Editable.

Position Y field Displays the position of the gradient along the Y axis. Editable.

Rotation field Displays the degree of rotation of the gradient. Editable.

NOTE You can also manipulate the position and rotation of the gradient directly in the image window if Show Widgets is available.

Scale field Displays the scale of the gradient. Editable.

Centre field Directional gradient only: Displays the offset value of the centre of the spline as a percentage of its position on the spline. Editable.

Amplitude field Directional gradient only: Displays the pixel length of the spline. You can also drag either end of the spline directly in the image window. Editable.

Roundness field Radial gradient only: Displays the relative shape of the inner and outer circular patterns. A value of 1 displays the full rounded shape, whereas a value of 0 indicates a square shape. You can also enter a negative value to produce a concave shape. Editable.

Sharpness field Point gradient only: Displays the level of edge sharpness between the gradient point colours. Editable.

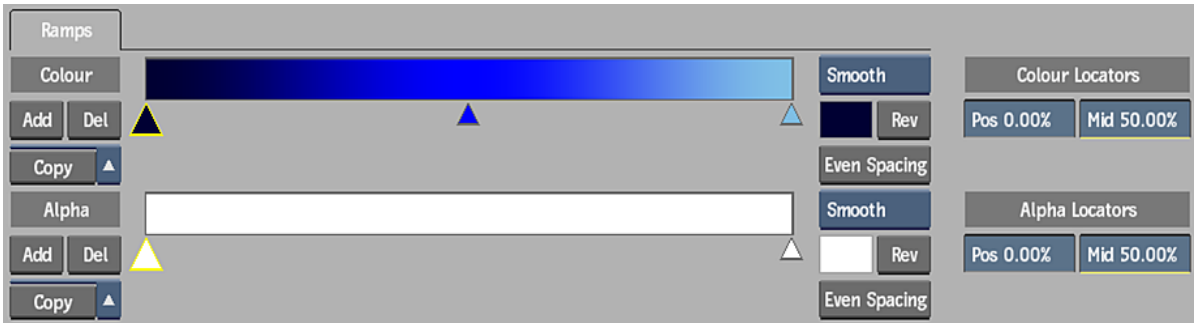
Softness field Displays the level of blur added to the gradient. A higher value produces a smoother transition between gradient colours. Editable.

Extrapolation box Select an extrapolation type for the gradient pattern.

Select:	To:
Constant	Keep the last value as a solid colour.
Repeat	Repeat the same gradient pattern.
Mirror	Repeat the gradient pattern in reverse order until the frame border is reached.

Offset Cycle field Displays the cycle position of the color creating the gradient. Editable.

Ramps Settings



NOTE The Colour and Alpha ramps are available for Directional, Radial and Spline gradients.

Colour Ramp Displays the gradations of the gradient colours. The triangles below the ramp represent each color in the gradient. You can add more colours to your gradient pattern.

A selected triangle's colour is displayed in the colour pot to the right of the ramp, and its position is displayed in the Position field. Also, when selecting a triangle, a smaller triangle appears representing the mid point between the colour and the next triangles' colour. You can move the triangles gesturally to achieve different gradations, and move the mid point triangle to shift the weight between the colours.

Colour Add button Adds a new colour triangle to the right of the currently selected triangle. Press Ctrl while clicking add to clone the selected triangle colour.

Colour Delete button Deletes the selected triangle.

Colour Ramp Copy list Select whether to copy the Colour ramp information to the Alpha ramp, or to preserve the Colour ramp information when switching gradient types.

NOTE The Alpha ramp under the Colour ramp is identical in functionality, except that it displays gradations in gradient transparency.

Alpha Ramp Displays the gradations of the gradient transparency. The triangles below the ramp represent each alpha colour in the gradient. You can add more alpha colours to your gradient pattern.

Alpha Add button Adds a new colour triangle to the right of the currently selected triangle. Press Ctrl while clicking add to clone the selected triangle colour.

Alpha Delete button Deletes the selected triangle.

Alpha Ramp Copy list Select whether to copy the Alpha ramp information to the Colour ramp, or to preserve the Alpha ramp information when switching gradient types.

Colour Interpolation Type box Choose a Linear or Smooth gradient interpolation curve.

Colour Colour pot Displays the colour of the selected triangle. Click to open the colour picker to select a different colour.

Reverse Colour button Click to reverse the colours of the gradient proportionally.

Colour Even Spacing button Click to space the ramp triangles evenly.

Alpha Interpolation Type box Choose a Linear or Smooth gradient interpolation curve.

Alpha Colour pot Displays the alpha colour of the selected triangle. Click to open the colour picker to select a different colour.

Reverse Alpha Colour button Click to reverse the alpha colours of the gradient proportionally.

Alpha Even Spacing button Click to space the ramp triangles evenly.

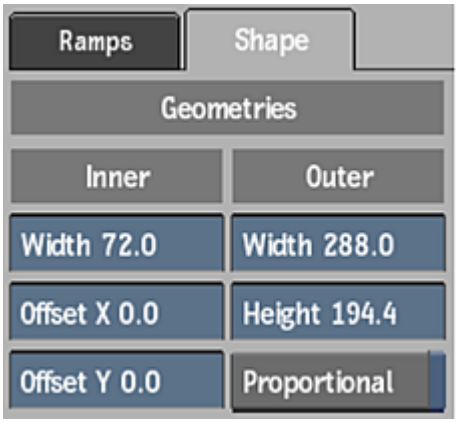
Colour Locator Position field Displays the location of the selected colour triangle along the colour ramp. Editable.

Colour Locator Mid field Displays the weight level between colours on the ramp (represented by the small triangle). Editable.

Alpha Locator Position field Displays the location of the selected alpha colour triangle along the alpha ramp. Editable.

Alpha Locator Mid field Displays the weight level between alpha colours on the ramp (represented by the small triangle). Editable.

Radial Shape Settings



Use these settings if you wish to further affect the shape of a radial gradient.

NOTE Radial Shape settings are available when Radial is selected from the Gradient Type box.

Inner Width field Displays the width of the inner radial circle. You can also gesturally drag the red square on the inner circle. Editable.

Inner Offset X field Displays the offset along the X axis of the inner radial circle. You can also gesturally drag the + inside the inner circle. Editable.

Inner Offset Y field Displays the offset along the Y axis of the inner radial circle. You can also gesturally drag the + inside the inner circle. Editable.

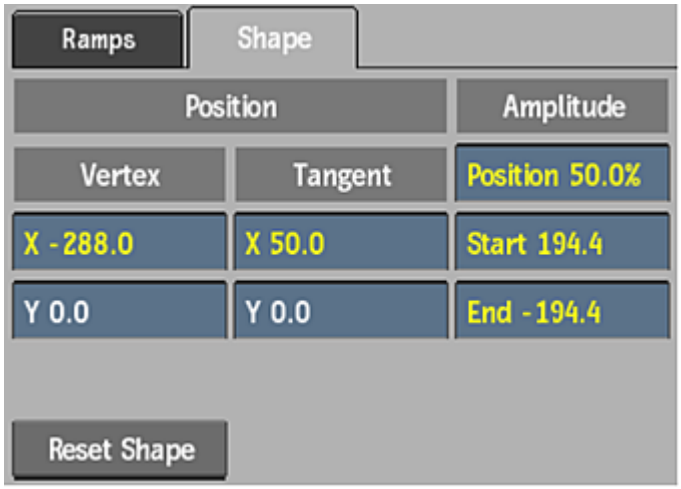
Outer Width field Displays the width of the outer radial circle. You can also gesturally drag the right-most red square on the outer circle. Editable.

Outer Height field Displays the height of the outer radial circle. You can also gesturally drag the left-most red square on the outer circle. Editable.

NOTE Drag the middle red square of the outer radial circle to affect both the width and the height at the same time.

Proportional button Enable to affect the outer radial width and height proportionally.

Spline Shape Settings



Use these settings if you wish to further affect the shape of a spline gradient. All of these settings have gestural equivalents in the image window. Ensure that Icons is enabled in the Gradient Setup menu.)

NOTE Available when Spline is selected from the Gradient Type box.

Vertex X Position field Displays the position along the X axis of the selected vertex. Editable.

Vertex Y Position field Displays the position along the Y axis of the selected vertex. Editable.

Tangent X Position field Displays the position along the X axis of the selected tangent. Editable.

Tangent Y Position field Displays the position along the Y axis of the selected tangent. Editable.

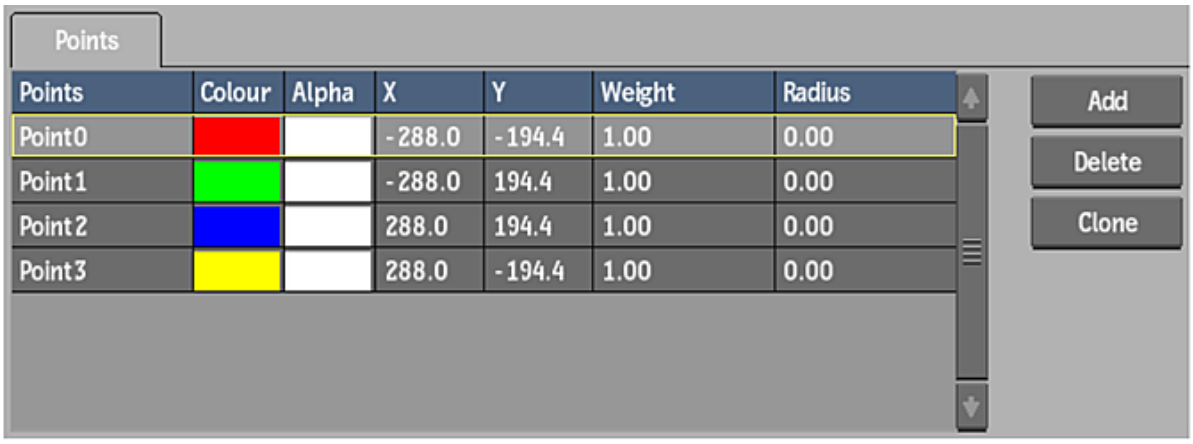
Amplitude Position field Displays the position of the spline as a percentage of the distance between the first and last vertex. Editable.

Amplitude Start field Displays the start position of the spline. Use to set the pixel length of the spline. Editable.

Amplitude End field Displays the end position of the spline. Use to set the pixel length of the spline. Editable.

Reset Shape button Resets the spline shape.

Points Settings



NOTE Available when Point is selected from the Gradient Type box.

Points List Use the rows to set the colour, position, and other settings for each color of a Point gradient.

Points Column Displays the number of the point. By default, four points are added to a gradient; however, you can add more.

Colour Column Displays the colour of the point. Click the colour pot to open the colour picker to select a different colour.

Alpha Column Displays the alpha colour of the point. Click the colour pot to open the colour picker to select a different colour.

X Column Displays the position of the point along the X axis. You can also gesturally drag the point in the image window to position it.

Y Column Displays the position of the point along the Y axis. You can also gesturally drag the point in the image window to position it.

Weight Column Displays the weight of the colour in relation to the nearest point colour in the image.

Radius Column Displays the radius of the point colour.

Add button Adds a new colour point to the list and image.

Delete button Deletes the selected colour point from the list and image.

Clone button Clones the selected colour point settings in a new point in the list and image.

Blending Settings

Screen option box Select a logical operation that can be used to blend the front clip and the result clip.

Transparency field Displays the percentage of blending when the result is composited on the front clip. Editable.

Gain field Displays the level of gain applied to the chosen blending option. Editable.

Result Output Settings

Output Mode box Select whether to output the combined result (with blending), or only the gradient effect itself.

Alpha Output Settings

Alpha Output box Select whether to output the combined alpha result, or only the gradient effect itself.

Interlace

Use Interlace to connect the odd and even scanlines of a clip.

To access the Interlace menu:

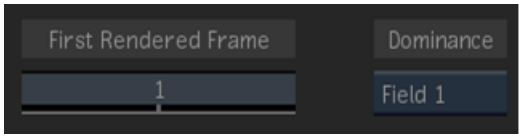
- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

The Interlace node accepts a front clip as input, and outputs a result.

For each pair of frames in the input clip, the Field 1 scanlines of one frame are interlaced with the even scanlines of the second frame to produce a single frame in the generated clip.

Interlace Menu Settings

General Settings



First Rendered Frame field Displays the first frame at which output is rendered from the node. Unrendered output does not display any media.Editable.

Field Dominance box Select Field 1 or Field 2 dominance, or Auto to have the application detect automatically the field dominance.

Keyer 3D

Use the Keyer 3D to key out a given range of colours.

To access the Keyer 3D menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Keyer 3D Menu Settings

General Settings

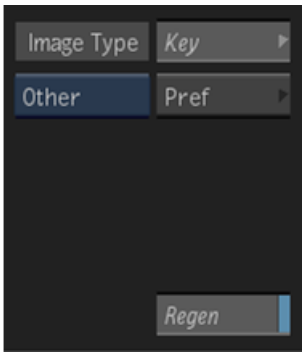


Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Key button Select to open the Key menu, where you set the elements to key in the RGB viewer, or the Pref menu, where you customize the RGB Viewer display properties.

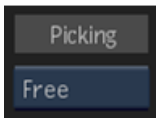
Regen button Enable to get dynamic updating of the image as you make changes.

Sampling Settings

Sampling box Select the type of key element value to sample in the image window.

Sampling Reset button Resets the values for the key element selected in the Sampling box.

Picking Settings



Picking box Select the key element to restrict selection in the RGB viewer. Select Free to select any element.

Rotation Axis Settings

NOTE The Rotation Axis Settings are available when you select the Pref menu from the Key button.

Rotation Axis box Select the centre of the RGB viewer or a key element as the axis of rotation of the RGB viewer.

Plot Regions Settings

NOTE The Plot Regions Settings are available when you select the Pref menu from the Key button.

Plot Region 1 button Enable to display the first plotted region.

Plot Region 2 button Enable to display the second plotted region.

Plot Region 3 button Enable to display the third plotted region.

Plot Point button Enable to display plotted colour value in the RGB viewer.

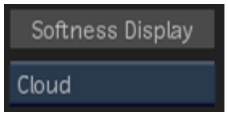
Tolerance Display Settings

Tolerance Display box Select to display the tolerance as an opaque, transparent, or wireframe ellipsoid.

Tolerance ConnexHull Settings

Tolerance Convex Hull option box Select to display the vertices, surfaces, or both the vertices and surfaces of the tolerance convex hull.

Softness Display Settings



Softness Display box Select to display the softness as an opaque, transparent, or wireframe ellipsoid.

Softness ConnexHull Settings

Soft Convex Hull option box Select to display the vertices, surfaces, or the vertices and surfaces of the softness convex hull.

Softness/Noise Settings

Colour box Select a YUV or RGB colour model.

Softness Scaling box Select to scale softness proportionally or to remove noise from softened areas.

Scaling field When Softness Scaling is set to Prop, adjust the X, Y, and Z-axis softness scaling values proportionally. When Softness Scaling is set to Min Noise, adjust the value to increase grain removal. (The scaling value resets to 1.00 after use.) Editable.

Scale X field Displays X-axis softness scaling. Editable.

Scale Y field Displays Y-axis softness scaling. Editable.

Scale Z field Displays Z-axis softness scaling. Editable.

Patches Settings

Patches box Select the Patch you want to sample.

Active button Enable to include the sample in the key.

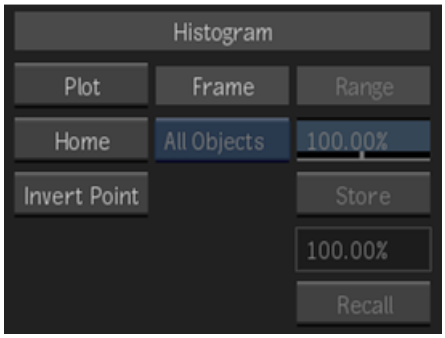
Display box Select to display the current patch as a wireframe, a convex hull, both a wireframe and convex hull, or a transparent cloud.

Colour field Enter the percentage of the colour value to use to render the patch. Editable.

Opacity field Enter the percentage of opacity of the patch. Editable.

Softness field Enter the softness value of the patch. Editable.

Key Histogram Settings



Plot button Activates a pick cursor which you use to select a plot value in the image window. If Plot Point is enabled in the Pref menu, a cube with a dashed outline surrounds the point in the RGB viewer.

Home button Restores the RGB viewer to its original settings and position.

Invert Point button Inverts the direction of scaling of the softness ellipsoid.

Frame option box Select an option to display the entire histogram, or a selected range of values.

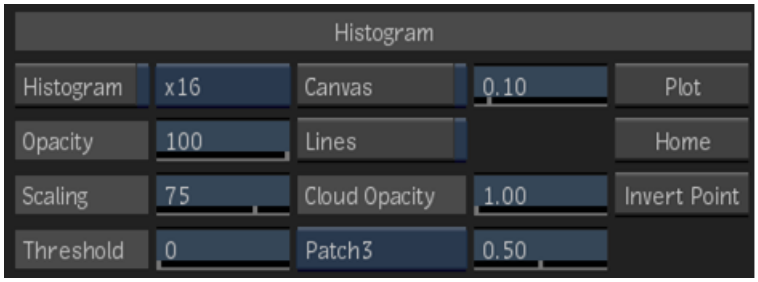
Range field Displays the percentage of the full range that is displayed in the histogram. Editable when the Frame option box is set to Free.

Store button Stores the value in the Range field in memory for later use. Active when the Frame option box is set to Free.

Buffer Value field Displays the Range value stored in the buffer. Non-editable.

Recall button Retrieves the last Range value that was stored in the buffer.

Pref Histogram Settings



NOTE The Pref Histogram Settings are available when you select the Pref menu from the Key button.

Histogram button Enable to display the 3D histogram in the RGB viewer.

Histogram Grid option box Select an option for the histogram grid resolution.

Opacity field Displays the percentage of opacity of the cubes. Editable.

Scaling field Resizes the cubes. Cubes representing less prevalent colours are scaled down more than those representing more prevalent colours. Editable.

Threshold field Enter a value to edit the display of colours that are less prevalent in the image. As you increase the threshold, less prevalent colours are increasingly removed from the histogram. Editable.

Canvas button Enable to display the canvas.

Canvas Light field Sets the lighting in the RGB viewer. The light source emanates from behind the viewer.

Lines button Enable to display a white outline of the borders of the RGB viewer, and a green wireframe around selected ellipsoids.

Cloud Opacity field Displays the overall opacity for all ellipses and patches.

Key Element box Select an element type to change its opacity.

Patch Opacity field Displays the opacity for the third patch in the RGB viewer.

Plot button Activates a pick cursor which you use to select a plot value in the image window. If Plot Point is enabled in the Pref menu, a cube with a dashed outline surrounds the point in the RGB viewer.

Home button Restores the RGB viewer to its original settings and position.

Invert Point button Inverts the direction of scaling of the softness ellipsoid.

Keyer Channel

Use the Keyer Channel to extract a key from a red, green, or blue channel, or from a custom value.

To access the Keyer Channel menu, use:

- Batch, then select a node from the Node bin.

- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Keyer Channel Menu Settings

General Settings

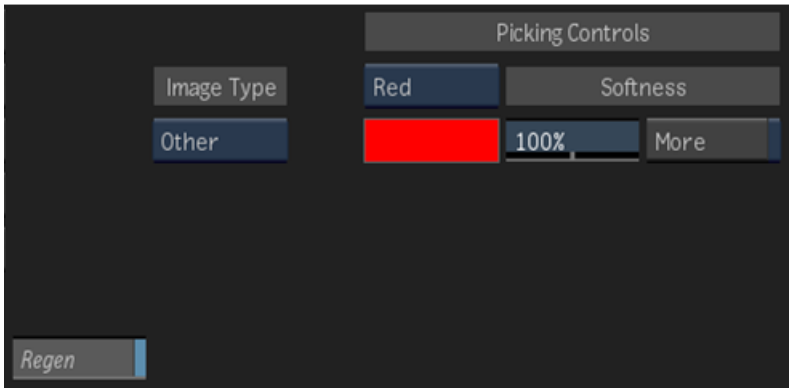


Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Channel Mode box Select a predefined or custom colour channel.

Key colour pot Activates the pick cursor. Use to sample an area of the image to display an average colour value for a custom channel.

Softness field Displays the percentage of transparency of the key-in clip. Editable.

More button Enable to enhance the keying effect.

Spread field The Spread field is available when Custom is selected from the Channel Mode box.Displays a value for the range of colours extracted from the key-in clip. Editable.

Regen button Enable to get dynamic updating of the image as you make changes.

Keyer HLS

Use the Keyer HLS to extract a key by adjusting tolerance and softness using hue, luminance, and saturation ranges.

To access the Keyer HLS menu, use:
- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).

■ The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Keyer HLS Menu Settings

General Menu Settings

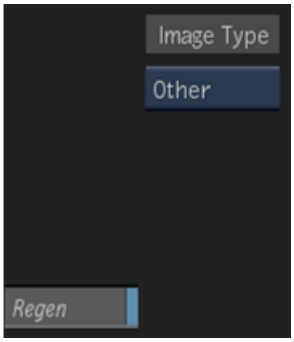
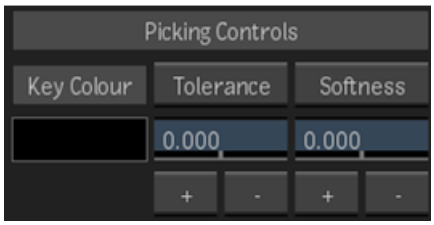


Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Regen button Enable to get dynamic updating of the image as you make changes.

Picking Controls Settings



Key Colour colour pot Activates the pick cursor. Use to sample an area of the image to display the average colour value.

Tolerance button Activates the pick cursor. Use to select the maximum and minimum values for the tolerance range in the image window.

Master Tolerance field Displays the value added to the existing tolerance of each channel. Resets to 0 after each use.

+ Increases tolerance values.

- Decreases tolerance values.

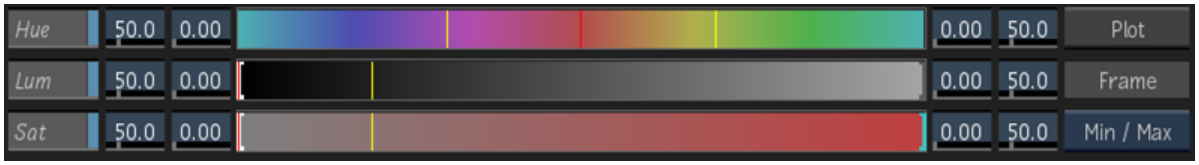
Softness button Activates the pick cursor. Use to select the maximum and minimum values for the softness range in the image window.

Master Softness field Displays the value added to the existing softness of each channel. Resets to 0 after each use.

+ Increases softness values.

- Decreases softness values.

Colour Control Settings



Hue button Enable to extract hue values for the key.

Minimum Softness field Displays the minimum hue value of the softness range. Editable.

Minimum Tolerance field Displays the minimum hue value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum hue value of the tolerance range. Editable.

Maximum Softness field Displays the maximum hue value of the softness range. Editable.

Luminance button Enable to extract luminance values for the key.

Minimum Softness field Displays the minimum luminance value of the softness range. Editable.

Minimum Tolerance field Displays the minimum luminance value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum luminance value of the tolerance range. Editable.

Maximum Softness field Displays the maximum luminance value of the softness range. Editable.

Saturation button Enable to extract saturation values for the key.

Minimum Softness field Displays the minimum saturation value of the softness range. Editable.

Minimum Tolerance field Displays the minimum saturation value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum saturation value of the tolerance range. Editable.

Maximum Softness field Displays the maximum saturation value of the softness range. Editable.

Plot button Activates the pick cursor. Use to sample an area of the image to display its colour value.

Frame option box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Keyer Luma

Use the Keyer Luma to extract a key from the luminance of a clip.

To access the Keyer Luma menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Keyer Luma Menu Settings

General Settings

Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Master Tolerance field Displays the value added to the existing tolerance of each channel. Enter 100 for an entirely opaque matte.

Master Softness field Displays the softness. Drag left or right to adjust the percentage value or enter a new percentage value.

Relative To box Select to calculate softness and tolerance ranges relative to 0 and 1 colour values, or relative to the maximum luminance.

Relative To field Displays the maximum luminance. This field is active if the Maximum Luminance option is selected in the Relative To box.

Get Maximum Value button Analyses the image to determine the maximum luminance value.

Plot button Activates the pick cursor. Use to sample an area of the image to display its colour value.

Frame option box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Keyer RGB

Use the Keyer RGB to extract a key by adjusting tolerance and softness using red, green, and blue ranges.

To access the Keyer RGB menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Keyer RGB Menu Settings

General Settings

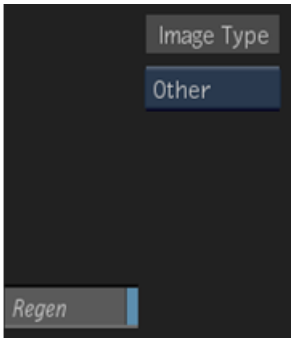
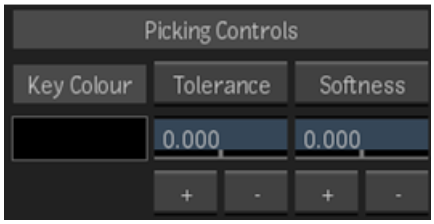


Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Regen button Enable to get dynamic updating of the image as you make changes.

Picking Controls Settings



Key Colour colour pot Activates the pick cursor. Use to sample an area of the image to display the average colour value.

Tolerance button Activates the pick cursor. Use to select the maximum and minimum values for the tolerance range in the image window.

Master Tolerance field Displays the value added to the existing tolerance of each channel. Resets to 0 after each use.

+ Increases tolerance values.

- Decreases tolerance values.

Softness button Activates the pick cursor. Use to select the maximum and minimum values for the softness range in the image window.

Master Softness field Displays the value added to the existing softness of each channel. Resets to 0 after each use.

+ Increases softness values.

- Decreases softness values.

Colour Control Settings

Red Channel button Enable to extract red channel values for the key.

Minimum Softness field Displays the minimum red channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum red channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum red channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum red channel value of the softness range. Editable.

Green Channel button Enable to extract green channel values for the key.

Minimum Softness field Displays the minimum green channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum green channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum green channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum green channel value of the softness range. Editable.

Blue Channel button Enable to extract blue channel values for the key.

Minimum Softness field Displays the minimum blue channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum blue channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum blue channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum blue channel value of the softness range. Editable.

Plot button Activates the pick cursor. Use to sample an area of the image to display its colour value.

Frame option box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Keyer RGBCMYL

Use the Keyer RGBCMYL to extract a key by adjusting tolerance and softness using red, green, blue, cyan, magenta, yellow, and luminance ranges.

To access the Keyer RGBCMYL menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Keyer RGBCMYL Menu Settings

General Settings

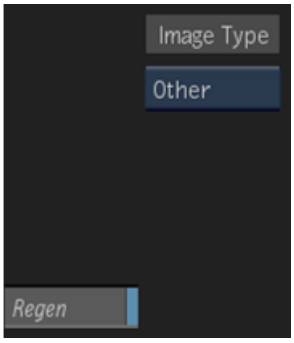
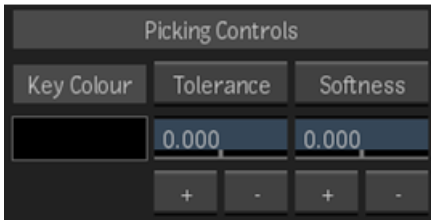


Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Regen button Enable to get dynamic updating of the image as you make changes.

Picking Controls Settings



Key Colour colour pot Activates the pick cursor. Use to sample an area of the image to display the average colour value.

Tolerance button Activates the pick cursor. Use to select the maximum and minimum values for the tolerance range in the image window.

Master Tolerance field Displays the value added to the existing tolerance of each channel. Resets to 0 after each use.

+ Increases tolerance values.

- Decreases tolerance values.

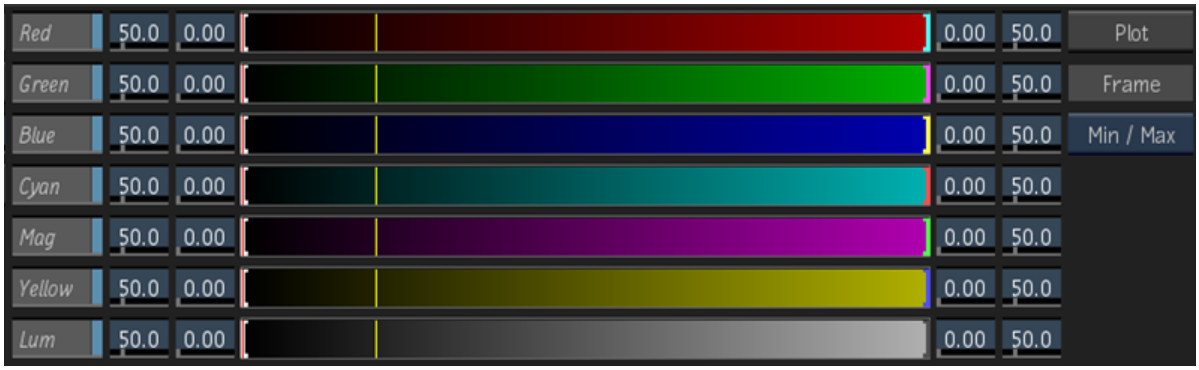
Softness button Activates the pick cursor. Use to select the maximum and minimum values for the softness range in the image window.

Master Softness field Displays the value added to the existing softness of each channel. Resets to 0 after each use.

+ Increases softness values.

- Decreases softness values.

Colour Control Settings



Red Channel button Enable to extract red channel values for the key.

Minimum Softness field Displays the minimum red channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum red channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum red channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum red channel value of the softness range. Editable.

Green Channel button Enable to extract green channel values for the key.

Minimum Softness field Displays the minimum green channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum green channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum green channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum green channel value of the softness range. Editable.

Blue Channel button Enable to extract blue channel values for the key.

Minimum Softness field Displays the minimum blue channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum blue channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum blue channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum blue channel value of the softness range. Editable.

Cyan Channel button Enable to extract cyan channel values for the key.

Minimum Softness field Displays the minimum cyan channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum cyan channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum cyan channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum cyan channel value of the softness range. Editable.

Magenta Channel button Enable to extract magenta channel values for the key.

Minimum Softness field Displays the minimum magenta channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum magenta channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum magenta channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum magenta channel value of the softness range. Editable.

Yellow Channel button Enable to extract yellow channel values for the key.

Minimum Softness field Displays the minimum yellow channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum yellow channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum yellow channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum yellow channel value of the softness range. Editable.

Luminance button Enable to extract luminance values for the key.

Minimum Softness field Displays the minimum luminance value of the softness range. Editable.

Minimum Tolerance field Displays the minimum luminance value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum luminance value of the tolerance range. Editable.

Maximum Softness field Displays the maximum luminance value of the softness range. Editable.

Plot button Activates the pick cursor. Use to sample an area of the image to display its colour value.

Frame option box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Keyer YUV

Use the Keyer-YUV to extract a key by adjusting tolerance and softness using luma (Y) and video component (U,V) ranges.

To access the Keyer YUV menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip and outputs a result.

Keyer YUV Menu Settings

General Settings

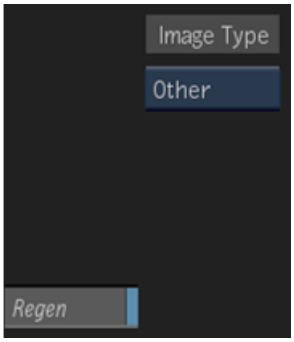
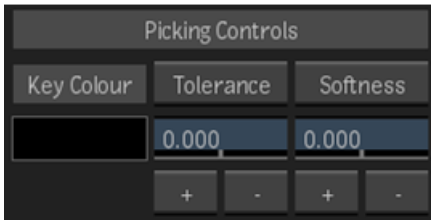


Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Regen button Enable to get dynamic updating of the image as you make changes.

Picking Controls Settings



Key Colour colour pot Activates the pick cursor. Use to sample an area of the image to display the average colour value.

Tolerance button Activates the pick cursor. Use to select the maximum and minimum values for the tolerance range in the image window.

Master Tolerance field Displays the value added to the existing tolerance of each channel. Resets to 0 after each use.

+ Increases tolerance values.

- Decreases tolerance values.

Softness button Activates the pick cursor. Use to select the maximum and minimum values for the softness range in the image window.

Master Softness field Displays the value added to the existing softness of each channel. Resets to 0 after each use.

+ Increases softness values.

- Decreases softness values.

Colour Control Settings

Y Channel button Enable to extract luminance channel values for the key.

Minimum Softness field Displays the minimum luminance value of the softness range. Editable.

Minimum Tolerance field Displays the minimum luminance value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum luminance value of the tolerance range. Editable.

Maximum Softness field Displays the maximum luminance value of the softness range. Editable.

U button Enable to extract blue-luminance difference values for the key.

Minimum Softness field Displays the minimum blue-luminance difference value of the softness range. Editable.

Minimum Tolerance field Displays the minimum blue-luminance difference value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum blue-luminance difference value of the tolerance range. Editable.

Maximum Softness field Displays the maximum blue-luminance difference value of the softness range. Editable.

V button Enable to extract red-luminance difference values for the key.

Minimum Softness field Displays the minimum red-luminance difference value of the softness range. Editable.

Minimum Tolerance field Displays the minimum red-luminance difference value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum red-luminance difference value of the tolerance range. Editable.

Maximum Softness field Displays the maximum red-luminance difference value of the softness range. Editable.

Plot button Activates the pick cursor. Use to sample an area of the image to display its colour value.

Frame option box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Lens Distort

Use the Lens Distort to rectify or simulate lens distortions caused by camera lens or perspective irregularities that can result in skewed angles.

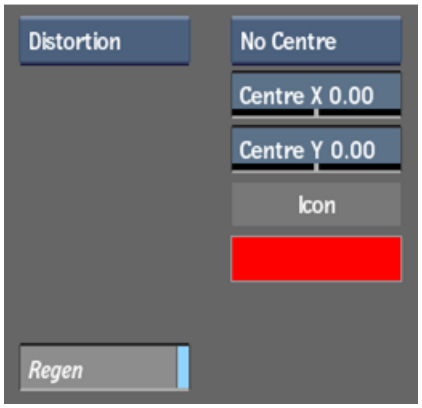
To access the Lens Distort menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).

This node accepts a front and matte clip, and outputs a result and an outmatte.

Lens Distort Menu Settings

General Settings



Distortion box Select whether to perform a lens distortion or rectification. All other radial distortion controls are implemented based on this selection.

Regen button Enable to get dynamic updating of the image as you make changes.

Centre box Select the size of the crosshair icon in your image.

Centre X field Displays the horizontal position of the centre of the lens. Editable.

Centre Y field Displays the vertical position of the centre of the lens. Editable.

Icon colour pot Displays the colour of the crosshair in your image. Editable.

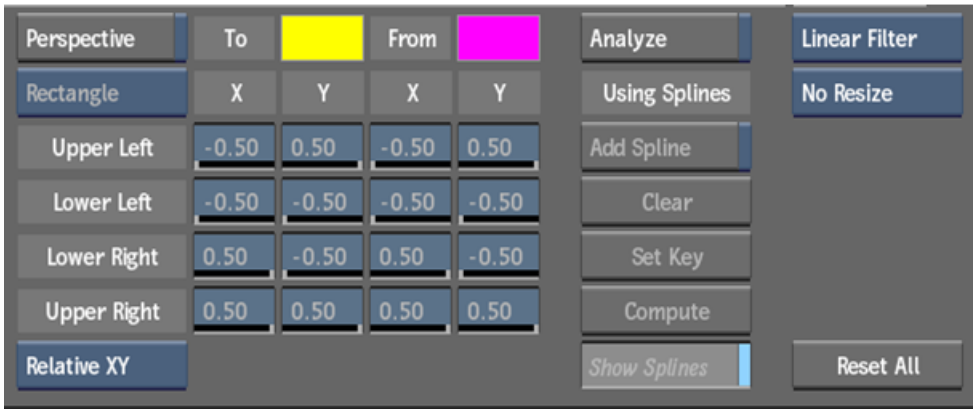
Radial Settings

Lens Mag field Displays the magnitude of radial distortion or rectification. Editable.

Lens Adj field Displays the radial distortion or rectification. Editable.

Anamorph field Displays the ratio of distortion or rectification along the X or Y axis. A value greater than 1.00 stretches more on the X axis, while a value less than 1.00 stretches more on the Y axis. Editable.

Lens Distort Controls



Perspective button Enable to perform a perspective rectification.

Perspective Type box Select an option to map the source anchor points to a rectangle, preserving either the horizontal or vertical edges of the image, or mapping one four-corner free-form shape to another. Active when Perspective button is enabled.

Range box Select whether to set the X and Y range from -0.5 to 0.5 (relative) or set the X and Y range to half of the image size (absolute).

Source colour pot Displays the colour of the source anchor points. Editable.

Destination colour pot Displays the anchor points of the destination shape. Editable.

Upper Left X field Displays the horizontal value for the upper left source anchor. Editable.

Lower Left X field Displays the horizontal value for the lower left source anchor. Editable.

Lower Right X field Displays the horizontal value for the lower right source anchor. Editable.

Upper Right X field Displays the horizontal value for the upper right source anchor. Editable.

Upper Left Y field Displays the vertical value for the upper left source anchor. Editable.

Lower Left Y field Displays the vertical value for the lower left source anchor. Editable.

Lower Right Y field Displays the vertical value for the lower right source anchor. Editable.

Upper Right Y field Displays the vertical value for the upper right source anchor. Editable.

NOTE The following fields are available when Free Form is selected from the Perspective Type box.

Upper Left X field Displays the horizontal value for the upper left destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Lower Left X field Displays the horizontal value for the lower left destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Lower Right X field Displays the horizontal value for the lower right destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Upper Right X field Displays the horizontal value for the upper right destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Upper Left Y field Displays the vertical value for the upper left destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Lower Left Y field Displays the vertical value for the lower left destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Lower Right Y field Displays the vertical value for the lower right destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Upper Right Y field Displays the vertical value for the upper right destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Analyse button Enable to use the spline controls.

Add Spline button Enable to draw a spline along a feature that you want to straighten.

Clear button Click to clear splines at the selected keyframe.

Set Key button Click to set a keyframe based on the result of the analysis.

Compute button Click to analyse the image based on the splines you added.

Show Splines button Enable to display splines in the image window.

Filter box Defines the type of filter for your distortion or rectification.

Resize box Assigns the resize option you choose to apply to your image (if any distortion or rectification is present).

Reset All button Click to restore default settings.

Map Convert

Use Map Conver to provide conversion transformation for image-based lighting (IBL) and environment maps.

You can use the results of Map Convert in Action as an IBL map (page 690).

To access the 2D Transform menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front and matte clip, and outputs a result an outmatte.

Map Convert Menu Settings

Input and Output Format Settings

Input Format	Output Format
Cubic	Spheric
Horizontal	

Input Format box Select the type of image you want to convert.

Cubic Type box Select horizontal or vertical to identify the type of cubic image you are inputting. Select Spliced Faces to stitch together multiple images to create a horizontal cubic image. Available when Cubic is selected in the Input Format box.

Output Format box Select the output format for the image you are converting.

The Map Convert effect accepts the following format types:

Type:	Description:
Spheric	Sometimes referred to as a chrome ball or mirrored ball. An image of a mirrored ball in an assumed orthogonal projection.
Angular	Similar to the mirrored ball format, except that the radial dimension is mapped linearly with the angle, giving better sampling around the edges.
Cylindrical	An image mapped onto an unfolder cylinder (also known as longitude and latitude maps).
Cubic (Horizontal or Vertical)	An unfolded six-sided cube.
Cubic (Spliced Faces)	A type of cubic map available as an input format using a different frames displayed as an unfolded cube showing six sides of the image. To use this input type, frames from the input clip are used as the 6 faces.
Polar	A coordinate system for locating points in which each point on a plane is determined by a distance from a fixed point and an angle from a fixed direction. When using this input format, only Cartesian is available as the output format.
Cartesian	A coordinate system for locating points on a plane by measuring the horizontal and vertical distances from an arbitrary origin to a point. When using this input format, only Polar is available as the output format.

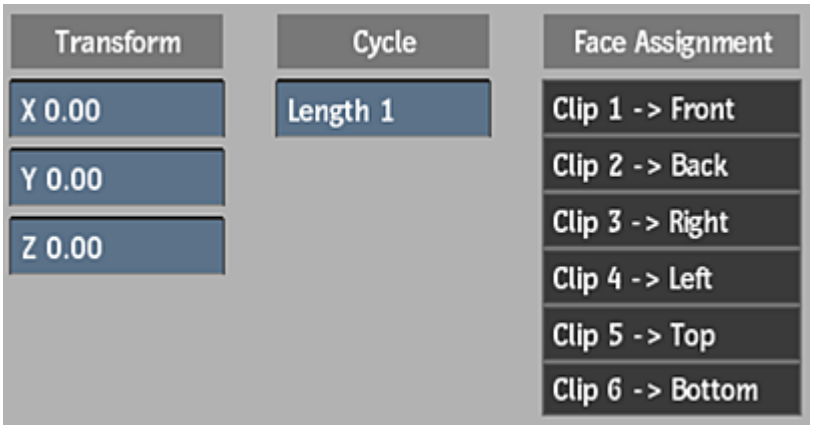
Transform Settings

These Transform settings apply to Cylindrical, Angular, Spheric, and Cubic, (Horizontal or Vertical) input formats.

Transform X Position field Displays the horizontal value of the image. Editable.

Transform Y Position field Displays the vertical value of the image. Editable.

Transform Z Position field Displays the depth value of the image. Editable.



In addition to the common transform settings, a Cubic (Spliced Faces) input format also contains the following settings:

Sequence Length field When creating a cubic map from spliced images, indicate how frequently images change in the sequence. Each change in the sequence gets mapped to a different face of the cube.

Face Assignments fields These locked fields display the order that the face assignment follows when creating the cubic map.

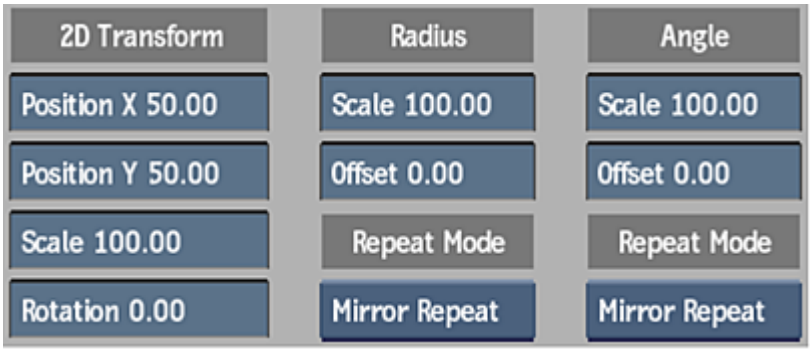
X Position field Displays the horizontal position of the output image. Editable.

Y Position field Displays the vertical position of the output image. Editable.

Scale field Displays the scale of the output image. Editable.

Rotation field Displays the rotation of the output image. Editable.

Repeat Mode box Select an option to fill the empty portions of the frame.



Radius Scale field Displays the scale of the radius for a cartesian to polar conversion. Editable.

Radius Offset field Displays the offset of the radius for a cartesian to polar conversion. Editable

Angle Scale field Displays the scale of the angle for a cartesian to polar conversion. Editable.

Angle Offset field Displays the offset of the Angle Scale for a cartesian to polar conversion. Editable.

Angular Repeat Mode box Select an option to fill the empty portions of the frame.

Output Size Settings

Output Size box Select whether to output the map at the size of the input image (Automatic), or select Custom to change the size of the output image.

Output Width field Displays the width of the output image. Editable when you select a custom output size.

Output Height field Displays the height of the output image. Editable when you select a custom output size.

Ratio field Displays the width to height ratio of automatic outputs. Non-editable.

Master Keyer

Use the Master Keyer to gesturally pull a key.

To access the Master Keyer menu, use:

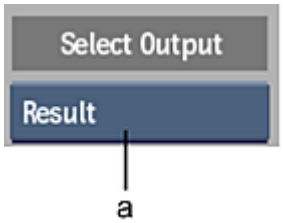
- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front, back and key in clip, and outputs a result and an outmatte.

Add a Master Keyer node to the pipeline when you need to create an accurate key of a clip. Use the Master Keyer node to automatically suppress colour spill, to colour correct, and to remove noise or grain. See Creating and Refining a Key in the Master Keyer (page 1032).

While there is no setup menu for the Master Keyer node, it can be saved as part of the Batch setup, or alternatively, as a Custom node. The Master Keyer is also available from the Keyer node.

Select an output type from the Select Output box.



(a) Select Output box

TIP In order for the keyboard shortcuts `M` and `D` to correspond to Matte and Degrain, the Master Keyer in Batch must have focus and the viewport cannot be set to a schematic view. If the schematic has focus, then `M` corresponds to Move and `D` corresponds to Delete.

The Master Keyer node processes gaps in clips set to No Media based on the input tab receiving the information.

Input	Result
Front	No media
Back	Black frames
Matte	White frames

Master Keyer Menu Settings

MasterK Settings

Primary Sample colour pot Activates a pick cursor. Use to sample an area of the image.

Mix field Displays the mix between the primary and secondary sample. Drag right to include more of the secondary sample or left to include less.

Secondary Sample colour pot Activates a pick cursor. Use to sample an area of the image where you do not want any softness in the matte.

Patch1 button Enable to isolate a range of colours to be included in, or excluded from, the key. This button is active when an area of the image is sampled and Patch1 is selected in the Sampling box.

Patch box Select the areas of the matte to which the patch is applied.

Patch Range field Displays the colour range value. Editable.

Patch Softness field Displays the softness value. Editable.

Patch2 button Enable to isolate a range of colours to be included in, or excluded from, the key. This button is active when an area of the image is sampled and Patch2 is selected in the Sampling box.

Patch box Select the areas of the matte to which the patch is applied.

Patch Range field Displays the colour range value. Editable.

Patch Softness field Displays the softness value. Editable.

Patch3 button Enable to isolate a range of colours to be included in, or excluded from, the key. This button is active when an area of the image is sampled and Patch3 is selected in the Sampling box.

Patch box Select the areas of the matte to which the patch is applied.

Patch Range field Displays the colour range value. Editable.

Patch Softness field Displays the softness value. Editable.

Degrain button Enable to modify the grain and de-sharpen the edges of the key.

Size field Displays the size of the grain in the image. Editable.

Edges field Displays the level of sharpness of edges in the image. Editable.

More button Enable to increase the overall Degrain effect.

Sampling box Select the sampling method required to refine the key.

Reset button Resets sampling parameters (excluding the mix value and key colour).

Primary Reference colour pot Activates a pick cursor. Use to sample an area of the image.

Secondary Reference colour pot Activates a pick cursor. Use to sample an area of the image where you do not want any softness in the matte.

Colour Settings

Luma field Displays the luminance value. Editable.

Edge Size field Displays the range for the blend. Both the Luma field and the Edge Balance trackball are affected. Editable.

Spill colour pot Displays the colour to suppress in the clip. Editable.

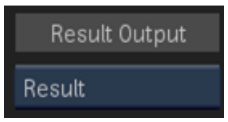
Range field Displays the range for the removal of colour spill along the edges of the key. Drag right to soften the edge and remove colour spill further into the key. Drag left to harden, or create a thinner, edge.

Hue field Displays the hue value. Editable.

Auto Colour Correct button Enable to apply colour settings to the front clip.

Reset button Resets the colour settings.

Result Output



Result Output button Select the output mode for the result.

Matte Curves

Use Matte Curves to adjust the luminance of input mattes.

To access the Matte Curves menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front, back and matte clip, and outputs a result.

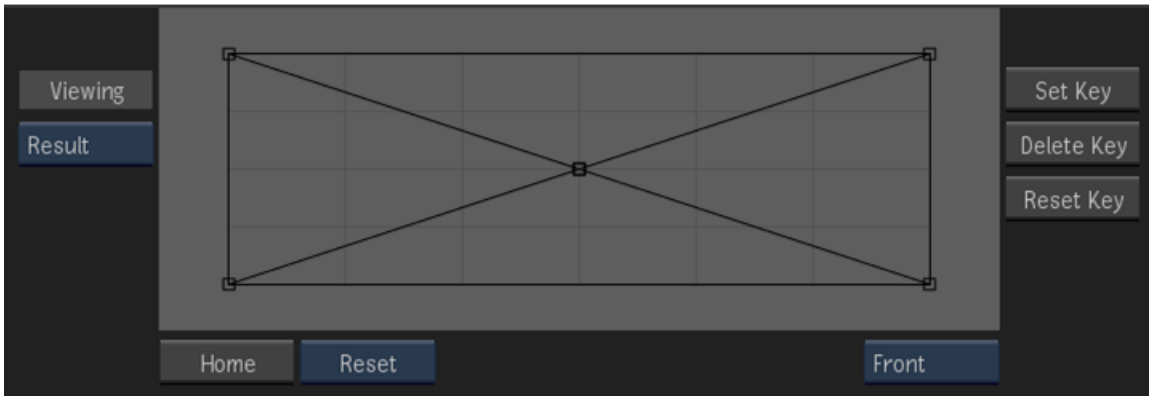
NOTE The Matte Curves node is the same as the Modular Keyer Matte Curves node.

The Matte Curves node processes gaps in clips set to No Media based on the input tab receiving the information.

Input	Result
Front	Black frames
Back	Black frames
Matte	No media

Matte Curves Menu Settings

General Settings



Result box Select the type of clip you want to process.

Home button Reverts to the original view.

Reset box Resets the curve view.

Matte box Select the matte curve you want to adjust.

Set Key button Sets a keyframe at the selected frame.

Delete Key button Deletes the selected keyframe.

Reset Key button Resets the curves at the selected keyframe.

Matte Edge

Use the Matte Edge effect to create an image based on the detected edges, or use it to modify the edges of a matte to help in creating a key.

To access the Matte Edge menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front and matte clip as inputs, and outputs a result. You can add the Matte Edge node to any part of the pipeline except the Front pipe of the CBLend node. The Matte Edge node is not appropriate for this pipe because its result is a matte.

Matte Edge Menu Settings

General Settings

Rendering Mode box Select whether to render in Automatic, Progressive or Interlaced mode.

Regen button Enable to get dynamic updating of the image as you make changes.

Edges Settings

Edges button Enable to create an image composed of the edges in an image.

Matte Edge Mode box Select whether to use the Basic, Advanced or Smooth edge-detection algorithm. Certain settings change based on this selection.

Basic Settings

Width field Displays the value that affects how the edge-detection algorithm determines whether each pixel forms part of the edge. Editable.

Minimum Input field Displays the lower limit of the luminance values. Pixels with lower values are mapped to black. Editable.

Maximum Input field Displays the upper limit of the luminance values. Pixels with higher values are mapped to white. Editable.

Advanced Settings

Minimum Input field Displays the lower limit of the luminance values. Pixels with lower values are mapped to black. Editable.

Maximum Input field Displays the upper limit of the luminance values. Pixels with higher values are mapped to white. Editable.

Inner Width field Displays the width of the inner edges of the matte. Editable.

Outer Width field Displays the width of the outer edges of the matte. Editable.

Width Proportional button Enable to affect the inner and outer width proportionally.

Inner Softness field Displays the level of softness on the inner edges of the matte. Editable.

Outer Softness field Displays the level of softness on the outer edges of the matte. Editable.

Softness Proportional button Enable to affect the inner and outer softness proportionally.

Smooth Settings

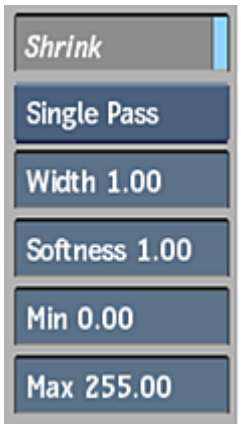
Smooth Minimum field Enter a lower limit for the edge detection. Editable.

Smooth Maximum field Enter the upper limit for the edge detection. Editable.

Smooth Softness field Enter a value for softness of the edge. Editable.

Smooth Gain field Enter a value for the softness gain of the edge. Editable.

Shrink Settings



Shrink button Enable to remove pixels from the edge of the matte.

Shrink Mode box Select whether to use shrink in a single pass or iterative mode. Single Pass mode allows you to control edge softness.

Shrink Width field Displays the width of the border that is removed from the edge of the matte. Editable.

Softness field Displays the amount of softness applied to the edges of the matte.

Minimum Input field Displays the lower limit of the luminance values. Pixels with lower values are mapped to black. Editable.

Maximum Input field Displays the upper limit of the luminance values. Pixels with higher values are mapped to white. Editable.

Erode Settings

Erode button Enable to blend the light and dark edges of the matte.

Erode Width field Displays the width of the matte border to soften. Editable.

Blur Settings

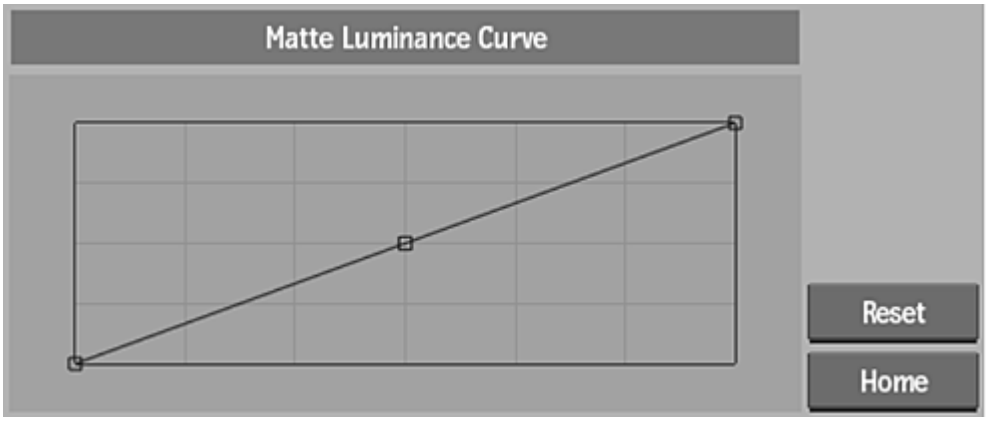
Blur button Enable to apply a softening Gaussian blur filter to the edge of the matte.

Blur Width field Displays the width of the blur applied to the edge of the matte. Editable.

Blur Height field Displays the height of the blur applied to the edge of the matte. Editable.

Proportional button Enable to constrain blur amount proportions.

Matte Luminance Curve Settings

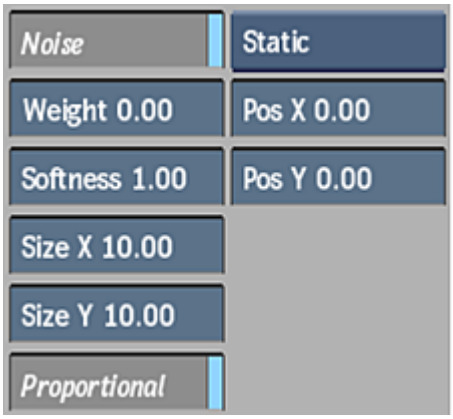


Matte Luminance Curve Controls the blending of the edges.You can add keyframes to the curve using Add mode, move keyframes with Move mode, and modify the curve's shape using tangent handles.

Reset box Resets the curve view.

Home button Reverts to the original view.

Noise Settings



Noise button Enable to add noise to the transparent areas of the matte.

Noise Mode box Select the mode of noise to apply throughout the length of the image. Static adds noise as a still frame; Impulse adds noise that changes at each frame.

Select:	To add noise:
Static	As a still frame. You can move the static noise using the Position X and Y fields.
Impulse	Changing at each frame, but always starting from the same seed.

Weight field Displays the level of noise in the clip. Values below 1 add more white noise, and values above 1 add black noise. Editable.

Softness field Displays the level of Gaussian softness added to the noise. Editable.

Position X field Displays the X position of the noise when using Static mode. Editable.

Position Y field Displays the Y position of the noise when using Static mode. Editable.

Size X field Displays the size of the noise along the X axis. Editable.

Size Y field Displays the size of the noise along the Y axis. Editable.

Proportional button Enable to affect the X and Y sizes proportionally.

Mono

Use Mono to generate a monochrome copy of the front clip.

To access the Mono menu, use:

- Batch, then select a node from the Node bin.

- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

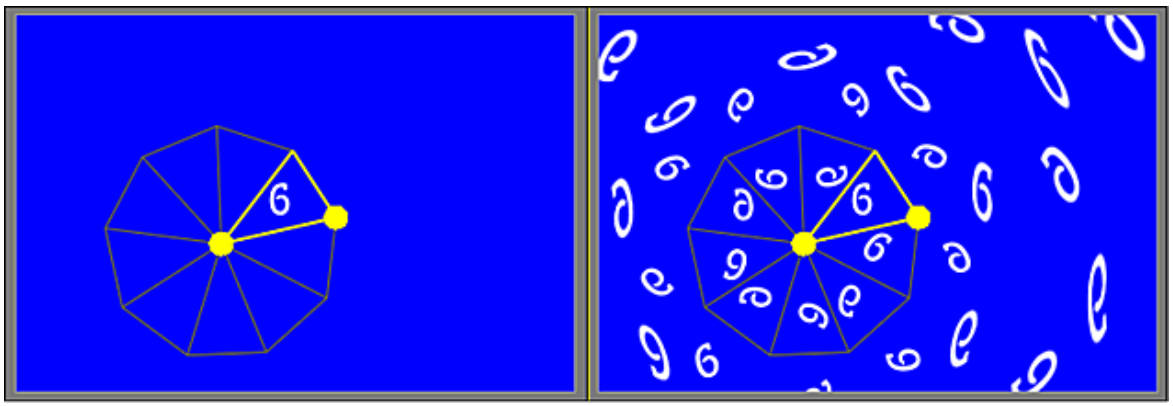
Mono Menu Settings

General Settings

Luminance box Select the channel you want to use to create the monochrome clip.

Motif

Use the Motif node to create a tiled symmetrical texture.



To access the Motif menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front and a matte input, and outputs a result clip and a matte output.

Source clips can be transformed before symmetrical effects are applied. Transformations are applied to both the front and matte clips simultaneously. You can select the type of symmetry mode to use on the transformation, and define its parameters.

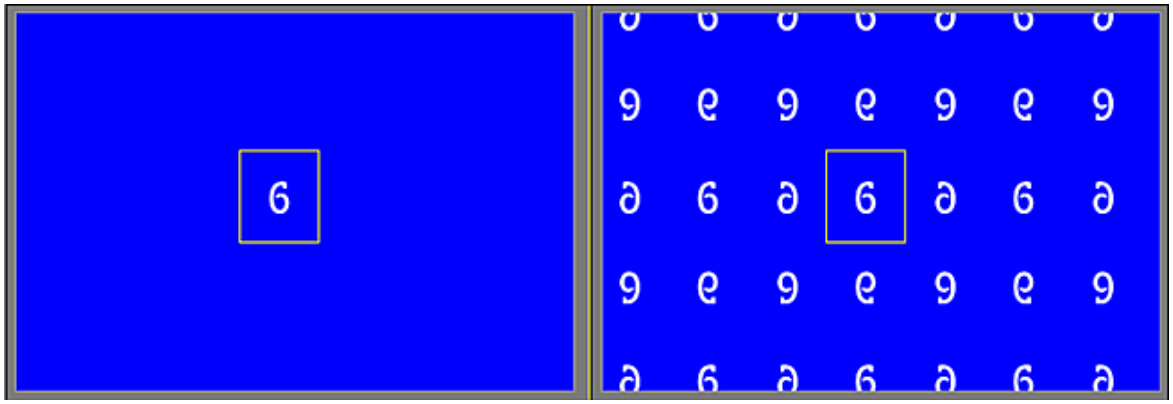
The radial symmetry mode displays a user-defined widget, which is a polygon with an equal number of sectors to the order of symmetry. You select the sector to use as the originating tile, the basis for a kaleidoscopic texture. Additional parameters allow you to define the center of symmetry of the image, and the angle and radius of the originating tile.

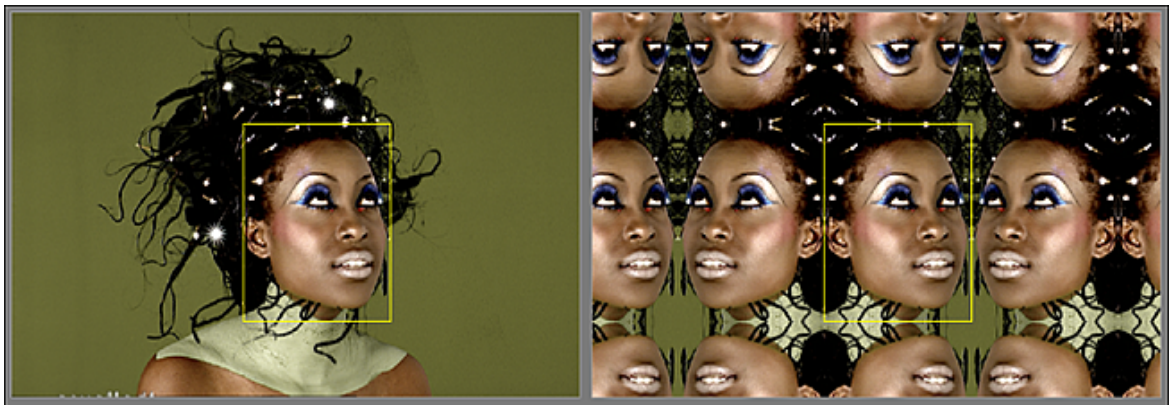
Front and Result views in radial symmetry mode

In Radial mode, you can also reuse the pixel on the edge of a sector to pad the space between outer tiles.

Result views with the repeating disabled and enabled

The region of interest (ROI) effect uses a user-defined rectangular or triangular selection as the originating tile to create a basic mirrored texture.





Front and Result views in region of interest symmetry mode

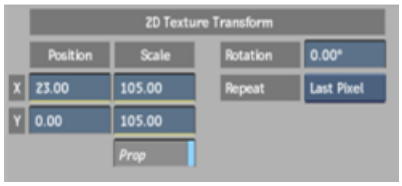
Image courtesy of The House

Motif Menu Settings

Motif Node

The Motif node displays 2D Texture Transform controls that allow you to simultaneously change the position, scaling, rotation, and type of image padding. The Symmetry Mode controls display a colour pot for the original tile selection.

2D Texture Transform settings



X Position field Displays the horizontal offset in pixels of the input and matte clips. Editable.

Y Position field Displays the vertical offset in pixels of the input and matte clips. Editable.

X Scaling field Displays the amount of horizontal scaling to apply to the front and matte clips.

Y Scaling field Displays the amount of vertical scaling to apply to the front and matte clips. Editable.

Scaling proportional button Enable to effect Scale X and Scale Y proportionally.

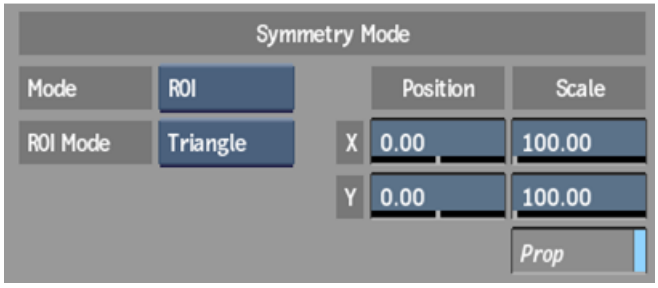
Rotation field Displays the angle of rotation of the front and matte clips. Editable.

Repeat button Select fill options to pad the empty portions of the frame with the last line of pixels, a repeated (rolled) image, or black pixels.

Symmetry Mode settings

Mode button Select the type of symmetry effect to apply to the transformation. Additional parameters are displayed in Radial mode.

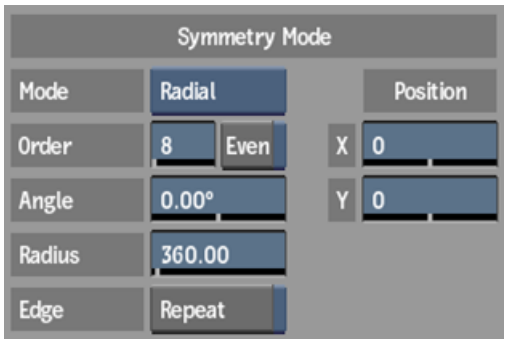
NOTE The following settings are available when the Symmetry Mode button is set to ROI.



Symmetry Mode box Select between a square or triangle region of interest mode.

Proportional button Enable to effect Scale X and Scale Y proportionally.

NOTE The following settings are available when the Symmetry Mode button is set to Radial.



Order field Displays the order of symmetry and indicates the number of sectors or sides of the polygonal widget.An order of 2 creates a basic mirrored image. An order of symmetry can have an odd number as a value. Tiles are mirrored in a counter-clockwise direction, therefor the sector to the right of the highlighted sector may not be its mirror image. Editable.

Even button Enable to maintain an even order of symmetry.If the order of symmetry is an odd value, an extra tile will be added, ensuring that adjacent tiles are mirror images of each other.

Angle field Displays angle of the axis of symmetry. Editable.

Radius field Displays the pixel length of the sector radius. Editable.

Edge Repeat button Enable to use the pixel colour on the sector's edge to pad the space between outer tiles. Displayed in Radial mode.

Position X field Displays the horizontal position of the centre of symmetry. Editable.

Position Y field Displays the vertical position of the centre of symmetry. Editable.

Softness settings

Softness field Displays the amount by which the motif effect is out of focus. Editable.

Display settings

Show Widgets button Enable to display the widgets in the clip.

Widget colour pot Select the colour used to highlight the originating tile. Editable.

Gestural Modifications

Modifications to the originating tile and symmetry mode parameters can be made gesturally in any view.

Drag:	To:
Outer edge of the region of interest	Change the width or height of the selection.
Corner of the region of interest	Change the width and height of the selection.
Inside the region of interest or widget	Move the selection.
The widget centre	Change the centre of symmetry.
Circle on the outer edge of the widget	Change the radius and the angle of symmetry.

Motion Analysis

Use Motion Analysis to analyse image displacement in a frame with respect to the frame before it.

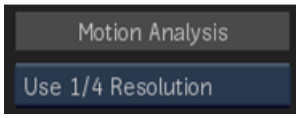
To access the Motion Analysis menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip, and outputs forward and backward vectors.

Motion Analysis Menu Settings

General Settings



Quality box Select Use Full Resolution to render the image at the current resolution, or an option with decreased motion analysis accuracy and increased rendering speed.

Motion Blur

Use Motion Blur to simulate the blur created by fast moving objects.

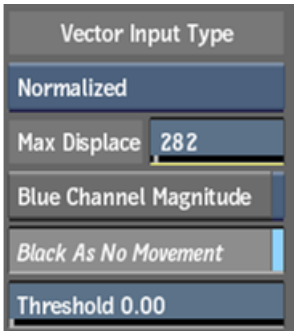
To access the Motion Blur menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts front, matte and forward vector clips and outputs a result, outmatte, or forward Vector clip.

Motion Blur Menu Settings

Vector Input Type Settings



Vector Input Type button Select whether the vector inputs are Absolute or Normalized.

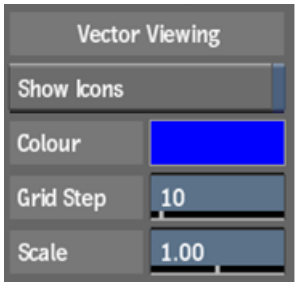
Max Dispersion field Displays the level of motion displacement in the image. Set to the same value that was set in the 3D application. Editable.

Blue Channel Magnitude field Enable to use the blue channel as a magnitude multiplication of the maximum displacement value.

Black As No Movement button Enable to set black pixels as no movement in the motion vector. When enabled, you can set a threshold value for near-black pixels.

Threshold field Displays the level of near-black pixels taken into account in the conversion.

Vector Viewing Settings



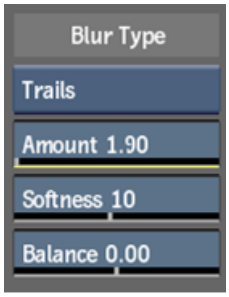
Show Icons button Displays the forward motion vectors in the Result view.

Colour pot Changes the display colour of the forward and backward motion vectors.

Grid Step field Displays the length of the pixel area used to calculate each vector.

Scale field Displays the size of the motion vectors.

Blur Type Settings



Blur Type button Enable this button to select between trails and samples. Choose trails for a pixel-based motion blur. Choose sample for an accumulation-based motion blur.

Blur Type amount field Displays the amount of motion blur applied to the image. Editable.

Blur Type softness field Displays the amount of softness applied to the trails. Only available when trails is selected as the blur type. Editable.

Blur Type balance field Displays the amount of blur that either precedes or follows the image. The more positive the number, the more blur precedes the image. The more negative the number, the more blur follows the image. Editable.

Artefacts Settings

Vector Softness field Displays the amount of softness that is applied to reduce the quality of vectors, and therefor solve precision artefacts. Editable.

Post Blur field Displays the amount of global blur applied after other effects have been applied. Apply conservatively. Editable.

Matte Fill field Display the amount used to fill in gaps in the matte. Best applied after softness adjusted. Only available when the blur type is trials. Editable.

Colour Interpolation Settings

Colour Interpolation button Enable this button to verify that the source is already premultiplied. In this case, the colour interpolation effect must treat the image differently for the effect to work properly. This button is only functional when there is an input matte.

Motion Opacity Curve

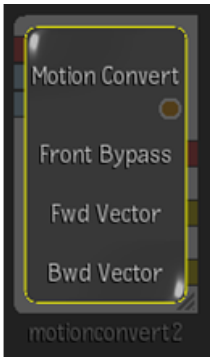
Motion Opacity Curve Allows you to control the opacity of the trail by shaping it using the curve.

Home button Reverts to the original view.

Reset box Resets the curve view.

Motion Convert

Use Motion Convert to convert normalized 2D vectors into absolute vectors that you can work with in Flame.



To access the Motion Convert menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).

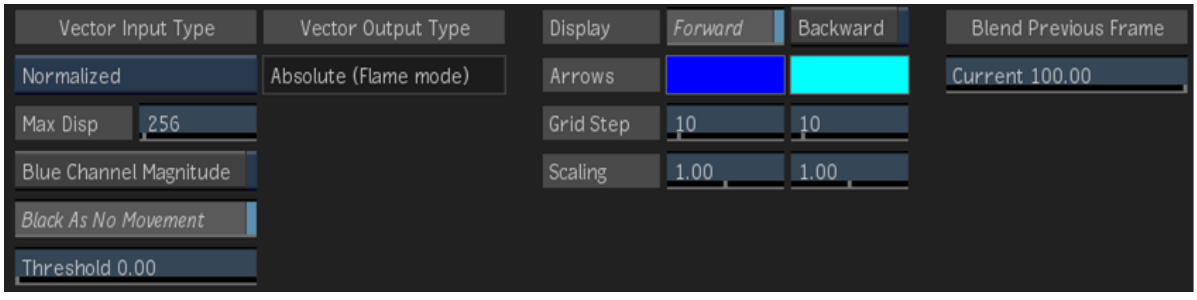
This node accepts a front clip, and forward and backward vectors. It outputs a result, as well as forward and backward vectors.

Motion Vectors are 2D vectors that represent the displacement in normalized pixel units of a pixel in the current frame to the next frame (forward motion vector), or its displacement from the previous frame (backward motion vector). Motion Vectors can be rendered by a 3D application when dealing with synthetic images, or produced through image analysis when images come from live action footage.

You can attach the forward and backward outputs from the Motion Analysis node or import motion data from another source.

Motion Convert Menu Settings

Motion Convert Node



Vector Input Type box Select whether the vector inputs are Absolute or Normalized.

Maximum Displacement field Displays the level of motion displacement in the image. Set to the same value that was set in the 3D application.

Blue Channel Magnitude button Enable to use the blue channel as a magnitude multiplication of the maximum displacement value.

Black As No Movement button Enable to set black pixels as no movement in the motion vector. When enabled, you can set a threshold value for near-black pixels.

Threshold field Displays the level of near-black pixels taken into account in the conversion. Editable.

Vector Output Type Displays the absolute vector output type mode.

Forward button Enable to display the forward motion vectors in the Result view.

Forward Arrow colour pot Displays the current colour of forward motion vectors. Editable.

Backward button Enable to display the backward motion vectors in the Result view.

Backward Arrow colour pot Displays the current colour of backward motion vectors. Editable.

Forward Grid field Displays the length of the pixel area used to calculate each forward motion vector. For example, enter 5 to sample an area of 25 pixels.

Backward Grid field Displays the length of the pixel area used to calculate each backward motion vector. For example, enter 5 to sample an area of 25 pixels.

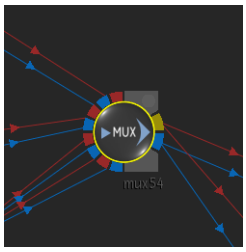
Forward Scaling field Displays the size of forward motion vectors. Editable.

Backward Scaling field Displays the size of backward motion vectors. Editable.

Blend Previous Frame field Displays the level of opacity of the current frame, which is overlaid on the previous frame. Set the value to 100 to display the current frame only.

MUX

The MUX (multiplexer) node is a tool that helps you create cleaner schematics by allowing you to have multiple RGBA inputs feeding your outputs. The selection of the active input can be changed over time, therefore MUX also acts as an animated switcher.



To access the MUX menu, use:

- The I/O tab in Batch.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts any number of front and matte clips, and outputs a result and outmatte.

Using the MUX Node

When entering Batch FX with existing Timeline FX or transitions, you may notice MUX nodes with multiple connections in the newly converted schematic. These MUX nodes are used to help recreate and represent your timeline in a flow graph environment.

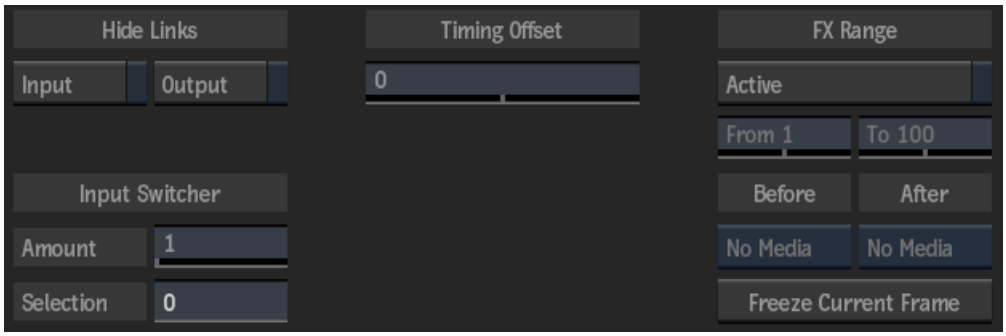
You can emulate this behaviour by adding your own MUX nodes to the schematic. If you need more inputs in your MUX node, use the Amount field in the MUX menu to set the number of inputs needed. To switch between inputs, use the Selection field to choose which input is active (this field can be animated with keyframes).

TIP If you add multiple inputs to a MUX node, you may have trouble seeing or connecting individual input sockets. In this case, you can expand the node by selecting it, and pressing `Shift` + `C`. Press the same keyboard shortcut again to collapse the node. You can also hide the input or output links of the MUX node to clean up a large schematic. Even when hidden, you can select a MUX node to display semi-transparent links.

In the case of timeline transitions, the MUX node also acts as a switcher to allow the proper inputs to be selected at the correct time.

In you enable Show Indicators in the Batch or Batch FX preferences, you can view the MUX Timing Offset and Range values directly under the node in the schematic.

MUX Menu Settings



Hide Links Settings

Input button Enable to hide all input links to the MUX node. Click the MUX node to display semi-transparent hidden links.

Output button Enable to hide all output links to the MUX node. Click the MUX node to display semi-transparent hidden links.

Input Switcher Settings

Amount field Displays the number of inputs on the node. Editable.

Selection field Displays the number of the input that has its output displayed in the Result window. Editable.

Timing Offset Settings

Timing Offset field Displays the number of frames by which the timing of the MUX pipeline is offset. You can animate settings in this field with keyframes. Editable.

FX Range Settings

Range Active button Enable to activate the range settings.

Range From field Displays the first impacted frame. Editable.

Range To field Displays the last impacted frame. Editable.

Range Before box Select an option to apply before the set frame range.

Range After box Select an option to apply after the set frame range.

NOTE There are two Ping Pong options included in the Range Before box and Range After box: Ping Pong and Ping Pong+. The Ping Pong repeat mode is inclusive, meaning that the last frame of the sequence is always repeated. For example, a five frame sequence would be: 1-2-3-4-5-5-4-3-2-1-1-2-3-4. The Ping Pong+ repeat mode is exclusive, meaning that the last frame of the sequence is never repeated creating a sequence that is always 1 frame shorter, but that does not create stuttering with a frame repeat.

Freeze Current Frame button Click to output the image of the currently selected frame for the duration of the clip.

Optics

Use Optics to add a glow effect to the clip in the process tree.

To access the Optics menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts front, back, and matte clips, and outputs a result.

The Optics node processes gaps in clips set to No Media based on the input tabs receiving the information.

Input	Result
Front	No media
Back	Black frames
Matte	No media

Optics Menu Settings

Optics Node Settings

Size field Displays the size of the glow effect. Editable.

Intensity field Displays the intensity level of the glow effect. Editable.

Noise field Displays the jitter level in the glow effect. Editable.

Transparency field Displays the transparency level of the front clip. Editable.

Optics Front button Enable to use the front clip.

Optics Back button Enable to use the back clip.

Invert button Enable to invert the matte clip.

Random button Enable to produce a different glow effect.

Interior colour pot Select a colour for the interior of the glow effect.

Exterior Colour pot Select a colour for the exterior of the glow effect.

Paint Node

About the Paint Node

The Paint node is a system that provides a scalable matte painting, retouching, or restoration workflow in Batch or Batch FX.

Due to its underlying technology, the Paint node automatically scales strokes when changing the resolution, ratio, or bit depth of input clips, or when switching from Full Resolution to Proxy mode. The Paint node supports “clipless” setups, which can be applied to any image input, while accurately reproducing the sequence of painted strokes.

The Paint node accepts a front and matte clip as input, and creates a result and output matte clip, respectively. You can paint on the result and output matte, with a selection of brushes in different paint modes.

The Paint node also allows you to connect multiple sources and use them to paint the contents of source images onto the result. This paint operation, applied with the Reveal paint tool, can be used with in-context overlay over the result image. A front and matte clip can be connected as a source by connecting the clips to a source node. The content of source front and source matte input can be used to create brush strokes on the result and output matte.

For information about the functionality of the Paint node, click here (page 1197).

The Paint node can be accessed from:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).

■ The Modular Keyer, then select a node from the Node bin.

The following views are available in the Paint node.

Select:	To display:
Front (F1)	The front clip or Batch or Batch FX tree input. The modifications to the front create the result clip.
Matte (F3)	The matte clip or Batch or Batch FX tree input. The modifications to the matte create the output matte.
Source Front (F1 F1)	The source front that is selected in the Sources list.
Source Matte (F3 F3)	The source matte that is selected in the Sources list.
Result (F4)	The result image. Paint can be applied to the result.
Output Matte (F4 F4)	The output matte image. Paint can be applied to the output matte.

Paint Node Menu Settings

Source Controls

Displays the front clip, matte clip, and sources. The Sources list manages the sources attached to the current node. A source is composed of a front source and a matte source. See Using Sources (page 1197) .

Add button Click to add a new source node, and select the source front and source matte. Ctrl-click to add a source node only.

Source Front option box Select Front Lock to use the current frame for the duration of the source front clip. Select Front On to unlock the clip and apply it in its original state.

Source Matte option box Select Matte Off to paint anywhere on the canvas, Matte On to limit painting to areas on the source matte, or Matte Invert to limit painting to areas outside the source matte.

Hide Strokes button Hides the strokes associated with the current source.

Clear Strokes button Clears the strokes associated with the current source.

Current Frame box Select to clear strokes for the Current Frame or the entire Sequence.

Sources List Displays the sources attached to the current node.

Brush Attributes

Sets the size of the brush and other brush attributes affecting how paint is applied to the canvas. See Brush Attributes and Attribute Modes (page 1202).

Size field Set the radius of the brush in pixels. Editable.

Rate field Set the rate at which to apply brush strokes to the canvas. Use a higher percentage value to produce a smoother continuous stroke. Editable.

Fixed Rate button Enable to make the stroke dependent on the speed at which you move the brush.

Pressure field Set the transparency on the pressure and direction of the pen. Editable.

Jitter field Set the level of dispersion. A lower value produces a greater concentration of paint. Editable.

Direction field Set the percentage of a complete rotation to rotate the brush around the Z-axis. Editable.

Roll field Set the percentage of a complete rotation to rotate the brush around the X-axis. Editable.

Brush Attribute Modes

Brush Attribute Modes option box Select an attribute as a reference value. Select a clip to use its luminance as a reference.

Brush Attribute Modes option box Select an attribute as a reference value. Select a clip to use its luminance as a reference.

Brush Attribute Modes option box Select an attribute as a reference value. Select a clip to use its luminance as a reference.

Brush Attribute Modes option box Select an attribute as a reference value. Select a clip to use its luminance as a reference.

Brush Attribute Modes option box Select an attribute as a reference value. Select a clip to use its luminance as a reference.

Brush Attribute Modes option box Select an attribute as a reference value. Select a clip to use its luminance as a reference.

Paint Mode

Sets the type of paint operation that is applied to the brush. Some brushes paint in a colour; others use contents of the result or a source. See Using Paint Modes (page 1211) and Using Blending Modes (page 1218).

Paint Modes box Select the type of effect you want to apply to the brush.

Opacity field Displays the opacity of the brush. Use a lower value to apply a more transparent colour. Editable.

Fade field Displays how quickly the stroke fades when the Fade brush attribute mode is selected. A higher value fades the stroke faster.

Blending Modes box Select the blending operation to apply to the brush colour components.

Paint On box Select to apply strokes to the current frame, from the current frame to the last frame, or to all frames in the sequence.

Scale field Displays the scale value of the reference image. Editable.

Rotate field Displays the angle of rotation to rotate the reference image. Editable.

X Offset field Displays the horizontal coordinate for the Clone offset. Editable.

Y Offset field Displays the vertical coordinate for the Clone offset. Editable.

Overlay Controls

Overlay button Enable to show a reference clip overlaid onto the canvas.

Reference box Select the view with the reference clip you want overlay.

Transparency field Set the transparency percentage of the reference clip. Editable.

Colour Palette and Brushes

Current colour pot Select the current brush colour. Editable.

Preset colour pots Select preset brush colours. Editable.

Brushes Select a default brush profile to set it as the brush stroke.

Matte Controls

Defines the area that can be used to paint. You can paint on the entire canvas or areas delimited by the matte properties. See Restricting Strokes with the Source Matte (page 1201) and Restricting Brush Strokes (page 1209).

Use Matte button Enable to limit brush strokes on the canvas to areas inside the matte.

Invert button Enable to limit brush strokes on the canvas to areas outside the matte.

Both button Enable to paint on the result and output matte simultaneously.

Canvas Controls

Controls clearing and wiping the canvas at the current frame. See Using the Canvas (page 1220).

Clear box Select whether to clear all strokes from the result image and output matte at the current frame or to clear strokes from all frames.

Wipe button Select to apply a uniform colour in a single brush stroke to the entire result image, the front only or the output matte only at the current frame.

Wipe colour pot Select the wipe brush stroke. Editable.

Use Source button Enable to use the current source to wipe over the image.

Rotation field Displays the angle of rotation used to rotate the canvas in the image window. Editable.

Node Setup

Active Enable to activate smooth filtering of pixels for enhanced display.

Pixel Spread

Use Pixel Spread to create a stretching effect from the image edges delimited by a matte. This can be useful to solve keying problems resulting from dark edges around the area to be keyed, for example.

To access the Pixel Spread menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front, back and matte clip, as well as a vector map clip, and outputs a result and an outmatte.

Pixel Spread Menu Settings

Type Settings

Spread Type box Select the type of pixel spread distortion effect to apply to the clip.

Spread Mode box Select whether to have the distortion effect work in expansion mode or contraction mode.Active when Spread Type is set to Stretch, Parallax, or Interpolate.

Spread Settings

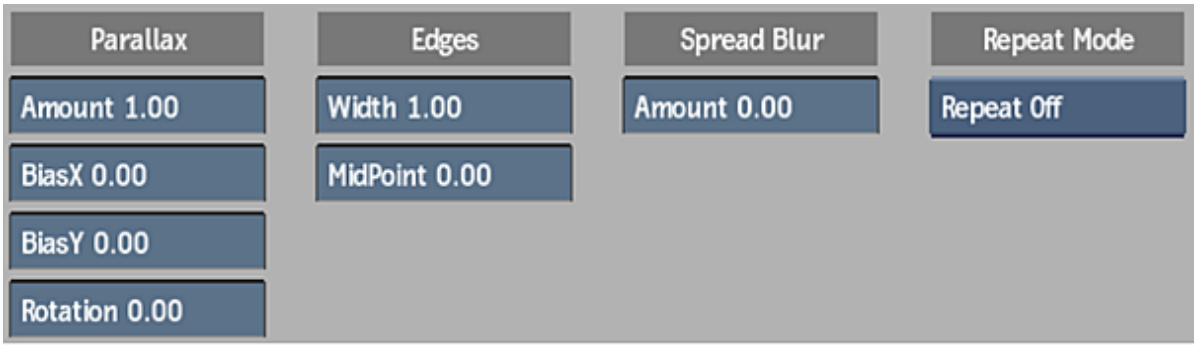
The following settings are available when Interpolate is selected from the Spread Type box.

Interpolation Softness Amount field Displays the amount of softness applied. Editable.

Edges Shrink Amount field Displays the amount of shrinking applied to the edges. Editable.

Edges Shrink Softness field Displays the softness of the edge. Editable.

Interpolation Source Premultiplied button Enable to verify if the source is already premultiplied, in which case the interpolation effect must treat the image differently for the effect to work properly.



The following settings are available when Parallax is selected from the Spread Type box.

Parallax Amount field Displays the amount of parallax distortion applied to the clip, in pixels. Editable.

Parallax X Bias field Displays the amount of horizontal offset applied to the parallax distortion effect, in pixels. Editable.

Parallax Y Bias field Displays the amount of vertical offset applied to the parallax distortion effect, in pixels. Editable.

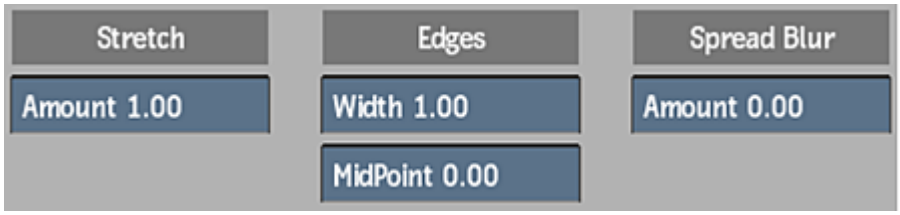
Parallax Rotation field Displays the degree of rotation applied to the pixel distortion. Editable.

Edges Width field Displays the width of the distortion edge, in pixels. Editable.

Edges MidPoint field Displays the offset of the distortion edge from the middle. Editable.

Spread Blur Amount field Displays the amount of blur applied to the clip following the direction of the edge. Editable.

Parallax Repeat Mode box Select an option to fill the empty portions of the frame.



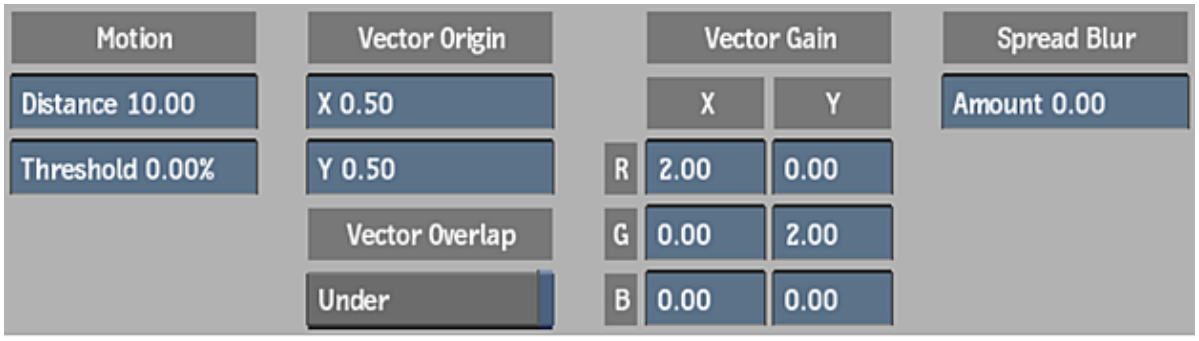
The following settings are available when Stretch is selected from the Spread Type box.

Stretch Amount field Displays the amount of stretching distortion applied to the clip, in pixels. Editable.

Edges Width field Displays the width of the distortion edge, in pixels. Editable.

Edges MidPoint field Displays the offset of the distortion edge from the middle. Editable.

Spread Blur Amount field Displays the amount of blur applied to the clip following the direction of the edge. Editable.



The following settings are available when Vector Warp is selected from the Spread Type box.

Motion Distance field Displays the amount of distortion to be applied to the image based on a given motion vector. A negative amount inverts the direction of the spread. Editable.

Motion Threshold field Displays the percentage for the cut-off point below which motion data is not applied to the image. Editable.

Vector X Origin field Displays the starting point of any horizontal motion, which can be used to offset values entered for X Gain. Editable.

Vector Y Origin field Displays the starting point of any vertical motion, which can be used to offset values entered for Y Gain. Editable.

Vector Overlap button Enable to invert the effects of the vector warp.

Red X Gain field Displays the amount of gain used from the red channel to augment the horizontal motion of the distortion effect on the image. Editable.

Red Y Gain field Displays the amount of gain used from the red channel to augment the vertical motion of the distortion effect on the image. Editable.

Green X Gain field Displays the amount of gain used from the green channel to augment the horizontal motion of the distortion effect on the image. Editable.

Green Y Gain field Displays the amount of gain used from the green channel to augment the vertical motion of the distortion effect on the image. Editable.

Blue X Gain field Displays the amount of gain used from the blue channel to augment the horizontal motion of the distortion effect on the image. Editable.

Blue Y Gain field Displays the amount of gain used from the blue channel to augment the vertical motion of the distortion effect on the image. Editable.

Spread Blur Amount field Displays the amount of blur applied to the clip following the direction of the edge. Editable.

Output Settings

Result Output box Select whether to output a combined result or the pixel spread effect only.

Alpha Output box Select Input to output the input matte, or Spread Matte to output the matte generated by the pixel spread.

Proxy Extract

The Proxy Extract tool is available from the Tools tab, under Clip.

Use the Proxy Extract tool to generate a proxy resolution clip from an existing clip in your project. This is useful when, for example, you have a 4K project, in which you've been working on 2K proxies and want to export a 2K version of your work.

To extract a proxy resolution clip:

1. Enable the Proxy Extract tool.
   You are prompted to pick a Front clip.
2. Pick a Front clip from the Desktop or the Media panel.
   You are prompted to pick a render location.
3. Pick a render location from the Desktop or the Media panel.
   The proxy resolution clip is generated.

NOTE

- Proxies must be enabled in the project for the Proxy Extract tool to be available.
- The proxy resolution is determined by the proxy settings in the Project and User Settings.

Proxy Regeneration

The Proxy Regeneration tool is available from the Tools tab, under Clip.

Use the Proxy Regeneration tool to regenerate updated proxy media for a single or multiple clips in your project, overwriting the original proxies. This is useful when, for example, your proxies were generated from soft-imported media and that original media has changed (colour corrected, for example).

To regenerate proxies:

1. Enable the Proxy Regeneration tool.
   You are prompted to pick a Front clip and the Proxy Regen Parameters menu appears.

2. Select the filter option, from the Proxy Quality box, to determine the quality of the proxy image.
3. Do one of the following:
   - To regenerate proxy media for a single clip, select Clip from the Proxy Regeneration Scope box.
   - To regenerate proxy media for all clips in a folder or reel, select Folder from the Proxy Regeneration Scope box.
4. Pick a Front clip from the Desktop or the Media panel.
   The proxy media is regenerated and the original proxy media is overwritten.

NOTE Proxies must be enabled in the project for the Proxy Regeneration tool to be available.

Proxy Regeneration Menu Settings

General Settings

Proxy Scale Field Displays the proxy scale ratio. For example, a value of 0.50 generates proxies that are 50% of the clip's resolution. This parameter is set in the Project and User Settings.

Proxy Bit Depth field Displays the bit depth for generated proxies. This parameter is set in the Project and User Settings.

Proxy Quality box Set the quality of the proxy image. Proxy quality affects rendering and proxy generation duration since lower qualities are faster to calculate. However, the quality does not affect the amount of storage space required.

Proxy Regeneration Scope box Set the proxy regeneration scope. Options are:

- **Clip**: Regenerates proxy media for the selected clip.
- **Folder**: Regenerates proxy media for all clips in the same folder or reel as the selected clip.

Pulldown

Use the Pulldown to remove or add pulldown to a clip.

To access the Pulldown menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front clip as input, and outputs a result.

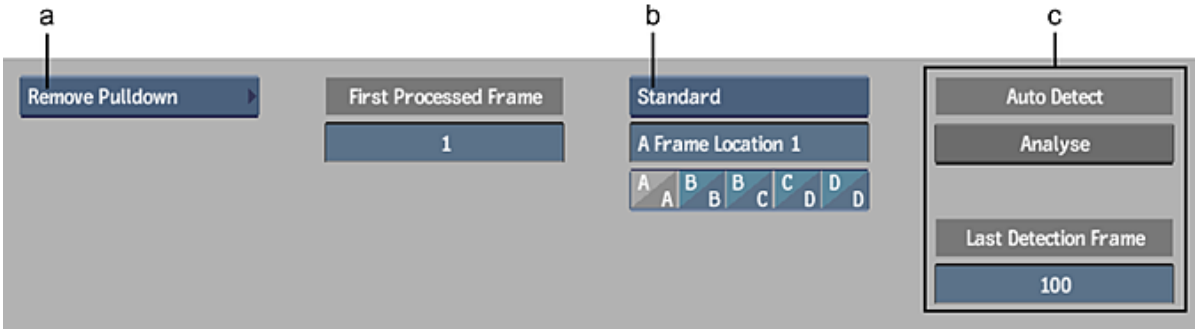
Additionally, the Pulldown node supports the following:

- 2:3 pulldown, also known as standard pulldown
- 2:3:3:2 pulldown, also known as advanced pulldown
- 24-to-25 fps conversion, also known as PAL pulldown

Pulldown Menu Settings

Pulldown Settings

When removing a pulldown, you can analyse the clip to automatically detect the type of pulldown and the A frame. If the analysis fails to determine the type of pulldown or the location of the AA frame, manually set the Pulldown and the A Frame Location options.



(a) Pulldown Mode option box (b) Pulldown Type option box (c) Remove Pulldown options; available when Pulldown Mode is set to Remove Pulldown

Pulldown type option box Select an option to add or remove pulldown from the clip.

First Rendered Frame field Displays the first frame at which output is rendered from the node. Unrendered output does not display any media.Editable.

Pulldown option box Select the type of pulldown process to apply to the clip.

A Frame Location field Displays the value of a reference (AA) frame. The selected frame becomes the frame of reference when adding or removing pulldown frames.Editable.

NOTE The following settings are available when Remove Pulldown is selected from the Pulldown Type option box.

Analyze button Click to determine the type of pulldown used and the A frame.

Last Detection Frame field Displays the value of the last frame of the clip used by the Analyse button. Selecting a subset of a clip speeds up the analysis. Editable.

NOTE When the node renders a transition, preceding and subsequent frames with no media are replaced with black frames.

Recursive Ops

Use Recursive Ops to perform a wide range of recursive, animation-based blending effects. Recursive Ops uses the processed result of the previous frame and blends it with the current frame using a selected blending mode. It features a built-in colour selection, which constrains the accumulation effect within that selection. Additionally, it accepts an external matte input which also conditions the way in which the internal blending occurs.

To access the Recursive Ops menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts front and matte clips, and outputs a result and outmatte.

Recursive Ops Menu Settings

Regen button Enable to dynamically refresh the image as changes are made to the settings.

Repeat Mode Settings

Repeat Mode options box Select an option to fill the empty portions of the frame.

Range Settings

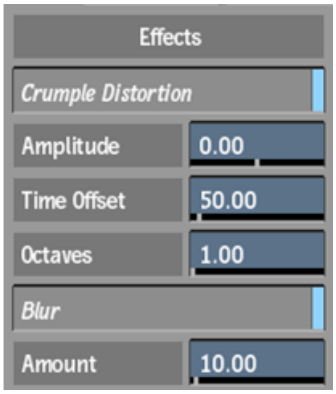
Range From field Displays the beginning of the range of frames that to be impacted by the effect. Editable.

Range To field Displays the end of the range of frames that to be impacted by the effect. Editable.

Range Before box Select an option to be applied before the set frame range. Bypass eliminates the effect, while Cycle repeats the effect.

Range After box Select an option to be applied afer the set frame range. Bypass eliminates the effect, while Cycle repeats the effect.

Effects Settings



Crumple Distortion button Enable to activate the crumple settings.

Amplitude field Displays the amount of distortion. Increase the value to increase the crumple effect. Editable.

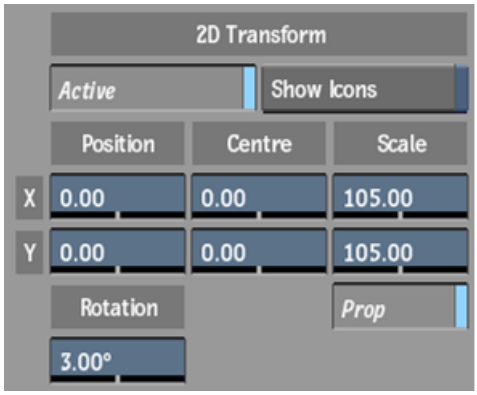
Time Offset field Displays the time offset interval of the crumpling. Editable.

Octaves field Displays the number of layers summed in the operation, from 0 to 10. Increase the value to increase the fractal crumple effect. Editable.

Blur button Enable to activate the blur settings.

Amount field Displays the amount of blur applied to the image. Editable.

2D Transform Settings



Active button Enable to activate the 2D Transform settings.

Show Icons button Enable to display the vertex editing tools in the image window.

Position X field Displays the horizontal position of the transformation. Editable.

Position Y field Displays the vertical position of the transformation. Editable.

Centre X field Displays the centre point value of the transformation along the horizontal axis. Editable.

Centre Y field Displays the centre point value of the transformation along the vertical axis. Editable.

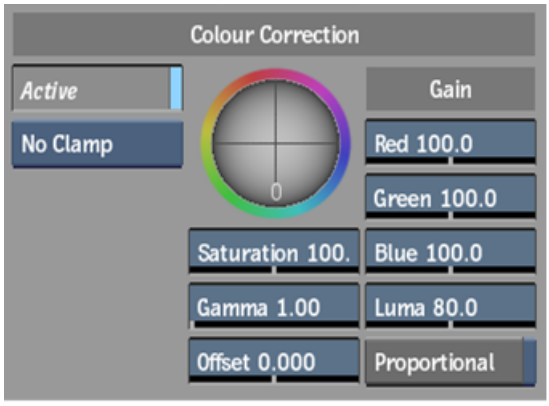
Scale X field Displays the horizontal scale factor. Editable.

Scale Y field Displays the vertical scale factor. Editable.

Proportional button Enable to scale X and Y values proportionally.

Rotation field Displays the rotation of the transformation. Editable.

Colour Correction Settings



Activate button Enable to activate the color correction settings.

Clamping box Select a clamping option.

Gain Trackball Adjusts the gain of the input.

Saturation field Displays level of colour purity in the image. Editable.

Gamma field Displays the gamma level. Editable.

Offset field Displays a value that modifies all of the colour parameters. Editable.

Red Gain field Set the percentage of colour values in the red channel. Editable.

Green Gain field Set the percentage of colour values in the green channel. Editable.

Blue Gain field Set the percentage of colour values in the blue channel. Editable.

Luma Gain field Set the percentage of luma gain value to display. Editable.

Proportional button Enable to adjust the gain of the colour values proportionally.

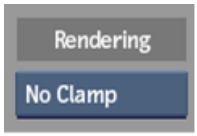
Blending Settings

Blending options box Select a logical operation that can be used to blend the front clip and the result clip.

Transparency field Displays the percentage of blending when the result is composited on the front clip. Editable.

Use Matte button Enable to apply the effect with the areas defined by the matte.

Rendering Settings



Clamp Render box Select a clamping option for colour and luminance values on output in the 16-bit floating point rendering pipeline.

Matte Output Settings

Matte Output options box Select an matte output option. Choose Selective to select the colour you wish to keep.

Selective Tolerance field Displays the tolderance level of the selected colour of the matte output. Editable. Available when Selective is chosen in the Matte Ouput options box.

Selective Colour box Click to enable the crosshair to select a colour in the image to be used as the matte output. Available when Selective is chosen in the Matte Ouput options box.

Regrain

Use Regrain to add grain from the RGB channels of a selected colour in an image.

To access the Regrain menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts front, back, and matte clips as input, and outputs a result.

Regrain Menu Settings

General Settings

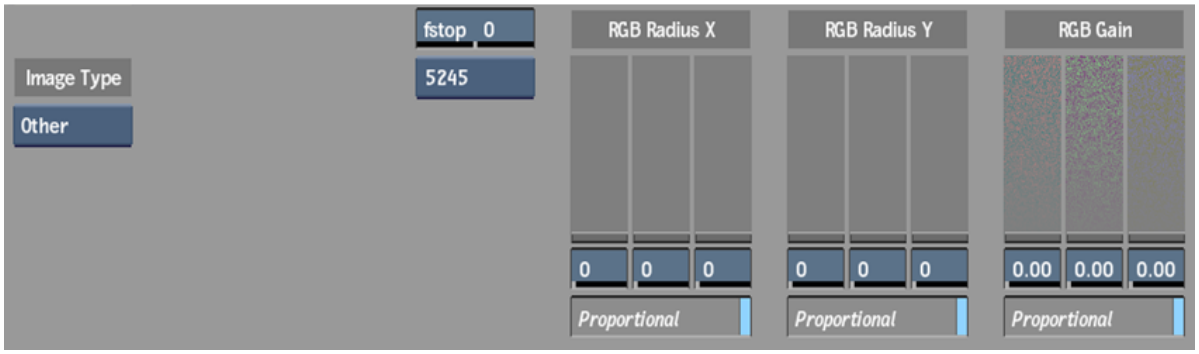


Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

fstop field Displays the relative exposure offset. Use to compensate for under or over exposure. Editable.

Grain Signature box Select a film stock grain to add to the clip.

Red X Radius slider Displays the value of the radius on the x-axis for the red channel. Editable.

Red X Radius field Displays the value of the radius on the x-axis for the red channel. Editable.

Green X Radius slider Displays the value of the radius on the x-axis for the green channel. Editable.

Green X Radius field Displays the value of the radius on the x-axis for the green channel. Editable.

Blue X Radius slider Displays the value of the radius on the x-axis for the blue channel. Editable.

Blue X Radius field Displays the value of the radius on the x-axis for the blue channel. Editable.

Proportional RGB Radius X button Enable to adjust the radius on the x-axis of the red, green, and blue channels proportionally.

Red Y Radius slider Displays the value of the radius on the y-axis for the red channel. Editable.

Red Y Radius field Displays the value of the radius on the y-axis for the red channel. Editable.

Green Y Radius slider Displays the value of the radius on the y-axis for the green channel. Editable.

Green Y Radius field Displays the value of the radius on the y-axis for the green channel. Editable.

Blue Y Radius slider Displays the value of the radius on the y-axis for the blue channel. Editable.

Blue Y Radius field Displays the value of the radius on the y-axis for the blue channel. Editable.

Proportional RGB Radius Y button Enable to adjust the radius on the y-axis of the red, green, and blue channels proportionally.

Red Gain slider Displays the grain value in the red channel. Editable.

Red Gain field Displays the grain value in the red channel. Editable.

Green Gain slider Displays the grain value in the green channel. Editable.

Green Gain field Displays the grain value in the green channel. Editable.

Blue Gain slider Displays the grain value in the blue channel. Editable.

Blue Gain field Displays the grain value in the blue channel. Editable.

Proportional RGB Gain button Enable to adjust the grain on the red, green, and blue channels proportionally.

NOTE The following options are available when Mono or Custom are selected from the Grain Signature box.

Curves Display the curves for the luminance and each RGB channel.

Histogram box Select to display the red, green, blue, or luminance histogram in the graph. Select Current Curve to display the histogram for the currently selected Curves Channel.

Curves box Select to display the red, green, blue or all curves in the graph.

Home button Restores the position of panned or zoomed curves to the default setting.

Reset button Resets to default curve settings.

Resize

Use Resize to change a clip's resolution, frame depth, and aspect ratio.

To access the Resize menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

You can also use Resize to pan and scan the destination image relative to the source. You can then select and animate the portion of the source clip that appears as the destination clip, and process to create the result image.

NOTE You can use Resize to output, for example, a HD project to a lower resolution for quick viewing.

Resize Menu Settings

Timeline FX Quick Menu Settings

To see the full Resize menu, click the Editor button.

Resize Quick Menu selector Displays the Resizing or Crop Box quick menu.

Source Settings

Use the Source settings to set or animate the position and size of the crop box.

Position/Scale Settings

X Position field Displays the horizontal position from the centre of the crop box relative to the centre of the source frame, in pixels. Drag left or right, or click to enter a new X Position value.

Y Position field Displays the vertical position from the centre of the crop box relative to the centre of the source frame, in pixels. Drag left or right, or click to enter a new Y Position value.

X Scale field Displays the horizontal scale of the crop box relative to the Crop Box Width field value, as a percentage. Drag left or right, or click to enter a new X Scale value.

Y Scale field Displays the vertical scale of the crop box relative to the Crop Box Height field value, as a percentage. Drag left or right, or click to enter a new Y Scale value.

Crop Mode box Select an option to determine the scaling behaviour of the crop box while repositioning or rescaling. Use Free to adjust the crop box freely. Use Prop to use the current Crop Box Width and Crop Box Height settings proportionally. Use Source or Destination to use the respective aspect ratio for the crop box.

Source Ratio field Displays the aspect ratio of the crop box in the source frame. Editable.

Crop Options Settings

Crop Box Width field Displays the current width setting of the crop box, in pixels. Editable.

Crop Box Height field Displays the current height setting of the crop box, in pixels. Editable.

Source Width button Uses the width settings of the source frame for the Crop Box Width field.

Source Height button Uses the height settings of the source frame for the Crop Box Height field.

Source Frame button Uses the width and height settings of the source frame for the Crop Box Width and Crop Box Height fields, respectively.

Destination Width button Uses the width settings of the destination frame for the Crop Box Width field.

Destination Height button Uses the height settings of the destination frame for the Crop Box Height field.

Destination Frame button Uses the width and height settings of the destination frame for the Crop Box Width and Crop Box Height fields, respectively.

Crop Box Border colour pot Displays the current colour of the crop box border. Editable.

Crop Box Line Style box Select the line style for the crop box.

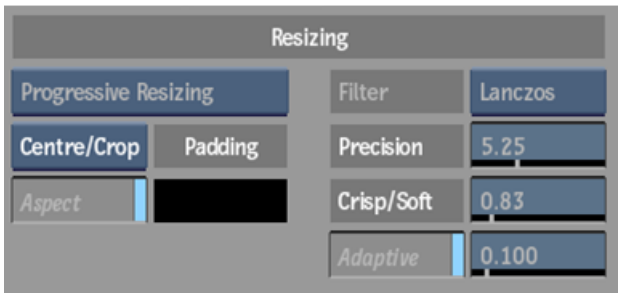
Source Pre-Rendering

Source Pre-Rendering buttons Select a conversion method to use for the resize. Use the Both Fields button when both source and destination formats are interlaced or progressive. Use the Field Merge button to combine the two fields of the source clip. Use the Deinterlace button to select one field, in which case the Source Pre-Rendering Field box is enabled.

Source Pre-Rendering Field box Select a field to use for the resize. Active when the Deinterlace Source Pre-Rendering button is enabled.

Resizing Settings

Use the Resize settings to change the size of a clip.



Resize Field Format box Select an option to determine whether the resize is performed on progressive or interlaced frames, if both source and destination clips are interlaced. If either clip is progressive, the resize operation is always progressive.

Fit Method box Select a fit method to be applied to the selected clip.

Precision field Displays the frequency cut-off point used during resize. Editable.

Crisp/Soft field Displays the amount of blurring used during resize. Editable.

Resize Filter box This option is available when Letterbox, Crop Edges or Fill is selected from the Fit Method box. Select the filter option to determine the quality of the interpolated resize result.

Aspect button This option is available when Letterbox or Crop Edges is selected from the Fit Method box. Enable to use non-square pixel formats. Active when Fit Method is set to Crop Edges or Letterbox.

Destination Padding colour pot Displays the colour used for padding the destination frame when the source image is smaller than the destination. Editable.

Adaptive Deinterlacing button Enable to use data from the adjoining interlaced field to improve the results of the resizing algorithm. This is most effective for clips containing stationary objects.

Adaptive Deinterlacing field Displays the amount of data used from the adjoining field for adaptive interlacing during resize. Active when Adaptive button is enabled.

Destination Settings

Use the Destination settings to define the format of the resized clip.

Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

Width field Displays the custom width resolution of the clip. Editable.

Height field Displays the custom width resolution of the clip. Editable.

Aspect Ratio Presets box Select a standard frame aspect ratio. Select the Set to w:h option to set the clip to use square pixels. Select Custom to define a custom frame aspect ratio in the Aspect Ratio field.

Aspect Ratio field Displays the custom render/output aspect ratio. Editable

Bit Depth box Select the render/output bit depth of clips.

Scan Mode box Select the scan mode of clips.

Crop/Lock Output button Enable to lock the destination resolution to match the crop box. Use this option to animate the resolution of a clip and change it on a per-frame basis. Available when you access Resize settings.

Sparks

Sparks are software plug-ins created by Autodesk or third-party developers.

To access Sparks, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

The Autodesk Sparks API Reference Guide is available in PDF format. If you are interested in marketing or selling your Sparks, you must apply to the system Sparks program. To request an application, send an e-mail message to *email Sparks Manager*.

The Sparks viewport uses the Viewing Rules for colour management. Enter the View settings panel to bypass the colour management or select a viewing rule other than the primary one for that media.

Using a Sparks Plug-in

A Sparks plug-in functions in the same way as other commands or modules. Before using a Sparks plug-in, you have to load it. After loading the plug-ins, you can replace them if all the Sparks buttons are already in use.

Loading and Replacing Sparks Plug-ins

To load a Sparks plug-in:

1. Do one of:
   - From the Tools menu, select the Plugins tab and then click one of the Sparks buttons.
   - In Batch or Batch FX, select the Sparks node and drag it into the schematic.
   - In the Timeline, select a clip and add a Sparks Timeline FX to it.
2. Once a plug-in is loaded, the Sparks file browser appears. Sparks buttons are labelled as follows:
   - L: the plug-in can be loaded.
   - E: the plug-in has settings.
   - S: you can enter the module using the same media from the previous session. Navigate to `opt/Autodesk/sparks`.

   You can also load Sparks from `opt/Autodesk/<product home>/sparks`.
3. Select a plug-in. You are returned to your previous location. The name of the plug-in appears on the selected button.

To replace a Sparks plug-in:

1. Press Alt and click the existing Sparks button.
2. In the Sparks file browser navigate to `opt/Autodesk/sparks` or `opt/Autodesk/<product home>/sparks` and select a Sparks plug-in. You are returned to your previous location, and the name of the new plug-in appears on the selected button.

Using Sparks in Batch or Batch FX

Two Sparks nodes are available:

- Sparks node: plug-ins are loaded via the Sparks file browser. The number of inputs and bit depth that a Sparks node accepts depends on the Sparks plug-in. A Sparks node has light grey source tabs, since each Sparks plug-in varies in the type of clip it uses. When working with a Sparks plug-in, missing media in front clips set to No Media can be converted into black frames while they are in use. When missing media is output from this node, it returns to a No Media state, regardless of whether it was set to display differently in the node. If the error message "SPARK IS NOT SUPPORTED" appears, contact the company that developed the Sparks plug-in for information on obtaining a compatible version.
- Sparks Load node: multiple Sparks nodes can be populated at the same time into a destination bin. Populating a bin with predefined Sparks nodes saves time since you do not have to access the Sparks browser each time you want to use one of the preloaded Sparks. Unlike the Sparks node, the Sparks Load node itself cannot be dragged to the schematic; only the predefined node populated into a bin can be dragged to the schematic. To create a predefined Sparks node in a bin:

  1. Drag the Sparks Load node on top of any tab in the ALL Tools bin except the ALL Nodes tab.
  2. In the Sparks Plug-in file browser that appears, select one or many plug-in or select multiple plug-ins.
  3. Click load.

  The selected Sparks are created as their own nodes in the destination bin. To use one, drag it from the destination bin to the schematic. You do not have to re-enter the Sparks browser to load a Sparks if you preloaded it into a bin.

Stabilizer

Stabilizing and Tracking

Use the Stabilizer to remove camera instability and motion jitter, and to track reference points in your clips. You can also use the Stabilizer to produce 2D or, in the context of Action, 3D motion, anchor a surface to the clip's background or anchor the UV points of the surface to features of a clip. With tracking, a point or points on the clip are tracked as they move through the scene. You can then apply the resulting motion path to an object on another layer so that it follows the same path as the object you tracked.

Stabilizing is the inverse of tracking. With stabilizing, the motion path is used to shift the scene so that the point that is tracked remains fixed at one position.

Tracking and stabilizing are often processes of trial and error. It is recommended that you track or stabilize using the default settings. If the tracker box strays from its original point, you can fine-tune the analysis.

Accessing the Stabilizer

Access the Stabilizer from:	To:
The Tools tab	Stabilize.
Action Axis	Track or stabilize.
Action Perspective Grid	Perform 4-corner tracking.
Action GMask node	Perform Cluster or Vertex tracking.
Action Surface	Perform Bilinear, Extended Bicubic, or Perspective tracking.
Action Analyzer	Provide a 2D tracking path for 3D manual tracking.
GMask	Track a garbage mask or the vertices of a GMask.
Distort	Track vertices or the axis of a spline when warping or morphing.
Warper	Track points or an axis of a mesh when warping or morphing.
Paint Tool	Track an AutoPaint stroke.
Blur and Glow	Track the center point of the radial blur.
2D Transform	Provide correction for 2D motion, rotation and scaling.

Stabilizer Menu Settings

Miscellaneous Buttons

Return button Returns to the previous tool.

Load button Loads a setup.

Save button Saves a setup.

Setup Name field Displays the name of the last saved setup.

Revert button Reverts to the last saved setup.

Colour pot Displays the colour to fill the area where the image has been shifted after stabilization (available if Shift or Letterbox is chosen in the Scale and Shift Option box). Editable.

Context button Enable to use the Action scene as the tracking reference.

Setup button Opens the Setup menu, where you can specify user interface and tracking preferences.

Animation button Opens the channel editor, where you can animate the various Stabilizer settings.

Expand/Collapse box Select whether to expand or collapse selected animation channels.

View box Select an option to set the view in the image window.

Auto Key button Enable to set a keyframe automatically each time you change a value at any frame.

Set Key button Sets a keyframe at the selected frame.

Delete Key button Deletes the selected keyframe.

Current Frame field Enter a frame number to jump to the corresponding frame.

Duration field Displays the duration of the clip in frames.

Reset box Select an option to reset shift, tracking, reference, or all data.

Reset All button Resets all parameters.

Grid button Accesses the overlay menu.

View button Accesses the viewing settings menu.

Undo button Undoes the last action performed.

Main Stabilizer Menu

Tracker box Select the Tracker to work with.

Add Tracker button Click to add a new tracker.

Active button Enable to activate the selected tracker.

Tracker colour pot Displays the colour of the selected tracker. Editable.

When you add a new tracker, the system automatically assigns it a unique colour so that you can easily distinguish between multiple trackers. You can change the colour of an individual tracker or of all the trackers at once.

Tolerance field Displays a value the Stabilizer uses to match reference points from frame to frame and to set keyframes. Editable.

Tracker Selection box Select which trackers are affected when you change a parameter.

Analyze button Click to generate stabilization or tracking data.

Step button Click to analyze a single frame and advance to the next frame.

Direction box Select to analyze forward or backward.

Snap button Click to redefine the reference at the selected frame.

Lock Key button Click to lock the selected keyframe as a point on the tracking path. Click again to unlock.

Delete Key button Click to delete the selected point on the tracking path.

Source option box Select whether to track in Progressive (frame mode) or Interlaced (field mode).

By default, the Stabilizer works in Progressive (frame mode). Select Interlaced from the Source Option box mode when working with interlaced images, or with images that display a lot of field jitter. In Interlaced (field) mode, the Stabilizer sets two keyframes for every frame: one for the even field and one for the odd field, and an asterisk appears in the image viewer's current frame display to indicate the second field for each frame.

Remove Vertical Jitter button Enable to remove vertical jitter.

Remove Horizontal Jitter button Enable to remove horizontal jitter.

Reference X field Displays the position of the reference box along the X axis. Editable.

Reference Y field Displays the position of the reference box along the Y axis. Editable.

Reference Width field Displays the width of the reference box. Editable.

Reference Height field Displays the height of the reference box. Editable.

Offset X field Displays the offset X axis value. Editable.

Offset Y field Displays the offset Y axis value. Editable.

Auto Update Reference button Enable to update the reference point at each frame during tracking. Disable to track the movement of the reference point specified in the reference frame. You can set the default for this button in the Stabilizer Setup menu.

Enabling Auto Update Reference is useful when tracking a pattern that changes considerably from the first frame to the last frame in the clip. For example, the pattern may be rotating or may change size or shape. In this case, disable the Auto Update Reference button. The reference point is then updated in each frame. In each frame of the analysis, the Stabilizer looks for the reference point from the previous frame.

When Auto Update Reference is disabled, the tracker box follows the movement of the reference point that you specified in the reference frame throughout the analysis.

NOTE Unless you are in Gang mode, you must set the Auto Update Reference button for each active tracker individually.

Tracker Y field Displays the position of the tracker box along the Y axis. Editable.

Tracker Width field Displays the width of the tracker box. Editable.

Tracker Height field Displays the height of the tracker box. Editable.

Import Track button Open the Import Stabilizer menu, where you to import a text file of saved tracking data.

Export Track button Click to open the Export Stabilizer menu, where you to export a text file of saved tracking data.

Shift X field Displays the difference between the position of the reference box (0,0) and the position of the tracker box in the current frame along the X axis. Editable.

Shift Y field Displays the difference between the position of the reference box (0,0) and the position of the tracker box in the current frame along the Y axis. Editable.

Import Shift button Click to open the Import Stabilizer menu to allow you to import a text file of saved shift data.

Shift Copy button Click to copy the selected Shift channel (including the aspect ratio of the clip).

Export Shift button Click to open the Export Stabilizer menu to allow you to export a text file of saved shift data.

Setup Menu

Pretracking button Enable to preview the motion path for a specified number of frames. Active when Path is enabled.

Before analysing, you can use the Pretracking option to preview the motion path for a specified number of frames. You can then adjust the tracker position, if necessary, to find the best reference point. The Pretracking option applies only to the selected tracker, regardless of whether you selected Solo, Selected, or Gang in the Tracker Selection box. If you move or resize a tracker with Pretracking enabled, the next frames are analysed.

Pretracking field Displays the number of frames to pretrack. Editable.

Auto Pan button Enable to allow the part of the image that is selected to stay in the image window when zoomed in.

Opacity field Displays the opacity of the reference image. Editable.

The reference image (the image where you placed the reference box) appears in transparency when you select a keyframe. Adjust the opacity of the image to make it more or less transparent. When the opacity is set to 0, the reference image does not appear. When the opacity is set to 100, the reference image is completely opaque.

Zoom field Displays the magnification factor of the reference box when selected. Editable.

By default, the reference box turns into a magnifying glass when you select it. Increase or reduce the magnification factor in the Zoom box.

Zoom:	Result:
0	No zooming. The crosshair appears in the tracker box after you analyse the clip.
1	No zooming and no crosshair.
2, 3, 4, 5	A magnification factor of 2, 3, 4, and 5, and a crosshair in the tracker box.

Default Auto Update button Select whether the Auto Update Reference button in the Stabilizer menu defaults to on or off.

Path button Enable to display the tracking path.

The tracking path is the path that the reference point makes as it changes position from frame to frame. By default, the tracking path appears in the image window. You can turn it off by disabling the Path button.

Colour Corrector Input button Enable to display any Colour Corrector settings on the clip.

Linetest button Enable to display the context view at a lower resolution.

Stereo Filter

Use Stereo Filter to process an anaglyph, interlaced or dual image clip containing one video track from either a left and right eye mono clip, or an existing stereo track.

To access the Stereo Filter menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a left eye mono clip and a right eye mono clip, and outputs an anaglyph, interlaced or dual image stereo clip. You can add a Stereo Filter node to any part of the pipeline except to the Front and Matte pipes of MBlend nodes and the Matte pipes CBLend nodes.

Stereo Filter Menu Settings

General Settings

Stereo Mode box Select a stereo mode to display the appropriate rendering method options. Different method settings appear based on the type selected.

Select:	To:
Anaglyph	Render a red/cyan clip.
Interlace	Render an interlaced RGB clip.
Dual Image	Render a clip containing adjacent images.

Method box Depending on the selected Stereo Mode, you have different options for modifying the effect.

Anaglyph Settings

Anaglyph Method box Select a method for your anaglyph result. For example, select Dubois to reduce the ghosting between the left and right eyes.

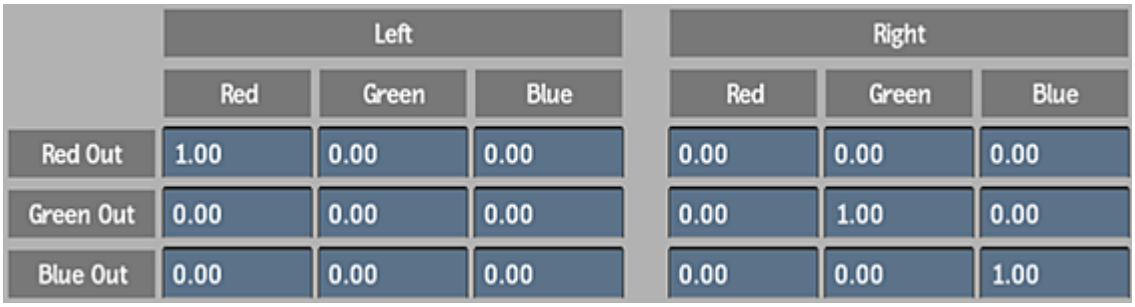
Select:	To:
Custom	Customize the RGB left and right gain factor values. With this option, you can create anaglyph results based on the 3D lenses that will be used. **NOTE** This option is available in Batch or Batch FXand the Modular Keyer.
Dubois	Reduce ghosting between the left and right eyes.
RGB	Create an anaglyph result based on the RGB values.
Mono	Remove the RGB values before creating an anaglyph result. With this option, you will see just the stereo effect.

Interlaced Settings

Interlace Method box Select whether your interlace result outputs the left eye input as field 1 or field 2.

Dual Image Settings

Output the left eye and right eye so that they are adjacent to each other in the same clip, either in a left and right or top and bottom orientation.



	Left			Right		
	Red	Green	Blue	Red	Green	Blue
Red Out	1.00	0.00	0.00	0.00	0.00	0.00
Green Out	0.00	0.00	0.00	0.00	1.00	0.00
Blue Out	0.00	0.00	0.00	0.00	0.00	1.00

Left and Right Custom Gain Factor Fields Displays the gain factor in the colour channel for the left eye and right eye. Enabled when Custom is selected in the Method box.

NOTE This option is available in Batch or Batch FXand the Modular Keyer.

Dual Image Method box Select whether your dual image result outputs the left eye and right eye so that they are adjacent to each other in the same clip, either in a left and right or top and bottom orientation.

Red in Red Output field Displays the red gain factor in the red channel for the Left eye. Editable.

Green in Red Output field Displays the green gain factor in the red channel for the Left eye. Editable.

Blue in Red Output field Displays the blue gain factor in the red channel for the Left eye. Editable.

Red in Green Output field Displays the red gain factor in the green channel for the Left eye. Editable.

Green in Green Output field Displays the green gain factor in the green channel for the Left eye. Editable.

Blue in Green Output field Displays the blue gain factor in the green channel for the Left eye. Editable.

Red in Blue Output field Displays the red gain factor in the blue channel for the Left eye. Editable.

Green in Blue Output field Displays the green gain factor in the blue channel for the Left eye. Editable.

Blue in Blue Output field Displays the blue gain factor in the blue channel for the Left eye. Editable.

Red in Red Output field Displays the red gain factor in the red channel for the Right eye. Editable.

Green in Red Output field Displays the green gain factor in the red channel for the Right eye. Editable.

Blue in Red Output field Displays the blue gain factor in the red channel for the Right eye. Editable.

Red in Green Output field Displays the red gain factor in the green channel for the Right eye. Editable.

Green in Green Output field Displays the green gain factor in the green channel for the Right eye. Editable.

Blue in Green Output field Displays the blue gain factor in the green channel for the Right eye. Editable.

Red in Blue Output field Displays the red gain factor in the blue channel for the Right eye. Editable.

Green in Blue Output field Displays the green gain factor in the blue channel for the Right eye. Editable.

Blue in Blue Output field Displays the blue gain factor in the blue channel for the Right eye. Editable.

Stereo Toolbox

About Stereo Toolbox

When working with Stereo3D content, it may happen that certain elements of the scene are present in one eye but not the other. When this is the case, you can apply a Floating Window that crops part of Stereo3D content that is only displayed in one eye, preserving the 3D illusion.

Specifically, the Floating Window tool has two main uses:

- To manipulate the Z-axis position of the 3D scene, without changing the overall depth bracket of the scene.
- To shape and position a floating window to mask out elements in the 3D scene.

NOTE You must be viewing the Stereo3D footage on a 3D monitor or on a 2D monitor in Anaglyph mode, in order to view the results of the Floating Window properly.

The controls allow you to position the corners of the floating window in Z-space. It also allows one, two or all four corners to be manipulated simultaneously.

There are 3 sets of parameters in the Floating Window menu:

1. The first set of controls consists of the Convergence slider. The convergence slider is used to set the initial horizontal offset between the left and the right eye. A higher convergence value results in a 3D scene with more depth along the Z-axis. A lower convergence value results in a 3D scene with less depth along the Z-axis.
2. The second set of controls consists of four numerical sliders labeled Top/Left, Top/Right, Bottom/Left and Bottom/Right.

   These controls allow you to "pull in" or "push out" the corresponding corner of the Floating Window. This translates to a black mask being drawn on the edges of the image in the left or the right eye. Depending on whether the slider value is positive or negative, they affect each eye's mask differently. For example:

   - Increasing the value of the Top/Left slider results in the Floating Windows's corner being "pushed out" away from the viewer. The mask is applied on the left edge of the right eye in that case.
   - Decreasing the value of the Top/Left slider results in the Floating Window's corner being "pulled in" towards the viewer. When the values are negative, the corner is in theater space (i.e. virtually in front of the monitor) and a black edge becomes visible on the left edge of the left eye.
   - The other controls affect the Floating Window in the same way. Increasing the value of a slider "pushes out" the corresponding corner of the Floating window, whereas decreasing the value results in the corner being "pulled in".
3. The third set of controls will affect the Post-Render-Shift (PRS) value, which effectively displaces the whole scene (including the Floating Window) along the Z-axis.

   The PRS is implemented as a horizontal translation of one, or both eyes. You can select which eye remains fixed by selecting a value from the Alignment box. 'Align Left' applies the PRS entirely to the right eye, while Align Right applies it to the left eye and Center applies it to both eyes.

   Aux and Offset values are simply additive extensions of the PRS value. For example, if you have PRS=1, Offset=2 and Aux=4, the overall PRS value is 7.

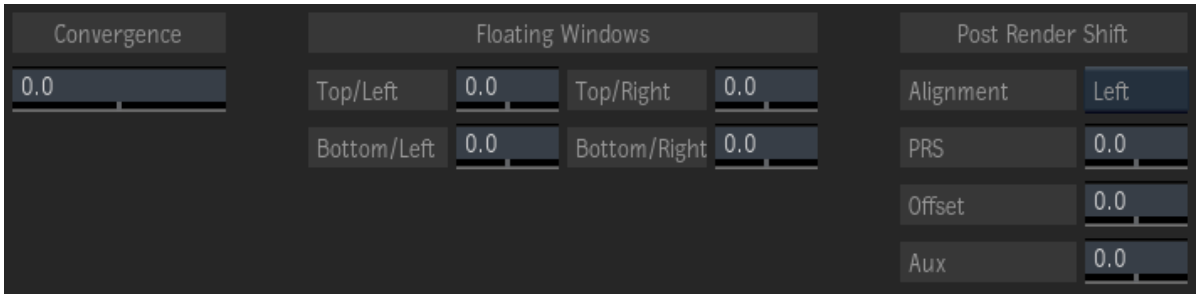
All the slider values can be animated.

Accessing the Stereo Toolbox

To access the Stereo Toolbox use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

Stereo Toolbox Menu Settings



Convergence Settings

Convergence field Set the initial horizontal offset between the left and the right eye. A higher convergence value results in a 3D scene with more depth along the Z-axis. A lower convergence value results in a 3D scene with less depth along the Z-axis. Editable.

Floating Windows Settings

Top/Left field Affects the displacement value of the Top/Left corner of the Floating Window. Increasing the value results in the Floating Windows's Top/Left corner being "pushed out" away from the viewer. Decreasing the value results in the Floating Window's Top/Left corner being "pulled in" towards the viewer. Editable.

Bottom/Left field Affects the displacement value of the Bottom/Left corner of the Floating Window. Increasing the value results in the Floating Windows's Bottom/Left corner being "pushed out", away from the viewer. Decreasing the value results in the Floating Window's Bottom/Left corner being "pulled in", towards the viewer. Editable.

Top/Right field Affects the displacement value of the Top/Right corner of the Floating Window. Increasing the value results in the Floating Windows's Top/Right corner being "pushed out", away from the viewer. Decreasing the value results in the Floating Window's Top/Top Right corner being "pulled in", towards the viewer. Editable.

Bottom/Right field Affects the displacement value of the Bottom/Right corner of the Floating Window. Increasing the value results in the Floating Windows's Bottom/Right corner being "pushed out", away from the viewer. Decreasing the value results in the Floating Window's Bottom/Right corner being "pulled in", towards the viewer. Editable.

Post Render Shift Settings

Alignment box Select whether the offsets affect the Left, Right, or both eyes (Centre).

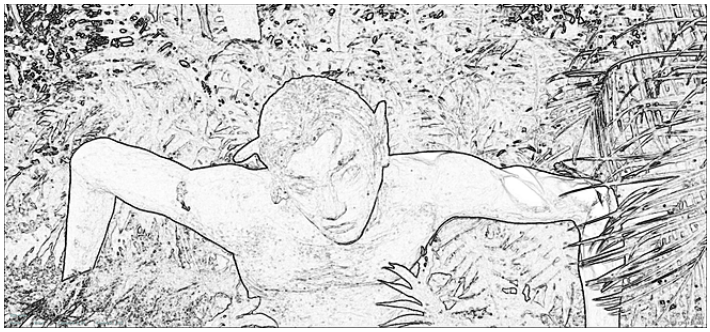
PRS field Affects the horizontal translation offset value applied between the two shots in the scene. This parameter is relative to the setting if the Alignment box. Editable.

Offset field Affects the horizontal translation offset value applied between the two shots in the scene. This parameter is relative to the setting if the Alignment box. Editable.

Aux field Affects the horizontal translation offset value applied between the two shots in the scene. This parameter is relative to the setting if the Alignment box. Editable.

Stylize

Use Stylize on an image sequence to create a wide range of visual styles, including painting, printing and sketching. Stylize allows you to build your look by stacking layers with different effects.



Stylize includes a collection of 10 core effects that are stacked and blended together as layers. The core effects fall into four major categories: canvas texture, patterns, colour fillings, and outlines. Each of the core effects has its own settings, which display when an effect is selected:

- Canvas
- Dots
- Hatch Pattern
- Palette Reduction
- Colour Smudge
- Drawing
- Selective Drawing
- Sketched Outlines
- Scribbled Outlines
- Sharp Outlines

Since the possible combinations of core effects are endless, over 80 presets are available to help you get started with Stylize. These presets can also help you learn how Stylize layer stacking works. The presets are divided into a number of categories and are easily added to your scene.

To access the Stylize menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a front, back and matte clip, and outputs a result and an outmatte result.

Stylize Menu Settings

Using Stylize Presets

Use the Stylize presets to create a specific look or to create a starting point to build your effect. Presets are available in 4 categories: Comic, Paint, Print, and Sketch.

Presets button Opens the Presets browser where you can select a preset.

TIP In the browser, switch to Proxies view (press **P**) to see a visual representation of the preset.

Presets dropdown list Select a preset from the dropdown list. Menu settings are changed to reflect the chosen preset.

Setup and General Settings

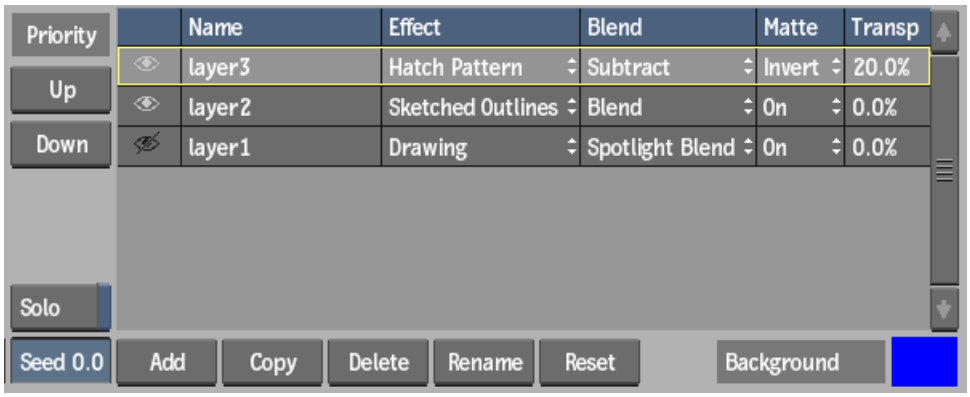
Clamp Input box Select a clamping option for colour and luminance values on input in the 16-bit floating point rendering pipeline.

Clamp Output box Select a clamping option for colour and luminance values on output in the 16-bit floating point rendering pipeline.

Regen button Enable to dynamically refresh the image as settings are changed.

Layer Table Settings

Use the Layer Table to organize the layers that make up your Stylize effect. The top layer in the Layer Table has the highest priority in the overall effect. Selecting a preset populates the Layer Table with the necessary layers and settings.



Visibility column Displays the visibility of each layer. Click the eye icon to change a layer from visible to invisible or vice-versa.

Name column Displays the name of the layer. Click the Rename button to change the name of a selected layer.

Effect column Displays the effect applied per layer. Use the scroll arrows to change the effect. Different settings appear based on the effect chosen.

Blend column Displays the blend operation per layer. Use the scroll arrows to change the blend value.

Matte column Displays whether a matte is on, off, or inverted per layer. Use the scroll arrows to change the value.

Transparency column Displays the transparency level per layer. Scroll to change the value, or click to display the calculator.

Priority Up button Click to move the selected layer up in priority in the layer list.

Priority Down button Click to move the selected layer down in priority in the layer list.

Solo button Enable to hide all other layers except the selected layer. You can also turn specific layers on or off using the eye icon on the left of each layer.

Seed field Displays the random seed variation value to use in the stylize effect. Editable.

Add button Click to add a new layer to the layer list.

Copy button Click to copy the existing layer to a new layer.

Delete button Click to delete the selected layer from the layer list.

Rename button Click to open the online keyboard to rename the selected layer in the layer list.

Reset button Click to reset all of the attributes associated with the selected layer and returns them to default values.

Background colour pot Displays the background colour used in a blending operation. Click to change the colour.

Canvas Effect Settings

These settings are available when a layer is selected in the Layer Table with Canvas as the effect.

Pattern Type box Select a pattern type to apply to the image.

Transparency field Displays the percentage of transparency applied to the Canvas effect. Editable.

Position X field Displays the position of the pattern along the horizontal axis. Editable.

Position Y field Displays the position of the pattern along the vertical axis. Editable.

Scale X field Displays the change in size of the pattern along the horizontal axis. Editable.

Scale Y field Displays the change in size of the pattern along the vertical axis. Editable.

Proportional button Enable to change the scale fields proportionally.

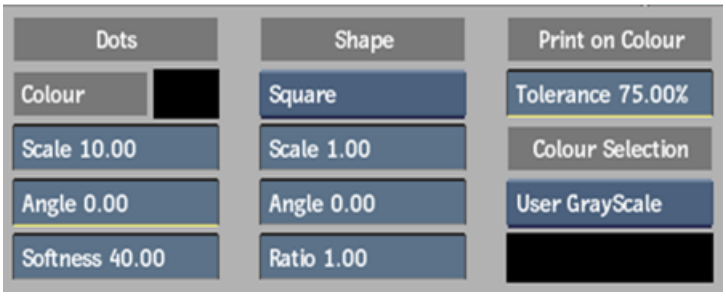
Rotation field Displays the angle of rotation of the pattern along the Z-axis, from its centre point. Editable.

Highlights colour pot Displays the colour applied to the light areas of the Canvas effect. Click to change the colour.

Shadows colour pot Displays the colour applied to the dark areas of the Canvas effect. Click to change the colour.

Dots Effect Settings

These settings are available when a layer is selected in the Layer Table with Dots as the effect.



Effect colour pot Displays the colour applied to the effect. Click to change the colour.

Scale field Displays the change in size of the effect. Editable.

Angle field Displays the change in the angle applied to the effect. Editable.

Softness field Displays the change in softness applied to the effect. Editable.

Shape box Select the shape of the Dots effect.

Scale field Displays the size of the Dots shapes. Editable.

Angle field Displays the change in angle of the Dots shapes. Editable.

Ratio field Displays the change in ratio of the Dots shapes. Editable.

Tolerance field Displays the tolerance level applied to the colour selection of the effect. Editable.

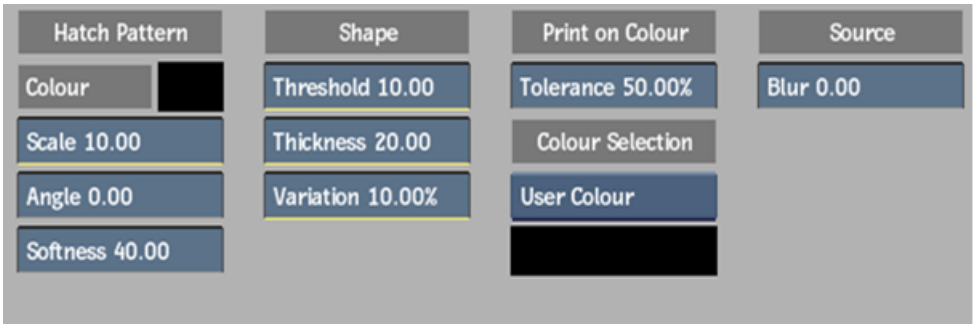
Colour Selection option box Select an option to apply a colour to the image.

Colour Selection colour pot Displays the colour applied to the image. Click to change the colour. Only active if User Colour or User GrayScale is selected.

Blur field Displays the amount of blur applied to the input image before other effects are applied. Editable.

Hatch Pattern Effect Settings

These settings are available when a layer is selected in the Layer Table with Hatch Pattern as the effect.



Effect colour pot Displays the colour applied to the effect. Click to change the colour.

Scale field Displays the change in size of the effect. Editable.

Angle field Displays the change in the angle applied to the effect. Editable.

Softness field Displays the change in softness applied to the effect. Editable.

Threshold field Displays the amount of colour constraint applied to the shape. Editable.

Thickness field Displays the amount of thickness applied to the shape. Editable.

Variation field Displays the percentage of variance applied to the shape. Editable.

Tolerance field Displays the tolerance level applied to the colour selection of the effect. Editable.

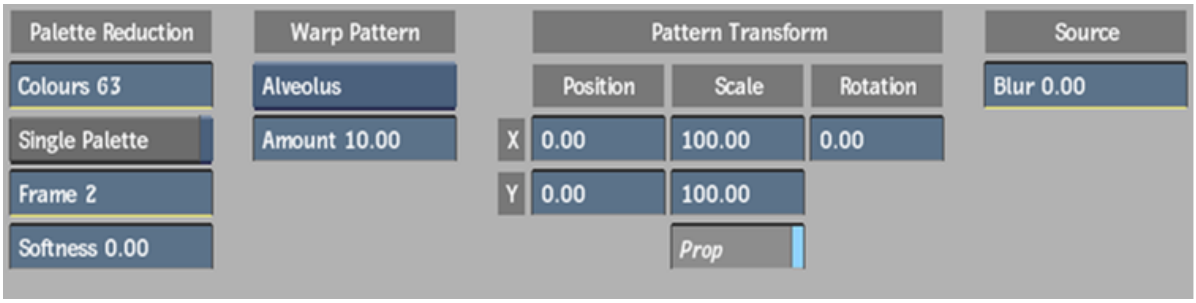
Colour Selection option box Select an option to apply a colour to the image.

Colour Selection colour pot Displays the colour applied to the image. Click to change the colour. Only active if User Colour or User GrayScale is selected.

Blur field Displays the amount of blur applied to the input image before other effects are applied. Editable.

Palette Reduction Effect Settings

These settings are available when a layer is selected in the Layer Table with Palette Reduction as the effect.



Colours field Displays the number of colours applied to the Palette Reduction effect. Editable.

Palette box Select between Single Palette to display image stability by using a single colour palette as a reference, and Dynamic Palette to display image transition by using a new colour palette at each frame.

Frame field Displays the frame selected when Single Palette is enabled in the Palette box. Editable.

Softness field Displays the amount of softness applied to the Palette Reduction effect. Editable.

Pattern Type box Select a pattern type to apply to the image.

Amount field Displays the amount of the selected warp pattern that is applied to the image. Editable.

Position X field Displays the position of the pattern along the horizontal axis. Editable.

Position Y field Displays the position of the pattern along the vertical axis. Editable.

Scale X field Displays the change in size of the pattern along the horizontal axis. Editable.

Scale Y field Displays the change in size of the pattern along the vertical axis. Editable.

Proportional button Enable to change the scale fields proportionally.

Rotation field Displays the angle of rotation of the pattern along the Z-axis, from its centre point. Editable.

Blur field Displays the amount of blur applied to the input image before other effects are applied. Editable.

Colour Smudge Effect Settings

These settings are available when a layer is selected in the Layer Table with Colour Smudge as the effect.

Amount field Displays the amount of Colour Smudge effect that is applied to the image. Editable.

Distortion field Displays the amount of distortion applied to the Colour Smudge effect. Editable.

Emboss colour pot Displays the colour applied to the embossing. Click to change the colour.

Amount field Displays the amount of embossing applied to the image. Editable.

Softness field Displays the amount of softness applied to the embossing. Editable.

Angle field Displays the angle applied to the embossing. Editable.

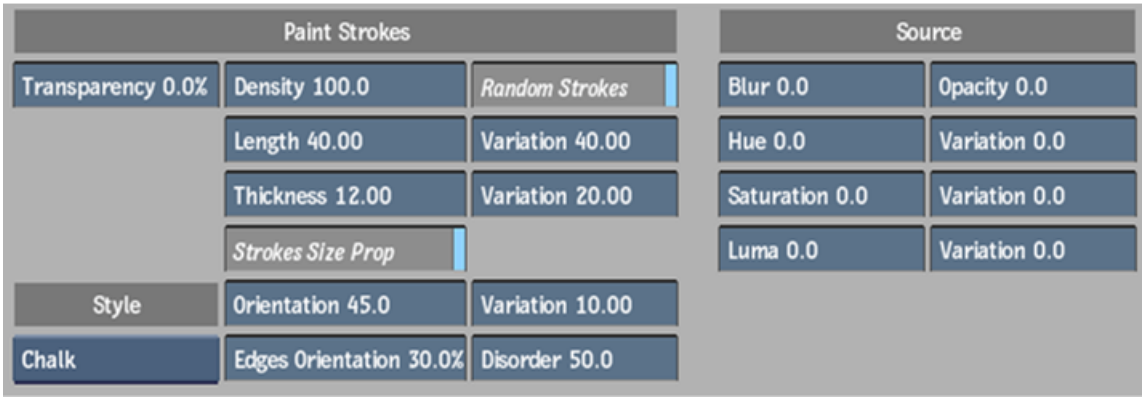
Anti-Aliasing Active button Enable to use anti-aliasing on the edges of the image.

Anti-Aliasing Samples box Select an option to determine the size of the samples. Available when anti-aliasing is enabled.

Anti-Aliasing Softness field Displays the amount of softness applied to the anti-aliasing. Available when anti-aliasing is enabled. Editable.

Drawing Effect Settings

These settings are available when a layer is selected in the Layer Table with Drawing as the effect.



Transparency field Displays the percentage of transparency applied to the paint strokes. Editable.

Style Type box Select the style type of the paint strokes.

Density field Displays the amount of density applied to the paint strokes. Editable.

Length field Displays the length applied to the paint strokes. Editable.

Length Variation field Displays the amount of variance applied to the length of the paint strokes. Editable.

Thickness field Displays the amount of thickness applied to the paint strokes. Editable.

Thickness Variation field Displays the amount of variance applied to the thickness of the paint strokes. Editable.

Paint Each Frame button Enable to apply a random variation in the length and thickness values.

Preserve Stroke Ratio button Enable to allow proportional values to be applied to the length and thickness of paint strokes.

Orientation field Displays the orientation applied to the paint strokes. Editable.

Orientation Variation field Displays the amount of variance applied to the orientation of the paint strokes. Editable.

Edges Orientation field Displays the percentage orientation applied to the edges of the paint strokes. Editable.

Disorder field Displays the amount of random distribution applied to the paint strokes. Editable.

Blur field Displays the amount of blur applied to the input image before other effects are applied. Editable.

Opacity field Displays the amount of opacity applied to the input image before other effects are applied. Editable.

Hue field Displays the amount of hue applied to the input image before other effects are applied. Editable.

Hue Variation field Displays the amount of variance applied to the hue. Editable.

Saturation field Displays the amount of saturation applied to the input image before other effects are applied. Editable.

Saturation Variation field Displays the amount of variance applied to the saturation. Editable.

Luminance field Displays the amount of luminance applied to the input image before other effects are applied. Editable.

Luminance Variation field Displays the amount of variance applied to the luminance. Editable.

Selective Drawing Effect Settings

These settings are available when a layer is selected in the Layer Table with Selective Drawing as the effect.

NOTE Most of the settings in the Paint Strokes section are the same as for the Drawing effect. Selective Drawing also has two extra settings.

Paint Strokes colour pot Displays the colour applied to the paint strokes. Click to change the colour.

Front Mix field Displays the amount of colour from the source image that is visible. Editable

Inclusion colour pot Displays the colour on which the paint strokes are created. Click to change the colour.

Inclusion Tolerance field Displays the percentage of tolerance applied to the selected colour. The higher the tolerance, the more paint strokes are created. Editable.

Exclusion colour pot Displays the colour on which the paint strokes are eliminated. Click to change the colour.

Exclusion Tolerance field Displays the percentage of tolerance applied to the selected colour. The higher the tolerance, the more paint strokes are eliminated. Editable.

Global Colour box Switch between the use of colour selections or grayscale versions of these colour selections.

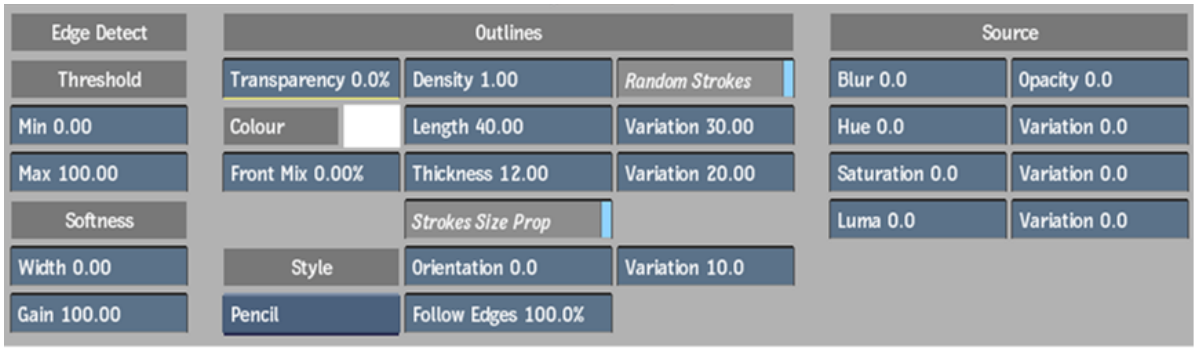
Global Length field Displays the percentage of length by which semi-transparent strokes are modified. Editable.

Global Softness field Displays the percentage by which the range of transparency of semi-transparent strokes is modified. Editable.

Global Thickness field Displays the percentage by which the range of thickness of semi-transparent strokes is modified. Editable.

NOTE The settings in the Front Controls section are the same as for the Drawing effect.

Sketched Outlines Effect Settings



These settings are available when a layer is selected in the Layer Table with Sketched Outlines as the effect.

Minimum Threshold field Displays the minimum amount of colour restraint applied to the outline edges. Editable.

Maximum Threshold field Displays the maximum amount of colour restraint applied to the outline edges. Editable.

Softness Width field Displays the amount of softness applied to the width of the edges. Editable.

Softness Gain field Displays the amount of softness applied to the gain of the edges. Editable.

Transparency field Displays the percentage of transparency applied to the outlines. Editable.

Outlines colour pot Displays the colour applied to the outlines. Click to change the colour.

Front Mix field Displays the percentage of colour from the source image that is visible. Editable.

Style Type box Select the style type of the outlines.

Density field Displays the amount of density applied to the outlines. Editable.

Paint Each Frame button Enable to apply a random variation in length and thickness values.

Length field Displays the length applied to the outlines. Editable.

Length Variation field Displays the variance applied to the length of the outlines.

Thickness field Displays the amount of thickness applied to the outlines. Editable.

Thickness Variation field Displays the variance applied to the thickness of the outlines.

Preserve Stroke Ratio button Enable to allow proportional values to be applied to the length and thickness for the outlines.

Orientation field Displays the orientation applied to the outlines. Editable

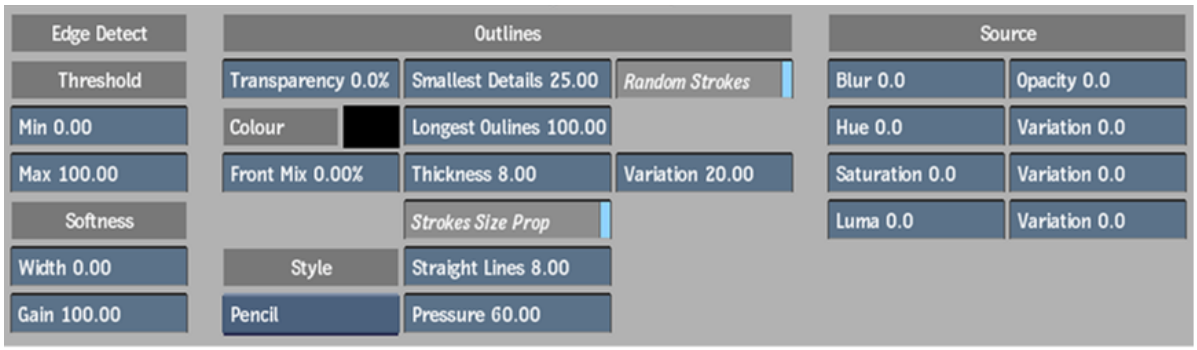
Orientation Variation field Displays the variance applied to the orientation of the outlines.

Follow Edges field Displays the percentage by which the outlines follow their original position. Editable.

NOTE The settings in the Front Controls section are the same as for the Drawing effect.

Scribbled Outlines Effect Settings

The Edge Detect, Softness, and Front Controls settings are the same as for the Sketched Outline effect. Some of the Outlines settings are similar also. The following Outlines settings are specific to the Scribbled Outlines effect.



Smallest Details field Displays the amount by which the smallest outlines will be filtered out. Editable.

Longest Outlines field Displays the amount by which the length of the longest outlines will be increased. Editable.

Straight Lines field Displays the amount of the straightest outlines that are visible. Editable.

Pressure field Displays the variation in the pressure applied to the outlines. Varies with the style of the outline. Editable.

Sharp Outlines Effect Settings

The Edge Detect, Softness, Outlines, and Front Controls settings are the same as for the Sketched Outline effect. The Pattern Transform settings are the same as for the Canvas effect. The following Warp Pattern settings are specific to the Sharp Outlines effect.

Pattern Type box Select a pattern type to apply to the image.

Amount field Displays the amount of warp applied to the image. Editable.

Substance Noise and Splatter

The Substance nodes are procedural noise pattern generators, allowing you to create highly-customizable, repeatable patterns such as cells or concentric shapes to use in your Batch or Batch FX process tree.

Substance Noise node: Use this node to generate a noise pattern based on a chosen preset to use as an input for other nodes.

Substance Splatter node: This node accepts front and matte inputs as the seed that you feed through a chosen preset pattern.

To access the Substance nodes:

1. Drag the Substance Noise node or Substance Splatter node to the Batch or Batch FX schematic.
   The file browser appears, pointing to the default location of the presets:
   - *opt/Autodesk/presets/<product home>/substance_presets/NOISE* or
   - *opt/Autodesk/presets/<product home>/substance_presets/SPLATTER*

   TIP Switch to Proxies view to see a visual representation of the presets.

2. Select a preset.

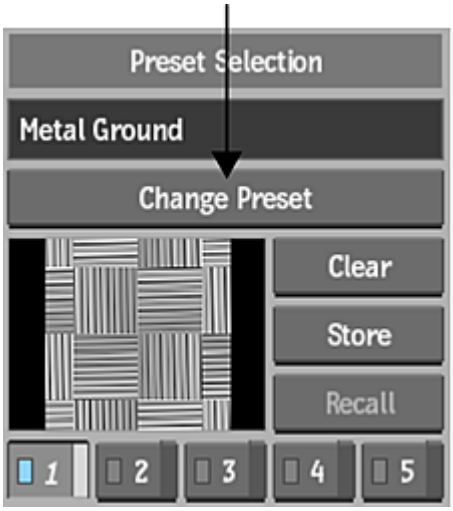
   NOTE The Splatter presets include normal and additive blend versions.

   The preset is automatically loaded into Batch or Batch FX.
3. If you are working with a Substance Splatter node, attach front and matte input clips.
4. Double-click the Substance node in the schematic to access the menu.
   Some of the settings in the menu are dependant on the preset you loaded.
5. Change the menu settings, as needed. Enable Regen to update the image as fields are updated.
6. Optional: Enable Force Tile to ensure that your resulting image tiling is repeatable. In this case, the Crop Output and Zoom options are greyed out.

NOTE If your rendering network is set up with GPU-enabled Burn nodes, you can burn Substance clips. See your *Autodesk Burn Installation and User Guide* for installation and configuration information.

To change a Substance preset:

1. Select the Substance Noise or Substance Splatter node in the schematic.
2. Click Change Preset.



The file browser appears, and you can select a different preset to load.

NOTE All menu parameters are reset when a new preset is loaded.

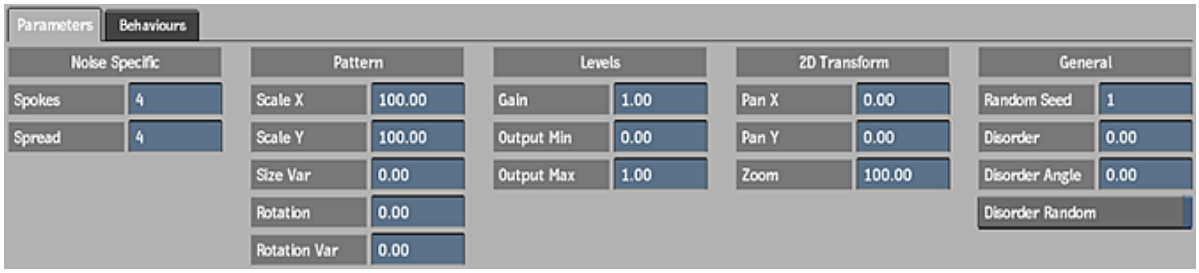
Setting Substance Parameters and Behaviours

Once you have loaded your preset, you can use the Parameters and Behaviours settings in the Substance menu to change and animate the pattern.

Some of the settings in the Parameters tab vary depending on the chosen preset, while other settings, such as Zoom, Random Seed, and Disorder are present in every Substance preset. You can get a quick description of each setting by viewing its tooltip.

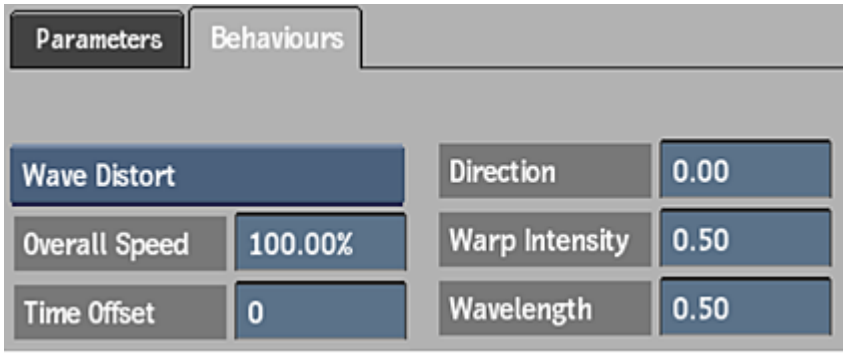
NOTE If Force Tile is enabled, the Zoom field is greyed out.

Example of Noise parameters



Example of Splatter parameters

Use the settings in the Behaviours tab to apply preset motion to your pattern, and to help you quickly animate your patterns. For example, you can choose a Flow behaviour and animate your pattern to resemble a flag waving.



The behaviours are divided into a number of categories, each with their own settings.

Organic Pulse Slow pulsing evolution of width and height of the pattern.

Wind Directional force that spins the pattern with intensity.

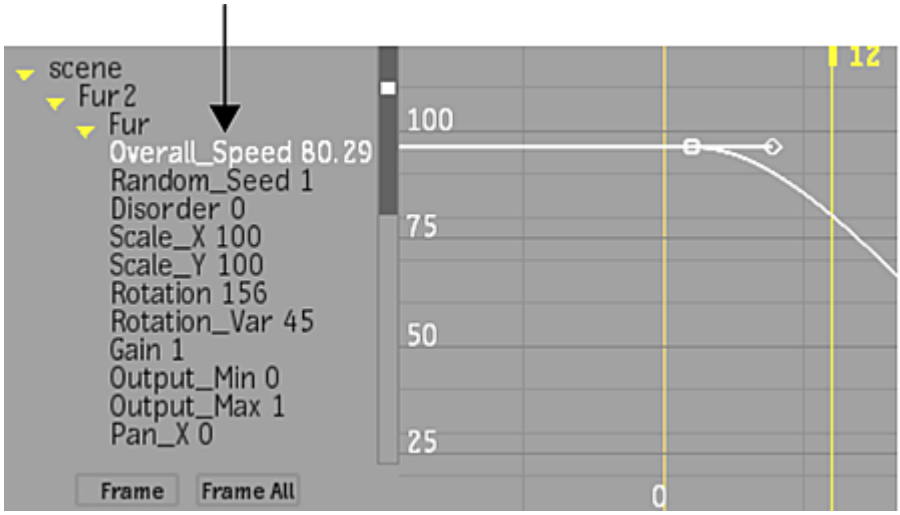
Flow Directional force that displaces the pattern in different direction.

Oscillation Evolution of size and luminance of the pattern.

Wave Distort Directional warping effect that deforms the pattern with different wavelength warps.

You can adjust the playback rate and timing of your chosen behaviour animation by using the Overall Speed and Time Offset settings (available for all behaviours).

You can also adjust the Overall Speed in the Channel Editor.



Substance Menu Settings

Preset Selection Settings

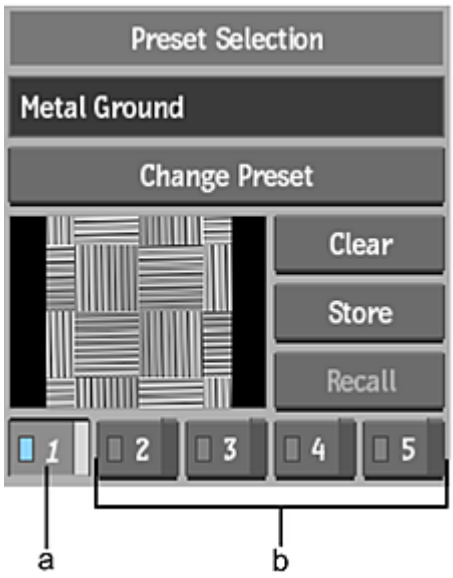
Preset Name field This locked field displays the name of the current preset.

Change Preset button Click to open the file browser to select a different preset.

Preset Memory Settings

The Memory section of the Noise or Splatter menu allows you to store up to five variations of the current preset. This is useful if you want to compare a certain preset type with different settings and animations.

Use the following preset memory buttons to clear, store, or recall menu parameters. A blue LED indicates that a preset memory is stored in the slot.



(a) Preset stored in this preset memory slot (b) No presets stored in these slots

Clear button Clears the selected preset memory.

Store button Stores the menu parameters in the selected preset memory.

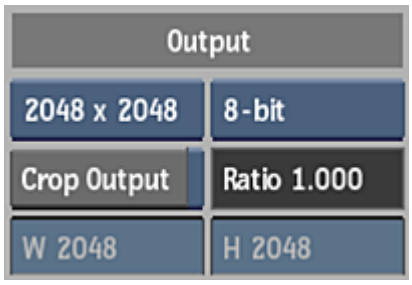
Recall button Recalls the last stored parameters for the selected preset memory.

Preset Memory buttons Use to clear, store, or recall menu parameters. A blue LED indicates that a preset memory is stored in the slot.

Proxy image window Displays a proxy of the stored preset memory.

Output Settings

Use the settings in the Output section to set the size and resolution of clips that are output from the Substance node.



Texture Resolution box Select the resolution of the pattern.

Bit Depth box Select the render/output bit depth of clips.

Crop Output button Enable to crop the output of the pattern by the amounts displayed in the Width and Height fields.

NOTE The crop options are greyed out if the Force Tile button is enabled.

Ratio field Displays the aspect ratio of the pattern clip. The ratio is calculated by W:H to output a square pixel ratio. Non-editable.

Crop Width field Displays the width of the cropped output. Editable.

Crop Height field Displays the height of the cropped output. Editable.

Splatter Pattern Resize Settings

Use the Pattern Resize settings to change the input size and filtering type of a Splatter pattern.

Pattern Resize box Select the input size of the Splatter pattern.

Filter box Select a resize filtering option.

Other Settings

Regen button Enable to automatically update the image as fields are updated.

Force Tile button Enable to ensure that your resulting image tiling is repeatable. In this case, the Crop Output and Zoom options are greyed out.

Behaviours Settings

Behaviour Type box Select the type of preset motion to apply to the pattern.

Overall Speed field Displays the rate at which the animation plays. Editable.

Time Offset field Displays the start point of the animation. With a value of 0, the animation starts at frame 1. With a value of 100, the animation begins as if it has been generating for 99 frames. You cannot animate this field. Editable.

Text

Use Text to make text spin, dance, and change colour over time.

To access the Text menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Tools, then select from the menu (See Accessing Tools (page 493)).

This node accepts a background clip. If a background clip is not connected to the Text node, you can select a resolution for the output of the Text node. Additionally, this node outputs a result and an outmatte.

NOTE The Prerender Text preference does not apply when you access Text from Batch or Batch FX.

Text is compromised of layers, paragraphs, and characters. You can create effects such as a text roll of credits, text crawls, bumpers, and a text that moves on a motion path. You can also use logo images in a text roll so that the logo of a sponsor appears in the credit list.

Text Menu Settings

Timeline FX Quick Menu Settings

To see the full Text menu, click the Editor button.

Alpha Rendering Mode box Select whether a front clip, matte, or both are used to create the text layer.

General Settings

Use Transparency button Enable to create a text effect in which the background clip is used as the text's fill. Also select RGBA from the Alpha Rendering Mode box.

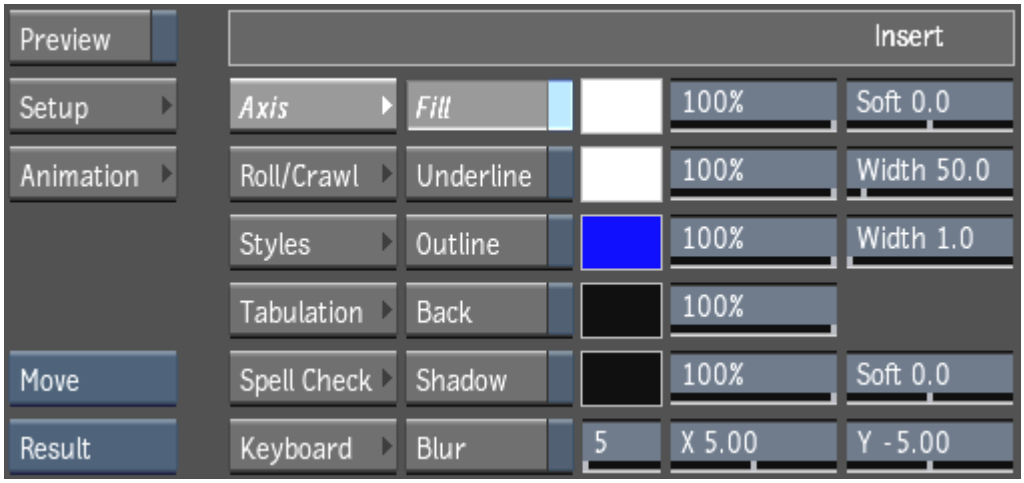
Rendering Mode box Select whether to render in Automatic, Progressive or Interlaced mode.

Preview button Enable to preview the text effect.

Preview Options box This box is available when you access the Text Menu settings from the Timeline.Select to enable Preview FX or Context, to preview the effect.

Setup button Displays the Text Setup menu.

See Setup and Rendering Options (page 1321).



(a) Setup button (b) Animation button (c) Attribute controls (d) Text Mode box

Attribute controls Changes the appearance of text by setting properties such as fill and shadow. See Creating Text Effects (page 1330).

Animation button Creates a text animation by setting different text properties at specific keyframes in a clip.

See Animating Text (page 1339).

Text Mode button Select a text mode to select text, edit text in layers, modify text attributes, or enter text strings.

Applies various text settings to layers, paragraphs, and characters. Use text modes to manipulate layers, add text to a clip, or edit existing text in a clip. See Modifying Layer and Character Properties (page 1330).

Result button Select an option to set the view to either the composited text, the matte of the text, or the original back clip.

Reset box Select an option to reset the selected text layer, the layer axis, the roll/crawl, or the style templates.

Reset All Click to reset the text tool to the default values set in Preferences.

Edit button Opens the Edit menu.

Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

Bit Depth box Select the render/output bit depth of clips.

Scan Mode box Select the scan mode of clips.

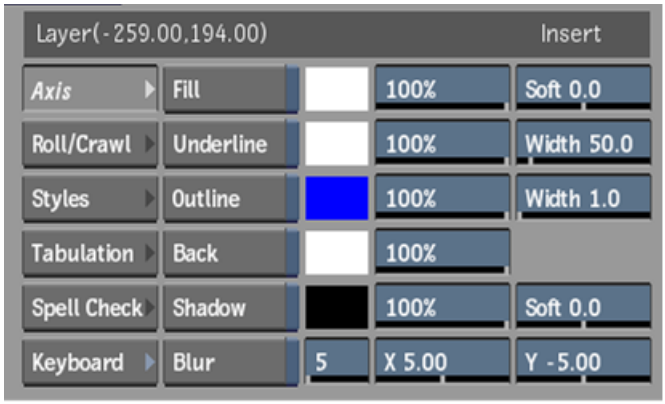
Width field Displays the custom width resolution of the clip. Editable.

Height field Displays the custom width resolution of the clip. Editable.

Aspect Ratio Presets box Select a standard frame aspect ratio. Select the Set to w:h option to set the clip to use square pixels. Select Custom to define a custom frame aspect ratio in the Aspect Ratio field.

Aspect Ratio field Displays the custom render/output aspect ratio. Editable

Insert Settings



Axis button Opens the Axis menu to set properties for layers and characters.

Roll/Crawl button Open the Roll/Crawl menu to set text motion properties for vertical or horizontal text scrolling.

See Creating Text Rolls and Text Crawls (page 1336).

Styles button Opens the Styles menu to create preset text style formats.

See Defining Styles (page 1345)

Tabulation button Open the Tabulation menu to set text tabs.

See Tabulating Text (page 1335)

Spell Check button Open the Spell Check menu to check text for spelling errors.

See Spell Checking (page 1336)

Keyboard button Opens the on-screen keyboard to enter text.

See Entering Text (page 1326)

Fill button Enable to use fill on the text selection.

Sets fill colour, transparency, and softness. Set fill softness using the Soft field. To increase the softness, enter a value close to the maximum of 100. To decrease the softness, enter a value close to the minimum of -100. Enter 0 for no softness.

Fill colour pot Displays the colour of the fill. Editable.

Fill Transparency field Displays the transparency level of the fill. Editable.

Fill Softness field Displays the softness level of the fill. Editable.

Underline button Enable to underline the text selection.

Underlines the characters using the associated colour, transparency, and width. Click the colour pot to choose a colour from the colour picker. To set underline transparency, enter a percentage in the Transparency field. To make the underline more opaque, enter a value close to the maximum of 100. To make the underline more transparent, enter a value close to the minimum of 0. To set the outline width, enter a value in the Width field.

Underline colour pot Displays the colour of the underline. Editable.

Underline Transparency field Displays the transparency level of the underline. Editable.

Underline Width field Displays the width of the underline. Editable.

Outline button Enable to outline the text selection.

Outlines the characters with a solid colour using the associated colour, transparency, and width. Click the colour pot to choose a colour from the colour picker. To set outline transparency, enter a percentage in the Transparency field. To make the outline more opaque, enter a value close to the maximum of 100. To make the outline more transparent, enter a value close to the minimum of 0.

To set the outline width, enter a value in the Width field. Use anti-aliasing rendering options with outlined text. These options are found in the Text Setup menu. See Setup and Rendering Options (page 1321).

Outline colour pot Displays the colour of the outline. Editable.

Outline Transparency field Displays the transparency level of the outline. Editable.

Outline Width field Displays the width of the outline. Editable.

Back button Enable to apply a solid colour background using the associated text layer colour and transparency to the text selection.

Click the colour pot to choose a colour from the colour picker. To set back transparency, enter a percentage in the Transparency field. To make the back more opaque, enter a value close to the maximum of 100. To make the back more transparent, enter a value close to the minimum of 0.

Back colour pot Displays the colour of the background. Editable.

Back Transparency field Displays the transparency level of the background. Editable.

Shadow button Enable to apply a drop shadow to the text selection.

Applies a shadow to text characters using the associated colour, softness, transparency, and Pos X and Pos Y fields. Click the colour pot to choose a colour from the colour picker. To set shadow transparency, enter a percentage in the Transparency field. To make the shadow more opaque, enter a value close to the maximum of 100. To make the shadow more transparent, enter a value close to the minimum of 0. To set shadow softness, use the Soft field. To increase the softness, enter a value close to the maximum of 100. Enter 0 for no softness.

To set the shadow position on the X-axis, enter a value in the X field. A positive value places the shadow to the right and behind the text characters. A negative value places the shadow to the left and in front of the text characters. To set the shadow position on the Y-axis, enter a value in the Y field. A positive value moves the shadow up. A negative value moves the shadow down.

Shadow colour pot Displays the colour of the shadow. Editable.

Shadow Transparency field Displays the transparency level of the shadow. Editable.

Shadow Softness field Displays the softness level of the shadow. Editable.

Shadow X field Displays the shadow position along the X-axis.

Shadow Y field Displays the shadow position along the Y-axis.

Blur Shadow button Enable to apply a blur effect to a drop shadow created with the Shadow button.

The type of blur applied depends on whether Box Blur or Gaussian Blur is selected in the Text Setup menu. Choose a higher value for a greater shadow blur effect. Using Blur Shadow with animated text decreases processing performance.

Blur Shadow field Displays the level of blur applied to the drop shadow. Editable.

Font Settings

Font field Displays the current font. Click to open the font browser to select a different font.

Font Reset button Resets to the default font.

Font Size field Displays the character size for the selected font. Editable.

Font Italic field Displays the angle of italicized text. Positive values make the characters slope to the right. Negative values make them slope to the left.

Font Kerning field Displays the amount of spacing between characters. Positive values increase the spacing. Negative values decrease the spacing.

Paragraph Settings

Justification box Select an alignment option for the paragraph.

Select Left, Right, Centre, or Justified.

Width field Displays the width of the layer. Editable.

Left Margin field Displays the amount of space in the left margin. Editable.

Leading field Displays the space between lines in the selected paragraph. Editable.

Indent field Displays the left indentation value for the first line in the paragraph. Editable.

Separation field Displays the space between the selected paragraph and the one above it. Editable.

Character Channels

Copy Attributes button Copies attribute settings from the selected text character.

Copy Transformations button Copies transformation settings from the selected text character.

Copy All button Copies both attribute and transformation settings from the selected text character.

Paste button Pastes any copied text character channel information.

Text on Path

On Path button Enable to place the selected text on a motion path.

Closed button Enable to close the path by connecting the first and last vertices. Text continues around the shape.

Offset field Displays the position of the text on the path. Animate the Offset value to make the text follow the path for the duration of the clip.

Invert button Reverses the order of the vertices on the path so that the text follows the opposite side of the path.

Text On Path Mode box Select an option to manipulate the selected path spline.

Clear button Resets the text path at the current frame. To reset the path for the entire duration of the clip, click Reset Layer.

Axis Settings

NOTE The Axis settings are available when the Axis button is enabled in the Insert settings.

Axis menu buttons Select whether to display the Layer Axis menu or the Characters Axis menu.

Show Axis button Enable to display the axis in the image window. This button is only available when the Character Axis menu button is selected.

Position X field Moves the layer or selected characters along the X-axis.By default, the value is 0 (for characters). Increase the value to move right and decrease it to move left. Editable .

Y Position field Moves the layer or selected characters along the Y-axis.By default, the value is 0 (for characters). Increase the value to move up and decrease it to move down. Editable .

Lock Position button Enable to lock the position of the text layer. You can lock text in either the Top/Left or Centre position using the box beside this button.

Centre Axis button Moves the axis to the center point of the character. This button is only available when the Character Axis menu button is selected.

X Position Character Axis field Sets the X-axis for the selected characters. Set each letter in a word spinning on its own axis in a different way. This button is only available when the Character Axis menu button is selected.

Using the Axis Pos X field, you set each letter in a word spinning on its own axis in a different way.

Y Position Character Axis field Sets the Y-axis for the selected characters. Set each letter in a word spinning on its own axis in a different way. This button is only available when the Character Axis menu button is selected.

Using the Axis Pos Y field, you set each letter in a word spinning on its own axis in a different way.

X Scale field Scales the layer or selected characters along the X-axis. This value is a percentage, so entering 50 means 50 percent. Use a larger value to increase the scale along the X-axis. Use a negative value to create a mirror image on the X-axis. Enter 0 to make a layer or selected characters disappear. The default is 100.Editable.

Y Scale field Scales the layer or selected characters along the Y-axis. This value is a percentage, so entering 50 means 50 percent. Use a larger value to increase the scale along the Y-axis. Use a negative value to create a mirror image on the Y-axis. Enter 0 to make a layer or selected characters disappear. The default is 100.Editable.

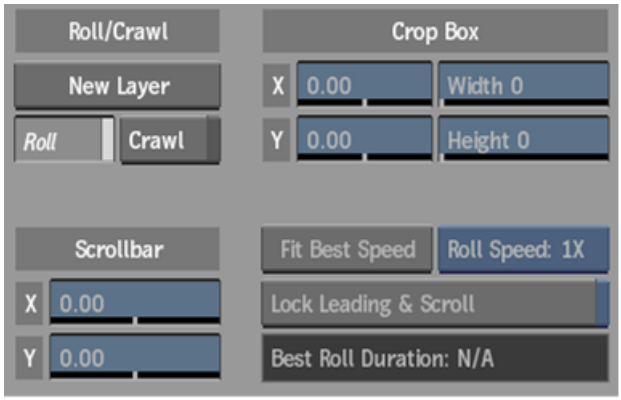
Proportional Scale button Proportionally changes the Scale X and Y values.

X Shear field Shears or slants the layer or selected characters along the X-axis. Use a positive value to slant right and a negative value to slant left. The maximum and minimum values are 60 and -60, respectively. The default value is 0.

Y Shear field Shears or slants the layer or selected characters along the Y-axis. Use a positive value to slant right and a negative value to slant left. The maximum and minimum values are 60 and -60, respectively. The default value is 0.

Rotation field Rotates a layer around its axis or selected characters around their axis.Use a negative value to rotate clockwise and a positive value to rotate counter-clockwise. The default is 0.Editable.

Roll/Crawl Settings



NOTE The Roll/Crawl settings are available when the Roll/Crawl button is enabled in the Insert settings.

Roll/Crawl Settings

New Layer button Creates a layer to contain a text roll or text crawl.

Roll button Enable to create text that scrolls vertically over an image.

Crawl button Enable to create text that scrolls horizontally across an image.

Scrollbar

Scrollbar X field Displays the X position of the roll or crawl layer inside the crop box. The value changes when you move the scrollbar at the right of the text layer for a roll or at the bottom of the layer for a crawl. Editable.

Scrollbar Y field Displays the Y position of the roll or crawl layer inside the crop box. The value changes when you move the scrollbar at the right of the text layer for a roll or at the bottom of the layer for a crawl. Editable.

Fit Best Speed button Creates a broadcast quality text roll based on the number of frames in the clip and the lines of text in the text roll. Enable this button to make corrections to a text roll without altering the speed or duration of the clip.

For NTSC and PAL, broadcast quality is calibrated at four pixels per frame. This rate ensures no flicker in the text roll.

Roll Speed button Select the speed of the text roll, in pixels per second.

Changes the speed of the text roll. For 1X, the speed rate is 120 p/s (pixels/second) in NTSC, and 100 p/s in PAL. For 2X, it is 240 p/s in NTSC, and 200 p/s in PAL, and so on.

Lock Leading & Scroll button Enable to lock the leading of the paragraph text so that you can insert another paragraph into the text roll. If the duration of the clip lengthens after you make modifications, click Fit Best Speed to adjust the length.

Best Roll Duration field Displays the suggested duration for the text roll. The duration is calculated using the start/end position of the roll and the currently selected speed. Non-editable.

Crop Box

Crop Box X field Displays the X position of the crop box inside the image window. The crop box determines the X coordinates of the text roll or text crawl on the clip. Editable.

Crop Box Y field Displays the Y position of the crop box inside the image window. The crop box determines the Y coordinates of the text roll or text crawl on the clip. Editable.

Width field Displays the width at which to crop the text within the text roll or crawl layer. Editable.

Height field Displays the height at which to crop the text within the text roll or crawl layer. Editable.

Styles Settings

NOTE The Styles settings are available when the Styles button is enabled in the Insert settings.

Apply Style 1 button Saves or applies Style 1.

Apply Style 2 button Saves or applies Style 2.

Apply Style 3 button Saves or applies Style 3.

Apply Style 4 button Saves or applies Style 4.

Apply Style 5 button Saves or applies Style 5.

Apply Style 6 button Saves or applies Style 6.

Apply Style 7 button Saves or applies Style 7.

Apply Style 8 button Saves or applies Style 8.

Apply Style 9 button Saves or applies Style 9.

Load Style button Opens the file browser where you can select a saved style to load.

Save Style button Opens the file browser where you can save a style.

Style Mode box Select whether to work with all styles or one style.

Style Save Number box Select the number of the style to save. Available when One Style is selected from the Style Mode box.

Style Name button Sets the name of a defined style.

Style Option box Select an option for creating styles. AutoStyle assigns styles from an existing layer.

Tabulation Settings

NOTE The Tabulation settings are available when the Tabulation button is enabled in the Insert settings.

Add button Adds a tab stop on the text layer ruler. By default, tab stops are set at every 100 pixels.

Delete button Removes the selected tab stop.

Previous button Selects the previous tab stop on the text ruler. Tab stops appear in yellow when selected.

Tab ID field Display the tab number in the current paragraph. When you click Next or Previous, the Tab ID field changes, showing the number of selected tab stops. Editable.

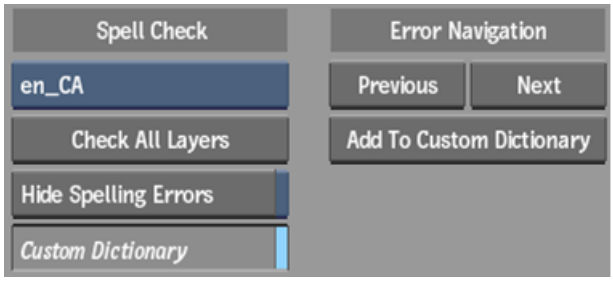
Next button Selects the next tab stop on the text ruler. Tab stops appear in yellow when selected.

Position field Displays the exact tab stop position on the X-axis, in pixels. This value is set in pixels. Editable.

Ruler button Enable to display the tabulation ruler in the text layer.

Justification box Select the text alignment at the tab stop.

Spell Check Settings



NOTE The Spell Check settings are available when the Spell Check button is enabled in the Insert settings.

Spell Check Settings

Language box Select the language for the spell checker.

Check All Layers button Runs the spell checker on the text layers. When the spell checker encounters a misspelled word, it draws a red line through it.

Hide Spelling Errors button Enable to hide the red strike through the line that appears in each misspelled word.

Custom Dictionary button Enable to use your custom dictionary with the spell checker. Disable this button to check all spelling. Disable to check all spelling.

Error Navigation Settings

Previous button Navigates to the previous misspelled word.

Next button Navigates to the next misspelled word.

Add To Custom Dictionary button Adds a word that the spell checker has flagged to the custom dictionary. The next time the spell checker encounters this word, it will be ignored.

Setup and Processing Options

Auto-Softness box Sets the auto-softness and software sampling level.

Auto-Softness mode is set by default and provides a softness equal to the display quality you see in the Text node.

Anti-Aliasing Softness field Displays the softness value of the anti-aliasing sample. Editable.

PreRender button Enable to accelerate rendering on a static layer, such as a text roll (has no effect on text layers with animation).

PreRender has no effect on text layers with animation.

Hardware Anti-aliasing Sample box Select a hardware anti-aliasing sampling level to accelerate edge anti-aliasing with no performance penalty. Available values are dependent on graphics card and project graphic bit depth.

The graphics hardware automatically renders the image at full speed with approximately the equivalent of up to 32 samples of anti-aliasing (depending on your graphics card and project graphic bit depth). Hardware anti-aliasing also gives anti-aliasing during normal interaction instead of only while rendering.

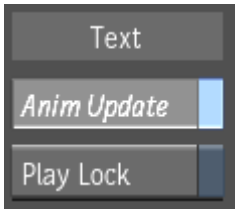
You can combine hardware anti-aliasing level with software anti-aliasing to obtain the desired level of image quality. For example, with hardware anti-aliasing set to 4 samples, and with 4 samples of software anti-aliasing selected, your results should be similar to selecting 16 samples of software anti-aliasing, but with a processing time much closer to that of 4 samples. You should experiment with different combinations to determine what works best for you.

Animation Update button Enable to update animated text dynamically in the image area when you play the clip.

Controls the playback of animated text. When enabled, animated text is updated dynamically in the image area when you play the clip.

Play Lock button Enable to update the animation settings according to the frame or timebar position as you move through the clip while keeping it locked at the current frame.

Controls the playback of frames and displays the playback of animated clips exclusively. When enabled, the first frame is locked while you play the clip.



Wireframe colour pot Displays the colour of the text path. Editable.

Points colour pot Displays the colour of the vertices on a text path. Editable.

See Animating Text on a Motion Path (page 1342).

Global/Layer Blur box Select whether to apply blurring globally or on a per layer basis.

Global Blur makes all shadows appear together, blurred with a single unique colour, on top of the background of all layers. Layer Blur blurs each layer individually according to its priority—background, blurred shadow, and fill.

When using Global Blur with prerendering, the layers are prerendered once and blurred on the front clip. These layers must not have any animation—they are static. For Layer Blur, there are two possible cases:

- All layers are static, without background. Only fill or shadow can be enabled for all layers—no blur. The layers are prerendered once and blurred on the front clip.
- Auto-Softness On, Progressive Rendering, or All Layers Static. Static layers (up to two) are prerendered once and blurred on the front clip as other layers are rendered.

Shadow Blur box Select whether to use a smooth blur with rounded edges (Gaussian), or a rectangular, rougher edged blur (Box).

Clear Undo Buffer button Resets the Undo entries.

Entering Text

Font Type box Select the type of font to display in the font browser list.

Font Preview window Previews the selected font. Click to enter text.

Exit Load Font button Exits the font browser.

Modifying Layer and Character Properties

Layer Axis box Select an alignment option for the selected text layer.

Select:	To move the axis:
Top/Left	To the upper-left corner of the text layer.
Centre	To the centre point of the text layer.

X Scale field Displays the scale of the text layer along the X axis. Editable.

This value is a percentage, so entering 50 means 50 percent. Use a larger value to increase the scale along the X-axis. Use a negative value to create a mirror image on the X-axis. Enter 0 to make a layer or selected characters disappear. The default is 100.

Y Scale field Displays the scale of the text layer along the Y axis. Editable.

This value is a percentage, so entering 50 means 50 percent. Use a larger value to increase the scale along the Y-axis. Use a negative value to create a mirror image on the Y-axis. Enter 0 to make the layer or selected characters disappear. The default is 100.

Proportional Scale button Enable to proportionally change the X Scale and Y Scale values.

X Shear field Displays the shear or slant value of the text layer along the X axis. Editable.

Use a positive value to slant right and a negative value to slant left. The maximum and minimum values are 60 and -60, respectively. The default is 0.

Y Shear field Displays the shear or slant value of the text layer along the Y axis. Editable. Use a positive value to shear up. A negative value near the minimum value -60 shears down. The maximum and minimum values are 60 and -60, respectively. The default is 0.

Rotation field Displays a rotation value for a layer rotating around its axis. Editable.

Use a negative value to rotate clockwise and a positive value to rotate counter-clockwise. The default is 0.

Timewarp

A timewarp is an effect in which the speed of the action taking place appears faster or slower than when it was originally recorded. A freeze-frame effect, in which a single frame is repeated, is another form of a timewarp. Timewarps can have a constant or variable speed. In some cases, such as when creating a fit to fill edit on the timeline, a timewarp is added automatically to a clip.

To access the Timewarp, use:

- Timeline, then add Timeline FX from the Effects ribbon (see Adding Timeline FX (page 475)).
- Timeline, then use Batch FX (see Creating Batch FX (page 496)).
- Tools, then select from the menu (See Accessing Tools (page 493)).
- Batch, then select a node from the Node bin.

In Batch or Batch FX, the Timewarp node accepts front and matte clips, as well as forward and backward vectors.

Motion Vectors are 2D vectors that represent the displacement in normalized pixel units of a pixel in the current frame to the next frame (forward motion vector), or its displacement from the previous frame (backward motion vector). Motion Vectors can be rendered by a 3D application when dealing with synthetic images, or produced through image analysis when images come from live action footage.

Timewarp Menu Settings

Timeline FX Quick Menu Settings

To see the full Timewarp menu, click the Editor button.

Timewarp Quick Menu selector Select which Timewarp quick menu to display.

NOTE There are also some Timeline Timewarp (page 1945) preferences that you can set in the Preferences menu.

Timing Info Settings

These settings are only available if you accessed Timewarp as a Timeline FX, and entered the full editor. They are informational settings only, to give you information about the segment on the timeline.

Source In field Displays the frame or timecode of the In point of the source segment. Non-editable.

Source Out field Displays the frame or timecode of the Out point of the source segment. Non-editable.

Timewarp In field Displays the frame or timecode of the In point of the timewarp result. Non-editable.

Timewarp Out field Displays the frame or timecode of the Out point of the timewarp result. Non-editable.

Record In field Displays the frame number or Record timecode of the In point of the segment in the sequence. Non-editable.

Record Out field Displays the frame number or Record timecode of the Out point of the segment in the sequence. Non-editable.

Length field Displays the length of the segment. Non-editable.

Mode Settings

Timewarp Type box Select a timewarp type: Speed, Timing, or Duration (Duration only available from timeline timewarp). Different settings appear depending on the type selected. Timewarp types are mutually exclusive, that is, animation settings are remembered for each type.

TIP On the timeline, you can quickly create speed or duration timewarps with the keypad and a keyboard shortcut. For speed timewarps, enter the speed on your keyboard, then press `Ctrl-T`. For example, select the clip, enter 50 on the keypad, then press `Ctrl-T` to automatically create a 50% timewarp. For duration, use the same method, but press `Alt-T`.

Speed field Displays the timewarp speed. Editable.

Speed Type box Select whether to display the speed as a percentage, or as frames per second.

Speed Timing field This locked field displays the timing animation value for the current time. Non-editable.

Reverse button Enable to create a reverse timewarp effect; that is, change the speed field value to negative or positive. In Batch, use in conjunction with the Offset field to manually set the desired end of clip.

Timing field Displays the source frame or timecode value for the current time. Editable. You can slip the source Media underneath a Timeline FX Timewarp, without affecting your keyframes, using the Timing field.

Duration field Displays the duration of the timewarp in frames or timecode. Editable.

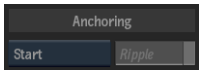
Duration Speed field This locked field displays the speed value corresponding to the duration set in the Duration field. Non-editable.

Strobe Settings

Repeat field Displays the number of frames or fields to be repeated to create a strobe effect. The strobe effect is applied after the timewarp effect, so any speed or timing animations are respected when applying the strobe effect. Editable.

Anchoring Settings

When you access Timewarp as a Timeline FX or within Batch FX, you can see these Anchoring settings:



Anchoring box Select where to set the anchor frame so that its timing is not affected when modifying a timewarp.

Select:	To anchor:
Start	The In frame value as the value not to change.
End	The Out frame value as the value not to change.
Start and End	Both the In and Out frame values. In this case, any speed changes affect the length of the media (and the state of the Ripple button has an effect).

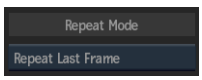
When you access Timewarp as a Batch node, you can see these Anchoring settings:

Source field Displays the anchor frame or timecode value of the timewarp. Editable.

Offset field Displays the amount of offset to apply to the anchor point. Editable.

Repeat Mode Settings

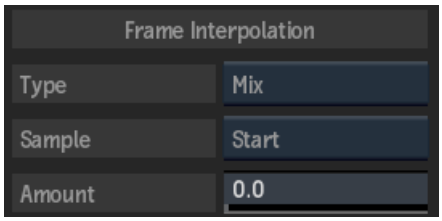
NOTE Not available if Timewarp is accessed as a Batch node.



Repeat Mode box Select how out of range timing values are handled.

Select:	To:
Lock to Segment Range	Only use timing values within the range of the segment. The Timing field is displayed in red when reaching a timing value that is impossible to achieve with the current source clip frame range (heads and tails).
Repeat Last Frame	Not clamp or limit the timing values, by repeating the last frame when reaching the last frame of the segment range.
Trim Segment	Trim the segment (based on the selected ripple settings), when any modification to the speed or timing cause the range of source frames to exceed the available frames.

Frame Interpolation Settings

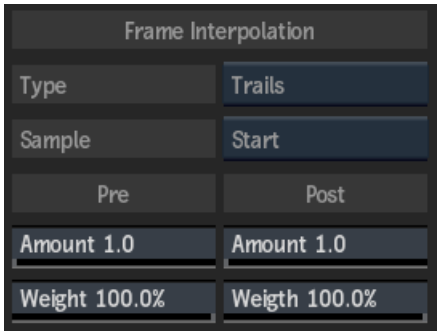


Frame Interpolation Type box Select which type of interpolation to use to blend timewarped frames. Different settings appear below depending on the type selected.

Type:	Description:
Mix	When mixing frames, you are combining multiple frames together to create one blended frame using an additive mix. Each result frame is a blend of an equal number of frames from both sides of the given frame.
Trails	With Trails blending, you have more control than in a simple mix. You can blend the frames before and after a given frame by different amounts. Do this to create a trail that comes before or after any motion in the clip. You can also set the amount of fall-off for the blend so that the trail fades away completely, partially, or not at all.
Motion	Motion estimation blends frames with subpixel accuracy, and is well-suited for particularly challenging timewarps. Motion estimation is good for processing slow-motion, constant, or variable speed timewarps of interlaced and progressive material. It is particularly good for material that introduces artefacts or for which you cannot get satisfactory results using Trail or Mix values.

Sample Mode box Select whether the interpolated position of the timewarp is set in relation to the beginning, middle or end of each frame or field. Available for Mix and Trails types.

Mix Amount field Displays the mix value for the timewarp. Each result frame is a blend of an equal number of frames from both sides of the given frame. Editable.

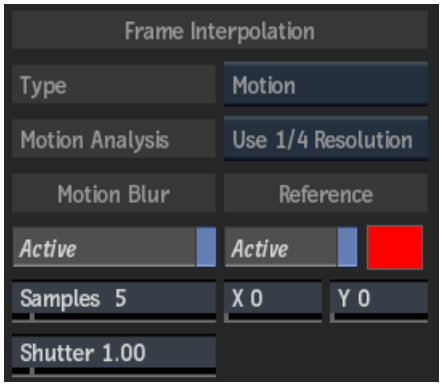


Pre Amount field Displays the value to mix the frames that precede the current frame with the current frame. Editable.

Pre Weight field Displays the fall-off value for the pretrail. When the value is 0%, the image in the current frame is at 100% intensity and the first image in the trail is at 0% intensity. When the value is at 100%, there is no fall-off. Editable.

Post amount field Displays the value to mix the frames that follow the current frame with the current frame. Editable.

Post Weight field Displays the fall-off value for the posttrail. When the value is 0%, the image in the current frame is at 100% intensity and the first image in the trail is at 0% intensity. When the value is at 100%, there is no fall-off. Editable.



Motion Analysis Quality box Select the level of motion analysis accuracy and rendering speed. Select Use Full Resolution to render the images at the current resolution. In Batch or Batch FX, this setting is unavailable if an external input is attached to the node's forward vector or backward vector tabs.

Motion Blur Active button Enable to apply a motion blur to the selected clip.

Sample field Displays the number of frames to sample when creating the timewarp. The samples include the current frame, and an equal distribution of past and future frames. Editable.

Shutter field Displays the number of frames for which the shutter stays open. For example, when the shutter value is set to 3, every third frame is as a sample. Editable.

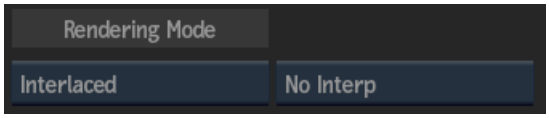
Reference Active button Enable to apply the timewarp only to objects that do not have the same relative motion as the selected pixel. When enabled, you can edit values in the X and Y fields.

Reference colour pot Displays the colour of the crosshair that marks the reference point. Editable

Reference X field Displays the horizontal position of the pixel to use as a reference point at the selected frame. Editable.

Reference Y field Displays the vertical position of the pixel to use as a reference point at the selected frame. Editable.

Rendering Mode Settings



Rendering Mode box Select whether to render in Automatic, Progressive or Interlaced mode.

Interpolation Mode box Select an interpolation mode (available if Interlaced or Automatic is selected in the Rendering Mode box). Interpolation helps remove field jitter artefacts that are produced when a reversal of fields occurs during the timewarp.

Select:	To:
No Interpolation	Use no field interpolation.
Half Interpolation	Interpolate only on displaced fields.
Full Interpolation	Interpolate on all fields. Note that this option has the least aliasing, but you may notice some softening of the image due to the blending of the fields.
Shift Up	Shift the image up by a single line. This may help reduce artefacts that appear in reverse timewarps of 100%.
Shift Down	Shift the image down by a single line. This may help reduce artefacts that appear in reverse timewarps of 100%.

Warper

Use the Warper to warp a clip or morph from a source clip to a result clip. Warping is free-form distortion of an image; it is a transition effect that matches the morphology of one clip gradually to that of another clip, such as when a human is morphing into an alien.

You can access the Warper tool through:

- Tools, then select from the menu (See Accessing Tools (page 493)).

Warper Menu Settings

Miscellaneous Buttons

Exit button Exits the Warper tool.

Load button Loads a setup.

Save button Saves a setup.

Setup Name field Displays the name of the last saved setup.

Revert button Reverts to the last saved setup.

Source option box Select a field rendering mode.

Player button Opens the Player to view the rendered clip.

Step Render field Displays the frame number interval to be rendered (to see intermediate results, for example). Editable.

Preview button Previews the distort effect without rendering.

Setup button Sets playback options, changes display colours, and inputs slip values.

Animation button Opens the channel editor.

View box Select an option to set the view in the image window.

Auto Key button Enable to set a keyframe automatically each time you change a value at any frame.

Reset All button Resets all parameters.

Setup Menu

Decimation field Displays the number of subdivisions for the main patches of the mesh. Editable.

Decimation is the number of subdivisions for the main patches of the mesh (not the small subdivided elements). The default decimation value is 10, which means that each patch is divided into 10 by 10 elements. You should increase this value when:

- You want to use smoother curves along the edges of the patches.
- A patch has been greatly subdivided. When the Decimation value is too low, the subdivided sections will appear to "swim" instead of locking to points and splines.
- A patch is twisted extensively. When the Decimation value is too low, the interior of the patch may appear to fold or break up unexpectedly. Increase the value to smooth the interior of the patch.

Keep the following points in mind when you are adjusting the decimation value:

- Increasing the Decimation value will cause interaction and processing speed to decrease dramatically.
- The Decimation value remains in effect during final processing.
- There is no relationship between the Decimation value and the TexPrec value. The Decimation value defines the precision of the geometry, whereas the TexPrec value affects the precision of the image that is mapped to this geometry.

Samples field Displays the anti-aliasing factor (the number of samples per pixel during final rendering). Editable.

The Samples field specifies the anti-aliasing factor. This number specifies the number of samples per pixel during final processing.

Texture Repeat button Enable to allow pixels to repeat when a source mesh goes outside of the borders of an image.

Texture Quality box Select a texture rendering option.

Provides three texture rendering options.

Select:	To:
High Quality	Use high-quality hardware texture rendering. This option always uses the highest level of precision. High Quality tiles images according to the available texture memory. This option is the default and is the recommended rendering option for film images or any image that is greater than the available texture memory.

Select:	To:
Low Quality	Use low-quality hardware texture rendering, specifically during interactive rendering mode or when you want to render an image quickly. This option uses the precision level that you set in the Precision field.
Pixel Rendering	Use polygon rendering.

Click Preview to hide the mesh and preview the resulting frame.

Texture Precision field Displays the speed/quality ratio for interaction, as well as the rendering quality when Low Quality is selected as the rendering option. Editable.

The higher the value, the higher the quality of the display during interaction, and the higher the quality of final processing when rendering under Low Quality.

Motion Update button Enable to use the same values for properties such as position, rotation, and colour in the scene. Disable to manually copy keyframe values from one frame to another.

When disabled, animated objects do not move, but keep the position of their current value. Disable Motion Update when you want to copy keyframe values from one frame to another.

Play Lock button Enable to update the animation settings according to the frame or timebar position as you move through the clip while keeping it locked at the current frame.

Front Slip field Displays the number of frames to slip the front clip. Editable.

Back Slip field Displays the number of frames to slip the back clip. Editable.

Front Matte Slip field Displays the number of frames to slip the front matte clip. Editable.

Back Matte Slip field Displays the number of frames to slip the back matte clip. Editable.

Icons button Enable to display splines, tangents, and vertices in the image.

Spline colour pot Displays the colour of splines in the image. Editable.

Tangent colour pot Displays the colour of tangents in the image. Editable.

Vertex colour pot Displays the colour of vertices in the image. Editable.

Shape Interpolation box Select an interpolation type.

Main Menu

Add Mesh button Enable to add a mesh or patch to the image.

Xcells field Displays the number of patches along the X axis. Editable.

Ycells field Displays the number of patches along the Y axis. Editable.

Proportional button Enable to keep the XCells and YCells fields proportional.

Delete Mesh button Deletes the selected mesh.

Front Clip Setup box Select an option for using the front clip.

Back Clip Setup box Select an option for using the back clip.

Matte Clip Setup box Select an option for using the matte clip.

Back Matte Clip Setup box Select an option for using the back matte clip.

Background Clip Setup box Select an option for using the background clip.

Warp button Displays the warping controls.

Morph button Displays the morphing controls.

Front Source button Enable to create a front source mesh.

Show Front Source button Enable to display the front source mesh in the image.

Front Source colour pot Displays the colour for the front source mesh. Editable.

Front Destination button Enable to create a front destination mesh.

Show Front Destination button Enable to display the front destination mesh in the image.

Front Destination colour pot Displays the colour for the front destination mesh. Editable.

Back Source button Enable to create a back source mesh in a morph.

Show Back Source button Enable to display the back source mesh in the image.

Back Source colour pot Displays the colour for the back source mesh. Editable.

Back Destination button Enable to create a back destination mesh in a morph.

Show Back Destination button Enable to display the back destination mesh in the image.

Back Destination colour pot Displays the colour for the back destination mesh. Editable.

Source Interpolation button Enable to use source interpolation in a morph.

Show Source Interpolation button Enable to display source interpolation in the image.

Front Source Mapping box Select the area that the front source is mapped to.

Front Destination Mapping box Select the area that the front destination is mapped to.

Back Source Mapping box Select the area that the back source is mapped to.

Back Destination Mapping box Select the area that the back destination is mapped to.

X Position field Displays the position of the control points along the X axis. Editable.

Y Position field Displays the position of the control points along the Y axis. Editable.

Z Position field Displays the position of the control points along the Z axis. Editable.

Curves button Enable to activate the interpolation and dissolve curves.

Dissolve field Displays the percentage change in the colour of the pixels at the current frame. Editable.

Interpolate field Displays the percentage change in the position of the pixels at the current frame. Editable.

Move button Enable to move a vertex, an edge, or a tangent handle.

Move Mode box Select an option to change the position, size, and orientation of part or all of the mesh.

Subdivide button Enable to divide patches in a mesh into smaller subpatches and split the edges into smaller segments.

Auto button Enable to reset broken tangent handles.

Select button Enable to select multiple control points to translate, scale, or rotate all or part of the mesh.

Split button Enable to add a vertex along the edge of a patch.

Copy button Copies the selected mesh.

Paste button Pastes the copied mesh.

Freehand button Enable to perform intricate mesh modelling by clicking to create a freehand mesh.

Break button Enable to break tangent handles.

Delete button Enable to delete edges or vertices of the mesh.

Magnet button Enable to use the Magnet tool to define the shape of the warped image.

Magnet Mode box Select a transformation type to use when Magnet is enabled.

Magnet Curve Editor Defines the weighted polarity from the centre to the edge of the magnet.

Magnet Curve Home button Resets the position of the magnet curve after panning.

Magnet Curve Reset button Resets the Magnet Curve Editor.

Magnet Curve Tools box Select an option to modify the magnet curve.

Magnet Load button Loads a custom magnet setup.

Magnet Save button Saves a custom magnet setup.

Stabilizer button Opens the Stabilizer menu to track movement in the clip to help in creating meshes.

Stabilizer Clip box Select the clip to be loaded into the stabilizer.

Tracking Mode box Select whether to track the axis or vertices of the clip.

Tracking Mesh box Select whether to apply the tracking data to the source or destination mesh.

Animating Keyframes

27

You can build animations simply by editing the values of parameters in any effect editor or menu. When you animate a parameter, you create an animation channel that you can use to control the values of the parameter over time.

To use auto key to build animations automatically in the editor for an effect or tool:

1. Enable Auto Key.
2. Go to the frame where you want to add the keyframe.
3. Change the value of the parameter. Enter a new value in the field using the calculator, or click and drag left or right to change the value.

   A keyframe is added to the animation channel for the parameter you edited.

   NOTE When auto key is enabled, you create keyframes whenever you change the value for a parameter. You also change the value of any parameter that you have already set.

To set keyframes manually in the editor for an effect or tool:

1. Disable Auto Key.
2. Go to the frame where you want to add the keyframe.
3. Change the value of the property. Enter a new value in the field using the calculator, or click and drag left or right to change the value.
4. Right-click the field and select Set Keyframe (Current Value).

To delete keyframes:

1. Go to the frame that contains the keyframe you want to delete.
2. Right-click the parameter that contains the keyframe and select Delete Keyframe.

   NOTE A yellow bar appears under values that are keyframed at the current frame.

To reset keyframe values:

1. Right-click the parameter and select one of the following options.

Select:	To:
Reset (Default Value)	Reset the value of the keyframe to the original value.

Select:	To:
Reset Channel (Current Value)	Reset the value of the entire channel to the current value.
Reset Channel (Default Value)	Reset the entire channel to the original value.

Keyframe Indicator Reference

A number of colours are used in numeric fields to indicate that keyframes are set for a parameter.

Indicator	Example
An asterisk in a field indicates that a value has changed from the default, but a keyframe is not set. Once a keyframe is set, the asterisk disappears.	
A blue line under a numeric field indicates that one or more keyframes are present on this channel, but not at the current frame.	
A yellow line under a numeric field indicates that a keyframe is present on this channel at this point in time.	
A dotted yellow line under a numeric field indicates that this channel is linked to another channel using an expression. In this case, the numeric value is also greyed out, as the value can not be modified. The name of the linking channel is also displayed as part of the tooltip.	

Expression / Keyframes indicators are also displayed above a Batch, Batch FX or Modular Keyer node. The indicators appear when the node contains at least one Expression or one Keyframe.

- The Expression indicator is represented by a yellow dotted circle:

- The Keyframe indicator is represented by a full light blue circle:

- The Indicators can be enabled / disabled using the Show Keyframe button located in Batch Prefs / MK Prefs > Preferences > Schematic or by using the "Toggle Batch Show Keyframe / Expression Indicators" keyboard shortcut. The shortcut is assigned to Shift-K in the Flame, Smoke Classic and Lustre profiles. The shortcut is unassigned in the Smoke FCP profile.
- Double-clicking on an indicator frames the node's channels in the Animation menu.
- The framing is applied in the Viewport, if one of the Animation views is already up.
- The framing is done in the Animation menu at the bottom, if no Animation view is currently displayed in a viewport.

Animating Keyframes in the Animation Channel Editor

The Animation Channel Editor offers more control over your animation than is offered when editing the values of parameters in an effect or tool editor. Use the Animation Channel Editor to fine-tune changes across all animation channels available for the effect or tool you are working on.

To display the Channel Editor:

1 Click Animation.

The Animation Channel Editor is displayed in the bottom area of the window.

To display the Animation Channel Editor in a viewport:

1 Click the View box.

2 Select one of the following from the Animation menu:

Select:	To display:
Channels	Animation curves.

Select:	To display:
Tracks	Animation channels as tracks. This view can help you visualize and align keyframes.
Info	A table displaying keyframe values in text format.

Panning and Zooming in the Animation Channel Editor

To center the Animation Channel Editor on your work, click one of the following:

- Frame: to center the display on the currently selected channel.
- Frame All: to center the display on all the animation channels that have at least one keyframe set on them.

  NOTE These buttons are located at the bottom left corner of the Animation Channel Editor.

To pan the Animation Channel Editor:

1. From the Tools box, select Pan.
2. Click and drag the cursor to pan the view.

To zoom the Animation Channel Editor:

1. From the Tools box, select Rect Zoom.
2. Click and drag a rectangle on the area you want zoom.

NOTE You can zoom x and y independently. Use the Zoom command from the Tools box. Click and drag up or down to zoom the y axis. Click and drag left or right to zoom the x axis.

Adding Keyframes to Channels in the Animation Channel Editor

To add keyframes automatically:

1. In the Animation Channel Editor, enable Auto Key.
2. Select the channel you want to add keyframes to.
3. Click the value next to the channel name to open the calculator.
4. Enter a value and click enter to close the calculator.
   A keyframe is added to the channel you selected.

To set keyframes manually:

1. In the Animation Channel Editor, disable Auto Key.
2. Select the channel or channels you want to add keyframes to.
3. Drag the positioner to the frame where you want to set the keyframe.
4. Click Set Key.

The current values for the selected channels are set at the current frame.

TIP It can be useful to set keys across multiple channels using the Track Editor since this view focuses on channels, and their composing keyframes, in time. To view the Track Editor, select Track from the Channel View box.

To add keyframes at different locations on the animation channel:

1. From the Tools box, select Add Points.
2. Click the spot on the channel where you want to add the keyframe.

   NOTE You can snap the keyframes you add to a frame (rather than in between frames). From the Curve Functions box, select FrmSnap.

To insert a keyframe:

1. Select the channel or channels you want to add keyframes to.
2. Drag the positioner to the frame where you want to set the keyframe.
3. Click Insert Key. The Insert Key button behaves differently depending on the current frame:
   - If the current frame is the last keyframe, a new keyframe is created <*duration*> number of frames after the last keyframe (*duration* is the value in the Duration field). The current frame advances to the new keyframe.
   - If the current frame is a keyframe other than the last keyframe, a new keyframe is created <*duration*> number of frames after the present keyframe. Other keyframes are moved by the same duration causing a ripple effect.
   - If the current frame is between keyframes, Insert acts the same as the Set button: a new keyframe is created and the duration is unchanged.

Changing Keyframe Values in the Animation Channel Editor

You can change keyframe values in the Channel or Track View of the Animation Channel Editor.

To gesturally change keyframe values:

1. In the Animation Channel Editor, select the channel that includes the keyframes you want to change.
2. Click Frame to view the channel.
3. From the Tools box, choose Select.

4 Select the keyframes. To select multiple keyframes, press Ctrl+drag to create a rectangular selection.

   NOTE To select keyframes in only the selected channel, press Ctrl+F and drag a rectangular selection around the keyframes you want to select.

5 Drag the keyframes to the new values. Press Alt to snap the frame value (vertical axis) to the nearest integer. Press Ctrl+Alt to snap to the nearest multiple of ten.

NOTE Enable Track Snap on the Filter tab to snap keyframes to frames (not in-between frames). Or, press Alt+Z to snap when Track Snap is disabled.

To set a specific value for a keyframe in the Animation Channel Editor:

1 Select the Channel that has the keyframe you want to change.
2 Set the positioner to the frame where the keyframe is located.
3 Click the value next to the channel name and enter a new value in the calculator. Or click the number and drag to change its value gesturally.

Copying and Pasting Channels or Keyframes in the Animation Channel Editor

You can copy and paste animation curves or keyframes between channels.

You can copy and paste folders provided the folder to which you are pasting has the same number of channels and folders as the folder you copied.

For example, in Action, you can copy the Scale folder from an axis and paste it into the Scale folder of another axis because they have the same channels (X scale, Y scale, for example). Trying to copy and paste the Camera folder to the Scale folder will not work because the two folders do not contain the same number and kinds of channels.

NOTE If an expression is applied to the selected channel, the expression is copied instead.

To copy and paste a selection of keyframes:

1 Select keyframes on a channel, and click Copy.

   TIP Keyframes do not have to follow one another to be copied and pasted to another channel.

2 Drag the positioner to a new frame.
3 Select the channel and the frame where you want to paste the keyframes, then click Paste.

   NOTE With the Select tool active, you can also `Shift+spacebar`-click a channel to paste keyframes starting from the location of the cursor. For example, `Shift+spacebar`-click frame 10 on a channel to paste the copied keyframes into the channel starting at frame 10.

   The keyframe values for the selected channels are pasted and set in the current frame.

To copy and paste a curve:

1 From the Channel View box in the Keyframe tab, select Channels. The Channel View box is to the left of the Insert Key button.
2 Select the channel containing the animation curve.

   TIP Select a folder to copy all the channels in a folder.

3 Click Copy.

4 Select the channel where you want to paste the copied curve.

5 Click Paste.

The channel values you copied are applied to the selected channel or folder.

Changing the Duration of an Animation in the Animation Channel Editor

You can change the duration of an animation by scaling its channel horizontally in the Channel Editor. This allows you to change the number of frames it takes for an animation to play through, without having to move each keyframe manually.

There are three ways to change the duration of an animation in the Animation Channel Editor:

- Use the X Scale tool to scale the channel using any keyframe as the origin of the scale.
- In the Tracks view, Drag the track handle to scales the track proportionally.
- Using the Duration field.

When changing the length of channels, you may want to adjust the number of frames of the clip in the Total Frames field. For example, if you shorten the length of all the clip's channels, you can then shorten the clips' frames accordingly. Or, if you scale the channels longer than the duration of the clip, you can then add frames to account for this.

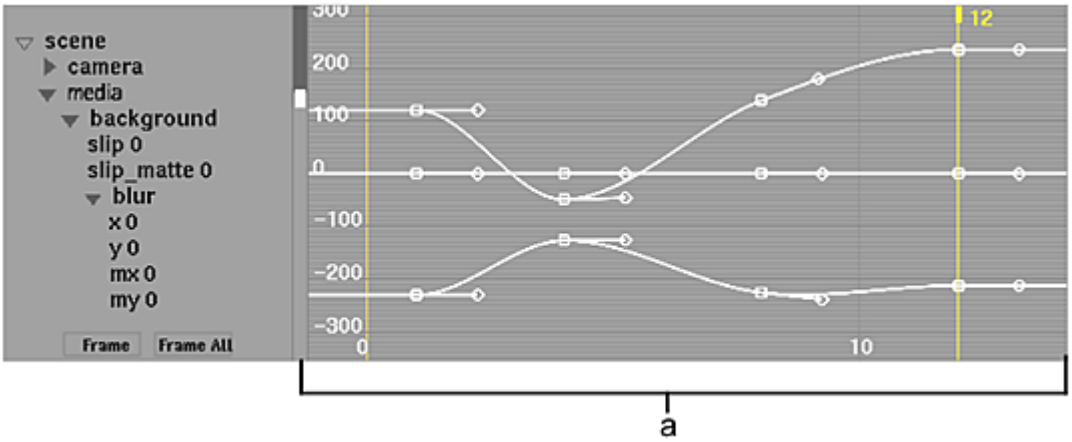
To change the duration of an animation in the Channel View:

1 From the Tools box, select X Scale.

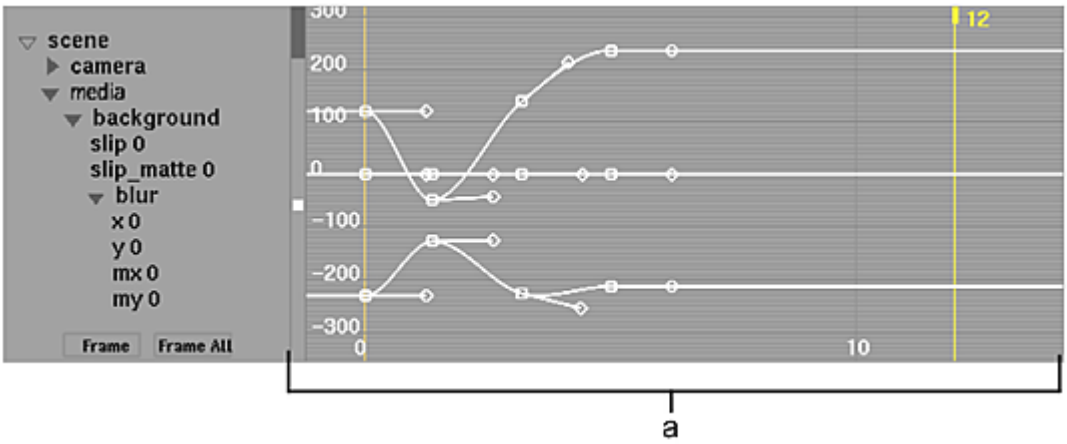
2 Select the keyframes or channels you want to scale.

You may have to click Frame to view the curve.

3 Click the keyframe that will act as the centre of the X Scale. Drag left to decrease the value or drag right to increase the value of the other keyframes along the X-axis, relative to the selected keyframe.



(a) Original animation curve



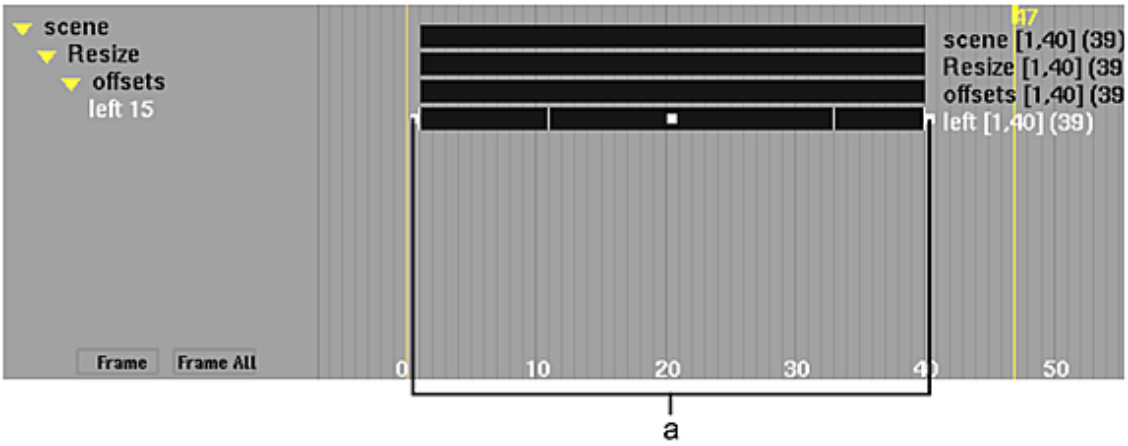
(a) XScale used to compress curves

To scale a channel in the Track Editor:

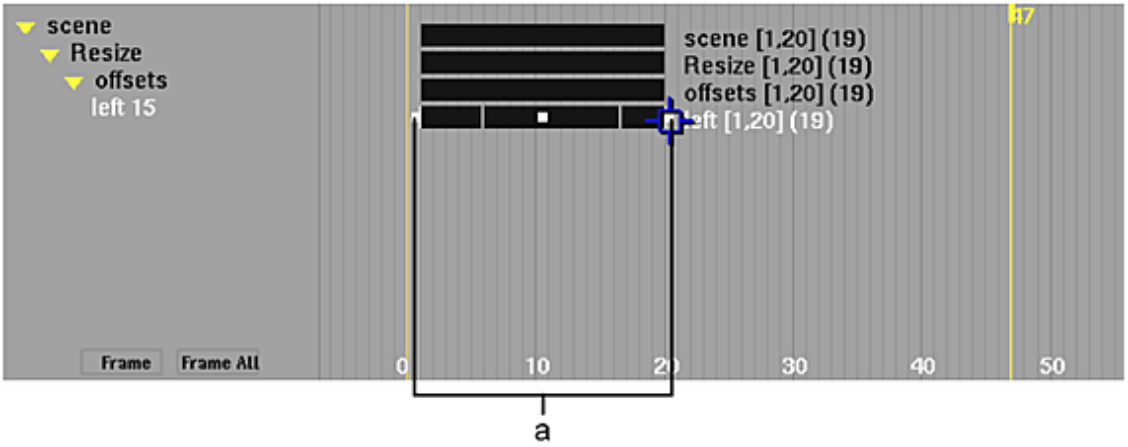
1. From the Channel View box, select Tracks.
2. From the Tools box, select Select.
3. Select the channels you want to scale.
4. Drag the last handle to the left to decrease the duration, or to the right to increase the duration.

   NOTE To avoid adding keyframes between frames when scaling, press Alt-Z while scaling. If you have already scaled the track, Select Frame Snap from the Curve Functions box to snap keyframes in the selected channel to the closest frame numbers.

   The track scales as you drag the handle. All keyframes are repositioned proportionally in time.



(a) Original animation track



(a) Dragging handle used to compress a track from 40 to 20 frames

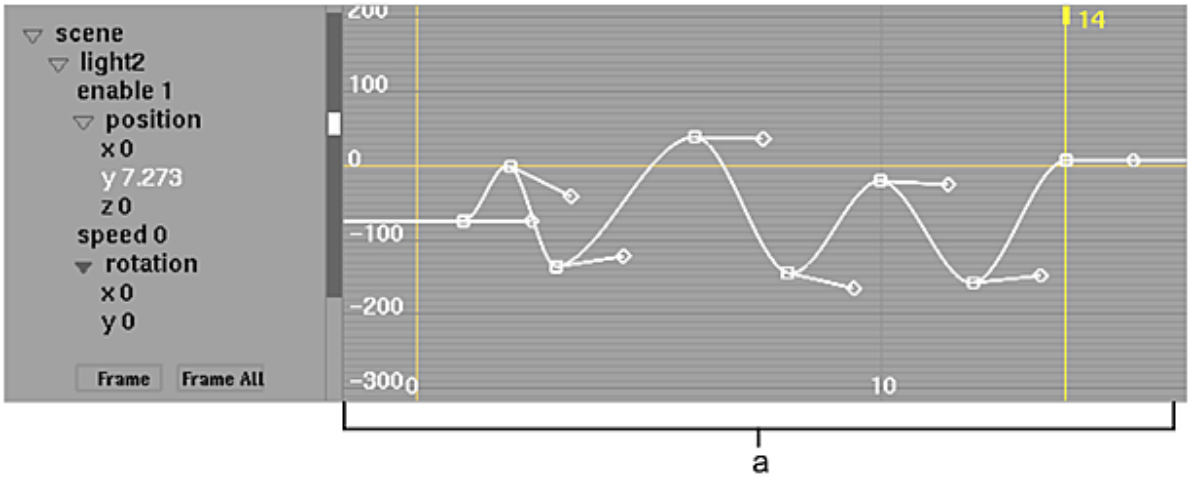
TIP You can move the handle of a folder to scale all channels in the folder. For example, moving the handle of the Scene folder (the top level folder) causes all the folders and channels in the hierarchy to be scaled by the same amount.

To change the number of frames between the current and next keyframe:

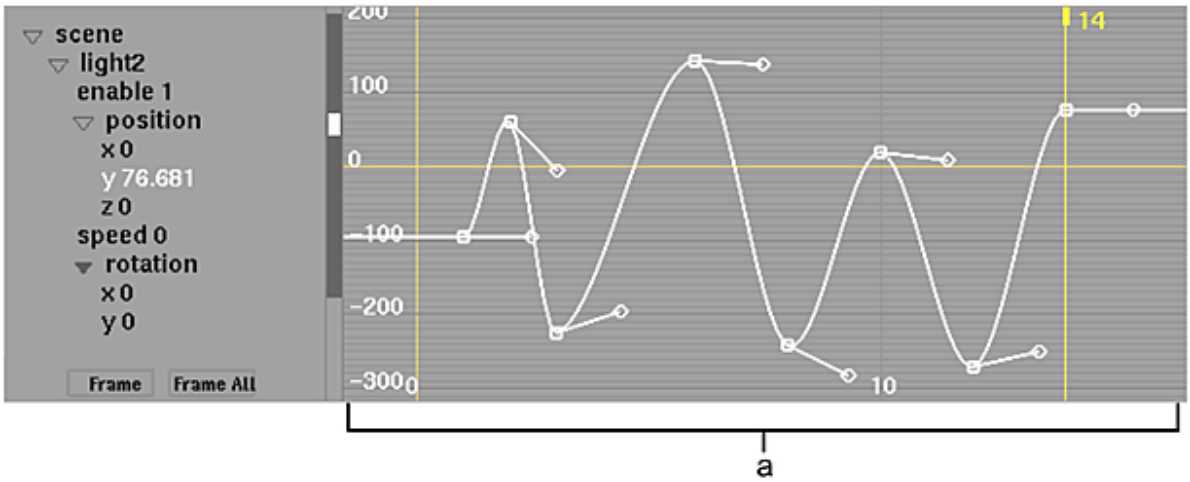
1. Select a keyframe.
2. In the Duration field, which is next to the Insert Key button in the Animation Channel Editor, enter a value for the number of frames between the selected keyframe and the next keyframe.

Changing the Values of Keyframes Proportionally in the Animation Channel Editor

In the Animation Channel Editor, you can change the vertical scale of an animation curve or a selection of keyframes and change keyframe values for a channel or a group of keyframes while maintaining each keyframe's relative frame position.



(a) Original animation curve



(a) YScale used to scale the curve based on a selected keyframe

To change the values of keyframes proportionally:

1. In the Animation Channel Editor, select a channel or group of keyframes.

   You can select more than one channel or a group of keyframes from more than one animation curve.
2. From the Tools box, select Y Scale.
3. Click the keyframe that will determine the centre of the Y Scale.
4. Drag up to increase the value or down to decrease the value of the other keyframes relative to the selected keyframe.

   YScale works differently if you select more than one curve. For the other selected curves, the number of the frame that you select is verified in all animation curves. If the animation curve has a keyframe at the selected frame, this keyframe is used as the centre of the Yscale. This keyframe then sets the proportional scale calculation. If the animation curve does not have a keyframe at the selected frame, the curve remains unscaled.

Offsetting Select Keyframes or Channels in the Animation Channel Editor

You can offset a curve or a group of keyframes on the horizontal or vertical axis.

To offset a channel:

1. Select the channel or keyframes to offset.
2. From the Curve Functions box in the Keyframe tab, select Translate X to offset the channel horizontally or Translate Y to offset vertically.
3. Enter a value in the Curve Value field.

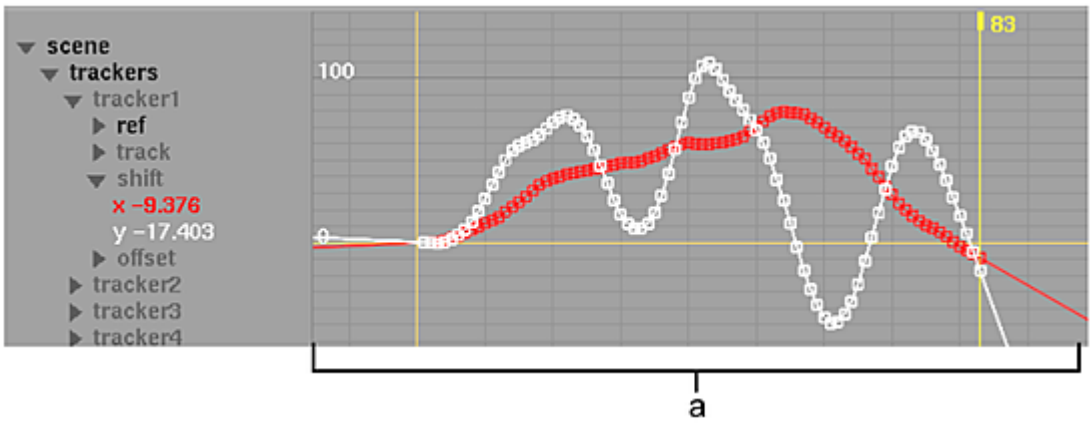
   A negative value offsets the channel or group of keyframes to the left (Translate X) or down (Translate Y). A positive value offsets to the right (Translate X) or up (Translate Y). The value in the Curve Value field produced by the drag returns to zero after release.

   The selected channel or keyframes are offset. The coordinates of each selected keyframe, relative to each other, remain unchanged. Translating along the Y-axis is only visible in the Animation Curve window.

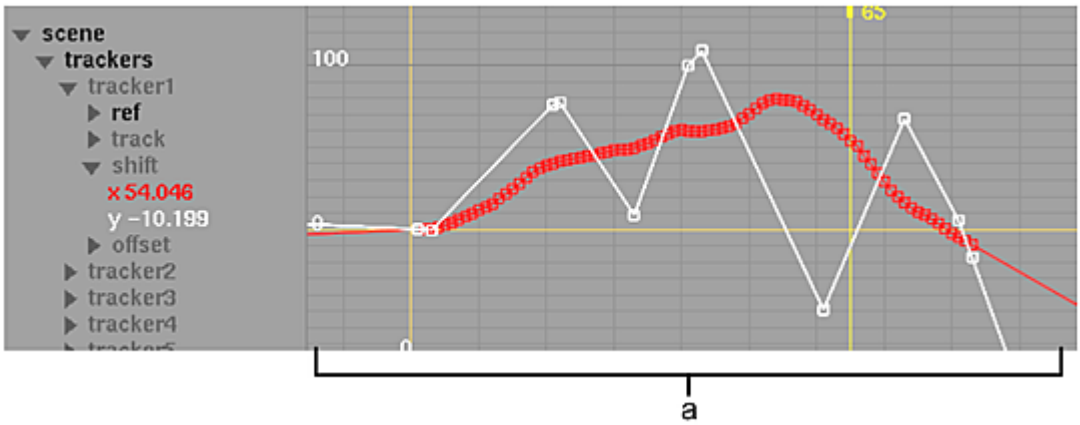
Simplifying the Number of Keyframes in the Animation Channel Editor

You can change the number of keyframes in your animation curve or in a group of keyframes. Using the Simplify function, the peaks and valleys in the selection are analysed and keyframes are generated with a duration between them as specified in the Curve Value field. A large value such as 20 or 30 decreases the number of keyframes between peaks and valleys, whereas a small value such as 1 or 2 increases the number of keyframes. The ideal simplify value varies between animation curves.

For example, the following figure shows an animation curve created using the Stabilizer. Simplify is applied with a value of 20. In sections where the slope of the curve is fairly constant in one direction, keyframes are removed between the peaks and valleys. You can adjust the simplified animation curve more easily because there are fewer keyframes to modify.



(a) The Y curve before applying Simplify has a keyframe at each frame



(a) The Y curve after applying Simplify with a simplification value of 20.

To simplify a channel or a group of keyframes:

1. Select a channel or group of keyframes.
2. From the Curve Functions box in the Keyframe tab, select Simplify.
3. In the Curve Value field, enter a simplification value.

   Use a large number to simplify the curve or group of keyframes. Use a small number to increase the number of keyframes. The value in the Curve Value field produced by the drag returns to zero after release.

Baking Keyframes

You can transform a curve or a group of keyframes by using the Bake function. This can be useful if you want to change an extrapolated cycle curve into a normal curve, with keyframes created along the cycle. You can also use the Bake function to remove the dependence of a curve linked to another curve with an expression.

To bake a cycled curve:

1. Select a curve that has a cycle extrapolation applied to it.
2. Set the current frame to the last frame that you want to be baked.
3. From the Curve Functions box in the Keyframe tab, select Bake.
4. In the Curve Value field, enter a bake value.

   Use a large number to simplify the curve or group of keyframes. Use a small number to increase the number of keyframes. The value in the Curve Value field returns to zero after you have finished baking.

Swapping Animation Curves

You can swap animation curves between two channels. Swap works in conjunction with Copy or Cut. You can also swap single keyframes but not a group of keyframes.

To swap two channels:

1. Select the first channel to swap.
2. Click Cut or Copy.

   The selection is copied to the clipboard.
3. Select the second channel to swap.
4. From the Curve Functions box in the Keyframe tab, select Swap.

   The selected channel is swapped with the channel in the clipboard.
5. Select the first channel and click Paste.

   The two channels are swapped.

Flipping Channels and Keyframes

Select Negate in the Curve Functions box to flip a channel or a group of keyframes vertically. Select Reverse to flip a channel or a group of keyframes horizontally. Typically, you reverse or flip keyframes so that you can create symmetry with an animation. For example, use Reverse on an image that is warping from a sphere to a flat plane so that it then goes from a flat plane back to a sphere again. Perform these functions in the Animation Curve window.

To negate or reverse a channel or group of keyframes:

1. In the Animation Channel Editor, select a channel or group of keyframes.
2. From the Curve Functions box in the Keyframe tab, select Negate or Reverse.

   The selection is flipped horizontally (negate) or vertically (reverse).

Removing Jitter from an Animation Curve

Select Jitter in the Curve Functions box to remove jitter from an animation curve or within a group of keyframes. Use the Curve Value field to specify the Over value. As a general rule, start with a large Over value over *n* frames to remove slow jitter, and a small Over value to remove fast jitter. The Jitter option is typically used on shift data in the Stabilizer to remove jitter while keeping camera movement. See Removing Jitter While Keeping Overall Motion (page 1074).

To remove jitter:

1. Select a channel or group of keyframes.
2. In the Curve Functions box in the Keyframe tab, select Jitter.
3. In the Curve Value field, specify the Over value.

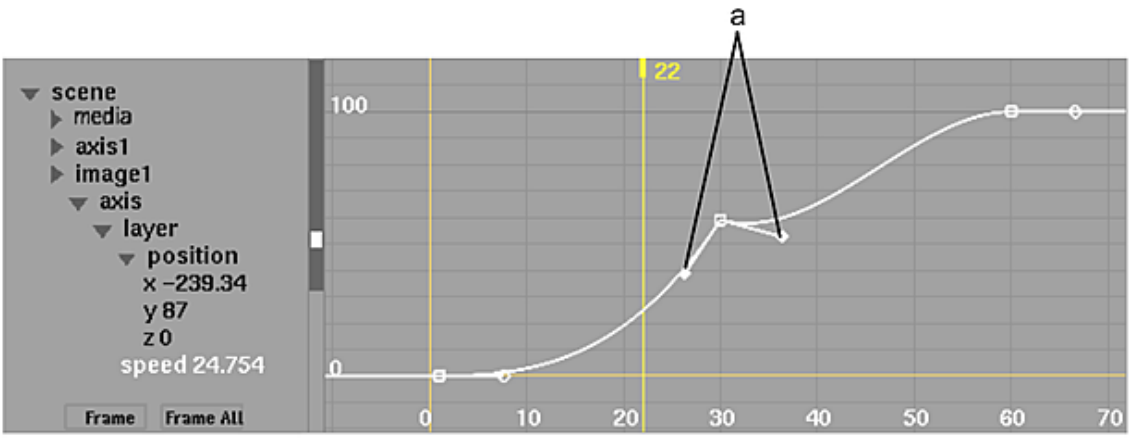
Applying an Average

Select Average in the Curve Functions box to remove jitter from a jittery animation track. While the Jitter option is used for removing jitter caused by trackers in the Stabilizer, the Average option is used to remove jitter from an animation track. Enter the number of keyframes used to calculate the average in the Curve Value field.

You can change the timing of a motion path animation without changing the motion path spline by adding a keyframe to the speed curve.

To average a channel:

1. Select a channel or group of keyframes.
2. In the Curve Functions box in the Keyframe tab, select Average.
3. In the Curve Value field, specify a value.
   The jitter is removed from the spline animation.
4. From the Tools box, select Break.
5. Click a keyframe on the speed channel.
   The keyframe breaks into two tangents.



(a) Keyframe tangents

6. From the Tools box, select Move.
7. Adjust the tangents.

In this example, the speed curve makes the apple accelerate rapidly until it reaches frame 29, where it stops and continues moving slowly.

You can also apply the preceding procedure to a group of keyframes.

Modifying Tangents to Reshape the Animation Curve

To modify the tangents in the Animation Channel Editor.

1. From the Tangents box, select Auto.
2. From the Interpolation box, select an interpolation type. Only animation curves that use Bézier, Hermite, or Natural interpolation have keyframes with tangent handles.
3. From the Tools box, choose Select and move the tangent handles.

   As you move the handles, the slope on each side of the keyframe is modified independently.

   NOTE You must break tangent handles to move them independently. From the Tools box, select Break Tangents and click the tangents you want to break.

To reset tangents to their default:

1. From the Tools box, select Auto Tangent.
2. Click the handle of the tangent you want to reset.

Changing the Value of a Tangent Handle by a Specific Amount

You can modify the shape of an animation curve by rotating its keyframe tangent handles to control the smoothness and speed of the animation.

To rotate a keyframe's tangent handles:

1. Select a channel or group of keyframes.
2. From the Curve Functions box in the Keyframe tab, select Tangent R to rotate each right tangent handle or Tangent L to rotate each left handle.

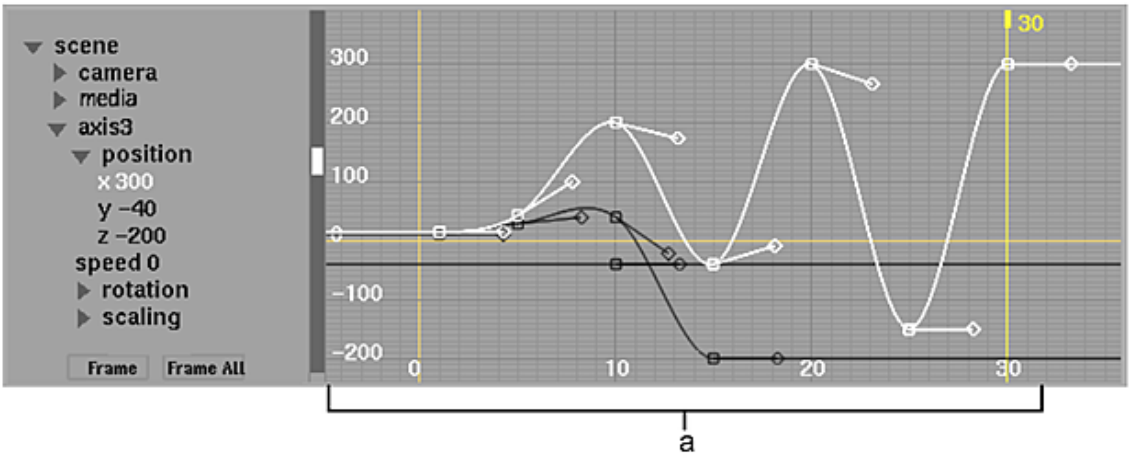
   NOTE Tangent R and Tangent L only work on curves and keyframes that use Bézier, Hermite, or Natural interpolation. Select your option from the Interpolation box.

3. Enter a value in the Curve Value field.

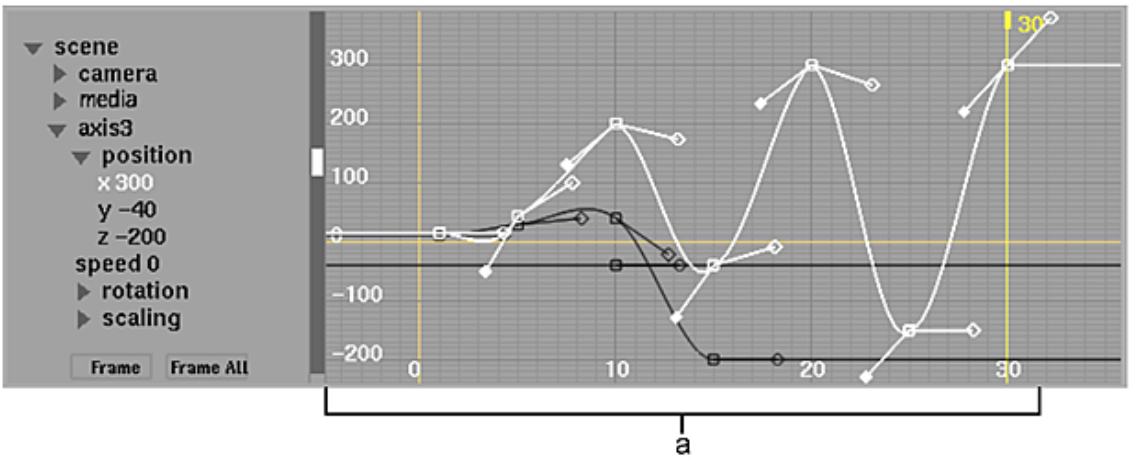
   A negative value rotates each tangent counter-clockwise. A positive value rotates each tangent clockwise.

   The tangent handle first breaks into two and then the handles are rotated.

   The following example shows the results of applying Tangent L with a rotation value of 36 to the keyframes of an entire curve. No keyframes were broken before Tangent L was applied.



(a) The Hermite curve before applying Tangent L. Note that none of the keyframes are broken.



(a) The same Hermite curve after applying Tangent L of 36. A break is applied to all keyframes before the tangents are rotated.

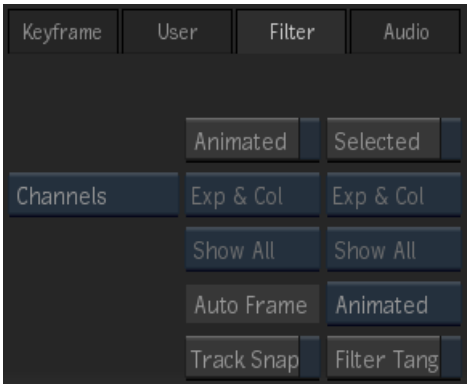
Setting Display Preferences in the Animation Channel Editor

Use the options in the Filter tab to control the display preferences in the Animation Channel Editor. The Animation Channel Editor display preferences can help you focus on the channels you are working on in the editor when many channels are displayed.

You can go in and out of the same tool in one session, and your display preferences are remembered. But if you restart, the your display preferences are not remembered. You must save and load your display preferences manually to access them when you restart the application.

To set display preferences:

1 In the Animation Channel Editor, select the Filter tab.



2 Select the type of channels you want to appear in the filtered view, animated or selected.

Select:	To:
Animated	Display channels that are animated.
Selected	Display channels that are selected.

3 Select how you want to expand or collapse channels that appear in the filtered view.

Select:	To:
Exp & Col	Expand animated or selected channels and collapse all others.
Expand Only	Expand animated and selected channels. All previously expanded channels remain selected.
No Expand	Not expand animated or selected channels.

4 Select how you want to display folders that contain channels in the filtered view.

Select:	To:
Hide All	Show only the animated or selected properties, for example, the X and Y values of an axis. All other properties in the same parent folder are hidden. All other folders are also hidden.
Hide Group	Show only the animated or selected channel, its parent folder, and all channels in the same group, where a group includes all folders in the same parent folder. All other parent folders and groups are hidden. **NOTE** If multiple channels are animated or selected, all corresponding parent folders and groups are shown.
Show All	Show all animated or selected channels.

5 From the Auto Frame box, select an option for framing your view of the Animation Channel Editor.

If you select All, Animated or Selected, your view of the Animation Channel Editor changes as you change your channel selection.

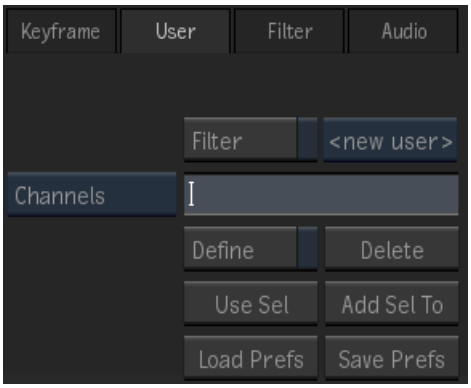
Select:	To frame:
All	All channels. This option is the same behavior as clicking the Frame All button in the Animation Channel Editor.
Animated	All animated channels.
Selected	All selected channels. This option is the same behavior as clicking the Frame button in the Animation Channel Editor.
None	No channels. This option enables the Frame and Frame All buttons in the Animation controls.

6 Enable Track Snap to force key frames to snap to whole frame numbers as you drag them.

7 Enable Filter Tangent to display tangents only on selected channels. This can help you cleanup the display.

To save display preferences:

1 Click the User tab.



2 Click Save Prefs.

To load display preferences:

1 Open the Animation Channel Editor from the tool where you saved your display preferences in.

2 On the User tab, click Load Prefs.

3 Click the Filter tab. Display preferences you saved are now activated on the Filter tab.

Using Selection Sets to Control the Channels and Folders You Want to Display

With a selection set, you can control the channels and folders you want to display in the Animation Channel Editor.

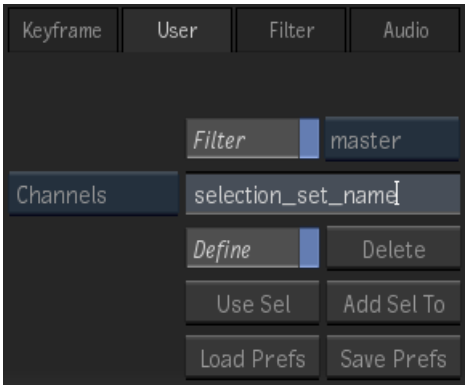
Selection sets can be particularly helpful if you access the Animation Channel Editor from a complicated Batch or Batch FX schematic, where all the selected nodes are displayed in the editor. With a selection set you can limit the number of channels or folders displayed to just those you are working on.

Selection sets are saved and available throughout a session. However, they are not saved between sessions. This means that you do not have access to the selection sets you create after you restart the application.

To create a selection set:

1. Select the User tab in the Animation Channel Editor.
2. Enable Filter.
3. Type a name in the Selection Set Name field and press `Enter`.

   The selection set is created and selected in the Selection Set box. Notice that the Define button is also enabled and the channels and folders in the Animation Channel Editor are colored red.



4. Select the channels or folders you want to in your selection set. If you are displaying the Batch or Batch FX schematic, you can select the nodes that you want in your selection set.

   Notice that channels and folders in Animation Channel Editor turn green as you make selections. This helps you identify the channels and folders in the selection set.

5. Do one of the following:
   - Click Use Sel to save the channels and folders you selected to a selection set.
   - Click Add Sel To, to add the selection to an existing selection set.
6. Disable Define.

To switch between selection sets:

1. Select the User tab.
2. Select a selection set from the Selection Set box.

To modify a selection set:

1. Select a selection set from the Selection Set box.
2. Enable Define.
3. Select the channels and folders you want to display in the selection set.
4. Click Add Sel To. The selection is added to the selection set and the selection set is saved.
5. Disable Define.

Generating Keyframes Based on Audio Analysis

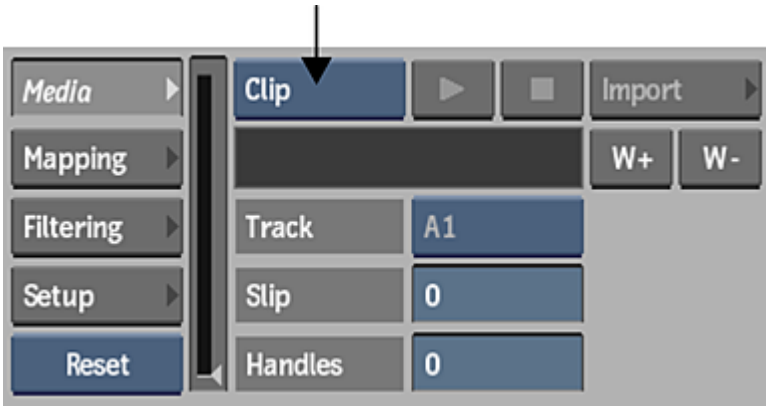
In the Channel Editor, you can use an audio clip to automatically generate keyframes on any video or audio channel based on the analysis of the audio.

To map keyframes based on audio analysis:

1. In the Channel Editor, select the Audio tab.

NOTE The Audio menu has a number of submenus with various settings. You may not need to use every setting to analyse your audio clip. See the sections after this procedure for an explanation of each setting.

2. From the Media Source box, select whether you want to analyse the current audio clip (Clip) or import an audio clip (File).

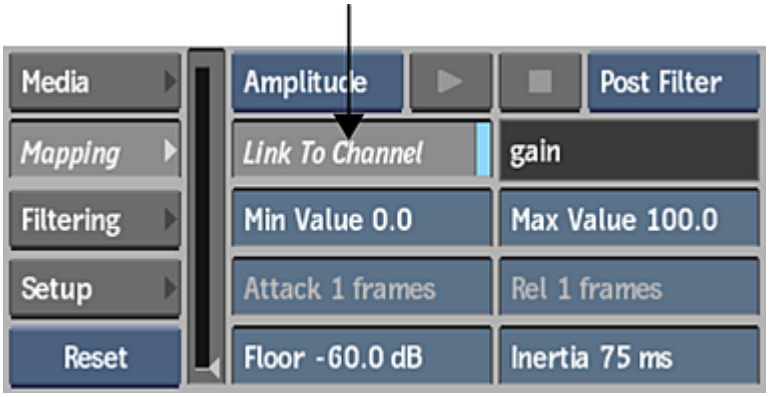


If there is more than one audio clip associated with your choice (for example, Front, Back, or Matte), you can choose which one to use in the Name box. If only one audio clip is available, the name of the clip appears in a locked field.

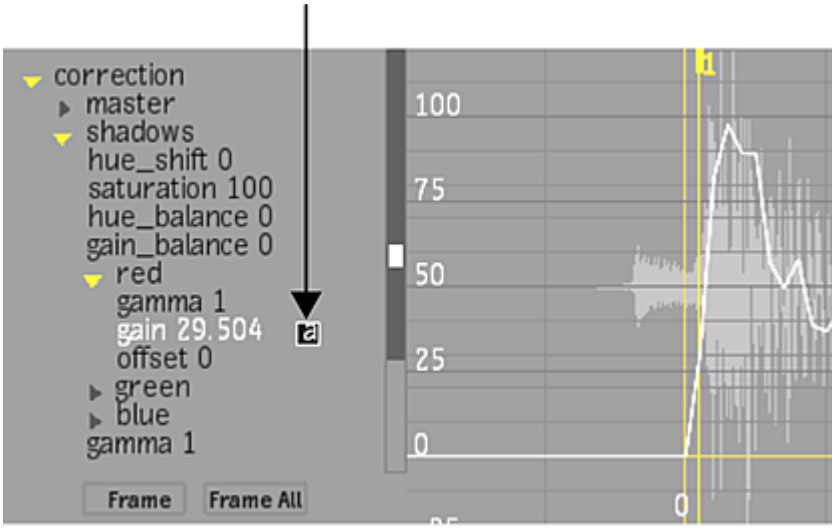
If you select File, click Import to open the Import Audio browser, and select an audio file to import.

3. In the Channel hierarchy, select the channel or channels you want to link the audio to.
4. Use the settings in the Media submenu to prepare the audio track you want to use. See Audio Keyframe Settings (page 1706).

5 From the Mapping submenu, enable Link To Channel.



The letter 'a' appears next to the channel in the Channel hierarchy, indicating that this channel is linked to the audio.



6 Use the settings in the Mapping submenu to determine how the audio track is mapped to the selected channel. Keyframes are adjusted automatically as changes are made to the settings.

7 Use the settings in the Filtering submenu to filter the exact frequencies you want to analyse. Keyframes are adjusted automatically as changes are made to the settings.

8 Disable Link To Channel or exit the module when you are satisfied with the analysis. The generated keyframes are automatically baked to the animation curve.

You can use different audio tracking settings on different channels. Disable Link To Channel and select a different channel in the hierarchy and restart the mapping process.

NOTE If you want to be able to tweak the settings, you can save the analysis settings in an audio tracker setup.

Audio Keyframe Settings

The animation controls Audio tab has a number of submenus with numerous settings to help you work with your audio clips.

Common Settings

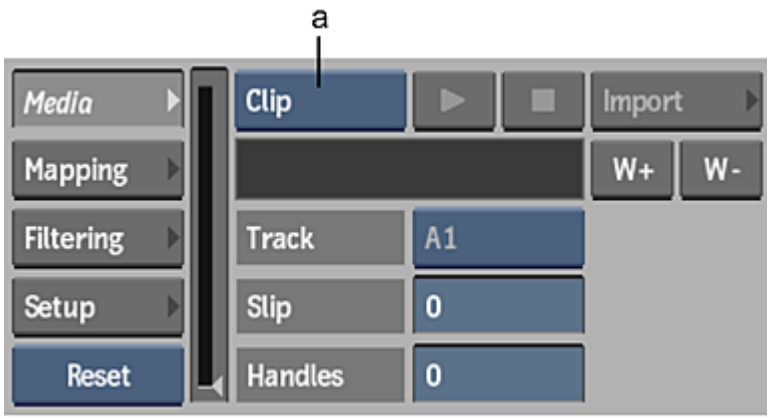
There are a few common settings found in the Audio submenus.

Play button Plays the audio media.

Stop button Stops the audio playback.

Reset box Available in the Media, Mapping, and Filtering submenus. In the Setup submenu, only Reset All is available.Select whether to reset the current Audio submenu, or all of the Audio settings.

Media Submenu



(a) Media Source box

Media Source box Select whether to analyse the current audio clip (audio attached to the front, back, or matte clips, for example), or the imported audio file.

Import button Opens the import audio browser to load a file.

Clip Name box If multiple audio clips are available, select which clip to use as source media. If only one audio clip is available, the name of the clip appears in a locked field.

W+ button Displays the selected media's waveform in the Channel Editor.

W- button Hides the waveform.

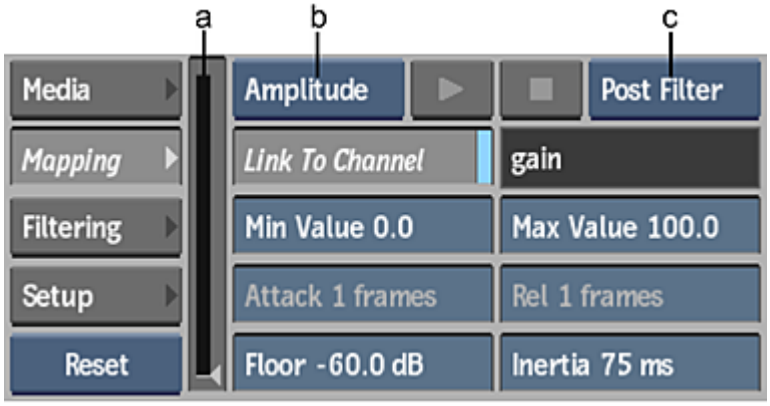
Channel box Select which audio channel to use as source media.

Slip field Displays the amount of frames to slip the media. Editable.

Handles field Displays the amount of handles (frames) to analyse before and after the current shot duration. Editable.

For example, with a shot of 100 frames, if you enter 30 in this field, the total number of frames becomes 160 (30 frames before and 30 frames after the original 100 frames).

Mapping Submenu



(a) Audio Level Indicator (b) Tracking Mode box (c) Listening Mode box

Tracking Mode box Select whether to use an Amplitude or Transient tracking mode. In Amplitude tracking mode, a keyframe is generated at each frame, following the envelope of the audio signal. In Transient tracking mode, a keyframe is generated for each audio transient (sudden rise in amplitude), provided that the signal rises above the specified threshold.

Listening Mode box Choose between Pre Filter (audio as it was imported) and Post Filter (audio with any changes made) monitoring when playing back audio media. Does not affect the analysis.

Link to Channel button Enable to map the tracked audio to the selected animation channel. An 'a' appears in the channel list next to the name of the linked channel. Multiple channels can be selected simultaneously.

Channel Name field This locked field displays the name of the selected channel being linked to the tracked audio. Multiple Channels is displayed if more than one channel is selected.

Min Value field Displays the minimum value at which keyframes can be set in the linked channel. Editable.

Max Value field Displays the maximum values at which keyframes can be set in the linked channel. Editable.

NOTE The minimum value can be higher than the maximum value to get an inverse mapping of the tracking mode in the animation channel.

Attack field Displays the number of frames needed for the channel to reach the Max Value before a transient. Available only in Transient tracking mode. Editable.

Release field Displays the number of frames needed for the channel to reach the Min Value after a transient. Available only in Transient tracking mode. Editable.

Threshold field (Not shown). Displays the value at which a keyframe is generated each time the signal rises above it. Available only in Transient tracking mode. Editable.

TIP Start with a high threshold, and lower it until all required peaks are detected.

Floor field Displays the minimum level of the audio media to be considered for the analysis (typically used to remove analysis noise between audio transients). Available only in Amplitude tracking mode. Editable.

Inertia field Displays the rate at which the signal decreases after a transient. Editable.

In Transient tracking mode, the Release field can be increased to avoid the generation of several consecutive keyframes when the transient is not clearly defined in the input signal. Conversely, if the Release field is set to 0 ms, each transient rising above the threshold generates a keyframe, even if each occurs within a few milliseconds of another.

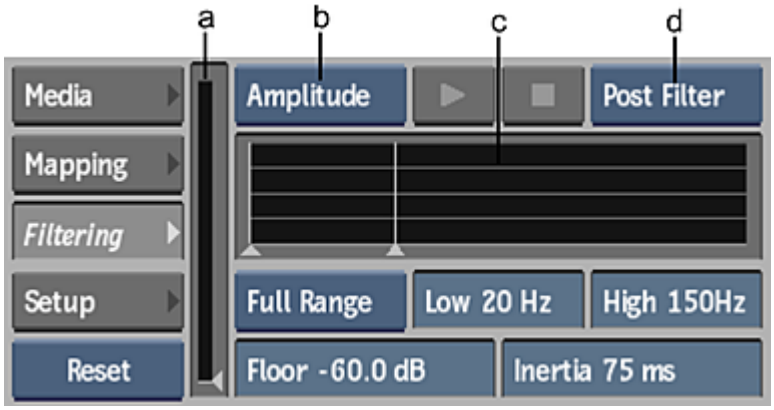
Audio Level Indicator A visual representation of the audio level and the current Threshold or Floor value.

Audio Level Indicator A visual representation of the audio level and the current Threshold or Floor value.

Reset box Select Reset to reset the Mapping submenu settings only. Select Reset All to reset all of the Audio tab settings.

NOTE Some of the settings in the Mapping submenu are repeated in the Filtering submenu so that you do not to have to switch tabs to change the settings. The same settings are reflected in both submenus.

Filtering Submenu



(a) Audio Level Indicator (b) Tracking Mode box (c) Frequency Graph (d) Listening Mode box

Frequency Graph A visual frequency representation of the audio signal being tracked. The audio display is always Pre Filter.

Frequency Range Preset box Select a frequency range preset to determine the cutoff frequencies for Low and High filters.

Low field Displays the lowest frequency of the input signal used in the analysis (also represented by a triangle under the frequency graph). Use the slider or field to adjust. Editable.

High field Displays the highest frequency of the input signal used in the analysis (also represented by a triangle under the frequency graph). Use the slider or field to adjust. Editable.

NOTE Some of the settings in the Filtering submenu are repeated in the Mapping submenu so that you do not to have to switch tabs to change the settings. The same settings are reflected in both submenus.

Setup Submenu

Load button Click to load a saved audio mapping setup.

Save button Click to save an audio mapping setup.

Setup Name field This locked field displays the name of the loaded setup

Animation Menu Settings

The Animation menu has some common settings, and a number of settings specific to certain operations, organized in submenus. For specific Audio submenu settings, see Audio Keyframe Settings (page 1706).

Common Settings

Regen button Enable to automatically update the scene when an animation change is made.

Channel View box Select a view of Channels, Tracks, or Info to control and coordinate your animation.

Auto Select button Enable to select the channel automatically when you modify a channel value in the field for an effect or tool.

Set Key button Sets the current values for the selected channels in the current frame (when Auto Key is disabled).

Delete Key button Deletes the selected keyframes or curve.

Keep button Deletes all keyframes except the current keyframe.

Cut button Cuts the selected keyframes or curve.

Copy button Copies the selected keyframes or curve.

Paste button Pastes the selected keyframes or curve.

Link button Creates an expression by linking one channel to another.

Expressions button Applies an expression to one or more channels.

Animation Reset box Select to reset the selected channel or all channels.

Keyframe Submenu

Curve Functions box Select an operation to apply to your animation.

Curve Value field Displays the value for the selection in the Curve Functions box. Editable.

Insert Key button Inserts a keyframe. Its behaviour changes depending on where the keyframe is inserted.

Duration field Displays the number of frames between keyframes. Editable.

Absolute/Relative box Select whether the value in the Duration field is absolute or relative to the present keyframe value.

Tangents box Select the behaviour of the keyframe tangents to help refine the shape of an animation curve between keyframes.

Tangent Mode	Tangent Behaviour
Auto	Tangents are fixed in an horizontal position to prevent the curve from over- or under-shooting keyframes. Moving a keyframe higher or lower than its neighboring keyframes alters their tangents so the curve remains as uniform as possible. The handles of tangents set to Auto are empty circles. Manually moving a keyframe's tangent voids this setting for that tangent. It will now perform as a Fixed tangent.

Tangent Mode	Tangent Behaviour
Fixed	Tangents are fixed in their current position so that moving a keyframe does not alter its tangents nor those of its neighboring keyframes. The handles of tangents set manually are filled circles.
Smooth	Legacy setting to ensure compatibility with previous versions of a setup. Tangents are not locked like when using the Auto setting: they are affected by the movement of adjacent keyframes. There are no other benefit from using this setting.

Interpolation box Select an interpolation type to define the shape of an animation curve between keyframes.

Extrapolation box Select an extrapolation type to define the shape of an animation curve before the first keyframe and after the last keyframe of the curve.

User Submenu

Filter button Enable to create and name a selection set (the channels, folders, and folder elements that you want to display as part of a set).

Selection Set box Select a user-defined selection set. Select <new user> to create a selection set.

Name field Enter a name for the selection set. Editable.

Define button Enable to display channels in red and green to indicate the channels contained in the current user-defined selection set.

Delete button Removes the currently selected selection set from the Selection Set box.

Use Selection button Displays the current selection set in the Channel Editor and at the same time saves the current selection set.

Add Selection To button Adds the current selection to the active selection set.

Load Filter Preferences button Loads the filter preferences last saved for the module.

Save Filter Preferences button Saves the current filter settings as preferences for the module.

Filter Submenu

Animated button Enable to display channels that have animation keyframes. Use in conjunction with the Animated Expansion and Animated Show/Hide boxes to further control the content displayed.

Selected button Enable to display only channels that are selected in the Channel Editor. Use in conjunction with the Selected Expansion and Selected Show/Hide boxes to further control the content displayed.

Animated Expansion box Select an expansion option for animated folders.

Selected Expansion box Select an expansion option for selected folders.

Animated Show/Hide box Select to display or hide animated folders.

Selected Show/Hide box Select to display or hide selected folders.

Auto Frame box Select an option to frame channels automatically. If None is selected, the Frame and Frame All buttons are enabled.

Track Snap button Enable to force keyframes to snap to frames when dragging tracks by their left or right handles.

Filter Tangents button Enable to display only the tangent handles of selected channels.

Advanced Animation: About Expressions

Use expressions to apply animation dynamically to one or more channels. For example, use expressions to easily simulate real-world forces such as gravity, momentum, and centrifugal force. Expressions save time since you can animate one channel, and then refer other channels to the first channel so that the other channels automatically behave in relation to the first one. You can use arithmetic operators, such as addition and multiplication, as well as predefined functions to precisely control the behaviour of an animation.

Expressions are available through all modules that use the Channel Editor.

Working with Expressions in the Channel Editor

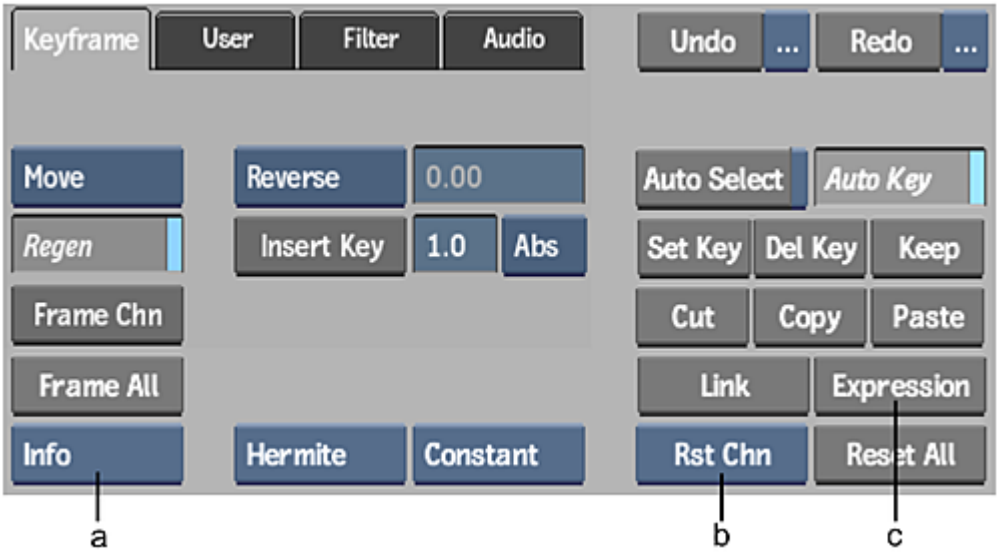
You can apply, modify and view expressions using the Channel Editor. You can use the copy and link functionality to quickly apply basic expressions to multiple channels.

WARNING Expressions override interpolation modes and previously set keyframes for a selected channel.

To apply an expression to a channel:

1 In any module menu, click Animation.

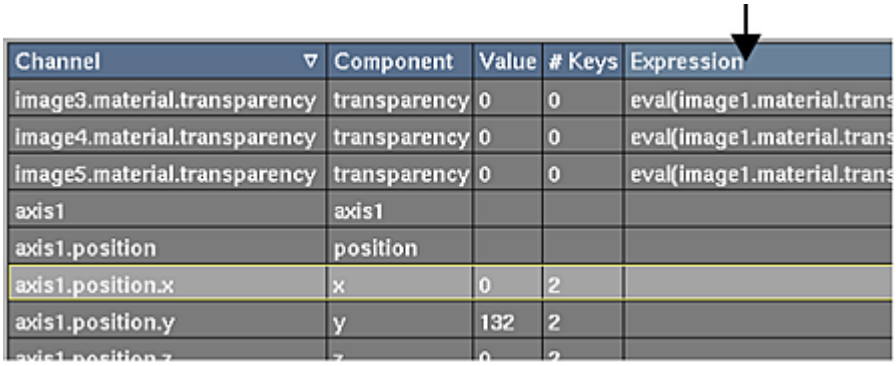
The Animation controls appear.



(a) Channel View box (b) Reset Channel button (c) Expressions button

2 From the Channel View box, select Info.

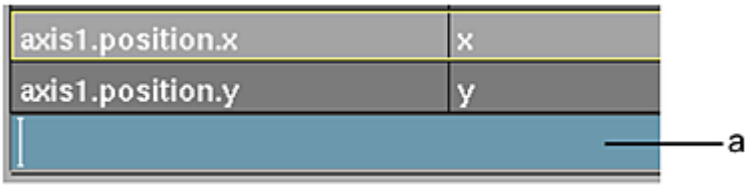
Channel information appears in a tabular grid for easier viewing of expressions and channel details. From Info view, you can click the Expressions column heading to sort by expressions and view them more easily.



3 Select the channel to which you want to apply an expression.

4 In the Animation controls, click Expression.

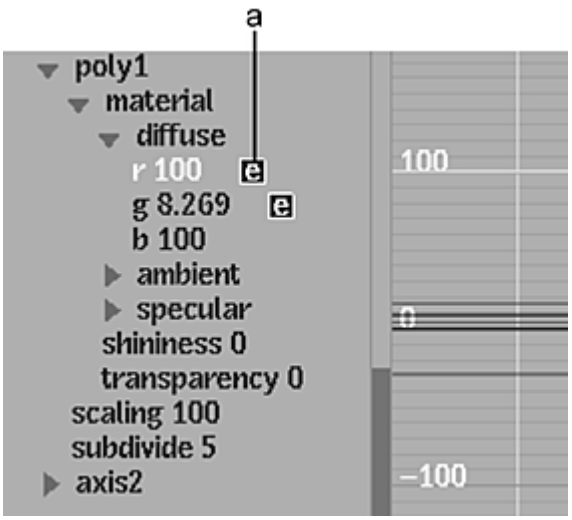
The Expression field appears below the Channel Editor.



(a) Expression field

5 Enter an expression according to the guidelines described in Operator Reference (page 1719) and Function Reference (page 1721) and press `Enter`.

The letter 'e' appears next to the channel in the channel hierarchy, indicating that this channel contains an expression.



(a) Expression indicator

NOTE Entering an invalid expression will result in the message “Error: Expression: parse error” and the entered expression will not be applied to the channel.

To modify an expression:

1 Click Animation to display the Animation controls.

2 In the channel hierarchy or Info view, select the expression that you want to edit.

3 In the Animation controls, click Expression.
The Expression field appears below the Channel Editor.

4 Modify the expression in the field and press Enter.
The modified expression appears in the table.

TIP When you are in the Expression field, you can press Up one or more times to retrieve its previous contents. This can be useful if you want to correct an invalid expression that you recently entered.

To remove an expression:

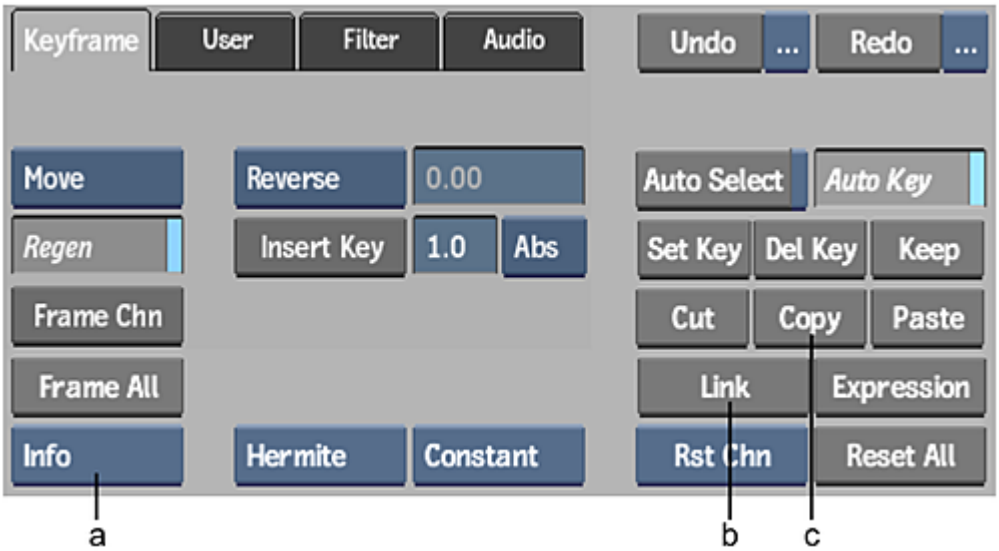
1 Click Animation to display the Animation controls.
2 In the channel hierarchy or Info view, select the channel with the expression that you want to remove.
3 Click Rst Chn to remove the expression.
The channel is reset.

Linking Channels

You can create an expression by linking the behaviour of one channel to another using Copy and Link. You can link different types of channels together. For example, make the scaling of a layer affect the rotation of another layer, or blur a layer by linking it to the position of an axis. Any change that occurs in the position of the axis is reflected dynamically in the layer blur.

To create an expression by linking one channel to another:

1 Click Animation to display the Animation controls.
2 From the Channel View box, select Info.



(a) Channel View box (b) Link button (c) Copy button

TIP Enable the Hierarchy View button

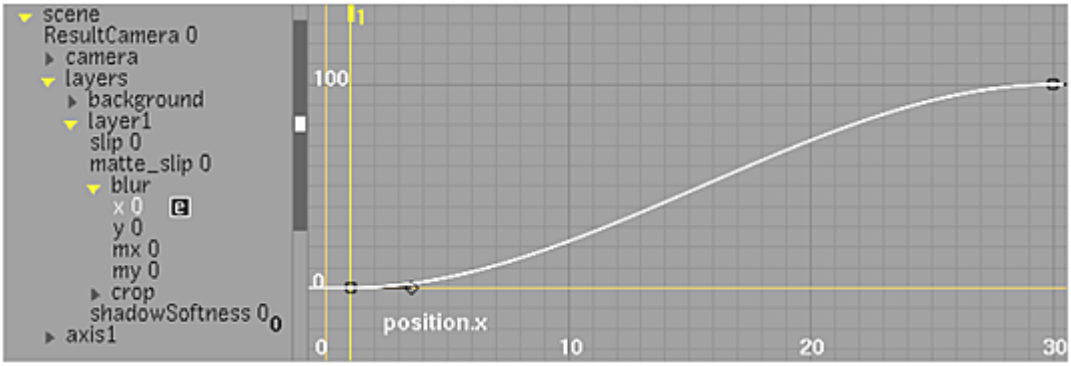
to better view the channel hierarchy.

3 Select a source channel from which to link and click Copy.

For example, copy the x position of axis1 and link it to the x blur value of a layer.

4 Select the destination channel and click Link.

The x blur value of layer1 is linked to the x position value of axis41, so as the position increases, so does the blur. This is shown in the following illustration.



Copying Nodes with Expressions

When working in Action, Keyer or Garbage Masks, if you copy a branch in Schematic view, or append to a setup by loading a second setup, any nodes with the same name as existing ones are renamed with unique names. Likewise, if expressions are associated with those nodes, the expressions are replicated and the associated channels are renamed to correspond with the new node names. This preserves the integrity of the expression within the copied branch or appended setup.

If you copy a portion of a branch that has expressions associated with it, the expressions related to the copied portion of the branch will be associated with the new copy. Expressions related to uncopied portions of the branch will maintain their association with the original branch.

Cascading Expressions

You can copy a single expression to multiple channels that build on the result in the previous channel using the *SelectionOrder* and *CascadeChannel* keywords. These keywords increment the channel number or name as follows:

- Each instance of SelectionOrder increments the channel number.
- Each instance of CascadeChannel replaces the occurrence of the channel name with the name of the preceding channel, relative to the order in which the channels are selected.

In general, experiment with the SelectionOrder and CascadeChannel keywords to determine which results are optimal for your animation.

To apply cascading expressions using SelectionOrder:

1 Click Animation to display the Animation controls.

2 From the Channel View box, select Info.

3 Select multiple channels to which you want to apply the cascaded expression, in the desired order.

As you select each channel, the selection order is shown next to the channel name in Info view.

TIP To select a range of channels, click the first channel that you want to include in the range. Then click the last channel to include in the range while holding `Shift`. To add to a range of files, hold `Ctrl` or the stylus button while selecting the channels from the Channel Editor.

4 Enter an expression that includes the keyword SelectionOrder.

Every instance of the keyword SelectionOrder will be substituted in your expression by a different number for each channel, which corresponds to the order in which you selected them.

To animate four channels using SelectionOrder:

1 In Action, add four axes named axis1, axis2, axis3 and axis4.
2 Animate axis1.position by creating a few keyframes.
3 Select the channels axis2.position, axis3.position and axis4.position in this order.
4 Enter the following expression: eval(axis1.position, frame - 5 * SelectionOrder).

The following expressions are applied:

- axis1.position: no expression
- axis2.position: eval(axis1.position, frame - 5 * 1)
- axis3.position: eval(axis1.position, frame - 5 * 2)
- axis4.position: eval(axis1.position, frame - 5 * 3)

To apply cascading expressions using CascadeChannel:

1 Click Animation to display the Animation controls.
2 From the Channel View box, select Info.
3 Apply an expression to a base channel to which the other channels will relate.
4 Select multiple channels, starting with the base channel, to which you want to apply the cascaded expression, in the desired order.
5 Enter an expression that includes the keyword CascadeChannel.

The base channel (the first channel selected) will remain unmodified. In the expressions for the other selected channels, every instance of the keyword CascadeChannel will be substituted by the name of the channel that preceded it, in the order in which you selected them.

To animate four axes using CascadeChannel:

1 In Action, add four axes named axis1, axis2, axis3 and axis4.
2 Animate axis1.position by creating a few keyframes.
3 Select, in this order, the channels axis1.position, axis2.position, axis3.position and axis4.position.
4 Enter the following expression: eval(CascadeChannel, frame - 5).

The following are applied:

- axis1.position: no expression
- axis2.position: eval(axis1.position, frame - 5)
- axis3.position: eval(axis2.position, frame - 5)
- axis4.position: eval(axis3.position, frame - 5)

NOTE This results in the same animation as the SelectionOrder example given above.

Expression Content

An expression is composed of numeric values, constants, channel references, or combinations of these used with arithmetic operators. Expressions are calculated from left to right, according to a specific order defined in Operator Precedence (page 1720).

For example, the following expression contains channel references and multiplies the transparency value of image1 by 2 to affect the transparency of image2 across all keyframes.

Channel	Expression
image2.material.transparency	image1.material.transparency* 2

Vectors

Certain channels such as Position, Rotation, Scale, and Shear are vector based since they represent 3D space and include X-, Y-, and Z-axes. A vector contains three elements that represent the individual components of the channel. Vectors are written using the convention (x, y, z) where x, y, and z represent separate scalar values.

Expressions can either affect the components of a vector individually or collectively. For example, the following expression sets the individual rotation for the x, y, and z axes to 30°, 45° and 90°, respectively.

Channel	Expression
axis1.rotation	(30, 45, 90)

The following expression sets the rotation for axis2 twice that of axis1, affecting the vector collectively.

Channel	Expression
axis2.rotation	axis1.rotation* 2

Functions

Flame includes many predefined functions that can be used to perform calculations in an expression. You pass function-specific values, called *arguments*, and they return another value back to the expression that called it. A function call in an expression begins with the function name, followed by an opening parenthesis, the arguments for the function separated by commas, and finally a closing parenthesis.

NOTE Function names are case-sensitive.

Arguments for functions can be either scalar values or vectors. When you use a function, make sure that you pass it the correct type of parameter. See Function Reference (page 1721) for information on the arguments and return values for each function.

You can nest function calls by using the return value of a function as one of the arguments of another function. When a nested function is used as an argument, it must return a value that conforms to the type and range that the argument requires.

You can define your own functions and use them in your expressions just as you would with any of the predefined functions. See Defining Your Own Functions (page 1741).

Examples

The following expression uses the noise function to create a random positioning effect for axis1.

Channel	Expression
axis1.position.x	noise(frame)*5

The following expression uses the eval function to make the position of axis2 the same as that of axis1, but delayed by 10 frames.

Channel	Expression
axis2.position	eval(axis1.position, frame - 10)

The following expression uses the eval function to make the animation of axis3 the same as that of axis1, but at half the speed.

Channel	Expression
axis3	eval(axis1, frame / 2)

Keywords and Constants

Use the keyword *frame* in an expression to get the value of the current frame number in your animation. This allows you to create an animation by specifying how a value changes over a sequence of frames.

Use the constant *PI* instead of the literal numeric value of π (3.1416...).

NOTE Keywords and constant names are case-sensitive.

Select:	To display:
PI	3.1416...
e	Euler's number (2.7182818284...)
phi	Golden Ratio (1.6180339887...)
c	Speed of light in vacuum (299792458 m/sec)
freefall	Standard free fall acceleration near the earth's surface (9.80665 m/sec^2)
echarge	Elementary charge (1.602176487e-19 Coulomb)

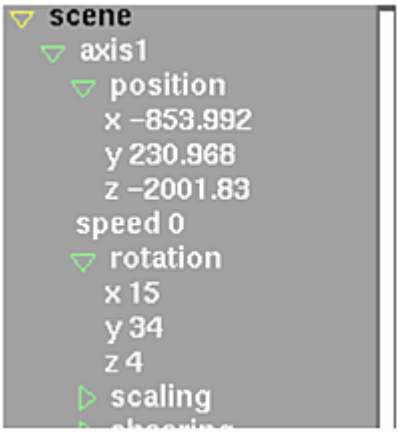
Simplified Expressions

A reference to another channel does not have to be fully qualified when it is at the same hierarchical level as the channel to which the expression is being applied. For example, the following simplified expressions

are valid when applied to the specified channel. The fully qualified versions of the expressions are also provided.

Channel	Simplified Expression	Fully Qualified Expression
axis1.position.y	x + 50	axis1.position.x + 50
axis1.rotation	position / 2	axis1.position / 2
image1.material.diffuse	specular * 3	image1.material.specular * 3

References to other channels also work when referring to the siblings of any parent. For example, consider the following structure.



You can apply the expression position.x to axis.rotation.x because position is a sibling of rotation, and rotation is a parent of rotation.x.

Operator Reference

Operators specify a mathematical or logical calculation to be performed between various elements of an expression.

Arithmetic Operators

Use the following arithmetic operators to perform basic mathematical operations.

Operator	Description
+	Addition
-	Subtraction
*	Multiplication
/	Division
%	Modulo (this is a binary operator)

Operator	Description
(*x, y, z*)	Vector where *x, y, and z* are scalar values

Comparison Operators

Use the following operators to compare two values with each other. When two values are compared using these operators, the result is 1 if the comparison is true and 0 if the comparison is false.

Operator	Description
==	Equal to
!=	Not equal to
<	Less than
<=	Less than or equal to
>	Greater than
>=	Greater than or equal to
&&	Boolean AND
||	Boolean OR
!	Boolean NOT

Operator Precedence

When you combine several operators in a single expression, the operations are performed in the following order.

Order	Operator	Description
1	!	Boolean NOT
2	-	Negation (as in -1)
3	*, / and %	Multiplication, division, and modulus
4	+ and -	Addition and subtraction
5	< , <=, >, and >=	Comparison
6	== and !=	Equivalence
7	&&	Boolean AND

Order	Operator	Description
8	||	Boolean OR

When operators with the same precedence are encountered, operators are evaluated from left to right. However, when part of a formula is enclosed in parentheses, it is evaluated first.

Function Reference

You can easily create very complex expressions by using the available functions. This function reference provides detailed information about each function. Function arguments in square brackets are optional. However, if you give a value to an optional argument, you must also give a value to every optional argument before that one.

Animation Functions

Use the following functions to animate channels in a general way.

NOTE Arguments shown in square brackets are optional. For example, in the syntax of the align function, the options [AxisToAlign] and [BankingAngle] are optional.

align

Returns a rotation vector such that a designated axis of an object is aligned with the direction of the object's movement. You can also bank the rotation around the axis. The result should usually be assigned to a rotation channel.

Syntax:	align(PosToFollow, [AxisToAlign], [BankingAngle])
Arguments:	■ *PosToFollow* is the vector representing the channel to align on, typically an animated position. ■ *AxisToAlign* is the vector representing the axis with which to align, by default the X-axis (1,0,0). ■ *BankingAngle* is the angle in radians by which to rotate the result about the AxisToAlign (performs banking). If the BankingAngle argument is specified, the AxisToAlign argument must also be specified.
Example:	align(axis.position, (0,0,1), frame * PI / 8) returns a rotation vector that points the object's Z-axis in the direction of its motion while the rest of the axis rotates along this axis.

lookat

Returns a rotation vector based on an object's position that points it towards a second moving object. The result should usually be assigned to a rotation channel.

Syntax:	lookat(TargetPos, ObserverPos, [AlignVector], [UpVector])

Arguments:	■ TargetPos is the vector of an object's position channel that you want the rotation vector to point towards. ■ ObserverPos is the vector of the position channel from which you are looking. ■ AlignVector is the vector of the direction that you want to have looking at the target, by default the Z-axis (0,0,1). ■ UpVector is the vector of the direction that you want to be pointing upwards, by default the Y-axis (0,1,0). The UpVector should be set to a different vector than the AlignVector. If the UpVector argument is specified, the AlignVector argument must also be specified.
Examples:	■ lookat(followed_axis.position, follows_axis.position) returns the rotation vector required so that follows_axis points its Z-axis towards followed_axis. ■ lookat(followed_axis.position, follows_axis.position, (0,1,0),(1,0,0)) returns the rotation vector required so that follows_axis points its Y-axis towards followed_axis, with its X-axis pointing upwards.

eval

Returns the value of a given expression at another point in time.

Syntax:	eval(Expression, FrameNumber)
Arguments:	■ Expression is the expression to be evaluated. This can be any valid channel value. ■ FrameNumber is the frame to simulate when evaluating the given expression.
Examples:	■ eval(axis1.position.x, 5) returns the value of axis1.position.xat frame 5. ■ eval(axis1.position, frame - 10) returns the axis1.positionvector at 10 frames behind the current frame. ■ eval(axis1, frame / 2) returns the entire axis1 channel at half the normal speed.

if

Returns one of two values based on the result of a conditional test. You can nest multiple functions inside each other to handle multiple results.

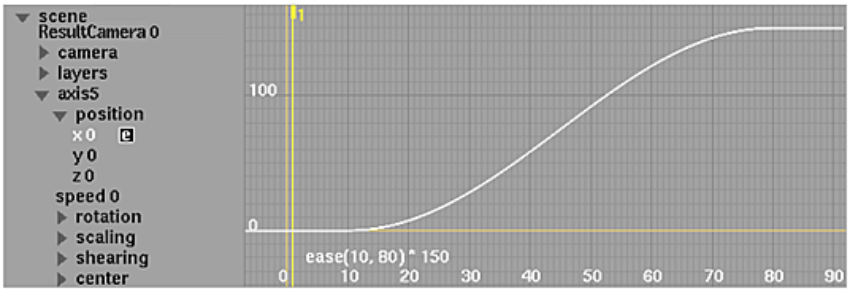
Syntax:	if(Condition, TrueValue, FalseValue)
Arguments:	■ Condition is any channel or expression. When comparison operators are used, a true expression evaluates to 1 and a false expression evaluates to 0. See Comparison Operators (page 1720). ■ TrueValue is the value to be returned for any non-zero result. ■ FalseValue is the value to be returned if Condition evaluates to 0.
Examples:	■ if(axis.position.y >= 100, 5, -5) returns 5 when axis.position.yis greater than or equal to 100, and -5 otherwise.

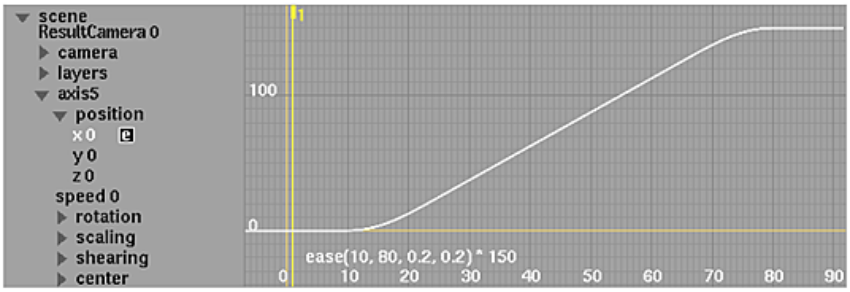
- if(frame < 10 || frame >20, 100, 200) returns 100 when the current frame is less than 10 or above 20, and 200 otherwise.
- if(axis.position.x, 6, 7) returns 6 when axis.position.x is something other than 0, and 7 when it is 0.
- if(axis.position.x == 100 && axis.position.y != 200,8, 9) returns 8 when axis.position.x is 100 and axis.position.y is not 200, and 9 otherwise.
- The next four examples all equivalently return 5 when neither axis.position.x nor axis.position.y are greater than or equal to 0, and -5 otherwise.

```
if(!(axis.position.x =>0 || axis.position.y =>0), 5, -5)
if(!(axis.position.x =>0) && !(axis.position.y =>0), 5, -5)
if(axis.position.x < 0 && axis.position.y < 0, 5, -5)
if(axis.position.x < 0, if(axis.position.y < 0, 5, -5), -5)
```

ease

Returns a number from 0 to 1 representing a smooth S-curve transition between a given range of frames. All frames before the start frame are assigned 0 and all frames after the end frame are assigned 1. A start weight and end weight specify how the S-curve is formed.

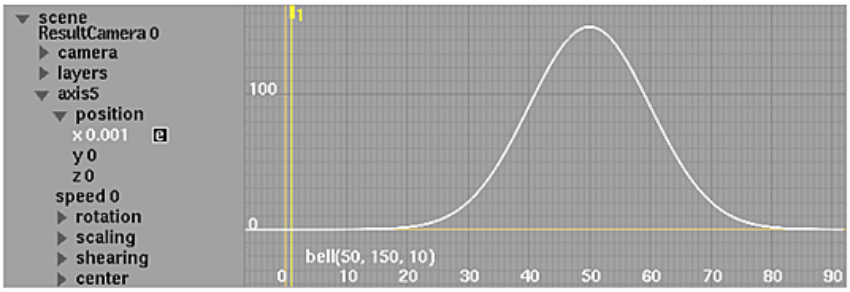
Syntax:	ease(StartFrame, EndFrame, [StartWeight], [EndWeight])
Arguments:	■ StartFrame and EndFrame are the frames at which the transition starts and ends respectively. ■ StartWeight and EndWeight are numbers that specify how curvature is distributed at the start and end of the curve respectively. To get an S-curve, their sum should be less than 1, otherwise a square curve will result. In addition, if either of these two arguments are specified, the other must also be specified.
Examples:	■ ease(10, 50) * 100 returns a standard S-curve from 0 to 100between frames 10 and 50. ■ 80 - ease(1, 40) * 20 returns a standard S-curve from 80 to60 between frames 1 and 40. ■ ease(1, 30, 0.2, 0.2) * 50 + 10 returns a tight S-curve from10 to 60 between frames 1 and 30. ■ ease(10, 80) * 150 yields the following curve:  ■ ease(10, 80, 0.2, 0.2) * 150 yields the following curve:



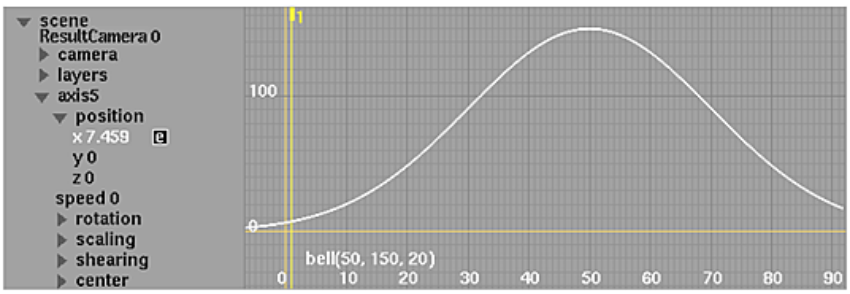
bell

Returns a set of values representing a bell (normal distribution) curve over time. You can specify the centre frame number and height and width characteristics of the curve.

Syntax:	bell(CentreFrame, Height, Width)
Arguments:	■ CentreFrame is the frame number at which the bell curve will reach its maximum height. ■ Height is the maximum height of the bell curve. ■ Width specifies the horizontal distribution of the curve.
Examples:	■ bell(10, 100, 3) returns a bell curve centred around frame10 with a maximum height of 100 and a width distribution of 3. ■ bell(50, 150, 10) yields the following curve:



■ bell(50, 150, 20) yields the following curve:



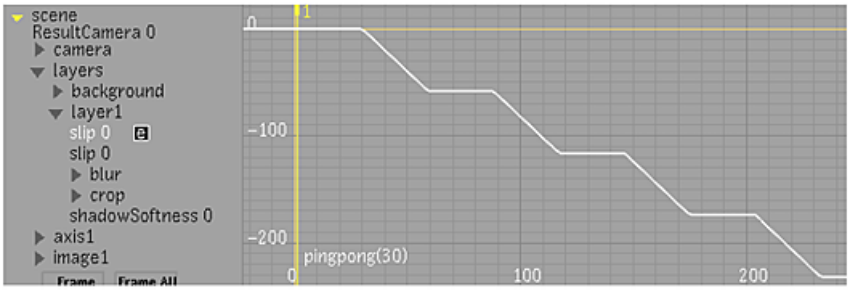
Slip Functions

Use the following functions to slip the starting frame of a clip backwards or forwards using the slip channel. In effect, the displayed frame in a layer's clip will be the animation's current frame number plus the slip value. For example, a slip value of -15 holds the clip at the first frame and repeats it 15 times before the clip begins. In this way, you can create a freeze frame effect with one layer while animating another layer. A slip value of 10 begins the clip at frame 11. See Modifying Surfaces (page 624).

NOTE Slip functions are actually implemented as user-defined functions in the functions file. See Defining Your Own Functions (page 1741).

pingpong

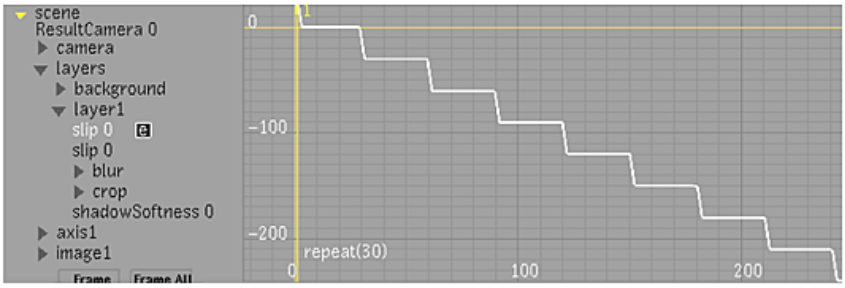
Returns a slip value that makes a clip continuously cycle forwards and backwards. The result should usually be assigned to the slip channel of a layer.

Syntax:	pingpong(length)
Arguments:	■ length is the number of frames to display before starting to play backwards. This is usually the length of the original clip.
Examples:	■ pingpong(10) returns a slip value that makes a clip continuously play in a forward and backward cycle from frame 1 to frame 10. ■ pingpong(30) yields the following curve: 

repeat

Returns a slip value that makes a clip continuously repeat forwards. The result should usually be assigned to the slip channel of a layer.

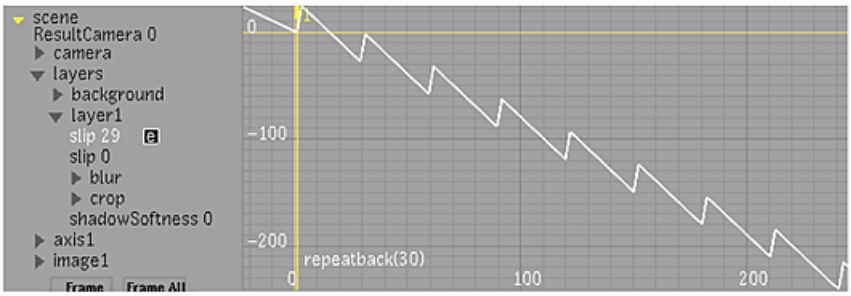
Syntax:	repeat(length)
Arguments:	■ length is the number of frames to display before repeating from the beginning. This is usually the length of the original clip.
Examples:	■ repeat(10) returns a slip value that makes a clip repeat forwards from frame 1 to frame 10. ■ repeat(30) yields the following curve:



repeatback

Returns a slip value that makes a clip continuously repeat backwards. The result should usually be assigned to the slip channel of a layer.

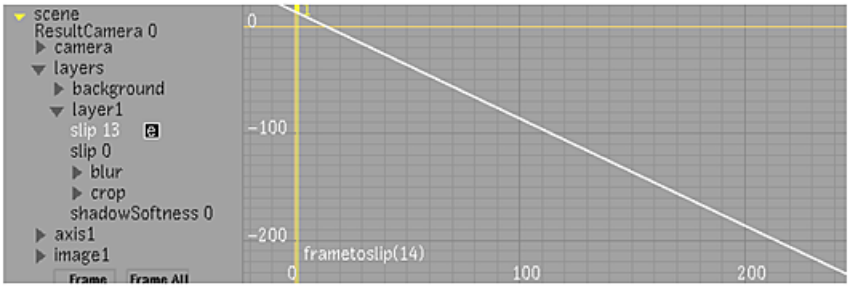
Syntax:	repeatback(length)
Arguments:	■ length is the number of frames from the beginning to display backwards before repeating. This is usually the length of the original clip.
Examples:	■ repeatback(10) returns a slip value that makes a clip repeat backwards from frame 10 to frame 1. ■ repeatback(30) yields the following curve:



frametoslip

Returns a slip value that makes a clip display at a given timebar frame. If the given frame number is constant, the resulting clips will freeze at that frame. The result should usually be assigned to the slip channel of a layer.

Syntax:	frametoslip(TimebarFrame)
Arguments:	■ TimebarFrame is the timebar frame at which to display the clip.
Examples:	■ frametoslip(10) returns a slip value that makes a clip freeze at frame 10. ■ frametoslip(frame / 2) returns a slip value that makes a clisplay at half the normal speed. ■ frametoslip(14) yields the following curve:



Simple Mathematical Functions

The following functions are useful for performing various simple mathematical calculations.

abs

Returns the absolute value of a given number. The absolute value is the positive value of any number.

Syntax:	abs(Number)
Arguments:	■ Number is the number of which you want the absolute value.
Examples:	■ abs(3) returns 3. ■ abs(-3) returns 3.

sign

Returns the sign of a given number. The sign function returns 1 if the number is above or equal to zero, and returns -1 if less than zero.

Syntax:	sign(Number)
Arguments:	■ Number is the number of which you want the sign.
Examples:	■ sign(5) returns 1. ■ sign(0) returns 1. ■ sign(-0.001) returns -1.

pow

Returns a number raised to the power of an exponent.

Syntax:	pow(Number,Power)

Arguments:	■ Number is the base number to be raised. ■ Power is the exponent to which the base number is raised.
Examples:	■ pow(3, 2) returns 9. ■ pow(4, 3) returns 64. ■ pow(5, 0) returns 1. ■ pow(-2.5, 4) returns 39.0625.

sqrt

Returns the square root of a given number.

Syntax:	sqrt(Number)
Arguments:	■ Number is the non-negative number of which you want the square root.
Examples:	■ sqrt(25) returns 5. ■ sqrt(abs(-25)) returns 5. ■ sqrt(0) returns 0.

max

Returns the larger of two given numbers.

Syntax:	max(Number1, Number2)
Arguments:	■ Number1 and Number2 are numbers of which you want to find the maximum value.
Examples:	■ max(5.9, 8.1) returns 8.1. ■ max(-14, -32) returns -14. ■ max(axis1.position.x, axis2.position.x) returns the larger of axis1.position.x or axis2.position.x.

min

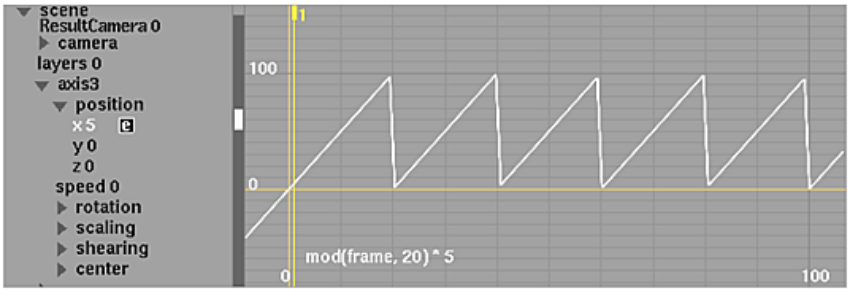
Returns the smaller of two given numbers.

Syntax:	min(Number1, Number2)
Arguments:	■ Number1 and Number2 are numbers of which you want to find the minimum value.
Examples:	■ min(5.9, 8.1) returns 5.9.

	■ min(-14, -32) returns -32. ■ min(axis1.position.x, axis2.position.x) returns the smaller of axis1.position.x or axis2.position.x.

mod

Returns the integer remainder from dividing one number by another. This function is useful for repeating an animation every given number of frames.

Syntax:	mod(Number, Divisor)
Arguments:	■ Number is the number to divide. ■ Divisor is the number by which you want to divide Number.
Examples:	■ mod(8, 3) returns 2 because 8 divided by 3 is 2 with 2 as the remainder. ■ mod(-8, 3) returns -2. ■ mod(8, -3) returns 2. ■ mod(-8, -3) returns -2. ■ mod(7.5, 2.25) returns 0.75. ■ mod(frame, 20) * 5 yields the following curve: 

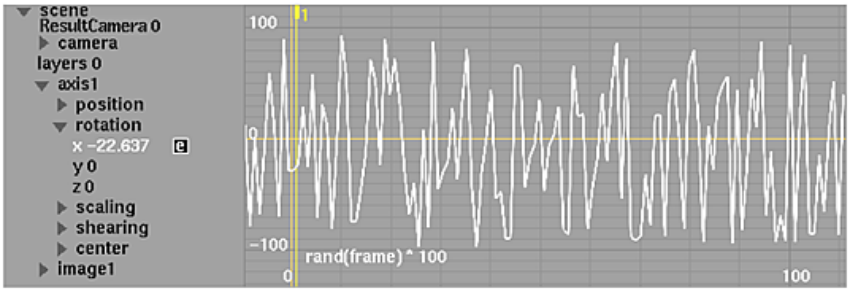
Random Number Functions

Use the following functions to create curves based on random values.

rand

Returns a random value between -1 and 1, based on the value used as an argument, called the *seed*. Using the same seed in the rand function for multiple channels produces the exact same results. To create completely random results, use the true rand function.

Syntax:	rand(Seed)
Arguments:	■ Seed is the value used to generate the random return value. This is usually set to a changing value such as the current frame number.

Examples:	■ rand(frame % 10) * 50 + 50 returns a set of random values between0 and 100 that repeats every 10 frames. ■ rand(frame) * 100 returns a random value between -100 and 100for every frame in the animation. The following curve shows the result of this function: 

truerand

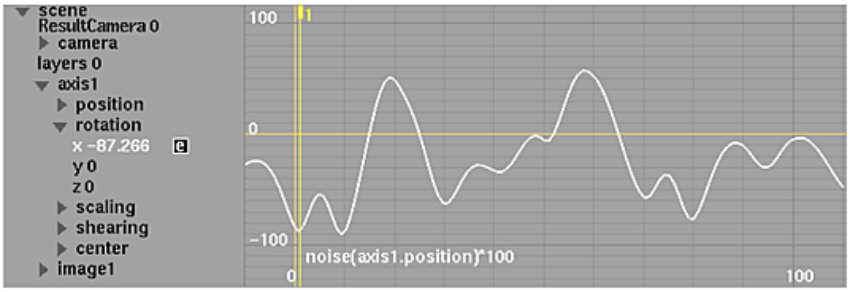
Returns a truly random value between two given numbers. The sequence of returned values will constantly change, never reproducing past results.

Syntax:	truerand(Low, High)
Arguments:	■ Low and High are the upper and lower bounds, respectively, of the random number to generate.
Example:	■ truerand(-5.5,10.8) returns a truly random value between -5.5and 10.8.

noise

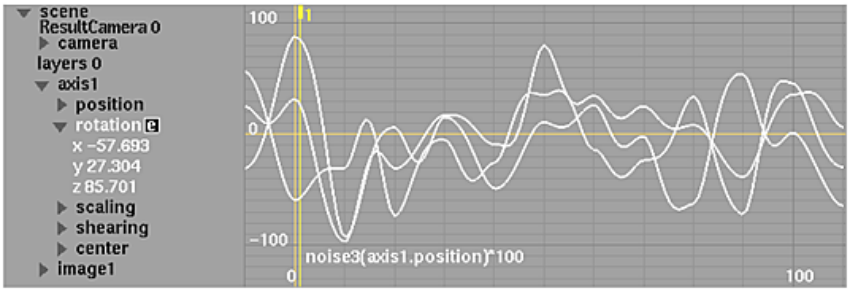
Returns a random value between -1 and 1, based on a given vector. If the parameter varies smoothly, this function will return a continuously changing value that also varies smoothly.

Syntax:	noise(Position)
Arguments:	■ Position is a vector used as a seed for the returned random value.
Examples:	■ noise(frame) * 5 returns a continuous random value between-5 and 5. ■ (noise(axis1.position) + 1) / 2 * 100 returns a continuous random value between 0 and 100. ■ noise(axis1.position) * 100 yields the following curve:



noise3

Returns a random vector for all elements in a vector between -1 and 1. If the parameter varies smoothly, this function will return a vector of continuously changing values that also vary smoothly.

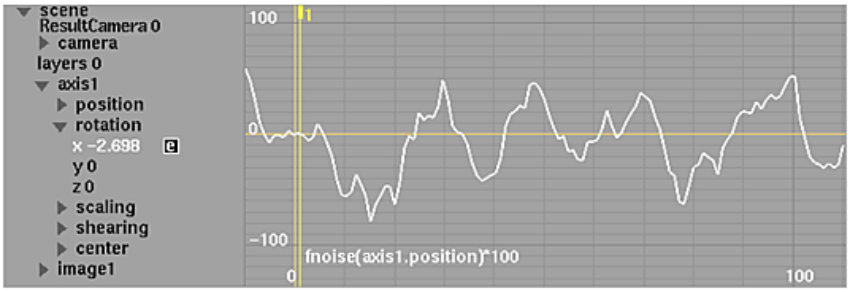
Syntax:	noise3(Position)
Arguments:	■ Position is a vector used as a seed for the returned random vector.
Examples:	■ noise3(frame) * 5 returns a vector of continuous random values between -5 and 5. ■ (noise3(axis1.position) + 1) / 2 * 100 returns a vector of continuous random values between 0 and 100. ■ noise3(axis1.position) * 100 returns the following three curves: 

fnoise

Returns a random value between -1 and 1, based on a given vector. If the parameter varies smoothly, this function will return a continuously changing value that also varies smoothly to a fractal pattern. This function is similar to the noise function, but it applies a fractal pattern to the result.

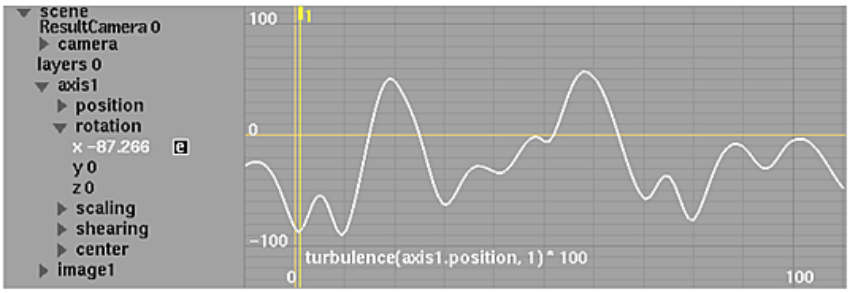
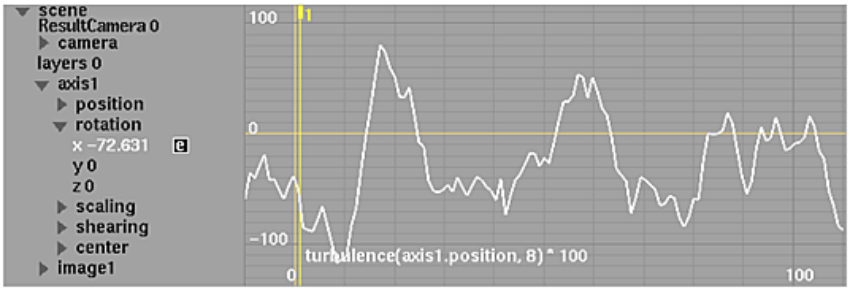
Syntax:	fnoise(Position)
Arguments:	■ Position is a vector used as a seed for the returned random value.
Examples:	■ fnoise(frame) * 5 returns a continuous random value between-5 and 5. ■ (fnoise(axis1.position) + 1) / 2 * 100 returns a continuous random value between 0 and 100.

- fnoise(axis1.position) * 100 yields the following curve:



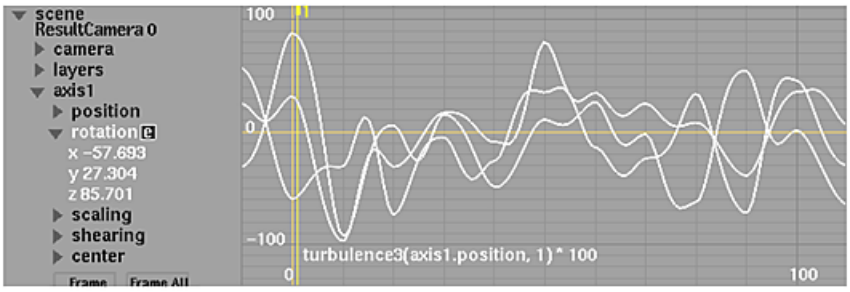
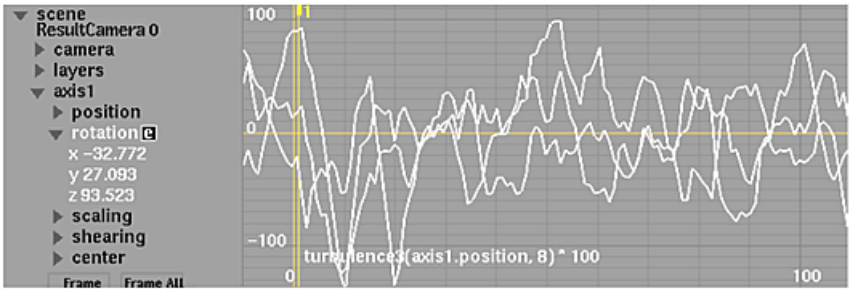
turbulence

Returns a random value between -1 and 1, based on a given vector and with the ability to control the level of smoothness for the resulting curve.

Syntax:	turbulence(Position, Level)
Arguments:	■ Position is a vector used as a seed for the returned random value. ■ Level is a positive integer where the greater the value, the greater the jitter applied to the resulting curve. If this argument is set to a value less than 1, the level will be treated as if 1 were used. If it is set to a number with a fractional component, the number will be rounded down to the nearest integer.
Examples:	■ turbulence(axis1.position, 1) * 100 yields the following curve:  ■ turbulence(axis1.position, 8) * 100 yields the following curve: 

turbulence3

Returns a random vector for all elements in a vector between -1 and 1, along with the ability to control the smoothness of the resulting curve. If the position parameter varies smoothly, this function will return a vector of continuously changing values that also vary smoothly.

Syntax:	turbulence3(Position, Level)
Arguments:	■ Position is a vector used as a seed for the returned random vector. ■ Level is a positive integer used to control the level of jitter of the resulting curve. If this argument is set to a value less than 1, the level will be treated as if 1 were used. If it is set to a number with a fractional component, the number will be rounded down to the nearest integer.
Examples:	■ turbulence3(axis1.position, 1) * 100 returns the following three curves:  ■ turbulence3(axis1.position, 8) * 100 returns the following three curves: 

Rounding Functions

The following functions are useful for performing various rounding calculations.

NOTE Although you can use these rounding functions as you would any other predefined function, they are actually implemented as user-defined functions in the functions file provided with the Flame installation. See Defining Your Own Functions (page 1741).

round

Returns a number rounded to the nearest integer.

Syntax:	round(Number)
Arguments:	■ Number is the number to round.
Examples:	■ round(2.8) returns 3. ■ round(-2.8) returns -3. ■ round(2.3) returns 2.

ceil

Rounds a number up to the next integer value regardless of its value.

Syntax:	ceil(Number)
Arguments:	■ Number is the number to round up.
Examples:	■ ceil(2.8) returns 3. ■ ceil(-2.8) returns -2. ■ ceil(4) returns 4.

floor

Rounds a number down to the nearest integer regardless of its value.

Syntax:	floor(Number)
Arguments:	■ Number is the number to round down.
Examples:	■ floor(2.3) returns 2. ■ floor(-2.3) returns -3. ■ floor(4) returns 4.

trunc

Returns the integer value of a number by truncating its fractional part.

Syntax:	trunc(Number)
Arguments:	■ Number is the number that you want to truncate.

Examples:	■ trunc(3.8) returns 3. ■ trunc(-3.8) returns -3. ■ trunc(PI) returns 3.

Trigonometric Functions

The following functions are useful for working with angles and performing various trigonometric calculations.

degrees

Converts angle units from radians into degrees.

Syntax:	degrees(Angle)
Arguments:	■ Angle is the angle in radians that you want to convert.
Examples:	■ degrees(PI) returns 180. ■ degrees(PI/2) returns 90.

radians

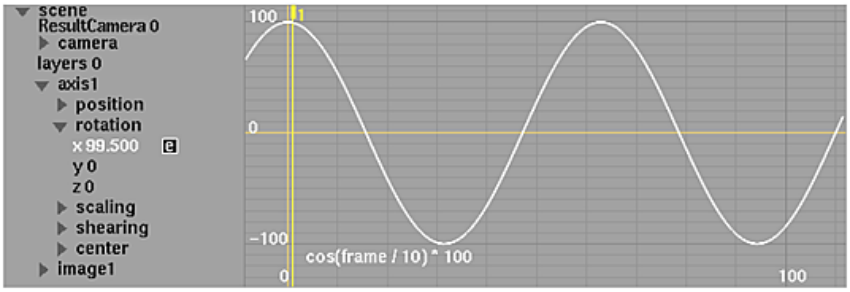
Converts angle units from degrees into radians.

Syntax:	radians(Angle)
Arguments:	■ Angle is the angle in degrees that you want to convert.
Example:	■ radians(225) returns 3.927 (5*PI/4).

cos

Returns the cosine of a given angle.

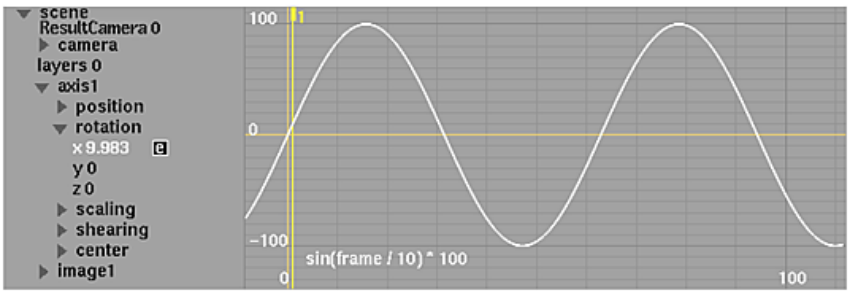
Syntax:	cos(Angle)
Arguments:	■ Angle is the angle in radians of which you want the cosine.
Examples:	■ cos(0) returns 1. ■ cos(PI / 3) returns 0.5. ■ cos(frame / 10) * 100 yields the following curve:



sin

Returns the sine of a given angle.

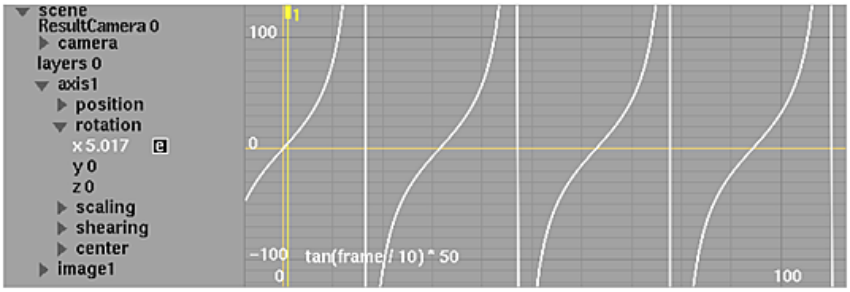
Syntax:	sin(Angle)
Arguments:	■ Angle is the angle in radians of which you want the sine.
Examples:	■ sin(0) returns 0. ■ sin(PI / 6) returns 0.5. ■ sin(frame / 10) * 100 yields the following curve:



tan

Returns the tangent of a given angle.

Syntax:	tan(Angle)
Arguments:	■ Angle is the angle in radians of which you want the tangent.
Examples:	■ tan(0) returns 0. ■ tan(PI / 4) returns 1. ■ tan(PI / 3) returns 1.7321. ■ tan(frame / 10) * 50 yields the following curve:



acos

Returns the arccosine—the inverse function of the cosine—of a given number. The returned angle is given in radians within the range 0 to PI.

Syntax:	acos(Number)
Arguments:	■ Number is the cosine of the angle you want and must be between-1 and 1.
Examples:	■ acos(0.5) returns 1.0472 (PI/3 radians). ■ degrees(acos(0.5)) returns 60.

asin

Returns the arcsine—the inverse function of the sine—of a given number. The returned angle is given in radians within the range -PI/2 to PI/2.

Syntax:	asin(Number)
Arguments:	■ Number is the sine of the angle you want and must be between-1 and 1.
Examples:	■ asin(0.5) returns 0.5236 (PI/6 radians). ■ degrees(asin(0.5)) returns 30.

atan

Returns the arctangent—the inverse function of the tangent— of a given number. The returned angle is given in radians within the range -PI/2 to PI/2.

Syntax:	atan(Number)
Arguments:	■ Number is the tangent of the angle you want.
Examples:	■ atan(1) returns 0.7854 (PI/4 radians). ■ degrees(atan(1)) returns 45.

atan2

Returns the arctangent of y/x, using the signs of both arguments to determine the quadrant of the return value. The arctangent is the angle from the origin to the vector (x,y). The returned angle is given in radians within the range -PI to PI.

Syntax:	atan2(x, y)
Arguments:	■ x and y are the components of the vector to be used in the function.
Examples:	■ atan2(1, 1) returns 0.7854 (PI/4 radians). ■ atan2(-1, -1) returns -2.3562 (-3*PI/4 radians). ■ atan2(1, 0) returns 1.5708 (PI/2 radians). ■ degrees(atan2(1,1)) returns 45.

Vector Functions

Use the following functions for performing various vector-related operations.

length

Returns the euclidean length (magnitude) of a given vector. The euclidian length is equivalent to the expression sqrt(pow(Vector.x, 2) + pow(Vector.y, 2) + pow(Vector.z, 2)).

Syntax:	length(Vector)
Arguments:	■ Vector is the vector of which you want the euclidean length.
Examples:	■ length((2, 0, 0)) returns 2. ■ length((1, 1, 0)) returns 1.4142. ■ length((-1, -1, -1)) returns 1.7321. ■ length(axis1.position - axis2.position) returns the distance between axis1 and axis2.

dot

Returns the scalar dot-product of two given vectors. The *dot-product* is the product of the lengths of two vectors and the cosine of the angle between them. If the two vectors are at a right angle (90 degrees), their dot-product is 0.

If the product of their lengths equals 1 and they point in opposite directions (180 degrees), their dot-product is -1. The dot-product is equivalent to the expression V1.x * V2.x + V1.y * V2.y + V1.z * V2.z.

Syntax:	dot(V1, V2)

Arguments:	■ V1 and V2 are the vectors of which you want the dot-product.
Examples:	■ dot((1, 1, 0), (0, 0, 1)) returns 0. ■ dot((2, 0, 0), (0.5, 0, 0)) returns 1. ■ dot((0, 2, 0), (0, -0.5, 0)) returns -1. ■ dot((2, 0, 1), (4, 5, 5)) returns 13.

cross

Returns the vector cross-product of two given vectors. The cross-product is the vector perpendicular to the plane containing the two vectors. In effect, there will be a right angle between the returned vector and the first given vector, as well as a right angle between the returned vector and the second given vector. The length of the resulting vector is equal to the product of the two vectors and the sine of the angle between them. The cross-product is equivalent to the vector (V1.y * V2.z - V1.z *V2.y, V1.z * V2.x - V1.x * V2.z, V1.x * V2.y - V1.y * V2.x).

Syntax:	cross(V1, V2)
Arguments:	■ V1 and V2 are the vectors of which you want the cross-product.
Examples:	■ cross((1, 0, 0), (0, 1, 0)) returns (0, 0, 1). ■ cross((1, 1, 0), (0, 1, 1)) returns (1, -1, 1). ■ cross((2, 0, 0), (0, 0.5, 0)) returns (0, 0, 1).

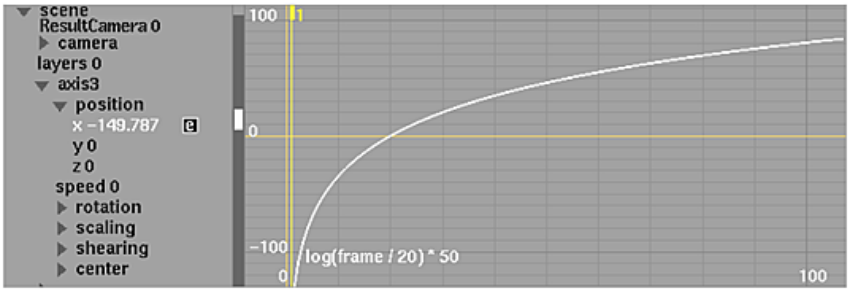
Logarithmic Functions

Use the following functions for performing various logarithmic calculations.

log

Returns the natural logarithm of a given number. The log function is the inverse of the exp function.

Syntax:	log(Number)
Arguments:	■ Number is the positive number of which you want the natural logarithm.
Examples:	■ log(1) returns 0. ■ log(2) returns 0.6931. ■ log(exp(5)) returns 5. ■ log(256) / log(2) returns 8. ■ log(frame / 20) * 50 yields the following curve:



log10

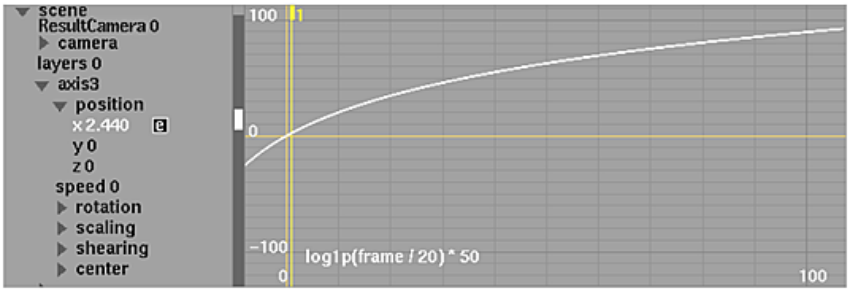
Returns the base-10 logarithm of a given number.

Syntax:	log10(Number)
Arguments:	■ Number is the positive number of which you want the base-10logarithm.
Examples:	■ log10(1) returns 0. ■ log10(10) returns 1. ■ log10(100) returns 2.

log1p

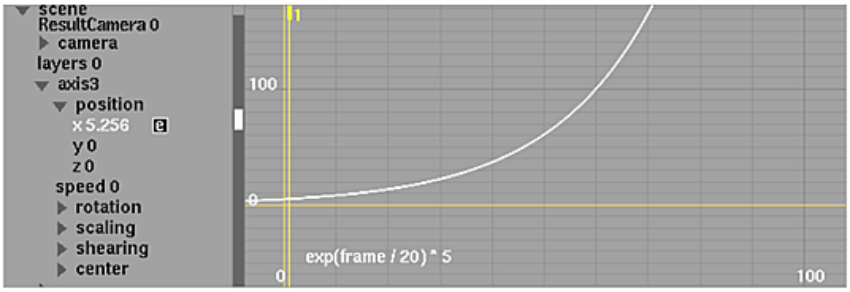
Returns the natural logarithm of 1 plus a given number. The log1p function is the inverse of the expm1 function.

Syntax:	log1p(Number)
Arguments:	■ Number is the positive number less 1 of which you want the natural logarithm.
Examples:	■ log(0) returns 0. ■ log(1) returns 0.6931. ■ log1p(expm1(5)) returns 5. ■ log1p(frame) returns the equivalent of log(1+frame). ■ log1p(frame / 20) * 50 yields the following curve:



exp

Returns the constant *e* (the base of the natural logarithm) raised to the power of a given number. The exp function is the inverse of the log function.

Syntax:	exp(Number)
Arguments:	■ Number is the exponent applied to the base *e*.
Examples:	■ exp(0) returns 1. ■ exp(1) returns 2.7182. ■ exp(2) returns 7.3890. ■ exp(log(5)) returns 5. ■ exp(frame / 20) * 5 yields the following curve: 

expm1

Returns the constant *e* (the base of the natural logarithm) raised to the power of a given number, minus 1. The expm1 function is the inverse of the log1p function.

Syntax:	expm1(Number)
Arguments:	■ Number is the exponent applied to the base *e*.
Examples:	■ expm1(0) returns 0. ■ expm1(1) returns 1.7182. ■ expm1(2) returns 6.3890. ■ expm1(log1p(5)) returns 5. ■ expm1(frame) returns the equivalent of exp(frame) - 1.

Defining Your Own Functions

Flame allows you to define your own functions and use them like any other predefined function. By defining functions in terms of existing functions, you can simplify the creation of complex expressions in your animations.

NOTE You should have a good understanding of how to use functions in expressions before you define your own functions.

The Functions File

There are two text files in which you can enter user-defined functions:

- Project-specific file: */opt/Autodesk/project/<project_name>/expressions/userfun.expressions*
- Template file: */opt/Autodesk/presets/<product_name>/expressions/userfun.expressions*

To define a function for your current project, add it to the project-specific file. Modifying this file will affect only that project and not any other existing or future projects.

If you decide that you want to use the function in future projects, copy it from the project-specific file to the template file. When you create a project, the template file is copied to your project directory as the project-specific file for that new project. This functionality allows you to modify user-defined functions in new projects without affecting the behaviour of your animations in older projects.

To edit the functions file:

1. Open a command window.
2. Use the mouse to position the cursor in the shell and type one of the following commands, depending on which file you want to edit.

Enter:	To:
nedit /opt/Autodesk/project/<project_name>/expressions/userfun.expressions	Edit the project-specific file.
nedit /opt/Autodesk/presets/<product_name>/expressions/userfun.expressions	Edit the template file.

   The functions file opens in a shell and you can edit it as you would any other text file.

Defining a Function

A function is composed of a function name, with the list of arguments, and the expression that forms its definition. You start a function by its name, followed by an opening parenthesis, the arguments for the function separated by commas, and finally a closing parenthesis. The following rules apply for function and argument names:

- The first character of a function name must be a letter, and subsequent characters can only contain letters or numbers. For example, sin100 is a valid function name, although 100sin is not.
- The first character of an argument must be a dollar sign ($), the second character must be a letter, and subsequent characters can only contain letters or numbers. For example, $arg1 is a valid argument name, although $1 and arg3 are not.
- Function names and arguments are case sensitive, meaning that lowercase and uppercase make a difference.

The function name and its definition are separated by a colon (:). The definition itself can be any valid expression, except that channel names are not accepted since user-defined expressions are not related to any particular setup. In the definition, the arguments can appear, complete with the dollar sign preceding it. Finally, the definition must end with a semicolon (;) and it can span several lines.

NOTE Errors in the functions file will be reported when it is first read. This file is read every time that an unknown function is used in an expression.

Comments

You can include comments in the functions file by starting the line with a number sign (#). The comments extend through the end of the line on which they appear and are ignored by Flame when the user-defined functions file is interpreted. Comments are useful for describing the purpose of a function and adding any notes you feel are pertinent. They can be introduced anywhere in the file, except for inside of function definitions that span several lines.

Sample Function Definitions

The following sample functions are found in the functions file that is installed with Flame.

- This function creates a sine curve for a given channel that oscillates between 0 and 100. The sin100 function takes one argument, named $arg1, that it uses in conjunction with the predefined sin function. The return value for sin100 is the sine of $arg1 times 50, plus 50.

  sin100($arg1) : sin($arg1)*50 + 50;

- This function creates a sine curve based on the specified frequency and amplitude. The sinfreqamp function takes three arguments, named $pos, $freq and $amp.

  sinfreqamp($pos,$freq,$amp) : sin($pos*$freq)*$amp;

- This function provides an approximation of the speed, or more precisely, the derivative (rate of change), of a given channel. The speed function takes one argument, named $channelName.

  speed($channelName) :eval($channelName,frame+0.1) -

  eval($channelName,frame-0.1))/0.2;

NOTE For information on the other example functions in the functions file, see Slip Functions (page 1725) and Rounding Functions (page 1733).

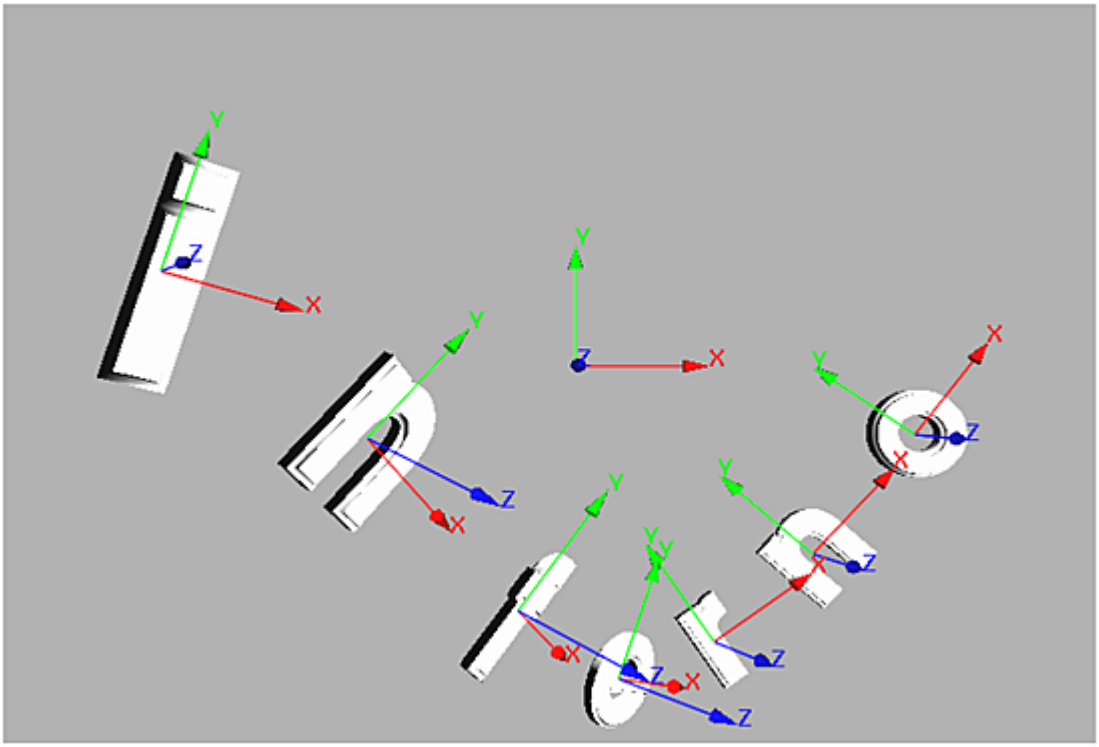
Sample Expression Setups

A few Action setups with sample expressions are provided in the directory */opt/Autodesk/presets/<product_name>/examples/action*.

To load an setup file from Action accessed from the Tools tab, click the Load button in the Action menu. To load an Action setup from Batch or Batch FX, select the Action node in the schematic, then click Node Prefs and Load Node.

Animated Text

To see this example, load the example setup named *expressions_text* into Action. This example involves multiple text objects that move along a path, with each letter following slightly behind the last.



Each letter's position and rotation are created through the expressions in the following folder. The letter 'o' rotates its X-axis towards the direction of its movement. The align function is used to generate the rotation vector.

Each preceding letter sets its position and rotation an increasing number of frames behind the letter 'o'. The channel dummy_axis.position.x is used here to store a value determining the distance between each letter. The eval function is used to determine the values of the o_axis.position and o_axis.rotation channels at different points in time.

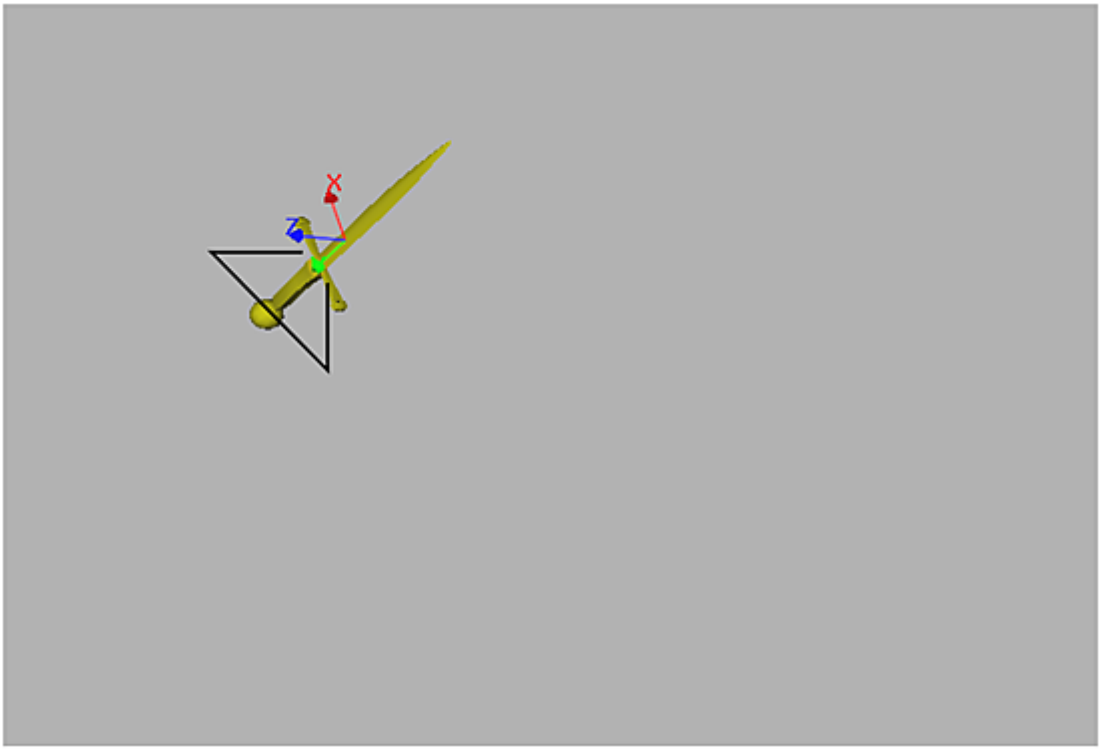
NOTE A series of expressions that differ only by a numeric value, such as in this example, can be entered quickly using the SelectionOrder keyword. See Cascading Expressions (page 1715).

Channel	Expression
o_axis.rotation	align(o_axis.position)
n2_axis.position	eval(o_axis.position, frame - dummy_axis.position.x * 1)
r_axis.position	eval(o_axis.position, frame - dummy_axis.position.x * 2)
e_axis.position	eval(o_axis.position, frame - dummy_axis.position.x * 3)
f_axis.position	eval(o_axis.position, frame - dummy_axis.position.x * 4)
n_axis.position	eval(o_axis.position, frame - dummy_axis.position.x * 5)
i_axis.position	eval(o_axis.position, frame - dummy_axis.position.x * 6)

Channel	Expression
n2_axis.rotation	eval(o_axis.position, frame - dummy_axis.position.x * 1)
r_axis.rotation	eval(o_axis.position, frame - dummy_axis.position.x * 2)
e_axis.rotation	eval(o_axis.position, frame - dummy_axis.position.x * 3)
f_axis.rotation	eval(o_axis.position, frame - dummy_axis.position.x * 4)
n_axis.rotation	eval(o_axis.position, frame - dummy_axis.position.x * 5)
i_axis.rotation	eval(o_axis.position, frame - dummy_axis.position.x * 6)

Spinning Dagger

To see this example, load the setup named *expressions_spinning_dagger* into Action. This example features a dagger moving in an arc. The dagger's Y-axis is aligned with the direction of its movement and the rest of the dagger spins around its axis.

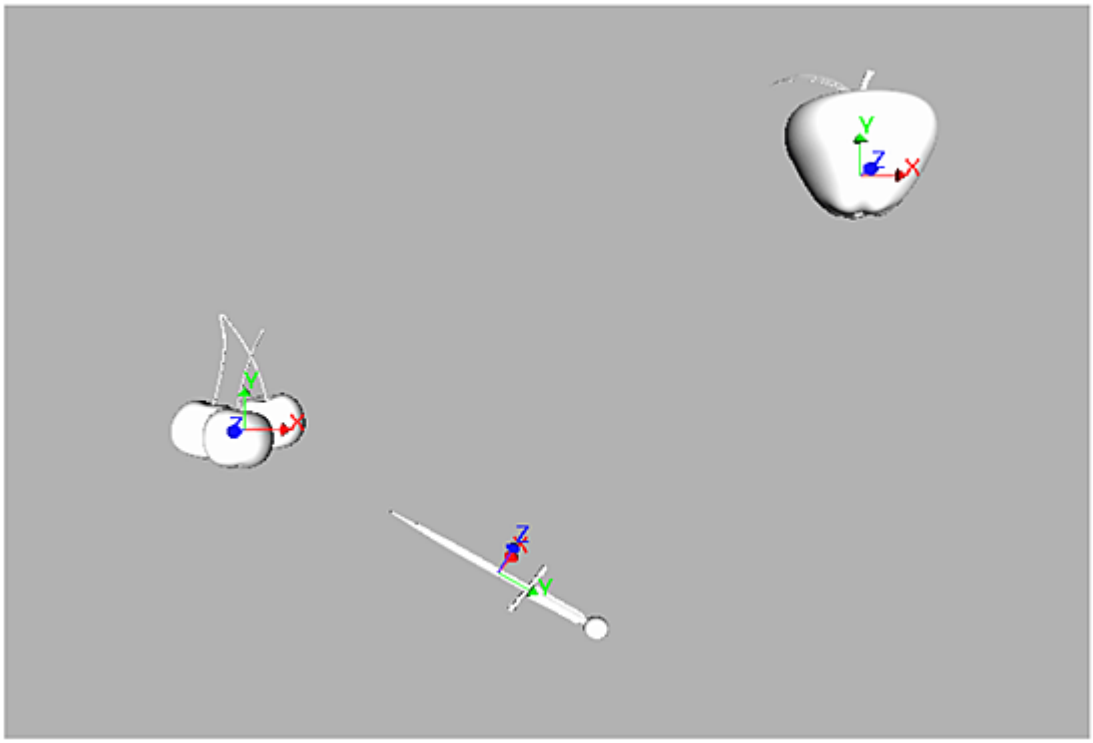


The dagger's rotation is created through the dagger's rotation of its negative Y-axis towards the direction of its movement and the rest of the dagger spins one full rotation around this same axis every 32 frames. The align function is used to generate the rotation vector.

Channel	Expression
dagger_axis.rotation	align(dagger_axis.position, (0,-1,0), frame * 2 * PI / 32)

Dagger, Apple, and Cherry

To see this example, load the setup named *expressions_dagger_apple_cherries* into Action. This animation points a dagger at either a moving apple or moving cherries, whichever is closer in position at the time.

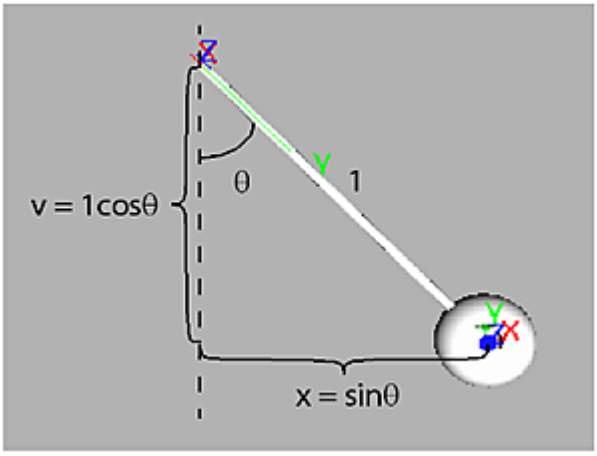


The dagger's rotation is created by the dagger's position as the dagger points its negative Y-axis in the direction of either the apple or the cherries, whichever happens to be closest. The length function is used to find the distance between the dagger and the other objects, while the if function is used to determine which is the shorter distance. The lookat function is used to generate the rotation vector.

Channel	Expression
dagger_axis.rotation	if(length(cherries_axis.position - dagger_axis.position) <length(apple_axis.position - dagger_axis.position), lookat(cherries_axis.position, dagger_axis.position, (0,-1,0), (1,0,1)), lookat(apple_axis.position, dagger_axis.position, (0,-1,0), (1,0,1)))

Dampened Pendulum

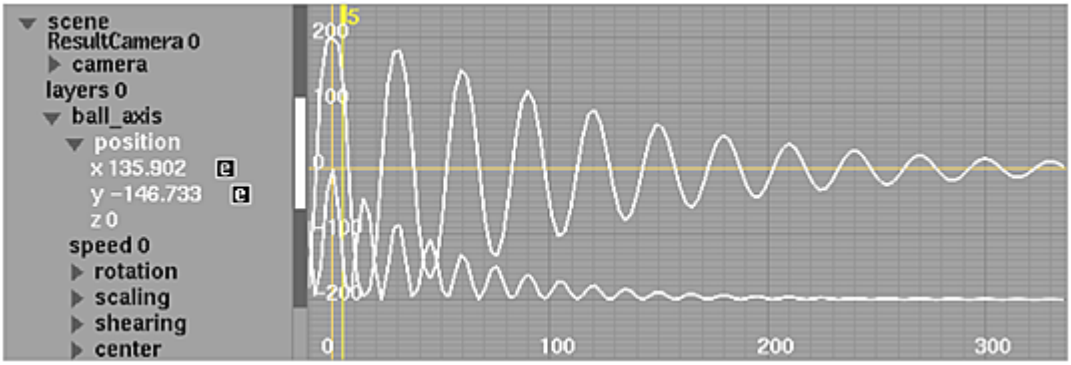
To see this example, load the setup named *expressions_dampened_pendulum* into Action. This examples involves a pendulum (a ball attached to a rope) that swings freely back and forth under the force of gravity until it comes to a stop.



The ball oscillates in a dampened harmonic motion that is created by animating the position of the ball and the rotation of the rope. The ball's x position is defined as the sine of the swing angle between the rope and centre axis, multiplied by the length of the rope. This angle is initially PI/2 and decreases logarithmically. In addition, the ball oscillates with a period of 30 frames. The sin function is used to calculate the sine of the swing angle and the exp function is used to create the logarithmic dampening effect. The cos function is used to create the oscillation.

The ball's y position is defined as the negative cosine of the swing angle between the rope and centre axis, multiplied by the length of the rope. This expression is formed similarly to that of the ball_axis.position.x channel. The rope rotates such that its Y-axis is always pointing toward the centre of the ball. The lookat function is used to generate the rotation vector.

Channel	Expression
ball_axis.position.x	sin(PI / 2 / exp(frame / 100) * cos(frame * 2 * PI / 30)) *200
ball_axis.position.y	-cos(PI / 2 / exp(frame / 100) * cos(frame * 2 * PI / 30))* 200
rope_axis.rotation	lookat(ball_axis.position, rope_axis.position, (0,1,0), (0,0,1))



Modifying Clips with Clip History

28

To create clip history while rendering:

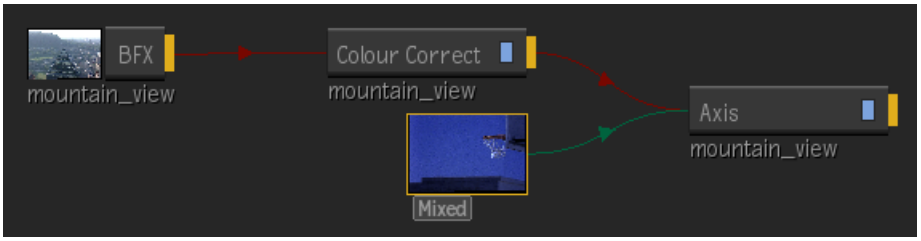
1. In the Preferences menu, click the General tab.
2. Under Default Rendering Options, enable Include History.

   NOTE In Batch and Tools editors, you also have this option in the Render box, but by setting it in the Preferences menu, you can make sure that history is enabled for all renders (including tools that render a result directly in the workspace without entering the editor).

To view clip history in the timeline and desktop clips:

1. Select a source or sequence clip with history.
2. From the Timeline View Mode box, select History.

   The view changes to a history view of the selected clip. Batch FX and Timeline FX applied to a clip are included in its history. From the history view you can see and edit all of the effects or tools applied to a clip. Double-click effects to re-enter the editor, or use the right-click context menu for other editing options.



   TIP To quickly enter the history view, double-click the "H" icon on a clip on a desktop clip.

3. To bypass history view, and quickly re-enter the last tool used on a desktop clip with history, press `Shift` and double-click the “H” icon on the clip. In this case, you enter the last tool, but if you make any changes, you need to re-render and create a new clip. If you click Exit after making changes, a message appears stating that all changes made since the last render will be lost.

TIP Hold `Shift` while double-clicking the "H" icon to create a new Batch group instead of replacing or appending the current Batch group.

To modify clip history through History View:

1. Display the clip history of a clip.

2 Double-click a node (in schematic view) or an entry (in list view) to open the corresponding tool or Timeline FX editor.

3 Edit the settings as needed.

4 If you loaded the clip into a tool editor, do one of the following:

- To save your changes, select Return from the Exit Options box.
- To exit without saving your changes, select Cancel from the Exit Options box, and then confirm.

5 If you loaded the clip into a Timeline FX editor, click Exit.

To replace sources in a clip's history:

1 Display the history of a clip.

2 Do one of the following.

Do this:	To swap:
Double-click a source	The original source with another.
Press `Ctrl` and double-click a tool node	All of the sources used in a tool.

The cursor becomes a coloured crosshair to indicate what type of source you can select.

3 Click the source clips you want to swap in to the clip history. The sources you swap must be of the same resolution and bit depth.

NOTE If the new source is longer than the original source, the new source is shortened to fit the duration of the original source. If the new source is shorter that the original, the last frame of the new source is repeated to the end of the segment.

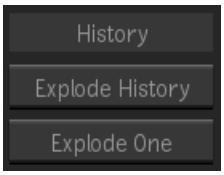
The sources are swapped and the Render button becomes active.

4 Click Render to render the swapped sources.

To explode clip history in Batch or Batch FX:

1 Double-click a clip with history in the Batch or Batch FX schematic to view its clip settings.

2 In the History section, select Explode History to expand the entire clip history into the process tree, or Explode One to expand only the most recent step of the clip's history.



Explode History and Explode One are also available from the right-click contextual menu on clip nodes with history.

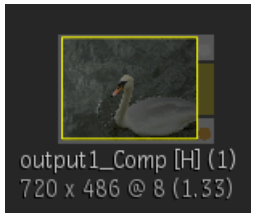
If the input clip to the last node also contains history, you can also expand the history for that clip.

NOTE These history options are not available for Batch or Batch FX clips with non-supported effects (the clip appears with a lowercase "h" next to its name in the schematic). In this case, if you double-click the clip again, you are returned to the timeline view. You can switch to History view to see its clip history.

About Clip History

Use clip history to track and update operations you have applied to a clip. The Timeline History view expands a clip, providing a view of your operations. You can also use the clip history in Batch or Batch FX as an access point for modifying clips. Clips with history have a "H" icon when displayed on the desktop. In the Batch or Batch FX schematic, a [H] appears beside the clip name under the clip proxy.

Desktop clip with history



Batch FX clip with history

Certain tools, including those that do not have an equivalent Batch or Batch FX node, such as Reverse, appear with a lowercase "h" icon when rendered. In this case, you can view the clip's history in the History view, but not make changes to that tool, or any upstream tools in the pipeline. To edit these effects, you need to rebuild the effect in the tool originally used to create it. You can use the Match button in the timeline History view to help you locate the clip to its source in the workspace.

Clips that have a clip history contain sources and intermediates. Sources are any of the original clips used to build an effect, for example, clips captured using an EDL. Intermediates are clips that are created when building a clip that uses multiple effects or operations.

A simple way of using clip history is to load a rendered clip back into the last editor that was used to create the clip. For instance, you load a clip into the Colour Corrector, modify the colour, and render a new clip. You decide that you want to change the colour slightly, so you press `Shift` and double-click the “H” icon of the new clip to go back into the Colour Corrector with the settings restored. You tweak the values slightly, and then render another clip (which has its own clip history).

If multiple tools were used to create a clip, you can load the result into a Batch or Batch FX setup, explode the history, and make modifications at any point in the process tree. For instance, you may have created a composite by bringing a colour-corrected back layer into the Keyer with a front and key-in layer that may also have been modified (for example, with a Flip). By loading the resulting clip into Batch or Batch FX, you can modify it at any point in the process tree, and then process a new clip.

NOTE You can also use a Create BFX clip (page 536) instead of, or in addition to, clip history for added flexibility in reusing or versioning complex pipelines.

Clip History Tips

- In the Batch or Batch FX preferences Rendering tab, you can enable Fill with Intermediates in the Cache section to so that each node's cache is filled with the intermediate clips contained in the history. Filling a history tree's cache increases interaction speed and avoids the need to reprocess effects that were already rendered. If an intermediate clip corresponding to a step in the history is deleted, the cache for that particular node is not filled.
- If you modify and then render an intermediate that is part of a clip history, all the clip history elements after the rendered event are also rendered. Additionally, all instances of the intermediate used in the clip history are modified. If the same intermediate appears in another clip history, however, it is not modified — once you modify an intermediate in a clip history, any copies you made of the intermediate in other clip histories are discrete.
- If you have a Paint or Stabilizer element in your clip history, all modules that precede the Paint or Stabilizer module will not be editable. You can, however, modify the elements before the Paint or Stabilizer

element by selecting the element immediately preceding the Paint or Stabilizer element in the history, and then using Match to create a clip with the same history in the Source Area. You can then use the result clip from the match. The history is expandable and editable.

- The duration of a clip created in the Paint tool is not retained. For example, enter the Paint tool with None selected, and then create a scene that is 30 frames. Then go into a tool such as Action and make some changes. When you view the clip history and click the Paint tool element, the clip will contain one frame only.
- When you enter a tool from clip history, in and out points mark the duration in the source clip that was used in the clip history (unless the entire clip was used). If you then make a modification and process the result, only the duration of the clip marked by in and out points is rendered, unless you park the cursor before the in point. If the cursor is parked before the in point, media is added to the head frames, starting from the cursor position.
- Note the following when accessing the history of Action clips:
  - When accessing the history of stereo output, you can change either the left or right eye output, or both.
  - When accessing the history of clips containing multiple outputs, you can change the output enabled in the Action Output List but you can only have one output enabled.
  - You cannot change an Output mode to Stereo if the previous output was not stereo.

Clip History Reference

Timeline History View

History View box Select a list or schematic representation of the clip's history.

History box Select to display the history of the selected source or sequence clip.

Render button Click to render sources that were replaced through the clip's history.

Match button Click to match the clip to its source in the workspace.

History View horizontal scroll bar Drag the horizontal scroll bar to pan the schematic.

History View vertical scroll bar Drag the vertical scroll bar to pan the schematic up or down.

Home button Returns the schematic to its default position.

Current Timecode field Displays the timecode of the current position. Editable.

Clip History Media Management Options

These settings can be found in the Timeline Gear menu's Commit section.

Delete History button Click to delete intermediates from selected timeline elements. If no elements are selected, intermediates for the entire edit sequence are deleted. Use when you want to free up disk space or shorten the time it takes to perform a wire transfer or archive clips with clip history.

Commit History button Click to commit a clip with history. When committing a clip, you delete the clip history without removing the clip or element's soft properties.

Clip History Preferences

These history-related preferences can be found in **Preferences ➤ General**.

Include History button Enable to include History when rendering.

Keep History Mode box Select whether Sources or Intermediates and Sources appear on the workspace when working with clip history.

Batch or Batch FX History Settings

These history options are available in the clip settings of a clip with history.

Explode History button Click to expand the entire clip history into the rendering tree.

Explode One button Click to expand only the most recent step of the clip's history. If the input clip to the last node also contains history, you can also expand the history for that clip.

The following setting is available in the Batch or BFX Prefs Rendering section, under Cache.

Fill with Intermediates button Enable to fill each node's cache with the intermediate clips contained in the history. If a clip was rendered in proxy mode, the history is not cached.

Colour Management 29

Colour management enables the desired colour values to be represented as accurately as possible throughout the finishing process. It is different from colour correction in that its intention is not to change colours but instead to preserve the perceived colour in different situations.

Colour Management Workflow

Colour management works by applying colour transforms and other operations at various stages of the pipeline. Typically, you use colour management when:

- Importing media. Images from different sources (such as digital cameras, scanned film, 3D renders, and matte paintings) often use different colour spaces and encodings, and you may want to convert them to a single working colour space.
- Outputting media. Different output formats require different encodings depending on the medium and expected viewing conditions.

You might also want to apply colour transforms to convert into and out of a specific colour space in order to perform a particular operation. For example, operations like compositing or lens blur work best in a linear colour space, while operations like tracking and edge detection work best in video or log colour space.

In addition, colour management is used to display images in Flame to ensure that the colours on the monitor and projector match what will be displayed on the final output as closely as possible.

All media in the application is now tagged with a colour space identifier to help deal with the proliferation of colour encodings used by the increasing variety of capture and display devices. Tagging your media correctly is central to using the colour management features.

Deciding whether to convert media on import

When importing media you may either convert it to a working space or leave it in its original colour space. Each approach has its advantages. If media will be used for extensive compositing in Batch and Action, it is usually best to convert to a common scene-linear working space. In other cases, it may be more convenient to leave it as is (and just tag it appropriately), for example if you have a reference track from offline editorial.

The key concept to realize is that because all media is tagged with a colour space, it is possible to easily work with material in a variety of colour spaces side by side.

Tagging Clips with a Colour Space

All media in the application is now tagged with a colour space identifier, for example, "Rec.709 video", "ARRI LogC", "ACEScg", "Matte", etc. Making the application colour space aware allows a mixture of different colour spaces to be used simultaneously in a convenient way by providing a number of benefits:

- The view transform setting of viewports is automatically updated so there is no need for manual readjustment as different clips, Timeline layers, or Batch nodes are selected.
- Accurate thumbnails and preview windows can be shown.
- The behavior of various algorithms may be customized automatically for the clip being processed. For example, the Image Type menu in keyers is now automatically set by default.
- OpenClips can describe the colour space of the media.
- There is the flexibility to either convert media upon import to a common colour space, or import in its native state and defer conversion until it may be needed.
- The application may provide warnings based on colour space.

Methods of Tagging

These benefits of tagging come with the responsibility of ensuring that media is correctly tagged upon import or creation. The application provides a number of methods to tag media such as:

- Use the Input Rules to automatically tag based on filename and path.
- Manually tag in MediaHub when importing.
- Modify Pre-processing settings after import.
- Insert a Colour Management Timeline Effect in Tag Only mode.
- Insert a Colour Management Batch node in Tag Only mode.
- Select a group of clips and use the Edit Colour Space option from the Timeline contextual menu.
- Set the Tagged Colour Space in generators such as Colour Source
- Add tagging metadata to OpenClip XMLs to integrate into a facility pipeline.

In some cases, the application assigns tags automatically:

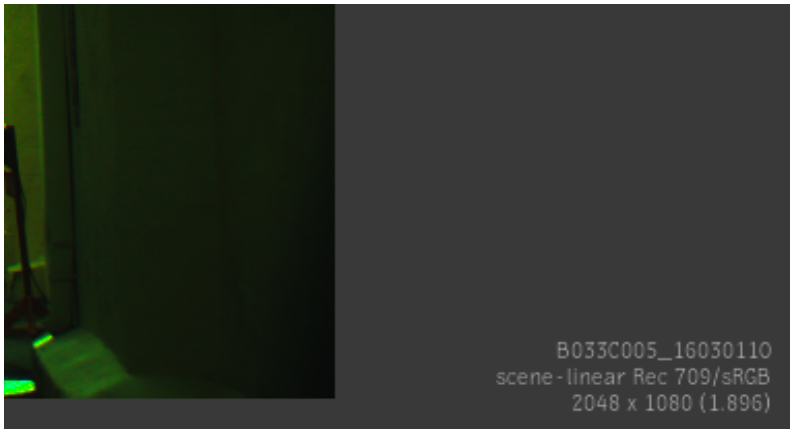
- When importing media in "From File or Rules" mode and the application is able to determine the colour space from the file. This is the case with camera raw formats such as RED r3d, ARRI Raw, and Sony Raw. Also for OpenEXR files that have the acesImageContainerFlag attribute set to 1.
- In Batch, the colour space of the Result is generally taken from the Front input. However the Result of nodes that produce mattes are tagged as such.
- In Action, the optional output passes (mattes, normals, etc.) are tagged appropriately.

Colour space tagging is preserved when Archiving and Wiring media.

Tagging as Unknown

When the colour space of a clip is not known, you may tag it "Unknown". This is what the application does when working with Archives or Wires from libraries created in releases prior to the introduction of colour space tagging. The "Unknown" tag may also be used temporarily to identify media that needs to be retagged later once more information is known about it.

Checking the Tagged Colour Space



The tagged colour space may be shown in various locations:

- There are file, input, and tagged colour space columns available in the media list view.
- The input and tagged colour space are shown in the MediaHub Preview panel at the top of the Clip Info.
- It is shown when you Alt-click on a clip or timeline segment.
- It may be written on top of clip thumbnails via the setting in User Interface Preferences
- It is written under many nodes in Batch.
- It is shown in the lower right corner of many viewports.

However, if the clip is tagged "Unknown", the colour space is generally not shown.

Mixing Colour Spaces within a Sequence

Each frame has its own colour space tag. Therefore via editing it is possible to create a Sequence or even a Segment that contains more than one colour space. The players and the Colour Management effect/node are able to respond to these changes at the frame level.

The fact that the player and viewports respond automatically to the tagging is often convenient when working with complicated Timeline Sequences. For example, your main track could have final scene-linear segments but you could have tracks or segments that are in video colour space as off-line references. However, care must also be taken when creating Sequences that mix colour spaces. For example:

- Unlike the players and viewports, Export is unable to apply different colour conversions on a per-frame basis. Keeping your Sequences uniform will avoid mixed colour space deliveries.
- Timeline blends and transitions such as dissolves should be done between elements that are in a common colour space. For example, if the colour space changes during a dissolve, the player view transform may switch causing an apparent artifact in the transition. For transitions such as fade-ins and fade-outs, you may use the Set Sequence Gap Colour Space option from the Timeline contextual menu if you need to override the default gap colour space of a Sequence.

Input Rules

Input Rules allow you to automatically tag clips on import based on their filename, extension, and path. Although the application ships with some example rule sets, because there are no widely used standard naming conventions, it is important for you to configure the rules to suit your needs.

The Input Rules are part of the colour Policy, so once you have configured them, it is easy to use them in other projects.

Creating Input Rules

Create an Input Rule as follows:

1. In the Colour Management Preferences, Input Rules tab, click Create Rule.
2. Press and hold in the Input Colour Space field, and select the colour space to use for this rule.
3. In the Pattern field, enter the text to search for in the file's name and path. Leave it as "*" to not consider the file's name and path for this rule.
4. In the Extension (Type) field, enter the text to describe the file extension (do not include the dot). For example, entering "dpx" will match files that end in ".dpx". Leave it as "*" to not consider the file's extension for this rule.
5. In the Rule Name field, enter a name for your rule.
6. Use the Up and Down buttons to set the priority of this rule relative to the other rules.

Rules that are organized higher in the rules table take precedence over lower ones. The first rule that matches is the one that will be used. At the bottom of the table is the Default rule that will be used if no other rule matches.

Pattern syntax The Extension and Pattern field strings use the Unix "glob" syntax. (There are many tutorials for this online.) Note that the matching is case-sensitive. For example, "dpx" will only match "dpx" whereas you may use "[dD][pP][xX]" to match "dpx", "Dpx", "DPX", etc. Here are some more details:

String	Pattern	Example
?	Any one character.	?pecular matches specular and Specular, as well as Xpecular and many others.
*	Zero or more characters.	*diffuse matches diffuse at the end of the name. diffuse* matches diffuse at the beginning of the name. *diffuse* matches diffuse anywhere in the name.
[*chars*]	Any one of the specified characters.	*[nN]ormal* matches normal or Normal anywhere in the name.
[*char1-char2*]	Any character in the range.	*[0-9] matches any name ending with a numeral.
[!*chars_or_range*]	Any character except those specified.	*[!0-9] matches any name that does not end with a numeral.
/	Escapes the next character, that is, treats the next character as a literal, instead of part of the pattern matching syntax.	*/[*/]* matches a pair of square brackets anywhere in the name.

Validating Input Rules

When you import media, the Input Rules will be applied when you select the "From File or Rules" option from the Input Colour Space menu when you are in Auto Convert mode and the Tagged Colour Space menu

when you are in Tag Only or Use LUT mode. Be aware that there are a few colour spaces are only compatible with one of the modes (Auto Convert or Tagging):

- Data colour spaces (Matte, Normals, etc.) and Unknown are only compatible with tagging since it is not sensible to try and Auto Convert them to a working colour space.
- The colour space Raw and colour spaces with "(no primaries)" at the end of the name are only compatible with Auto Convert since these are specialized ways of converting to a working space while ignoring gamut information and hence are not specific enough for use as tags.
- Regardless of whether Auto Convert or Tag Only is selected in the MediaHub, if you import in From File or Rules mode and the matched rule is not compatible with the currently selected mode, the import for that clip will "fall back" to using the compatible mode. For example, if the MediaHub is in Auto Convert mode and the matched rule for a piece of media identifies it as a Matte, the media will be imported in Tag Only mode (rather than trying to convert it to a working space, which would not make sense).
- If the mode is set to Auto Convert but the Working Space is set to From Project and the project working space is set to Unknown, the media will be imported in Tag Only mode.
- If a fall back to Auto Convert is ever not possible, the clip will be imported in Tag Only mode and tagged as Unknown.

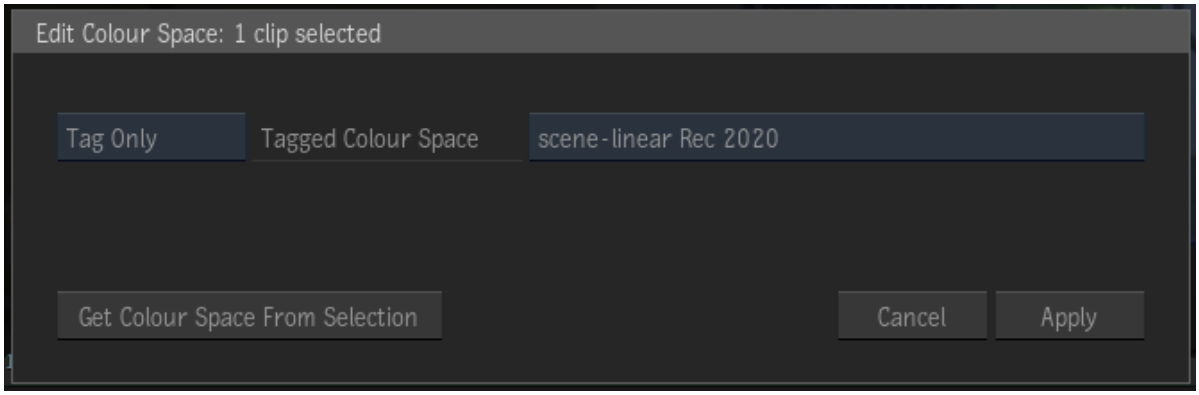
One way to test Input Rules, is to use List View in the MediaHub and enable the File Colour Space, and Colour Space columns. The File Colour Space shows the colour space being returned by the Input Rules for each clip. The Colour Space column shows the resulting tagged colour space based on the current mode (Auto Convert or Tag Only) and any fall-backs that were applied.

Updating Input Rules

When you import media in From File or Rules mode, the tagged colour space is computed and assigned at the time of import. If you later change your Input Rules, this does not change the colour space of media that was already imported. You will need to update the colour space of those clips manually, if desired. This may be done by going into the Pre-processing options in Timeline. Using the Colour Space column in the media list is a convenient way to check tagging. You may also search for media that uses a specific colour space. The Edit Colour Space option in the Timeline contextual menu is a convenient way to change the colour space of many sources at once.

Bulk Editing of Colour Space Tags

The Edit Colour Space option from the Timeline contextual menu allows you to modify the Pre-processing options of all the currently selected sources. (This tool does not affect Sequences, only sources.)



Tool Mode button Choose whether to convert your media into a different colour space or only tag them.

Tagged Colour Space menu Choose the colour space to tag your media with (this does not modify pixel values, but it does affect how viewports display your media).

Input Colour Space menu Select the source colour space of your media. "From Source" uses the the currently tagged colour space, which allows you to easily convert a set of clips in various colour spaces to a common working space.

Working Colour Space menu Select the destination colour space of your media. Set the Working Space first in order to filter the list of Input Colour Spaces to only show the compatible options. "From Project" takes the value from the Colour Management Preferences.

Get Colour Space From Selection button The tool starts with the colour space setting that was last used. Click this button to initialize with the colour space of your primary selection.

Applying Colour Management to Clips

There are two ways to apply colour management to clips:

- Automatic Conversion: By specifying options such as the source and destination colour spaces, a transform will be automatically calculated to perform the conversion.
- Manual Conversion: Use the traditional "LUT Editor" tools to to create or load a colour transform (or LUT).

This changes the underlying colour values of the clips' pixels. You should do this when you want the changes to flow though the pipeline to the final output. If you want to apply colour management to the display only, see Applying Colour Management to Displayed Images (page 1777) instead.

NOTE Regardless of whether you apply automatic or manual conversion, the resulting media must be tagged with a colour space. When using automatic conversion, the result is automatically tagged. When using manual conversion, select the colour space to tag the output of your LUT from the Tagged Colour Space menu.

Automatic Colour Conversion

The goal of automatic colour conversion is to simplify colour management by allowing you to focus on higher level concepts such as source and destination colour spaces rather than low-level concepts such as importing or creating LUTs. There are two automatic conversion tools provided:

- The Input Transform tool
- The View Transform tool

See also: Available Colour Space Options (page 178)

Input Transform

Use the Input Transform tool to convert from an input colour space to a working colour space by:

1. Select a destination working colour space from the Working Space menu. Choose "From Project" to have the working space set automatically from the project default.
2. Select a source or input colour space from the Input Space menu. Choose "From Source" to have the input space set automatically from the tagging of the input clip.

NOTE By choosing the Working Space first, the list of Input Space options will be filtered to show you options that are most compatible with the given working space. For example, pairs of colour spaces that would not require a view transform as part of the conversion.

By enabling the Invert button, you are able to convert from a working space back to an input space. This is useful when you need to deliver material in the same colour space as the original.

Adding a User-defined Input Colour Space

By choosing "+Add New" from the Input Colour Space menu, you may add your own LUT (or colour transform) as a User Colour Space. The following information must be provided:

Transform File box Browse to select the LUT or colour transform file to use for your new colour space.

Transform Name box Enter the name you want to see in menus for this new colour space.

Destination Colour Space menu Choose the colour space that your LUT outputs. This allows the system to calculate the transform needed to connect your LUT to any of the Working colour spaces. If you choose "Raw", the system will not apply and conversion after your LUT (the result will simply be tagged with whatever the select Working space is).

View Transform

The View Transform tool allows you to bake in the same colour processing the viewports apply on the fly. Use the View Transform tool to use a viewing transform to convert from a tagged or working colour space to a display colour space by:

1. From the Tagged Colour Space menu, select the colour space of the incoming clip. Choose "From Source" to have the input space set automatically.
2. Select the display colour space you want to convert to from the Display menu. Choose "Current Broadcast Monitor" or "Current Graphics Monitor" to use the settings configured in Colour Management Preferences.
3. Select the view transform you want to apply from the View Transform menu. Select "From Rules" to use the highest priority viewing rule for the given Tagged Colour Space and Display. Select from the Viewing Rules category to choose one of the other compatible rules of lower priority. Or select any other view transform.

NOTE By choosing the Tagged and Display colour spaces first, the list of viewing rules will be filtered to show you only the compatible options (just like in the viewports).

The automatic conversion algorithm will first convert from the tagged colour space to the colour space expected by the view transform. Then the view transform will be applied. Then the automatic conversion algorithm will convert from the output space of the view transform to the requested display space.

By enabling the Invert button, you are able to convert from a display space to a working colour space via an inverse view transform. This is sometimes useful for bringing video material into a scene-linear working space. When these scene-linear values later play out through the forward view transform, the result will look the same as the original material.

Adding a User-defined View Transform

By choosing "+Add New" from the View Transform menu, you may add your own LUT (or colour transform) as a User View Transform. The following information must be provided:

Transform File Browse to select your LUT or colour transform file.

View Transform Name Enter the name you want to see in menus for this new view transform.

Incoming Colour Space menu Choose the colour space that your colour transform or LUT expects as input. This allows the system to calculate the transform needed to connect media in a given tagged colour space to the colour space expected by your transform. Choose "Raw" if you do not want the system to insert this transform (in other words, to turn off auto-conversion).

Display Colour Space menu Choose the colour space that your colour transform or LUT produces as output. This allows the system to adjust the output to be appropriate for whatever display hardware has been configured as the Graphics or Broadcast monitors. Choose "Raw" if you want the output of your transform to be sent directly to the monitor without any adjustment.

3D LUT Interpolation menu If your transform includes any 3D LUTs, choose the type of interpolation that will be used when evaluating your LUTs. Trilinear is somewhat faster, Tetrahedral is somewhat better quality.

Adding a User-defined Display Transform

By choosing "+Add New" from the Display menu, you may add your own ICC Profile (or colour transform) as a User Display Transform. The following information must be provided:

Transform File Browse to select your LUT or colour transform file.

View Transform Name Enter the name you want to see in menus for this new view transform.

Input Colour Space menu Choose the colour space that your colour transform or ICC profile expects as input. Usually this should be CIE-XYZ. Choose "Raw" if you do not want the system to insert a transform to connect the output of a view transform to the input of your display transform.

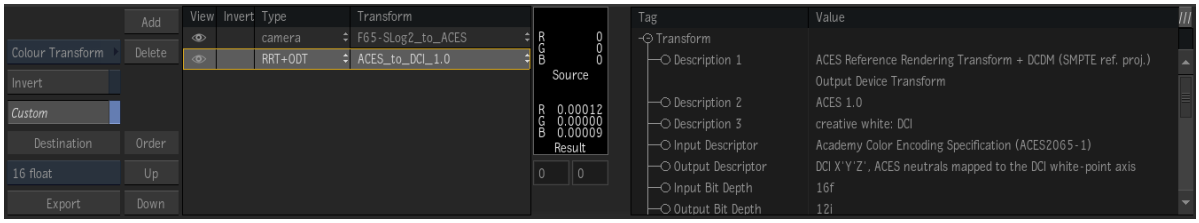
Display Type menu Select the type of display you are creating. This is used by the viewing rules to optionally limit certain rules to certain types of displays. (HDR = High Dynamic Range monitors, SDR = Standard Dynamic Range, conventional monitors.)

Manual Colour Conversion

Manual colour conversion allows you to apply a specific LUT or colour transform to a clip. A variety of tools are provided for importing, creating, and editing a transform. Note that when working with the manual tools, your result will be tagged with the colour space selected in the Tagged Colour Space menu.

Building Custom Colour Transforms

You can build a chain of colour transforms using the Custom mode of the LUT Editor (page 1760). The transforms are applied in order from beginning to end. For example, you can build a colour transform that first modifies the gamma and then changes the primaries.



1. In the LUT Editor, set the LUT type box to Colour Transform.
2. Enable the Custom button.
3. Click Add to insert an empty row.
4. Click in the Type column and select a type:
   - **Shared:** The location for shared custom colour transforms (available to all applications that use Autodesk Colour Management).
   - **Project:** The project transform folder (saved and archived with the project).
   - **Autodesk:** The default location for preset colour transforms supplied with the application. See Autodesk Color Transform Collection (page 1808).
   - **Import:** Browse for colour transforms on your file system. You can select native .ctf format files, as well as several other supported LUT and transform formats.
5. After you've selected a colour transform type, click in the Transform column and select a transform.

   Information about the transform and the operations it contains appears in the LUT Editor.
6. To add more transforms to the end of the chain, repeat steps 3 to 5.

   You can also:
   - Click in the Type or Transform columns again to change a transform in the chain.
   - Select a row and click Delete to remove a transform from the chain.
   - Use the Up or Down buttons to reorder the chain. Moving a transform up applies it earlier in the chain, and moving it down applies it later.
   - Use the visibility icon to mute a transform in the chain.
   - Enable the Invert button to invert the entire chain of transforms, or use the Invert column to invert individual transforms in the chain.

     NOTE Some colour transforms are impossible to invert perfectly. This happens when they map multiple input colours to the same output colour, such as when transforming from a large space to a smaller space. It is analogous to the fact that you cannot convert from a 12-bit signal to 8 bits and back to 12 bits again without data loss. In these situations, the Invert option maps the original output colour back to one of the original input colours, but applying the forward transform followed by the inverse transform will inevitably change some colours.

TIP You can create subfolders in the Shared or Project locations to define custom colour transform types.

Specifying a Colour Management Operator

To apply colour management to a clip, you must first specify the type of operation to perform in the LUT Editor (page 1760). You can:

- Apply a preset LUT and then modify its values.
- Choose an operator type. Depending on the type you choose, you can either import a baked operator or modify its values.

Applying a Preset

1 Select an option from the Presets box in the LUT Editor.

2 Confirm that you want to replace the existing settings.

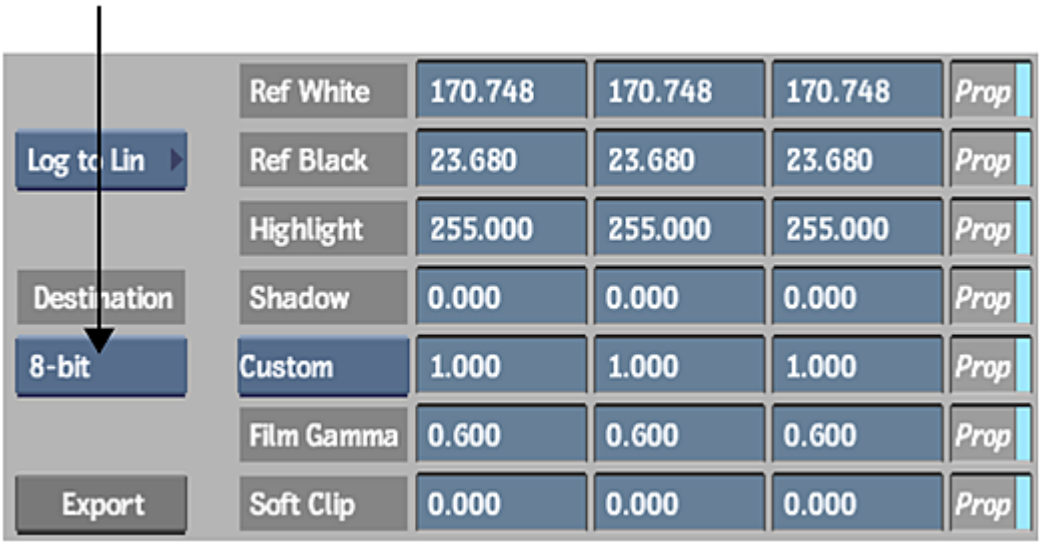
The Conversion LUT Type and the field values are all updated with preset conversion parameters. In Batch or Batch FX, a note is added with a description of the preset's usage.

3 If desired, modify the curve values as described in Modifying Basic Conversion LUT Curves (page 1766) and Advanced Editing of the Conversion LUT Curve (page 1772).

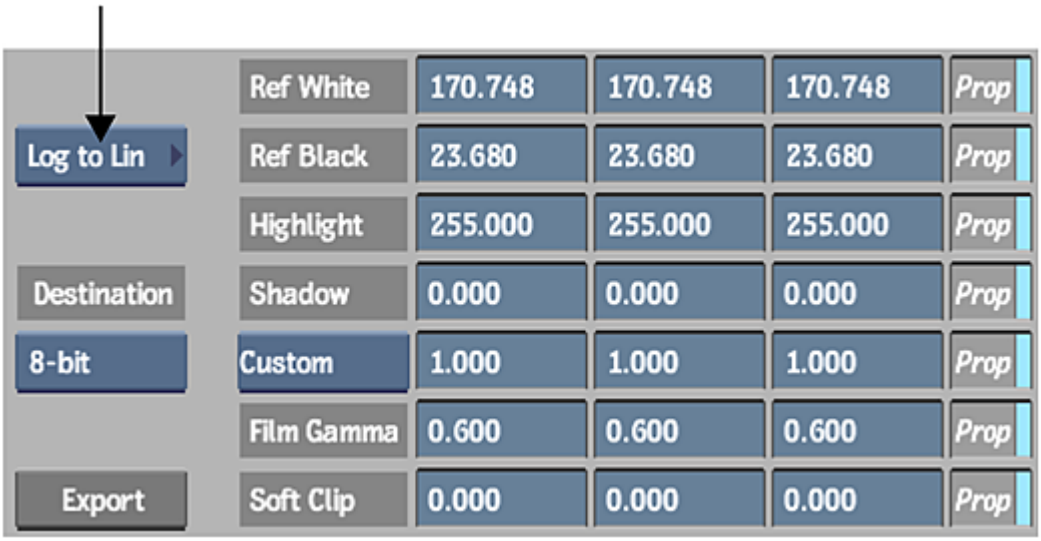
TIP To make your own preset available, save a setup to the /lut/presets subfolder of the application home directory. This does not work with 3D LUTs.

Choosing an Operator Type

1 Select an output bit-depth option from the Destination box in the LUT Editor.



2 Select an option from the LUT Type box.



Select:	To:
Colour Transform	Apply a colour transform based on the Academy/ASC colour transform XML format, as well as several other supported LUT and transform formats.
PhotoMap	Begin with a default curve for tone mapping.
3D LUT	Import a 3D LUT. Note that 3D LUTs are not available for floating-point values.
1D LUT	Import a 1D LUT. Note that some 1D LUTs are intended for integer values and others are intended for floating-point values.
EXR Display	Begin with a default curve for tone mapping.
Gamma	Begin with a default curve for modifying linear data.
Lin to Log	Begin with a default curve for converting linear or video data to logarithmic data.
Log to Lin	Begin with a default curve for converting logarithmic data to linear or video data.

3 Do one of the following:

- If you chose 1D LUT or 3D LUT, import a LUT operator.
- If you chose Colour Transform, you can import an colour transform operator. Alternatively, you can define a custom colour transform — see Building Custom Colour Transforms (page 1762).
- If you chose any other option, you can modify the curve values as described in Modifying Basic Conversion LUT Curves (page 1766) and Advanced Editing of the Conversion LUT Curve (page 1772).

Importing an Operator

1 Click Import.

The file browser appears.

2 Locate and select the file that you want to import. See Colour Management Files and Locations (page 1784).

When importing a colour transform, see Autodesk Color Transform Collection (page 1808) for a description of the available categories.

You are returned to the LUT Editor. The LUT name appears in the field next to the Import button.

- When you import a 1D LUT, the conversion curves appear in the LUT Editor. You can modify them only with the Advanced settings.
- When you import a colour transform, information about the transform and the operations it contains appears in the LUT Editor.

Modifying Basic Conversion LUT Curves

NOTE The procedures in this section apply only to Gamma, Lin to Log, and Log to Lin operators.

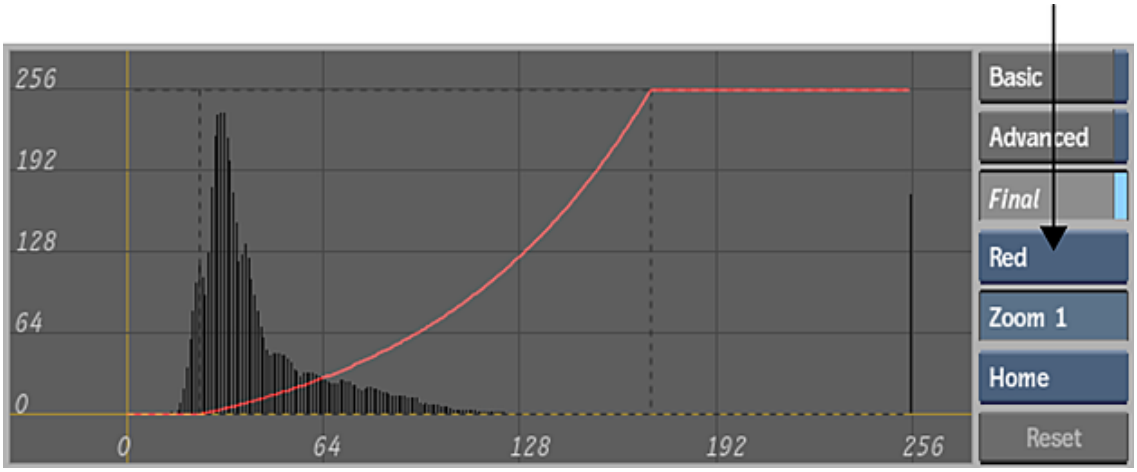
Once you define a basic LUT type in the LUT Editor (page 1760), you can modify:

- The reference white and reference black values
- The highlights and shadows
- The gamma of the conversion curve
- The gamma of the incoming film negative
- The degree to which the shoulder at the upper end of the conversion curve is softened

When you modify these values, the basic LUT curves are updated to reflect the changes. For all procedures, you can monitor the R, G, and B channels independently while modifying the LUT.

To monitor the R, G, and B channels independently:

1 From the Edit Curve box, select Red, Green, or Blue to monitor the red, green, or blue channels independently.



Modifying Reference White and Reference Black Values

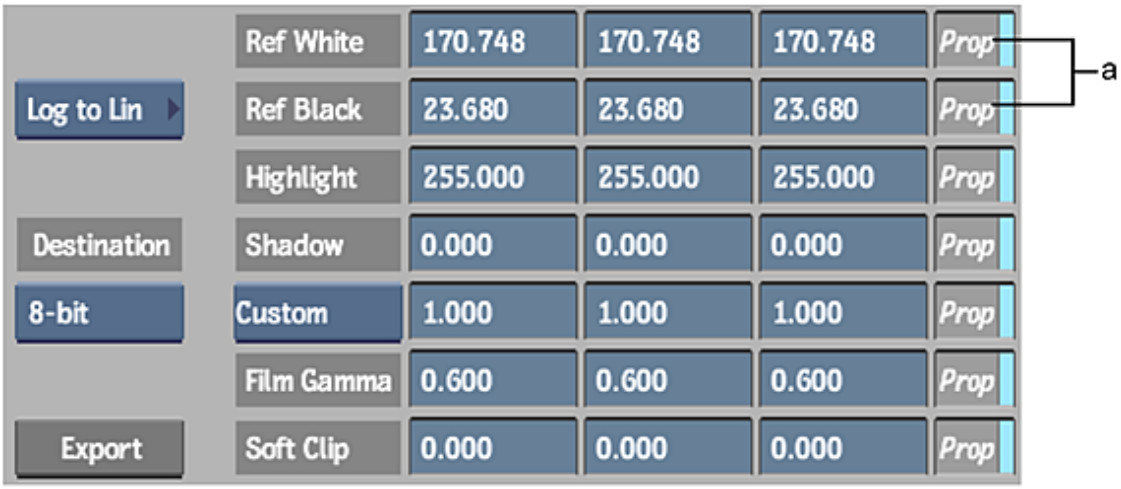
The reference white and reference black values define the intensity levels at which incoming pixels are considered white or black respectively. For example, selecting a logarithmic-to-linear basic conversion LUT curve sets the reference white to 685 and the reference black to 95. (These have been shown to be good

values for Kodak™ film stock.) As a result, source pixels with values from 685 and 1023 will produce white pixels, while those from 0 and 95 will be set to black.

Reference white and black are affected by the values assigned to highlights and shadows. White pixels are mapped to the value set for highlights, while black pixels are mapped to the value set for shadows. See Modifying Highlights and Shadows (page 1768).

To modify reference white or black values using the Ref White and Ref Black fields:

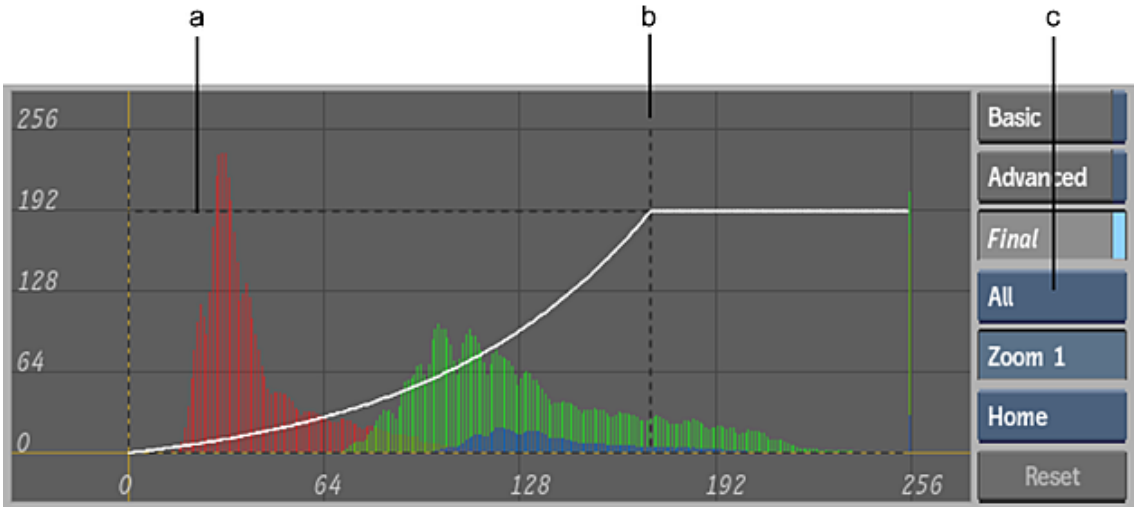
1 Do one of the following:
- To set proportional R, G, and B values, enable Prop and then enter the value in one of the (left-to-right) R, G, and B Ref White and Ref Black fields.
- To set independent R, G, or B values, disable Prop and then enter the value in the corresponding (left-to-right) R, G, or B Ref White and Ref Black fields.



(a) Ref White/Black fields and Prop (Proportional) buttons

To modify reference white or black values using the LUT Editor graph:

1 Do one of the following:
- To set proportional R, G, and B values, select All from the Edit Curve box, and then drag the vertical reference white or reference black line left or right.
- To set independent R, G, or B values, select Red, Green, or Blue from the Edit Curve box, and then drag the vertical reference white or reference black line left or right.



(a) Highlight line (b) Reference White line (c) Edit Curve box

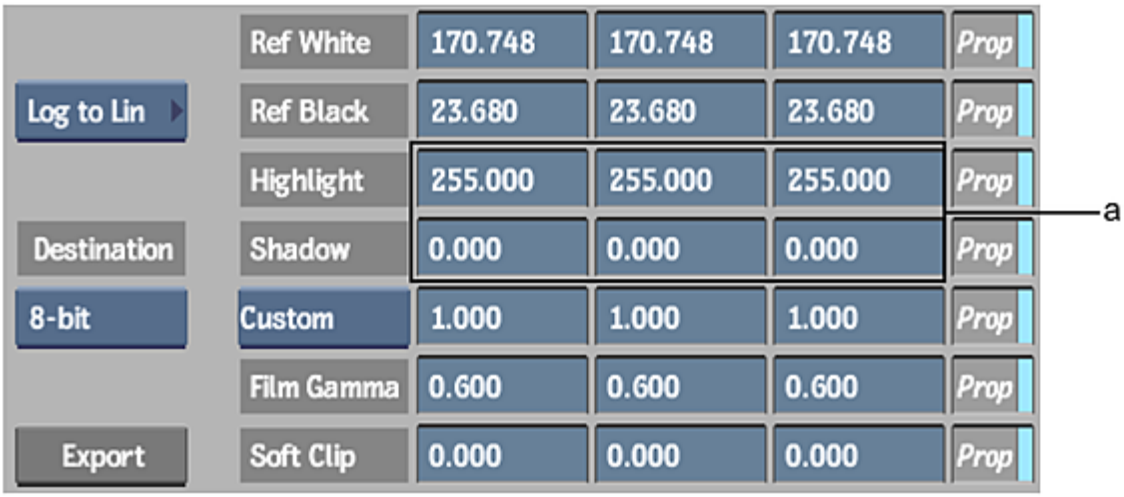
TIP If you already entered independent values and want to modify the R, G, and B values proportionally, you must drag the reference white or reference black line that corresponds to the corresponding channel. To view the selected R, G, or B luma values as a greyscale luma image in the Import Image menu, enable the Exclusive button.

Modifying Highlights and Shadows

Highlight and shadow values define the maximum and minimum values permitted for each channel. All values above the highlight are clamped down to it. Similarly, all values below the shadow are clamped up.

To modify highlight and shadow values using the Highlights and Shadows fields:

1 Do one of the following:
- To set proportional R, G, and B values, enable Prop and enter a value in one of the (left-to-right) R, G, and B Highlights and Shadows fields.
- To set independent R, G, or B values, disable Prop and enter the value in the corresponding (left-to-right) R, G, or B Highlights and Shadows fields.



(a) Highlight/Shadow fields

To modify highlight or shadow values using the LUT Editor graph:

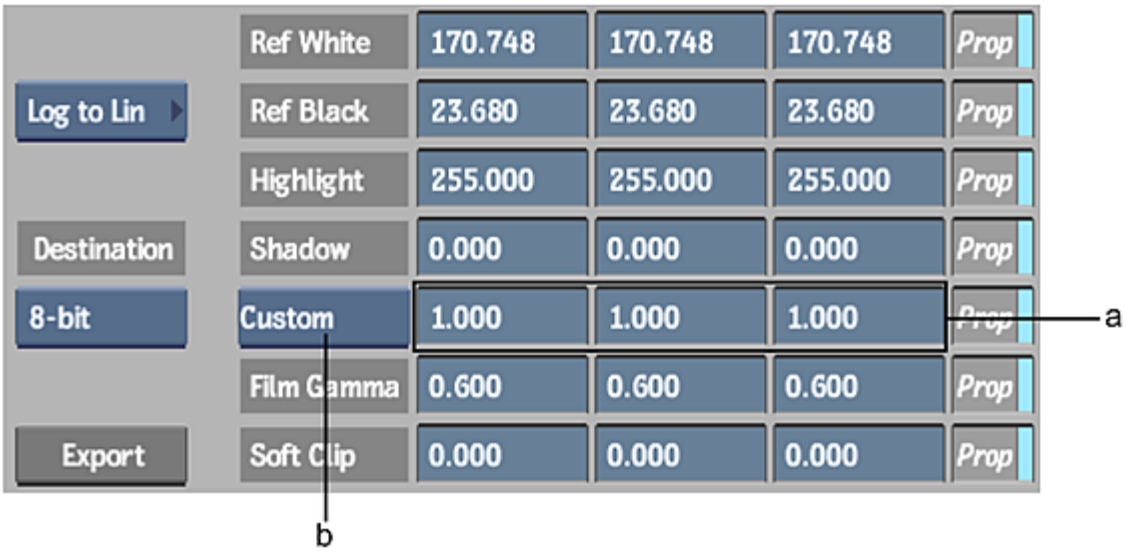
1 Do one of the following:
- To set proportional R, G, and B values, select All from the Edit Curve box, and then drag the horizontal highlight or shadow line up or down.
- To set independent R, G, or B values, select Red, Green, or Blue from the Edit Curve box, and then drag the horizontal highlight or shadow line up or down.

Modifying the Gamma of the Conversion Curve

Set the gamma correction value to correspond to the display gamma you set your system to use. You can select a preset from the Gamma Correction box or enter a custom value.

To modify the gamma correction curve using the Gamma Correction fields:

1 Select an option from the Gamma Correction box.



(a) Gamma Correction fields (b) Gamma Correction box

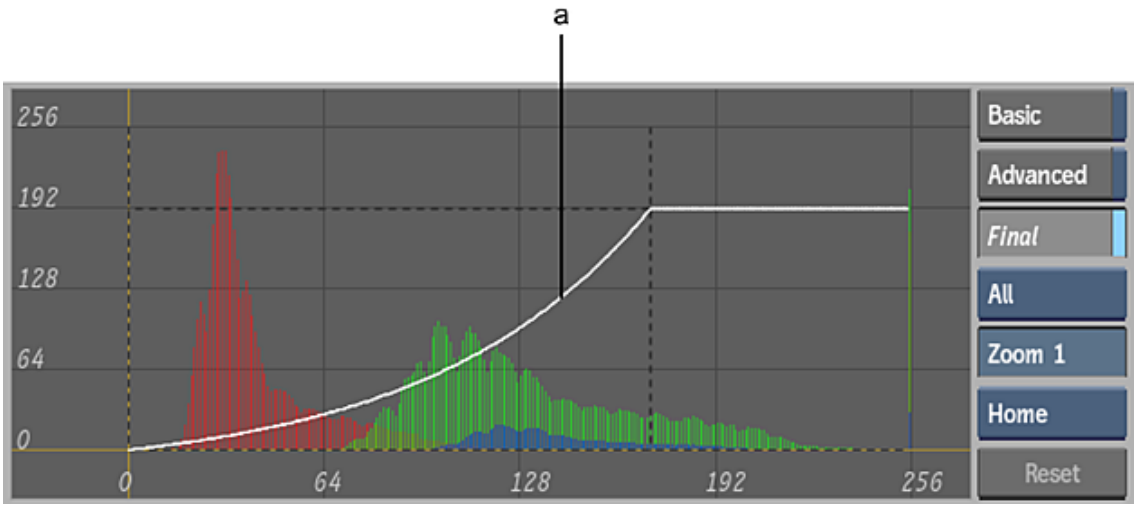
Select:	To:
Video Display	Set video gamma correction presets.
Custom	Enter custom gamma values.

2 If you selected Custom, do one of the following:
- To set proportional custom R, G, and B gamma correction values, enable Prop, and then enter the value in one of the (left-to-right) R, G, and B Gamma Correction fields.
- To set independent custom R, G, or B gamma correction values, disable Prop, and then enter the value in the corresponding (left-to-right) R, G, or B Gamma Correction field.

To modify the gamma correction curve using the LUT Editor graph:

1 Do one of the following:
- To modify the combined R, G, and B gamma correction curves proportionally, select All from the Edit Curve box, and then drag the gamma correction curve left or right.

- To set independent R, G, or B values, select Red, Green, or Blue from the Edit Curve box, and then drag the gamma correction curve left or right.



(a) Gamma correction curve

Modifying the Gamma of the Incoming Film Negative

The values in the Film Gamma fields affect the gamma correction curve inversely to the Gamma Correction fields. This is because the gamma correction performed using these controls is based on the gamma of the original film negative. Typically, the gamma of a film negative is 0.6. Only adjust this value if your film negative is of a non-standard gamma. You should not have to modify this value by very much.

To modify the gamma correction curve using the Film Gamma fields:

1. Do one of the following:
   - To set proportional custom R, G, and B gamma correction values, enable Prop and then enter the value in one of the (left-to-right) R, G, and B Film Gamma fields.
   - To set independent custom R, G, or B gamma correction values, disable Prop and then enter the value in the corresponding (left-to-right) R, G, or B Film Gamma field.

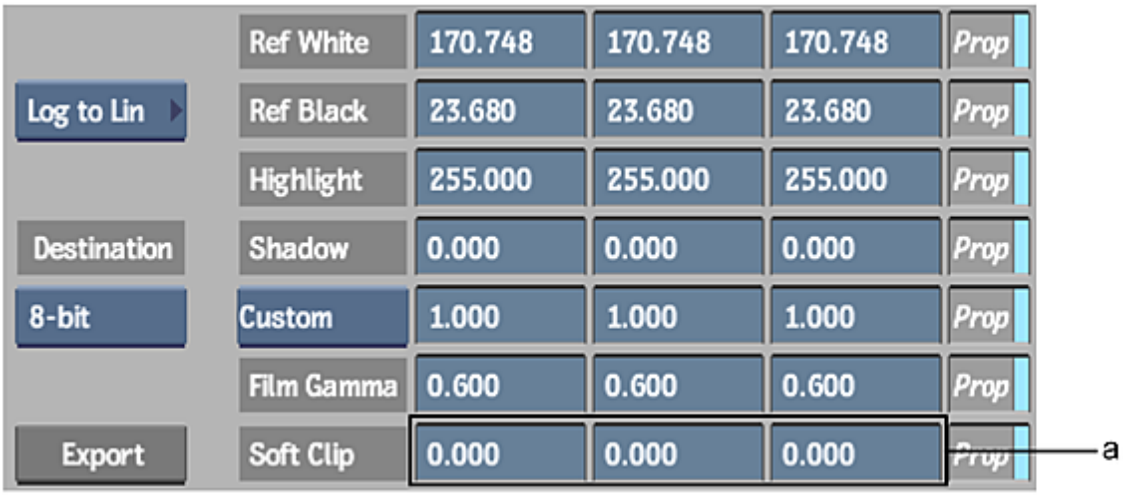
(a) Film Gamma fields

Adjusting the Softclip Values

You can soften the shoulder of the gamma correction curve by adjusting the Softclip values. When you soften the shoulder of the conversion curve, you soften the transition of colours toward the reference white value. This results in less harsh highlights in your clip.

To soften the shoulder of the gamma correction curve using the Softclip fields:

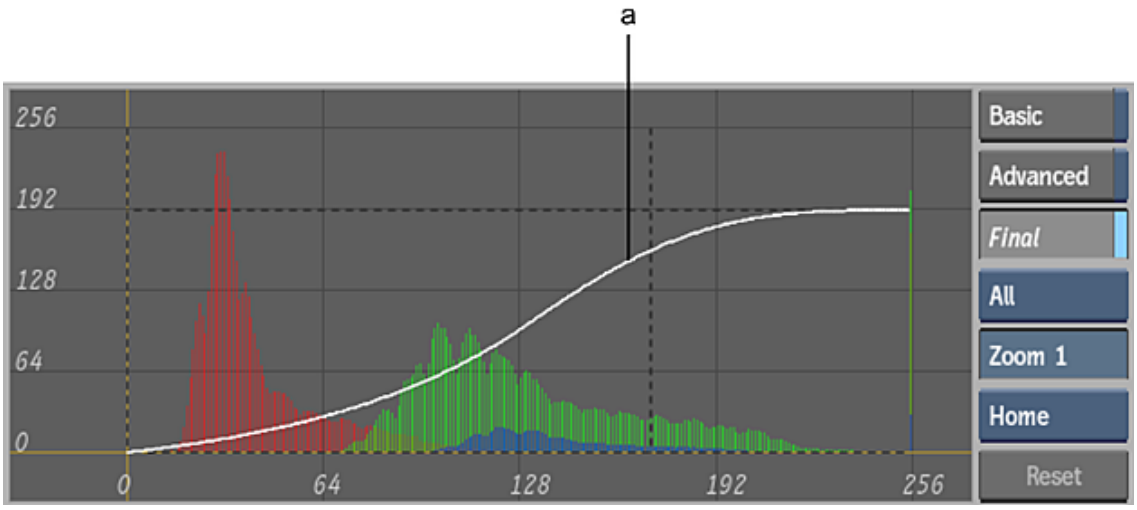
1. Do one of the following:
   - To set proportional custom R, G, and B gamma correction values, enable Prop, and then enter the value in one of the (left-to-right) R, G, and B Softclip fields.
   - To set independent custom R, G, or B gamma correction values, disable Prop, and then enter the value in the corresponding (left-to-right) R, G, or B Softclip field.



(a) Soft Clip fields

To soften the shoulder of the gamma correction curve using the LUT Editor graph:

1. Do one of the following:
   - To soften the shoulder of the combined R, G, and B gamma correction curves proportionally, select All from the Edit Curve box, and then `Shift`-drag the gamma correction curve left or right.
   - To soften the shoulder of the R, G, or B curves separately, select Red, Green, or Blue from the Edit Curve box, and then `Shift`-drag the gamma correction curve left or right.



(a) Softened shoulder

Advanced Editing of the Conversion LUT Curve

NOTE The procedure in this section does not apply to colour transforms or 3D LUTs.

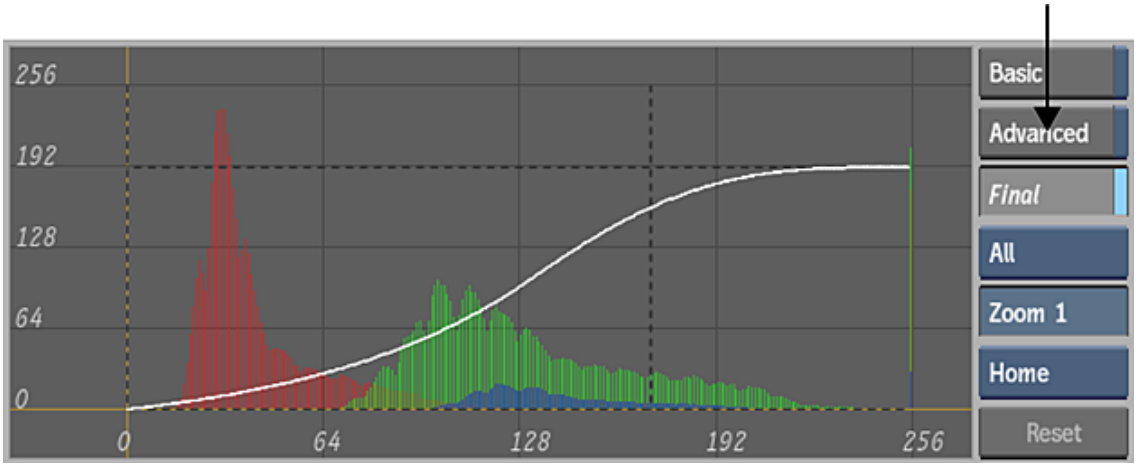
After you modify the basic conversion curves in the LUT Editor (page 1760), you may want to further refine the final result. To do so, you can use advanced editing tools. With advanced editing, you modify additional curves for the red, green, and blue channels to fine-tune the final conversion LUT and bring out or hide detail in specific areas. These curves have editable points, adding flexibility in how they affect each channel in the final conversion LUT.

You can modify the shape of each advanced editing curve using control points. Modifications you make to these curves influence the curve of corresponding channels in the final conversion LUT. For example, you can add points to the advanced editing curve for the red channel, and by dragging the points, affect the slope of the red channel's final conversion LUT curve.

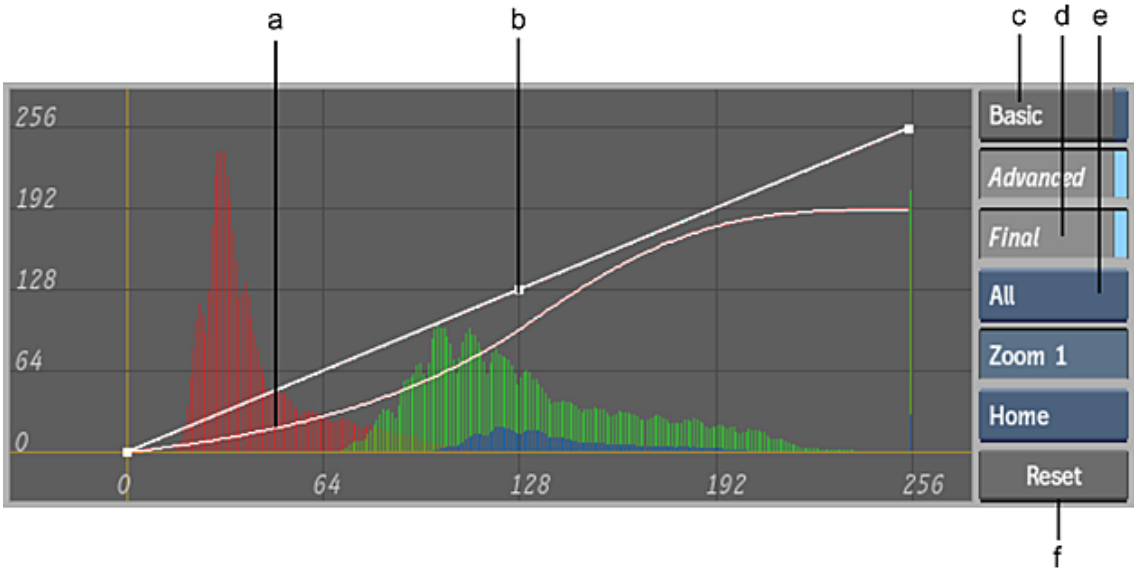
The modifications you make to the basic conversion LUT curves and the advanced editing curves are independent. The final LUT that you generate, however, is a single curve (or set of R, G, and B curves) that consists of basic LUT curves and adjustments you make with advanced editing curves.

To perform advanced editing of the conversion LUT curve:

1 In the LUT Editor, click Advanced.



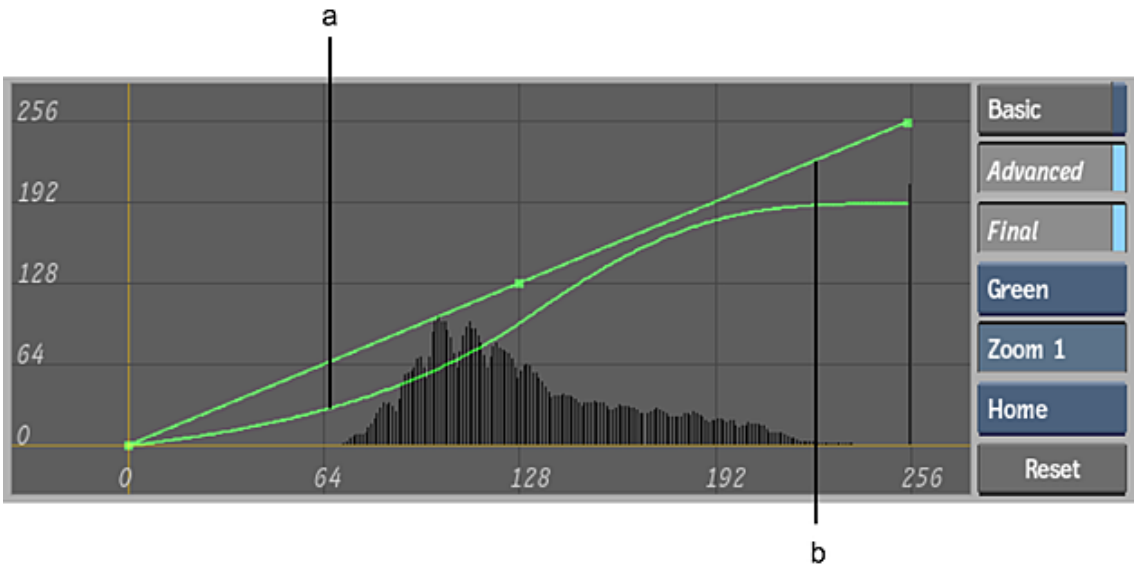
Additional advanced editing curves appear for the red, green, and blue channels. RGB curves use blending to distinguish overlapping curves. For example, when all RGB curves overlap, a white curve is produced, whereas green and blue overlapping curves produce a cyan curve.



(a) Red, blue and green overlapping advanced editing curves (b) Control point (c) Basic button (d) Final button (e) Edit Curve box (f) Reset button

2 To adjust the R, G, and B curves more easily, select one of the channels from the Edit Curve box.

The corresponding channel's basic and advanced editing curves are displayed. The histogram only displays values for the selected channel.



(a) Color channel selected in Edit Curves box (b) Color channel's advanced editing curve

3 To modify the advanced editing curves, use the Tools box.

Select:	To:
Add	Add control points to an advanced editing curve. With Add selected, click either the red, green, or blue advanced editing curve.
Delete	Delete control points from an advanced editing curve. With Delete selected, click a point on the red, green or blue advanced editing curve.
Move	Move control points. With Move selected, drag the points, or press Ctrl-drag to select multiple points. **TIP** Since the advanced editing curves use B-spline interpolation, the control points you manipulate will not necessarily rest on the curve itself. Points have a weighted influence on the curve depending on the position of the other points on the curve.
Zoom	Zoom in on the curves. With Zoom selected, drag over the curves right or left to zoom in or out.
Rect Zoom	Zoom in on a section of the curves. With Rect Zoom selected, drag a selection box to zoom in on the area of the curves contained by the selection box.
Pan	Pan the curves. With Pan selected, drag over the curves to pan the curves in any direction.

Dragging control points of advanced editing curves affects the basic curves and the colour effect is immediately reflected in the clip.

4 To view the basic and final conversion LUT curves, toggle the Basic LUT and Final LUT buttons.

5 To reset the advanced editing curves only, click Reset.

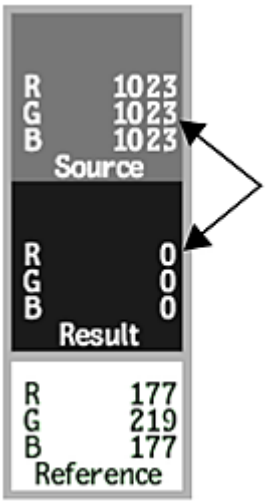
Comparing Colour Values Using Colour Patches

NOTE The procedures in this section do not apply to 3D LUTs.

While using the LUT Editor (page 1760), you may want to sample image channel values in the source, result, and reference clips to compare values of similar colours, or even corresponding pixels. The colour patches appear to the right of the LUT Editor graph.

To sample colours in the LUT Editor accessed from the Import Image menu:

1 Click the Source or Result Colour patch.



The cursor changes to a colour picker.

2 Click the image.

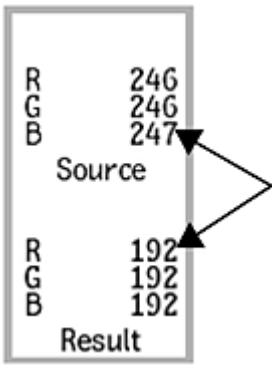
Both source and result colours are sampled simultaneously. The RGB values for the selected pixel are displayed, and the sampling boxes take on the corresponding colour.

3 To sample a set of reference values, click the Reference sampling box and use the colour picker to sample the image.

The values are taken from either the result or reference image, depending on which clip display option is currently selected.

To use colour patches in the LUT Editor menu in Batch orBatch FX:

1 Click the Source or Result sampling box.



The cursor changes to a colour picker.

2 Click the image.

Both source and result colours are sampled simultaneously. The RGB values for the selected pixel are displayed, and the sampling boxes take on the corresponding colour.

Roundtripping Clips Between Colour Spaces

In Batch, Batch FX, and the Modular Keyer, you can use the **Shift+T+click** combination to copy Colour Transform settings from one node to another while toggling the Invert option. For example, this makes it easy to convert content from ARRI LogC and other colour spaces to a single working space such as linear Rec. 709 or ACES, and convert back to ARRI LogC for grading and finishing, or to convert from your usual working space to a different one to perform a specific operation and then convert back again.

1 Set the Colour Management attributes in any node that supports them (i.e., Clip node, LUT Editor, Render node, Write File node, etc.).
   - Make sure that a colour transform or a custom chain of colour transforms is selected.
   - Make sure that Custom is enabled.
2 If you haven't already done so, add another node that supports the Colour Management attributes.
3 Make sure that the destination node is selected.
4 Press Shift+T and click on the node that you want to copy Colour Management attributes from.

Some colour transforms are impossible to invert perfectly. This happens when they map multiple input colours to the same output colour, such as when transforming from a large space to a smaller space. It is analogous to the fact that you cannot convert from a 12-bit signal to 8 bits and back to 12 bits again without data loss. In these situations, the Invert option maps the original output colour back to one of the original input colours, but applying the forward transform followed by the inverse transform will inevitably change some colours.

To check that the inverse is as accurate as possible, compare the result of applying just the forward transform to the result of applying the forward transform, followed by the inverse transform, followed by the forward transform again. If they match, then the inverse is accurate within the range of colors that may be inverted.

Exporting LUTs and Colour Transforms

Exporting a LUT or colour transform from the LUT Editor (page 1760) is different from saving a setup:

- PhotoMap, EXR Display, Gamma, Lin to Log, and Log to Lin are exported as baked 1D LUTs (*.lut* files). When imported, you can modify their curves only with the Advanced settings.
- A chain of colour transforms is exported as a single *.ctf* file that losslessly preserves each component operation. When imported, you cannot modify it.

Exporting a LUT

1 Access the LUT Editor.
2 Click Export.

The file browser appears.

3 Type a file name for the LUT in the Export field.
4 If you want to generate an inverse LUT with the *_inv* suffix appended to the file name, make sure that the Generate Inverse Lut option is active.
5 Click Export.

The LUT is exported to the specified location.

Exporting a Colour Transform

1. Access the LUT Editor.
2. Click Export.

   The file browser appears.
3. Type a file name in the Export field. The *.ctf* extension is added automatically.
4. If desired, type a nickname for the colour transform in the Nickname field. The nickname is displayed in the metadata panel of the LUT editor, and is also shown in the Viewer Colour Management pop-up (in the lower left of image windows) when loaded in your LUT preferences.

5. If desired, type a description for the colour transform. The description is displayed in the metadata panel of the LUT editor.
6. Select the destination location using the location shortcut under the EXIT Export LUT button:
   - **Shared** is a useful location for storing transforms that can be used by other Autodesk applications on the same workstation that use Autodesk Colour Management.
   - **Project** (/opt/Autodesk/project/<project name>/synColor/transforms) is a useful location for storing the transforms that are specific to the current project, for example, for archiving.
   - Saving transforms to the **Autodesk** location is not recommended.
7. Click Export.

   The colour transform is exported to the specified location.

Applying Colour Management to Displayed Images

When you apply colour management to displayed images, the colours on the monitor or projector are affected but the underlying colour values of the clips' pixels are not changed.

The colour management for displaying images is based on the concept of Viewing Rules. These rules allow the system to apply the correct viewing transform for media tagged with a given colour space.

Note that the colour management is now applied not only to most viewports but also to preview windows, thumbnails, the colour picker, and many colour pots/swatches. (Although Sparks and the Stabilizer are now colour managed, there are still a few other legacy viewports that are not.)

Viewing Rules

The Viewing Rules allow the system to automatically apply the correct view transform when displaying media. This is enabled by the fact that all media in the application is now tagged with a colour space identifier.

Viewing Rules are entirely customizable project settings that are configurerd in Colour Management Preferences. They are part of the colour management Policy that you may export to configure new projects.

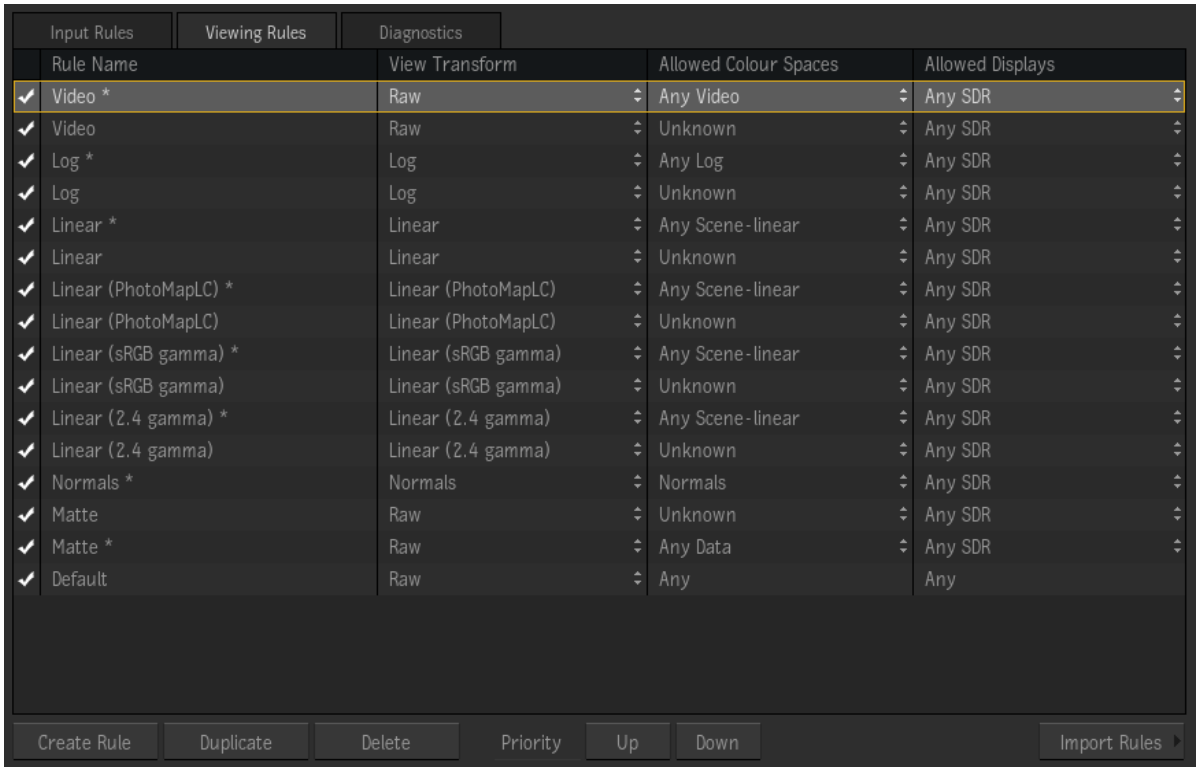
For a given view transform, a viewing rule specifies two essential characteristics:

- Which tagged colour spaces it is appropriate for

■ Which display devices it should be used with

When the application needs to display an image, the Viewing Rules table is searched for rules that are appropriate for the tagged colour space of the image and the given display being used (graphics or broadcast monitor). This list of applicable rules are shown in the viewport menu as the available options. (In other words, it is ok to define more than one rule for a given colour space and display pair.)

Viewing Rules that are organized higher in the rules table take precedence over lower ones. For any given tagged colour space and display, the highest priority rule is called the Primary Viewing Rule. The Primary rule is the one that is used by default.



Create a Viewing Rule as follows:

1. Press Create Rule to create a new rule.
2. Press and hold in the View Transform field and select the view transform to use. Your selection will auto-populate the other fields with default values.
3. Press and hold in the Allowed Colour Space field and select the tagged colour space the viewing transform should be used for. If you select one of the "Any" options, the rule will be used for any colour space in that category (e.g., "Any Log" would be used for any logarithmic colour spaces).
4. Press and hold in the Allowed Displays field and select the display to use the viewing transform with. Your may select one of the category options to use the rule for a family of displays of that type (e.g., High Dynamic Range video monitors, Standard Dynamic Range cinema projectors, etc.). If you select Current Broadcast or Graphics monitor, the field will be filled with the currently selected device.
5. Type a name for your rule in the Rule Name field. This is the name you will see in viewport menus.
6. Use the Up and Down buttons to set the priority of this rule relative to the other rules.

If a rule specifies one of the "Any" options in the Allowed Colour Space field, then the view transform will be available for use with a family of tagged colour spaces. If the colour space that the given view transform expects as input does not match that of the given image, the system automatically inserts the necessary transform to convert between the actual and expected colour space. Likewise, if one of the "Any" options is

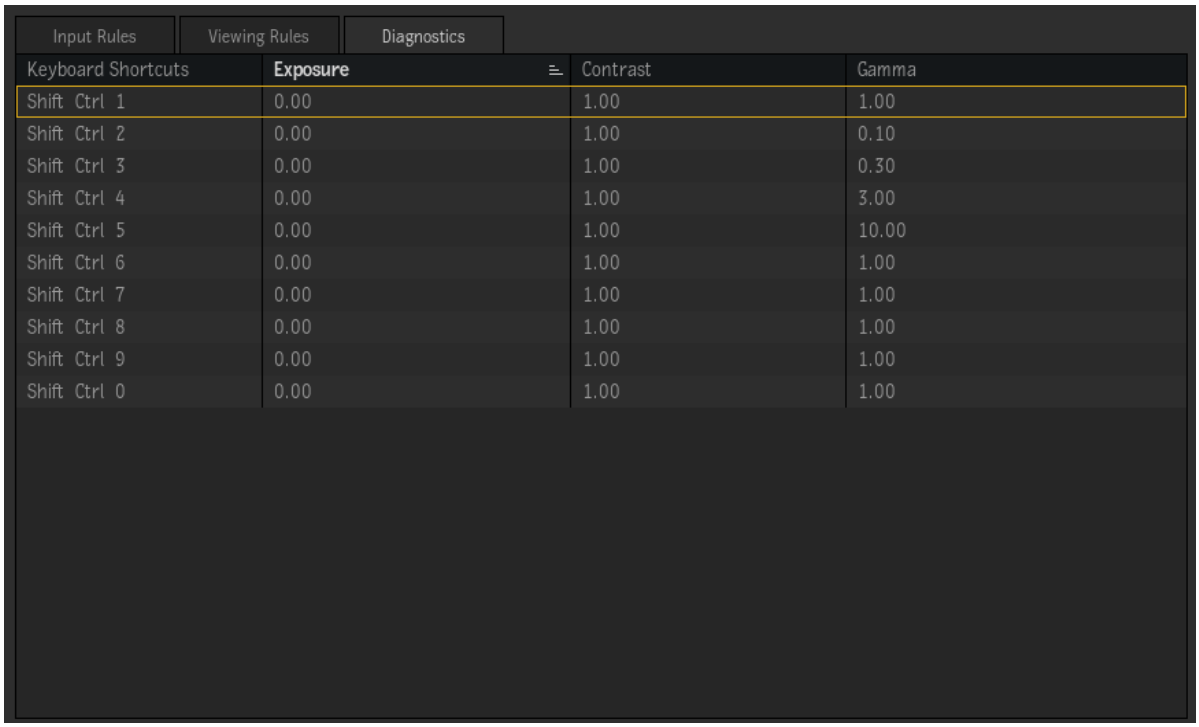
specified in the Allowed Displays field, then if the colour space produced by the given view transform does not match the display hardware currently being drawn to, the system automatically inserts the necessary conversion transform.

Note that the list of Allowed Colour Space options will be filtered based on the currently selected View Transform. Only colour spaces that are compatible with what the selected viewing transform expects are shown. (This is done to simplify the nature of any automatic conversion that would be needed from the selected Allowed Colour Space to the colour space expected as input by the given view transform.)

A rule may be temporarily bypassed by clicking in the left column to turn off the check mark.

The rules table always contains a Default rule at the bottom that matches Any colour space and Any display. The Default rule is only used if no other rules match.

Setting Shortcuts for Exposure, Contrast, and Gamma



In the Diagnostic tab of Colour Management Preferences, you may configure ten Exposure/Contrast/Gamma user presets for use in the Player and Viewports. The shortcut for these presets is Ctrl+Shift+# (1 to 10). These settings are saved as part of your User. (This feature is not supported in some of the older viewports.)

Controlling the Look in Viewports

When some transforms are applied as viewing transforms in a viewport, you can toggle portions of the transform on and off interactively. This can be helpful when working with looks.

Looks are basic colour grades that are created by a cinematographer to establish a visual mood for a scene during principal photography. However, the look must not be baked into the raw footage or visual effects because it is only the starting point for the final grading. Therefore, it is useful to be able to toggle the look on and off during finishing.

The Look on/off switch is available when:

- The current viewing transform in a viewport contains a reference to the Default Look Transform set in LUT Preferences.
- The current viewing transform is an ASC colour decision list (.cdl) file.
- The current viewing transform is a custom Colour Transform (.ctf) file that contains one or more operators with the LOOK_SWITCH dynamic parameter.

You can click directly on the Look switch at the bottom of the viewport to toggle the look on and off. Alternatively, you can use the Enable Look Transform button in the Image Display Viewer settings.

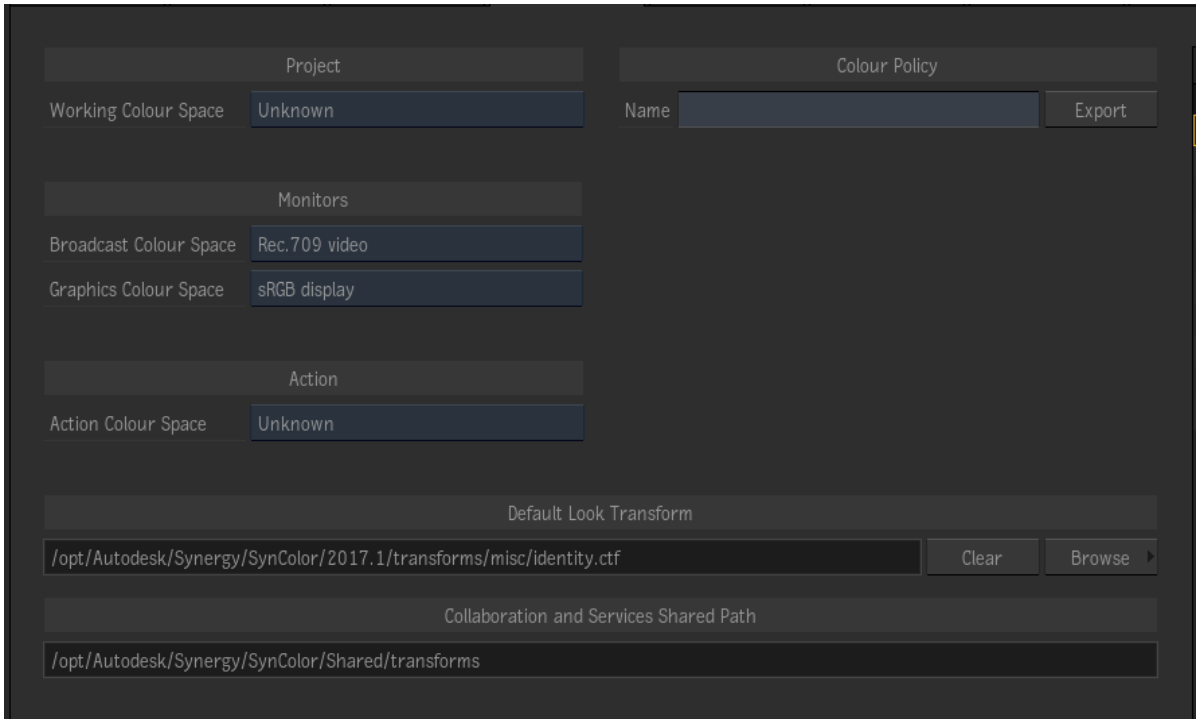
Setting Up the Default Look

1. In LUT Preferences, specify a transform as the Default Look Transform. You can select a transform file in any supported format.
2. In LUT Preferences, add one or more viewing transforms that reference the default look to the 3D LUT and Colour Transform list. The transforms that reference the default look include the following:
   - The ACES Output transforms in the view/ and RRT+ODT/ directories of the Autodesk location are available for displaying ACES material on various devices.
   - The ADX10_to_current-monitor and ADX16_to_current-monitor transforms in the film/ADX/ directory are available for displaying scanned film on the system or broadcast monitor.

   NOTE The default_look transform in the misc/ directory simply applies the default look transform, and the default_look-ACESproxy transform applies the default look transform in the ACESproxy color space that is typically used on set (first converting from the ACES color space and then back again). These transforms are not intended to be used directly as viewing transforms, but should be used as building blocks for creating your own custom chain of transforms.
3. Display an image in a viewport, and apply a viewing transform.

   The Look switch appears at the bottom of the viewport.

Project-Based Colour Policies



A Colour Policy provides a way to save colour management settings in a way that may easily be used to configure new projects. A Colour Policy contains the following:

- **Input Rules:** A way of automatically tagging sources with a colour space based on the file name.
- **Viewing Rules:** A way to have viewports automatically apply the correct viewing transform for a given colour space and display.
- **Project Working Space:** The default Working Colour Space for the project.
- **Action Colour Space:** The default Colour Space for Action output. This also defines the colour space used to convert Substance generated textures, as well as textures brought in via Action's Import node.
- **User Colour Spaces:** User-defined Colour Spaces, Viewing Transforms and Displays, created by you or your facility that are made available to the project.

Example Colour Policies

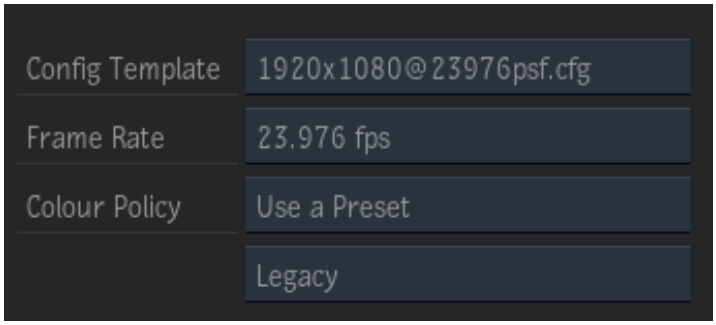
The application supplies several example Colour Policies:

- **Legacy:** A policy which is used for legacy projects and allows you to adopt the new colour management workflow at your own pace. The viewport behaves as before and media is tagged as Unknown (untagged). This is the default setting.
- **ACES 1.0:** An example of an ACES-only project configuration.
- **Rule Examples:** An example containing a wide range of different Input and Viewing Rules for you to experiment with.
- **Simple Linear Workflow:** A policy that configures for a basic de-gamma/re-gamma type workflow. This is also useful for projects that are mostly starting with Rec. 709 source material rather than camera Log or Raw.

NOTE We recommend you choose the example closest to how you want to work and then customize it (particularly the Input Rules) for your needs.

Selecting a Colour Policy when you Create a Project

When creating a project, you can select one of the example Colour Policies shipped with the application, use a policy you've previously exported, or copy the policy from one of the listed projects.



Colour Policy Mode button Select "Use a Preset" to configure colour management settings using a policy selected from the menu below. Select "Copy from another Project" to copy the colour management settings from the selected project.

Colour Policy List menu Initialize the colour management settings from one of these presets or copy the settings from another project.

Modifying a Project Colour Policy After Project Creation

All of the components of a Policy may be edited individually from within the Colour Management Preferences. However, if you need to completely replace the current settings with another Policy, you may do that by editing the project.

Note that when you create or edit the project, the files that comprise the chosen Policy are copied into your project directory (overwriting any existing files). Since the project settings use a local copy, any changes made to the source presets will not have any effect on existing projects.

Colour Management and Data Management

Archiving

When performing a Project Archive, all colour management settings are archived & restored.

When performing a Clips/Sequence Archive, the content's colour management settings are archived. When the content is restored to a different project, the colour management data of the content is also restored and the project colour management settings are updated accordingly:

- If the content requires a User Colour Space not available in the project, the original Colour Space is created and the project is updated.
- If the content uses an already existing Colour Space, the Colour Space of the current project is used.

NOTE If for any reason, the User Colour Space that was archived is different from the User Colour Space of the current project, the latter is used. This allows colour scientists or system administrators to update User Colour Spaces over time and not have artists manually changing the original User Colour Spaces on each clip & sequence. Modifying a User Colour Space referenced by already existing content may have side effects, such as:

- Changing a User Colour Space used by a clip with caching will not invalidate the cached media so you must manually flush the cache and then re-cache the media.
- Re-archiving content that is already part of an archive that references an updated User Colour Space is seen as the original archived content and is not re-archived.
- Network Transfer of clips and sequences that reference unavailable User Colour Spaces recreates the User Colour Spaces on the destination project.
- Media Export Presets: When you create an Export preset using a User Colour Space, a CTF of the User Colour Space is saved alongside the export preset.
- Media Export using a remote Wiretap Gateway: When exporting content using a remote Wiretap Gateway server, the Export jobs contain all the data to complete the encoding tasks on the remote server. There is no need to copy User Colour Spaces to the remote server.
- Rendering content using Burn: In a distributed rendering environment, where all Burn nodes share the same file system mount-points, no extra configuration is required.

User Colour Spaces from Legacy Versions

Because User Colour Spaces from the 2017 release can be used in 2017 Extension 1, but cannot be managed (i.e. archived), the naming of these Colour Spaces is displayed with a suffix of "non-archival". These Colour Spaces may appear in menus in:

- The Media Hub (Auto convert mode).
- The Colour Management node (Auto convert mode).
- The Input Rules (Input Colour Space menu).

Using a Centralized Location for a Workgroup

When working in a Workgroup with Flame Family products or collaborating with Wiretap-based services (i.e. Wiretap server, Wiretap Gateway, Media Converter), it is important to share the same colour management policies (including User Colour Spaces) for all the workstations to have access to the same data. There are many advantages of using a centralized location for colour management, such as:

- Project creation is easier because all Colour Policies are centralized.
- Remote applications and services share the same User Colour Spaces, Rules, etc.

NOTE Collaboration between workstations sharing content that do not use a Centralized Location might not work as expected. The following operations will not work:

- A Lustre workstation grading a sequence from a remote Flame will not be able to display content with project-based User Colour Spaces, unless the User Colour Spaces are copied to the project location.
- Burn.
- Media Converter used on a remote Wiretap Gateway will not be able to encode content with the specified colour management.

Configuring a Centralized Colour Management Location

1. Create a shared folder on a server. This can be /opt/Autodesk/ (which should already be exported) or any other location.

2 Make sure this folder is exported (modify /etc/exports and /etc/fstab) and that its permissions are set to full read/write.

3 On every workstation and server that is part of a workgroup, do one of the following:

- Use the Colour Management tab of the Flame Setup application to define the centralized shared location.
- Alternatively, you can edit the SynColor configuration file to set SharedHome to the centralized shared location, found here:

  ```
  /opt/Autodesk/Synergy/SynColor/<version>/synColorConfig.xml
  ```

  However, note that in this case you need to append "/transforms" to the actual intended centralized location.

4 Restart background services (Stone+Wire and Wiretap Gateway).

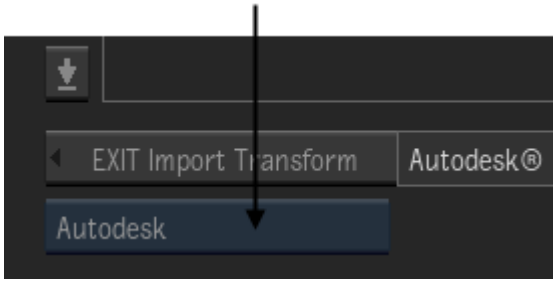
5 Start the applications and create a project.

NOTE

- Although the User Colour Spaces continue to be stored under the project, symbolic links are created in the centralized shared location (via /hosts/IP address/PROJECTHOME/synColor/policy/user_color_spaces) to ensure the content is available on all workstations and servers that are part of the workgroup.
- User Colour Spaces for any newly created projects will be available from the centralized shared location.
- Shared User Colour Spaces are also part of a Project Archive, even when using a centralized shared location.

Colour Management Files and Locations

When importing and exporting colour transofrms and LUTs you can use the location shortcut button at the lower left of file browsers to switch between the standard file locations.



You can use files located anywhere on your file system, but it is recommended you store your custom LUTs and colour transforms in either the **Project** or **Shared** location.

Colour Transforms

Colour transforms are based on the Academy/ASC XML colour transform format and have the *.ctf* extension. They are stored in the following locations:

- **Shared** (/opt/Autodesk/Synergy/SynColor/Shared/transforms by default) is a useful location for storing transforms that can be used by other applications that use Autodesk Colour Management.
- **Project** (/opt/Autodesk/project/<project name>/synColor/transforms) is a useful location for storing the transforms that are specific to the current project, for example, for archiving.
- **Autodesk** (/opt/Autodesk/Synergy/SynColor/<version>/transforms) contains the transforms installed with Flame. See Autodesk Color Transform Collection (page 1808).

As of the 2017.1 release, the Autodesk Transform Collection is a superset of all previous releases. So it is no longer necessary to keep previous transform collections installed on a workstation in order to load archives created in previous releases.

1D LUTs

Legacy Discreet 1D LUT files should have the *.lut* extension should be stored in the **Project** (/opt/Autodesk/project/<project name>/lut) location.

3D LUTs

Legacy Discreet 3D LUT files have the *.3dl* extension.

The 3D LUTs installed with Flame are in the **Lustre Colour** (/opt/Autodesk/Lustre_Color/lut/Lustre_Color_3DLUTs) location. Use the **Project** (/opt/Autodesk/project/<project name>/lut) location to store 3D LUTs specific to a particular project.

Graphics and Broadcast Monitor Settings

Graphics and Broadcast monitor settings are considered workstation (rather than project) settings and are stored in:

- /opt/Autodesk/Synergy/SynColor/<VERSION>/synColorConfig.xml.

(This file is system specific even with Central Install.)

Example Colour Policies

The example Colour Policies shipped with the applications are located here:

- /opt/Autodesk/presets/2017.1/colourmgt/policies (Linux)
- /Applications/Autodesk/ presets/2017.1/colourmgt/policies (Mac OS)

Project Colour Policy Files

The Colour Policy files for a project are stored in the **Policy home folder:**

<PROJECTHOME>/synColor/policy/

- **policy.cfg:** This file defines the Working Colour Space and Action Rendering Colour Space.
- **synColorFileRules.xml:** This file contains the Input Rules.
- **synColorViewingRules.xml:** This file contains the Viewing Rules.
- **user_colour_spaces:** This folder contains all the User-defined Colour Spaces created for the project, for example, when creating a new User Colour Space or when importing content that contains a User Colour Space (i.e Network Transfer, Archiving, etc).

These files are all editable in a text editor.

See also: Colour Management and Data Management (page 1782)

Autodesk Color Management

About Autodesk Color Management

Autodesk Color Management (also known as the Synergy color management component, or SynColor) is a shared technology component that is integrated into several Autodesk applications. This allows for consistent processing, interpretation, and communication of colors throughout a mixed pipeline.

Autodesk Color Management is designed to support a variety of color management methodologies, including ACES, ICC, OpenColorIO, and ASC CDL. It allows you to work with different color spaces and encodings so that you can adopt new workflows or emulate legacy ones.

Autodesk Color Management consists of a color engine together with a collection of transformations suitable for input, output, display, and other situations. The transforms are provided as separate files in .ctf format, an extension of the Academy/ASC XML color transform format. You can combine multiple files to create complex transformations, and in addition you can author your own files for custom purposes.

The color engine supports a wide variety of color operations, including 1D look-up tables (LUTs), 3D LUTs, gamma, log/antilog, exposure-contrast, matrix multiplication, and more.

In addition to native .ctf files, Autodesk Color Management can import many common color transform file formats, including the legacy Autodesk .lut and .3dl formats as well as third-party formats such Cinespace, Iridas, Pandora, and Nuke.

Implementation in Specific Products

Not all features are available in every application that supports Autodesk Color Management. In addition, the implementation and workflows are different for different applications.

For example, in Flame Premium and other Creative Finishing products, you can specify the exact color transform or chain of transforms that you want to apply. You can then export a chain as a single file for reuse. However in Maya, you generally specify the desired color spaces, and the system applies the transforms that are necessary to convert between those color spaces.

For information about how to work with the features available in a specific Autodesk application, see the documentation for that application.

Color Management Concepts

Color management enables enables colors to be reproduced as accurately as possible across different devices and media. Unlike color correction, the intention of color management is not to change colors but instead to preserve perceived colors in different situations.

What Is a Color Space?

A color space allows people and software to communicate colors unambiguously using a numeric representation.

A triplet of code values such as [0.506, 0.266, 0.266] by itself is not enough to specify a color. Those code values must be interpreted with respect to a particular color space. The color represented by those three numbers will be different in different color spaces.

There are two categories of color space:

- Device-independent color spaces describe colors in absolute terms.

- Device-dependent color spaces depend on the characteristics of specific hardware.

Device-independent Color Spaces

To fully match a triplet of code values to a specific color, a device-independent color space must define the following characteristics:

- The meaning of the three primary values in terms of CIE colorimetry.
- One or more data types and encodings.
- The image state.
- The associated viewing conditions.

Examples of device-independent color spaces include ACES and the ICC Profile Connection Space.

Primary Values

The primary values can be thought of as the coordinate axes used to define a color "point" in a color space. Device-independent color spaces define their primaries with reference to CIE colorimetry values — in that respect, CIE colorimetry provides a kind of universal reference frame or "world" coordinate system for converting between color spaces. In a given viewing environment, two colors with the same colorimetry will look the same to a typical human observer.

Some examples of different systems of primaries include:

- The coordinates of red, green, and blue specified by ITU-R BT.709 (also known as "Rec. 709") for HD video. These primaries are also used for sRGB (which has a different gamma).
- The coordinates specified by ITU-R BT.601 (Rec. 601) for SD video.
- The "P3" primaries specified by DCI and SMPTE for digital cinema projectors.

Data Type & Encoding

To interpret the numeric code values, it is necessary to know the data type of the numbers, for example, whether they are meant to be 8-bit, 10-bit, 12-bit, or 16-bit integers, or floating-point values. In addition, it's necessary to know the values' encoding, that is, whether the code values represent intensities on a linear scale or a logarithmic scale, and whether gamma has been applied.

Image State

The notion of "image state" is a standard framework (ISO 22028-1) for grouping color spaces that share similar characteristics and which require similar processing. There are three main image states.

- Scene-referred images are high-dynamic-range images. They use code values that are proportional to the luminance or radiance in the scene, whether that is a live set or a virtual scene in a 3D application like Maya. No tone-mapping has been performed, and code values greater than 1.0 are allowed. If the code values are encoded on a linear scale, then the images are called scene-linear. Most OpenEXR files are scene-linear.
- Output-referred images are normal dynamic range images. They have been tone-mapped, for example, using an S-shaped curve to compress super-whites as well as increase contrast to compensate for viewing conditions. The maximum code value is 1.0 (after normalization in the case of integers), and the values are not proportional to luminance in the original scene. Output-referred images are theoretically ready for display. However, this does not necessarily mean that they are ready for display on a specific device — for example, they may have been tone-mapped but still require a specific gamma for display on a particular monitor. Examples of output-referred images include sRGB, HD video, digital cinema (DCI), and so on. Output-referred images are also called "display-referred".

- Intermediate-referred images are somewhere between scene-referred and output-referred. They have had some color processing performed, so the code values are not proportional to scene luminance, but they are still not ready for display. Examples of intermediate-referred images include log encodings like Cineon-style film scans, Academy Density Exchange (ADX), some digital cinema camera outputs, and so on.

There is some confusion about how to convert between image states in the context of a "linear workflow" for CG rendering and compositing. Much of the confusion comes from the word "linear" — there are actually different kinds of linear encodings: scene-referred and output-referred. It is extremely important to understand the difference between scene-linear images and output-linear images (also called "linearized output-referred").

In both cases, the encoding is proportional to luminance — in other words, no gamma encoding has been applied. However, in scene-linear images, the values are proportional to the luminance of the scene but in output-linear images, the values are proportional to the luminance of the display. The mathematics that are used to render computer graphics assume the color space is linear, and almost always this means a scene-linear color space rather than an output-linear one.

To prepare a scene-linear image for display, you need to do more than simply apply a gamma encoding. Because the scene-linear image has a high dynamic range and will be viewed on a device with a limited dynamic range in a different viewing environment, you need to apply a tone map before the gamma encoding to produce an image that looks correct.

Conversely, to convert a video image to scene-linear, it is not sufficient to simply remove the gamma encoding. You also need to apply an inverse tone map to restore the luminance values of the original scene.

There is one notable exception: if an output-referred image is used as a texture to control diffuse reflectance or a similar property, then it might not be suitable to apply an inverse tone map. See Color Managing Textures and Maps (page 1801).

To make matters more confusing, video images are also sometimes called "linear" (as opposed to "log"). Video images are actually output-referred with gamma, so they must have their gamma encoding removed to become output-linear, and then must be "untone-mapped" to become scene-linear.

Viewing Conditions

Because human vision is adaptive, the appearance of color stimuli depends on the viewing environment. For example, a piece of paper will appear to be "white" under both bright daylight and a dimmer tungsten light bulb, even though it is lit by different amounts and hues of light.

Aspects of the viewing environment that control the appearance of a color include:

- The absolute luminance level of the image or scene. For example, a white shirt seen outdoors might have a luminance of 30,000 candelas per square meter, but its reproduction in a cinema might be only about 30 candelas per square meter — a factor of 1000 times dimmer.
- The "surround", that is, the color and brightness of objects in the field of view around the image. For example, the surround is dark in the case of cinema in a theater, dim for television in a home environment (or rather, it should ideally be dim), and normal (or none) for a real-world scene instead of an image.
- The adaptive white point. This is the color that is considered "white" after an observer has adapted to a given viewing environment.

The huge difference in absolute luminance level and surround between a typical outdoor daylit scene and a cinema or television viewing environment is one of the reasons that tone-mapping must include a contrast boost to scene-linear colors to make them look good on a projector or display.

The adaptive white point can be specified in one of several ways. One way is to refer to the chromaticity of a standard illuminant, such as illuminant A or the D series (D50, D55, D65, and D75) all specified by CIE. Another way is to refer to the correlated color temperature (CCT) as measured in kelvins (K). A third way is to specify the chromaticity coordinates — for example, the DCI/SMPTE calibration white for digital cinema is CIE {x = 0.314, y = 0.351}.

To compensate for differences in the adaptive white point between two environments, a chromatic adaptation transform is used to preserve the color appearance. For example, a chromatic adaptation that converts colors intended for a D65 display to the equivalent colors for a 9300K display must increase the saturation of the reds.

Device-dependent Color Spaces

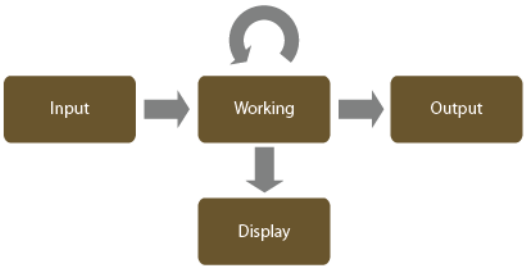
Device-dependent color spaces rely on the characteristics of a particular camera, monitor, projector, printer, or other device. Sending the same numeric color code value to a digital cinema projector as well as to a motion picture film recorder will result in different colors.

However, devices can be characterized. Characterization involves precisely measuring their responses in terms of absolute colorimetry. In this way, characterization provides a means to convert between device-dependent and device-independent color spaces. sRGB and AdobeRGB are essentially virtual device-dependent spaces that have been characterized well enough to use them as if they were device-independent.

In order for the characterization to remain valid, a device must be calibrated. Calibration involves adjusting the device to meet the "aim" (that is, the intended primaries, white point, and gamma) corresponding to that characterization. This process must be repeated periodically because devices' responses drift with use over time. For more information, see Calibrating Your Monitor (page 1797).

Color Management Workflow

Color management involves converting color values between color spaces. This conversion happens at specific points along a color pipeline.



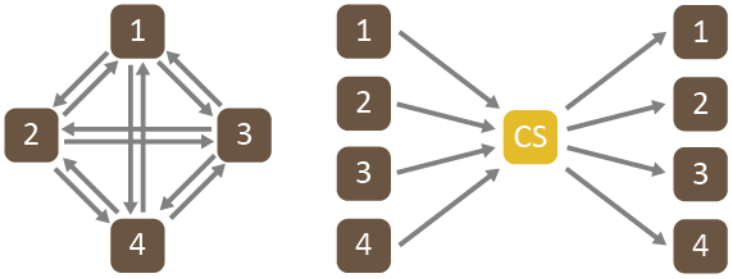
- When importing media, color values must be converted to a common working space so that color information can be properly combined with other sources. For example, to combine on-set footage with 3D CG rendered elements and a painted matte background, the images should be in the same color space. If the images originated in different color spaces, they require different input transforms to convert them to a common working space.
- While working, color values in your current working space must be converted to a color space that is appropriate for your system's monitor. For example, if you are working in a scene-linear color space, then you need to apply a transform that tone-maps the images and converts them to values that are appropriate for your display. This display transform is not baked into the underlying color values, but is only applied on-the-fly for previewing.
- Also while working, you may want to switch color spaces to perform specific operations. For example, some operations like compositing and blurring work best in a scene-linear space, but other operations like noise and grain operations, or tracking and stabilization, work best in a video or log space. When performing conversions like these, it's best to use an invertible transform to preserve the original color information. In addition, you need to change your display transform to match the new working space.

- Finally when outputting, you need to convert colors from your working space to the color space that is appropriate for your deliverables. For example, SD video, HD video, and digital cinema all use different color spaces, and require different output transforms.

Converting Between Color Spaces Using Connection Spaces

It is often necessary to convert between color spaces several times along an image pipeline. For example, you may need to convert several inputs from different spaces to a common working space so that they can be composited and blended, and then convert to another space for output.

To minimize the number of different color transforms that are needed to convert from each color space to every other one, it is common to use one color space as an intermediate or *connection space*.



For example, to directly convert back and forth between 4 color spaces, you would need 12 different transformations. Adding a fifth color space would require 8 more, or 20 in total. In general, adding an Nth space requires an additional 2(N – 1) transforms, or N^2 – N transforms in total.

However if you use a connection space, you only need 8 transformations to convert back and forth between 4 color spaces. Each additional color space requires only 2 additional transforms, or 2N transforms in total. But note that each conversion now requires two steps: the first to convert from the source space to the connection space, and the second to convert from the connection space to the target space.

Autodesk Color Management includes transforms that allow you to convert back and forth between color spaces using one of two connection spaces:

- ACES can be used as a connection space for scene-linear images.
  - The camera/ directory contains transforms for converting from various digital camera formats.
  - The film/ADX/ directory contains transforms for converting to and from Academy Density Exchange format.
  - The interchange/ACESproxy/ directory contains transforms for converting to and from the ACESproxy system used on set.
  - The RRT+ODT/ directory contains transforms for converting to and from various output-referred spaces.

- CIE XYZ can be used as a connection space for output-referred images, that is, for images that do not require any tone-mapping. The transforms in the collection include any chromatic adaptation necessary to convert to and from the CIE XYZ white point of D65, if required.
  - The display/ directory contains transforms for converting to and from various RGB color space encodings for specific display devices.
  - The interchange/ directory contains transforms for converting to and from color spaces commonly used for digital image exchange, including sRGB and AdobeRGB.

NOTE The transforms in the primaries/ directory convert only the primaries. They expect and produce a linear color space encoding, and they can be applied to either scene-linear or output-linear values.

ACES

The Academy Color Encoding System (ACES) is an image interchange framework being developed by the Science and Technology Council of the Academy of Motion Picture Arts and Sciences.

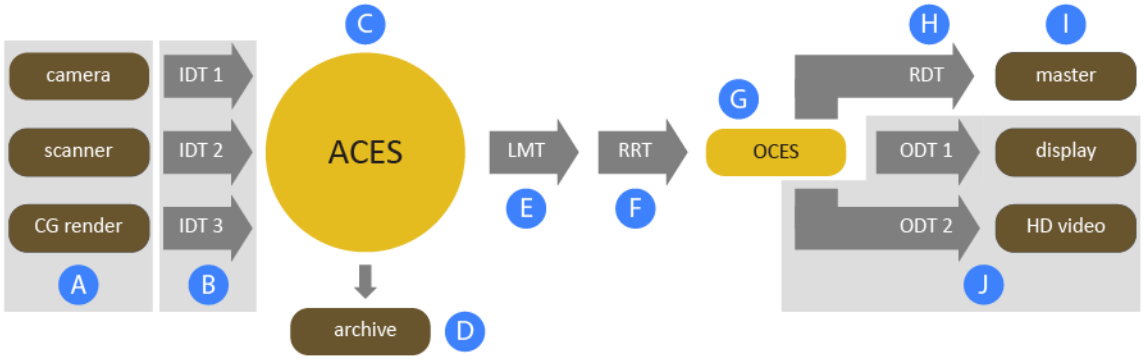
At the core of the ACES system is the Academy Color Encoding Specification (also known as ACES). This is an extremely wide-gamut scene-linear space that can encode any visible color without using negative values.

The ACES system also includes a set of color transforms that can provide an innovative color managed workflow. In particular, the ACES system provides the first standard high-quality tone-mapping algorithm for converting scene-linear images to output-referred color spaces for viewing.

The ACES color space, the ADX color space, and the constrained OpenEXR container specifications are published in the 2065 family of SMPTE standards.

The ACES Workflow

The ACES workflow uses standardized input and output transforms for each device.



A Inputs from different sources may use different color spaces and encodings.

B Input Device Transforms (IDTs) convert images to ACES. IDTs may be supplied by device vendors, software vendors such as Autodesk, other third parties, and AMPAS itself. In addition, you can author your own.
The ADX to ACES transform is known as the "universal un-build". Its inverse is the "universal build".

C The Academy Color Encoding Specification (ACES) is a scene-linear encoding and can be used as both a connection space and working space. The reference viewing environment is outdoor daylight with a white point of D60.

D ACES images can be saved for exchange or archiving in a constrained, or limited, version of the OpenEXR file format with extra metadata. The only allowable channel layouts are

stereo and non-stereo RGB and RGBA. This is a SMPTE standard.

E The Look Modification Transform (LMT) is optional. It may be inserted before the RRT to provide an aesthetic effect, such as a shot-specific color grade that needs to be shared throughout a workflow. It is typically established by the cinematographer on set. It can be baked in to the final output, or simply used for dailies and as a reference for the final color grade.

F The reference rendering transform (RRT) applies a transformation as a first step in preparing the images for viewing. Among other things, the RRT:

- accommodates for the difference between outdoor and cinema viewing environments.
- applies color adjustments associated with pleasing image reproduction.
- corrects for normal amounts of flare in images.

There is only one RRT allowed in the ACES 1.0 standard. However, the Autodesk Color Management transform collection also includes beta versions of the RRT to support older projects that may be using them.

G The Output Color Encoding Specification (OCES) represents the desired output colors if there were no dynamic range limitations. These values are the colors you would project in a cinema viewing environment if you had an idealized wide-gamut, high-dynamic range display device.

H The reference device transform (RDT) converts images from OCES so that they are viewable on a SMPTE reference projector. The RDT is used as a reference for developing ODTs for other devices.

I The output of the RDT forms the digital cinema distribution master (DCDM).

J Different output device transforms (ODTs) are applied to prepare the images for display on other devices. As with IDTs, the ODTs may be supplied by device vendors, software vendors such as Autodesk, other third parties, and AMPAS itself.

ACES in On-set "Look" Workflows

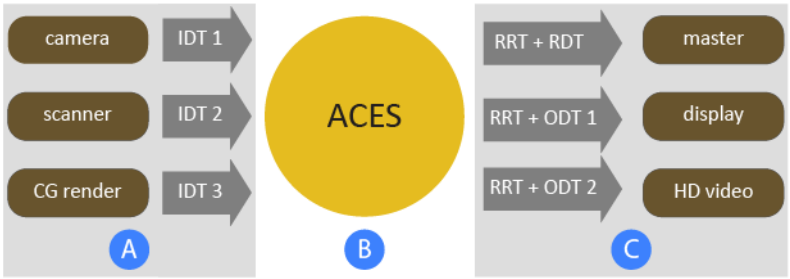
When working on set, it is generally not possible to use floating-point values for monitoring. To solve this issue, the ACES system includes an integer encoding that can be transmitted over an HD-SDI link for on-set monitoring, called ACESproxy. This is a lower-quality ACES encoding that should never be stored, but only used for on-the-fly previews.

Also on set, it is becoming common for the cinematographer to establish a "look", or basic color grade, so that the production team can see an approximation of the intended final grade early in the process. It is typically baked into dailies and footage for editorial. This look also serves as an initial reference for the final grade during the digital intermediate (DI) process. Because the effect of the look transform depends on what color space it is applied in, it is important to specify the intended space as part of the workflow.

The ACES system includes a standard for applying the look on-set in ACESproxy space, and then applying it again in DI such that the original look is preserved. This is the intended workflow for using ASC CDL looks.

ACES in Autodesk Color Management

The output transforms supplied with Autodesk Color Management combine the RRT and ODT, as well as the optional LMT, to convert directly to a format suitable for specific devices.



A	Autodesk Color Management ships with a variety of IDTs. These transforms can be found in the camera and film categories. In addition, you can use IDTs from other sources.
B	ACES can be used as a working space, or as a connection space for your chosen working space.
C	There is a variety of combinations of RRT and ODT available in the RRT+ODT/ directory. The combination of RRT and RDT to render the DCDM is the ACES_to_DCI-D60 transform.

Most of the transforms in the RRT+ODT/ directory include a reference to the look transform (LMT). The transform currently set as the defaultLook alias in the Autodesk Color Management preferences is applied before the RRT. The look gets applied in ACESproxy space as recommended by the ACES standard.

All internal processing is performed at 32-bit floating-point precision, so the conversion to ACESproxy for application of a look transform (and then back to ACES) avoids the quantization and clipping imposed by the integer-based implementation of ACESproxy used on set.

The RRT+ODT/ transforms apply the look transform in such a way that the look is not applied for the final output, but can be activated for viewing. Some Autodesk applications, such as Smoke and Flame Premium, allow you to toggle the look transform on and off interactively for display. This provides a way to quickly preview the on-set look in VFX, in editorial, and as a starting point for final color grading.

If you need to bake in the look for output, you can use a custom color transform to apply the default_look-ACESproxy.ctf transform from the misc/ directory before the RRT+ODT transform. See Custom Color Transforms (page 1805).

Choosing a Working Color Space

Autodesk Color Management lets you use any color space as your working space, and provides transforms to support many of the common and standard color spaces. You can use these transforms to convert images to your chosen working space on import, and to convert images to a specific working space for particular operations.

In general, it's best to use a wide-gamut color space for working. This ensures that you can properly represent as many colors as possible. For example, digital cinema (DCI) contains colors that cannot be represented using the Rec. 709 primaries (used by sRGB and HD video) without using negative values. Autodesk Color Management is able to preserve negative values, but other tools may not. In spite of this drawback, the Rec. 709/sRGB primaries are still the most commonly used for scene-linear working spaces.

ACES is an extremely wide-gamut color space that can represent any visible color. It can be used as a working space, but some color operations may give unexpected results because the gamut is so wide.

The P3, ProPhoto-RIMM, or UHDTV (Rec. 2020) primaries are good in-between choices. They have wide gamuts, but they are not too wide.

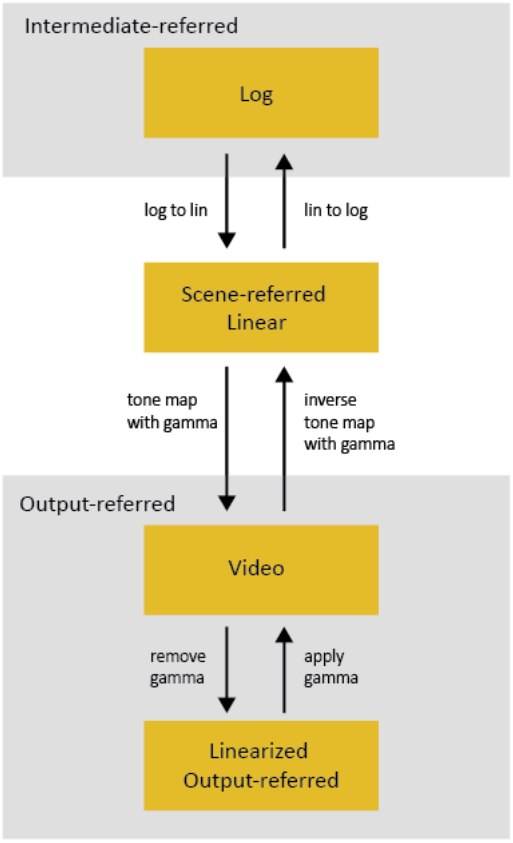
Some operations work best with colors in a specific type of space: scene-linear, video, or log. As you work, you may want to switch color spaces to perform certain operations, and then switch back to your main working space or a different space for other operations. For these situations, it's best to use a fully invertible transform to preserve as much color information as possible.

In general:

- You can convert scene-linear to video by applying a tone map such as PhotoMap_gamma_2.4 in the tone-map/ directory. In general, a tone map uses an S-shaped curve to increase the contrast and compress highlights.
- You can convert video to scene-linear by applying an inverse tone map such as inversePhotoMap_gamma_2.4 in the tone-map/ directory. This will not be exact if the footage was originally shot as video or converted with a different tone map, but it can be a good approximation that works well in many cases. In particular, note that simply removing the display gamma from video images produces linearized output-referred values rather than scene-linear values - there are no values above 1.0.

  NOTE There are special considerations for images that are used as textures and other maps. See Color Managing Textures and Maps (page 1801).

- You can convert scene-linear images to log space using one of the Lin_to_Log transforms in the film/LogLin/ directory.
- You can convert log images to scene-linear using one of the Log_to_Lin transforms in the film/LogLin/ directory.
- You can convert video images to log space by first converting to scene-linear and then to log.
- You can convert log images to video space by first converting to scene-linear and then to video.



Operations That Work Best with Scene-linear Colors

Some operations work best with scene-linear colors, that is, with code values that are directly proportional to light energy in the scene. These operations include:

- compositing and blending
- optical effects, including lens blur and defocus operations
- motion blur
- anti-aliasing
- resizing
- sub-pixel repositioning
- 3D rendering
- lighting and re-lighting

Operations That Work Best with Video or Log Colors

Some operations work best with video or log colors. These operations include:

- many color correction operations
- vectorscopes and histograms
- tracking and stabilizing
- grain and noise operations
- unsharp masking

- video transitions
- making gradients

Operations That Require Video Colors

Some operations require video colors, that is, colors that are restricted to the range [0, 1]. These operations include:

- color inversion
- converting RGB to HLS, HSV, or YCbCr

Common Color Management Scenarios

Although there's an unlimited number of possible situations that you may encounter in color management, they can be divided into categories that require similar treatment.

Color Managing Images on Input

When importing images, you should apply color transforms to convert them to a connection space and then to your chosen working space. Different types of images require different transforms.

In general, you need to apply two transforms to input images:

1. Convert images from their source space to a connection space. Several transforms are available:
   - Use the transforms in the camera/ directory to import images from various digital cinema cameras and convert them to ACES.
   - Use the transforms in the film/ directory to import scanned film plates in ADX or another Cineon-style log encoding and convert them to ACES.
   - Use the transforms in the display/ directory to import video images in HD, SD, or DCI formats and convert them to linearized output-referred CIE XYZ.
   - Use the transforms in the interchange/ directory to import other common color spaces and convert them to linearized output-referred CIE XYZ.
2. Once images have been imported and converted to ACES or CIE XYZ, you can use the other transforms to convert them to your working space. (However, keep in mind that a tone-map transform is needed when converting between scene-linear and output-referred color spaces.)

For example, suppose that you want to work with scene-linear values using the ProPhoto primaries. First convert the inputs to ACES, then apply ACES_to_CIE-XYZ followed by CIE-XYZ_to_ProPhoto-RIMM from the primaries/ directory. If desired, you can export a custom .ctf file that applies all three transforms in order.

On the other hand, if you want to import raw digital cinema camera footage and work with it using video-encoded values, you can import the images using the appropriate transform from the camera/ directory and then apply one of the transforms in the RRT+ODT/ directory.

If you need to convert Digital Cinema Distribution Master (DCDM) values to HD video, you can apply DCI_to_CIE-XYZ from the display/DCDM/ directory followed by CIE-XYZ_to_HD-video from the display/broadcast/ directory assuming that the DCDM was mastered to the DCI calibration white point. However if this results in unequal RGB values for colors that are supposed to be neutral, then you may need to replace the first transform with one that is relative to a different white point, such as DCI-D65_to_CIE-XYZ (see White Point Conversion (page 1803)).

Color Managing Images for Display

When working with images, you need to transform them from your working space so that they appear correctly on your display. This involves defining your monitor transform and then converting from your working space. In addition, your monitor should be calibrated periodically.

Defining the Monitor Transform

The display/ directory contains transforms that you can define as your graphicsMonitor alias (and broadcastMonitor alias, if appropriate) either in your application preferences or in the SynColor configuration file.

As an alternative to the transforms in that directory, you can use an ICC (International Color Consortium) profile as your graphicsMonitor or broadcastMonitor alias. ICC profiles are the standard method of color managing monitors in the computer industry. This is especially true on MacOS and Windows where it is integrated with the operating system, but even on Linux it is typically possible to obtain an ICC profile for a given monitor. Third-party monitor-calibration products also typically generate ICC profiles.

Because ICC profiles are so common, it is not only convenient to use them as your monitor transform, but it also provides an easy way of interoperating with other software that uses ICC color management (such as Adobe products).

Converting from the Working Space

Once you have defined your graphicsMonitor and broadcastMonitor aliases, you can then apply a viewing transform that references the currentMonitor alias. The currentMonitor alias will use either graphicsMonitor or broadcastMonitor, depending on which display is being drawn to.

The following transforms are available:

- display/broadcast/HD-video_to_current-monitor
- display/CIE-XYZ_to_current-monitor
- film/ADX/ADX10_to_current-monitor
- film/ADX/ADX16_to_current-monitor
- RRT+ODT/ACES_to_current-monitor

If your images are not in the appropriate working space for one of those transforms, then you can export a custom .ctf file that performs the required conversions:

- If your images are scene-linear but not in ACES, then you can convert to ACES and then apply ACES_to_current-monitor. For example, if you are working with scene-linear ProPhoto values, use ProPhoto-RIMM_to_CIE-XYZ followed by CIE-XYZ_to_ACES from the primaries/ directory, and then apply ACES_to_current-monitor.
- If your images are in log space, apply one of the transforms from the film/ directory to convert them to scene-linear before converting them from scene-linear as above.
- If your images are video-encoded, convert them to CIE XYZ and then use the display/CIE-XYZ_to_current-monitor transform.
- To view a DCDM, apply DCI_to_CIE-XYZ from the display/DCDM directory, followed by CIE-XYZ_to_current-monitor. Note that you might need to use a different transform such as DCI-D60_to_CIE-XYZ if a different creative white point was used for the DCDM.

Calibrating Your Monitor

To ensure that the displayed images are an accurate representation of the intended colors, your monitor must be calibrated. Calibration must be performed periodically because the monitor's response drifts with use over time.

With respect to calibration, there are two classes of computer monitors:

- Some high-end monitors have internal calibration circuitry. These monitors can be calibrated to emulate another monitor that uses a different set of primaries. You use the manufacturer's software to adjust these monitors to your desired aim.
- Other monitors must be calibrated externally using a third-party product to generate an ICC profile that you can use as your monitor transform.

When working in a video color space for video deliverables on a wide-gamut monitor, there are two possibilities for getting an accurate preview of the final output:

- If your monitor has internal calibration circuitry, you can calibrate to a video aim. For example for HD video, you can calibrate your monitor to the Rec. 709 primaries, D65 white point, and 2.4 gamma. In this case, you do not need to apply any color management for display.
- Alternatively, you can leave your monitor calibrated to a wide-gamut aim, and use the HD-video_to_current-monitor transform from the display/broadcast/ directory as your viewing transform.

Calibrating Autodesk Systems

For Autodesk Creative Finishing products that ship with an EIZO monitor (for example, CG245, CG246, or CG277), you can use the manufacturer's software for calibration. The monitor transforms in the display/Eizo directory assume that the monitor has been calibrated to the factory defaults: native primaries, D65 white point, and a gamma of 2.2.

However, these are wide-gamut monitors, and it is sometimes desirable to calibrate to a different aim. In such cases, you should apply a color transform that converts from the working space to the monitor's current aim, if that aim is different from the working space.

Older Autodesk Creative Finishing systems have a monitor that relies on a probe using procedures described in the *Lustre Color Management User Guide*. These monitors may have different default settings.

Color Managing Images for Output

If your working color space does not match the color space of your deliverables, you can apply a transform on output.

If your images are scene-linear, you can convert them to ACES if necessary using one of the transforms in the primaries/ directory. After that:

- If you are targeting video or digital cinema, use one of the transforms in the RRT+ODT/ directory.
- If you are targeting a film recorder, use one of the ACES_to_ADX transforms in the film/ADX/ directory.

If your images are in a log space and you are not outputting to film, apply one of the transforms in the film/ directory to convert them to scene-linear and then continue as above.

Managing Gamut for Digital Cinema

Digital Cinema Distribution Masters (DCDMs) require images in the DCI/SMPTE X'Y'Z' color space. The RRT+ODT/ACES_to_DCI transforms produce images in this color space (with variations for different creative white points).

However, this color space is much larger than that of any real projector. A typical cinema projector has a gamut closer to that of the DCI/SMPTE reference projector (P3 primaries).

Therefore, it is sometimes desirable to limit the gamut of colors placed in the X'Y'Z' color space to that of a more representative projector. This has two advantages:

- The gamut mapping is performed by the ODT (which preserves hue) rather than by the projector (which typically clips).

- Assuming that the content was graded and approved on a projector with a typical P3 gamut, limiting the colors to that gamut assures there will be no unpleasant surprises if the content is later viewed on a device with a wider gamut.

To achieve this:

1. Select the RRT+ODT for the device whose gamut you want to limit to, for example, RRT+ODT/ACES_to_P3-D60.
2. Then add transforms to convert to DCI/SMPTE X'Y'Z', for example, display/projector/P3-D60_to_CIE-XYZ followed by display/DCDM/CIE-XYZ_to_DCI-D60.

Color Managing Images from Scanned Film

Images from scanned film are typically supplied as DPX files with Cineon-style log encodings. However, the Cineon encoding was never fully standardized, and various vendors use slightly different implementations.

Academy Density Exchange (ADX) is a standard developed by AMPAS to eliminate any confusion and standardize on a single implementation. ADX is part of the ACES system, and is included in the SMPTE 2065 set of documents.

There are a couple of transforms for importing ADX-encoded images as scene-linear values. If you are importing DPX files from a properly calibrated ADX scanner, use one of these transforms:

- film/ADX/ADX10_to_ACES.ctf for 10-bit integer files
- film/ADX/ADX16_to_ACES.ctf for 16-bit integer files

If the images did not come from an ADX scanner, try importing them with one of the ADX transforms anyway as a first resort. If you are not satisfied with the result, you can try one of the Log_to_Lin transforms in the film/LogLin/ directory. However, be aware that these transforms modify each color channel independently (similar to 1D LUTs), and do not attempt to account for the channel cross-talk in film like the ADX_to_ACES transforms do.

Alternatively, you can try using one of the print-film emulation LUTs in the Lustre Color collection, if they are available to you. These transforms are installed with the Autodesk Creative Finishing products, as well as with Autodesk Composite. The transforms are supplied in the legacy .3dl format, and can be imported by Autodesk Color Management. Refer to the *Lustre Color Management User Guide* for more information about the transforms in that collection.

NOTE DPX files may contain values other than Cineon-style log-encoded colors. For example, they may contain video colors. When working with DPX files in general, you need to know the specific color encoding used. There may be metadata available in the files, but even when metadata is present it can be incorrect or misleading.

Color Managing Digital Cinema Camera Footage

The camera/ directory contains transforms for importing and converting footage from various digital cinema cameras to ACES. Use the transform that matches the specific camera model and lighting conditions (e.g. daylight or tungsten), as appropriate.

The input transforms that specify the illuminant are not designed to perform white balancing — for example, using a transform for tungsten illumination will not turn whites blue-ish. Instead, the different transforms take into account the different spectral power distributions of the different illuminants.

Once the footage has been converted to ACES, you can use the other transforms in the collection to convert it to your working space, if necessary.

NOTE The camera/Canon/Technicolor_CineStyle_to_HD-video.ctf is a special transform for viewing and outputting video that was shot using the Canon CineStyle picture style.

Camera Black

Footage from digital cinema cameras can contain negative values in the noise around black. These negative values can cause problems during image processing, especially when adjusting hue and saturation.

All of the input transforms supplied with Autodesk Color Management for input from various cameras map negative values to small positive values losslessly. If you prefer to undo this correction and deal with the negative values in another way, you can apply the remove_camera_black transform after the camera input transform.

In other cases, you may receive images that have already had a third-party input transform applied, but still have negative values. In these cases, you can use the apply_camera_black transform to map the negative values to small positive values.

Color Managing Video Footage

When importing video, you can use the transforms in the gamma/ directory to remove the gamma that has been applied. The resulting color values are linear, but they are still output-referred and should not be combined with scene-referred linear images.

There are a few options to convert these images to a scene-referred linear state:

- If the footage is HD (Rec. 709), you can apply HD-video_to_ACES from the RRT+ODT directory.
- Alternatively, you can apply an inverse tone map such as tone-map/inversePhotoMap_gamma_2.4. This will not be exact if the footage was originally shot as video or converted with a different tone map, but it can be a good approximation that works well in many cases.

SMPTE Legal Video Levels

Digital video standards use integer pixel encodings, and typically define a reference black and white level such that there is some headroom and footroom left for values to exceed these levels without being clipped. For example, SMPTE 10-bit digital video places reference black at 64 and reference white at 940. Broadcasters often have requirements that content provided to them not exceed these "legal" limits. (Note that the precise definition of "legal" varies depending on the video format and broadcaster.)

Software applications that work with integer pixel encodings often follow a different convention which places black and white at the minimum and maximum values allowed by the integer (e.g. for the 10-bit case, black at 0 and white at 1023).

When converting between video and computer encodings, it is necessary to decide whether to keep the headroom and footroom (in which case black and white are not quite where they are expected on the computer), or to discard the headroom and footroom and map the reference video black/white to the computer black/white.

Typically, this choice is made as part of the video I/O process and is applied in the video hardware. However, it is sometimes useful to also have color transforms available so that the conversion may be performed elsewhere in the workflow.

The full-range_to_legal_10bit transform in the levels/ directory scales, offsets, and clamps computer color values (i.e. 0 to 1023) so that they are within the legal SMPTE video range for broadcasting (i.e. 64 and 940). The legal-range_to_full_10bit transform inverts that operation.

Color Managing Rendered 3D CG Images

Images rendered from 3D scenes created in an application such as Maya are typically scene-referred linear RGBA images in a floating-point format such as OpenEXR.

To composite these images in a scene-linear working space such as ACES, you typically need to use one of the transforms in the primaries/ directory to convert from the native primaries, for example, from the Rec. 709 primaries.

If the linear CG images have incorrectly had a gamma applied but have not actually been tone-mapped, you can use one of the transforms in the gamma/ directory to remove it first.

The alpha channel represents the pixel coverage and should remain unmodified by any transform.

Color Managing Matte Paintings

All compositing should be done in a scene-linear color space, and this includes composites with painted matte backgrounds. However, matte paintings can be produced using different techniques, and these affect how you should process the images.

First, you must ensure that the images are scene-linear:

- If the image was created using a tone map for display as part of a color-managed workflow, then it is probably scene-linear already. In addition, the primaries are probably correct for the chosen working space, but you can change them if they are not.
- If the image was not created using a tone map for display, then it is probably output-referred with gamma-encoded values, for example, sRGB. To convert such an image to scene-linear, you must remove its gamma and apply an inverse tone map. These two operations are often combined in one transform. Ideally, you should use the inverse of the tone map that will be used for output and display. For example, if you will be using the ACES RRT tone map, then you can use sRGB_to_ACES from the RRT+ODT/ directory as an inverse tone map. Alternatively, inversePhotoMap_gamma_2.4 from the primaries/ directory is a generic transform that works well in many cases.
- It is also possible to receive a matte painting as a log-encoded image. In this case, use one of the transforms in the film/ directory to convert it to scene-linear. See Color Managing Images from Scanned Film (page 1799) for additional considerations.

Once the image is scene-linear, you can convert the primaries to your working space using one or more of the transforms in the primaries/ directory. For example, if you converted the image to scene-linear ACES and you want to work in scene-linear ProPhoto-RIMM, you can apply ACES_to_CIE-XYZ followed by CIE-XYZ_to_ProPhoto-RIMM.

Color Managing Textures and Maps

3D CG elements should be rendered using a scene-linear working space. However, images that are used for textures and other maps should be processed in various ways, depending on both the image state and the type of map.

Material Colors

Maps that are used for material colors should be scene-linear, but restricted to a 0-to-1 range. This is because they represent the proportion of light that is reflected or transmitted. These include diffuse, specular, and ambient reflectance colors, as well as transparency color.

First, you must ensure that the images are scene-linear:

- Output-referred images, such as video and sRGB, should have the gamma removed and an inverse tone map applied. These two operations are often combined in one transform. Ideally, you should use the inverse of the tone map that will be used for output and display. For example, if you will be using the ACES RRT tone map, then you can use sRGB_to_ACES from the RRT+ODT/ directory as an inverse tone map. Alternatively, inversePhotoMap_gamma_2.4 from the primaries/ directory is a generic transform that works well in many cases.

  NOTE Although it is common to simply remove the gamma from output-referred images, this is not enough to convert images to scene-linear — an inverse tone map is always required. However if a digital photograph was not flatly lit, or if a digital painting uses overly bright colors, then an inverse tone map may create values greater than 1.0. In these cases, the best approach may be to use one of the remove_gamma transforms in the gamma/ directory without applying an inverse tone map, even though this is not ideal.

- Log-encoded images should have the log encoding removed using one of the transforms in the film/ directory. See Color Managing Images from Scanned Film (page 1799) for additional considerations.
- Scene-linear images, such as images prepared using a tone map for display, can be used as-is if the primaries match your working space.

Once the images are scene-linear, you can convert the primaries to your working space using one or more of the transforms in the primaries/ directory. For example, if you converted an image to scene-linear ACES and you want to render using the UHDTV (Rec. 2020) primaries, you can apply ACES_to_CIE-XYZ followed by CIE-XYZ_to_LinearUHDTV.

Light Sources

Maps that represent light sources should be scene-linear but can contain values greater than 1.0. These include reflection maps, environment maps, and backplates, as well as some other effects such as incandescence.

- Output-referred images, such as video and sRGB, should have the gamma removed and an inverse tone map applied. After that, you can convert the primaries if necessary.
- Log images should have the log encoding removed, and then the primaries converted if necessary.
- Scene-linear images only need to have the primaries converted if they don't match your working space.

Non-color Data

If images are used to control non-color attributes, such as normals, depth, and so on, then the values should not have any color transform applied.

Color Management and Image-based Lighting (IBL)

The algorithms used to render 3D CG graphics require scene-linear colors, and images intended for image-based lighting are typically supplied as HDR or OpenEXR images that are scene-linear already. However, the images might still require some color management.

If the primaries used to encode an image do not match the working space used for rendering, then the image should be converted using one or more of the transforms in the primaries/ directory. For example, if the image uses the Adobe RGB primaries and you want to render using the UHDTV (Rec. 2020) primaries, then you can apply LinearAdobeRGB_to_CIE-XYZ followed by CIE-XYZ_to_LinearUHDTV.

White Point Conversion

The white point determines the color balance of an image. It should be adjusted depending on both the illumination of the original scene and the conditions under which the output is viewed — this process is also variously known as chromatic adaptation, white balancing, gray balancing, or neutral balancing.

It is sometimes necessary to convert media intended for one white point to a different one because the white point varies:

- Devices such as projectors and monitors are calibrated to specific white points.
- Cameras have different settings for various types of illumination.
- Well defined color spaces have a specified viewing environment that determines the observer's state of chromatic adaptation.

Converting Between Different White Points

In most cases, you do not need to separately adjust the white point of images yourself — it is simply a matter of choosing the appropriate transform. The transforms supplied with Autodesk Color Management automatically make the proper white point adjustments for display and output, as well as when converting between color spaces with specified white points. For example, the RDT+ODT/ACES_to_sRGB transform includes a conversion from D60 (ACES) to D65 (sRGB). The description tags in the metadata of the color transform files provides detailed information about the white point that each transform is intended for.

However, you may need to compensate for a projector or display calibrated to a different white point, or mix images that are intended for different adaptive white points. In these cases, you can apply the transforms in the whitepoint/ directory. Before doing this, first make sure to convert the images to the CIE-XYZ color space (either scene-referred or output-referred linear encoding).

Calibration Versus Creative White Point

White point issues for digital cinema projectors (and hence DCDMs) are particularly complex since there is both a calibration and a creative white point. The calibration white point is the white point that the device is calibrated to. However, in a dark theater there are no other light sources to affect adaptation, so the creative white point may be different.

For example, many people find the DCI white point used in digital cinema standards too green. So while theaters must still calibrate their projectors to this white point, neutral objects in a scene may be portrayed to another ("creative") white point such as D60. You can achieve this by using a transform such as RRT+ODT/ACES_to_DCI-D60 as the output transform, or by explicitly converting the white point yourself using the transforms in the whitepoint/ directory.

Sometimes this technique is also used on video monitors when you want to match the white point of another device without actually recalibrating the monitor to that other white point. For example, if you have a monitor in a dark projection room, the colors on the monitor will not match the projected colors because the white points are different. To make the monitor colors match, you must convert the white point from, for example, D65 to DCI. Once this is done, neutral colors on the monitor no longer correspond to equal code values but rather to whatever code values are required to make the desired white despite the different calibration white.

Using Diagnostic Color Transforms

There are some transforms in the levels/ directory that you can apply to the display to show potential problem areas in images.

- show_negs highlights pixels with negative color values. Other values are mapped towards middle values.

- show_out-of-range highlights pixels with negative color values, and darkens values that are above 1.0. Values in the [0, 1] range are mapped towards middle values.

Converting Images Between 12i or 16i and 16f Encodings

It is often useful or necessary to convert 12-bit integer (12i) or 16-bit integer (16i) to 16-bit floating-point (16f) encoding or vice-versa. When round-tripping between encodings like this, it's important that the transforms are as lossless as possible to avoid image degradation.

In Flame Premium and Smoke, it is typically desirable to promote 16i media to 16f as part of the import process in order to maximize the precision available for compositing and other operations. If you need to convert back to 16i for your output, you should do so only when exporting to preserve the maximal precision throughout the processing.

Similarly, when using 12i media in Flame Reactor, it will get promoted to 16f.

Converting from 16i Log Encoding

There are many varieties of 16-bit integer log encodings, but few standards. As such it is difficult to make general purpose recommendations on how to convert these images.

To convert 16i images from log space to scene-linear, there are several options:

- If you know the images are in ADX16 color space, you can use the ADX16_to_ACES transform from the film/ADX/ directory, which will take into account film channel crosstalk and provide more accurate colors. Alternatively, you can use the Log_to_Lin--adx16 transform from the film/LogLin/ directory, which is a simple 1D-LUT-style transform that is slightly more invertible for round-tripping back to 16i.
- Alternatively, try one of the other Log_to_Lin transforms in the film/LogLin directory. However, be aware that these operate on each channel independently. This means that they are less accurate in terms of color reproduction, but are more invertible and give smaller errors than the ADX transforms when roundtripping.

Converting from 12i and 16i Video Encodings

To convert 12i or 16i images from a video-like space to scene-linear 16f and then back to 16i while preserving the original values as much as possible, there are some special transforms in the bitdepth/ directory.

- The 12i_to_16f and 16f_to_12i transforms are designed to provide a lossless round-trip.
- The 16i_to_16f and 16f_to_16i transforms are designed to minimize quantization loss — the maximum error when round-tripping is 2 code values in 65535.

However, because 12i_to_16f and 16i_to_16f are very different from typical inverse tone maps, you should use 16f_to_12i or 16f_to_16i respectively as your display transform as well as your output transform (when round-tripping back to 12i or 16i).

In addition, note that these transforms are not appropriate for use with all operations. For example, resizing and other operations that involve spatial filtering give very different results in scene-linear space than in video space. In particular, the Lanczos filter has large negative coefficients that can cause artifacts like rings or haloes around high-contrast edges. In these situations, you should consider alternative transforms to perform the conversions:

- Use a log-to-linear transform such as Log_to_Lin--jzp in the film/LogLin/ directory for import, and its inverse such as Lin_to_Log--jzp for viewing and export.
- Use a transform such as remove_2.40_gamma in the gamma/ directory for import, and its inverse such as apply_2.40_gamma for viewing and export. This minimizes the haloes, but results in more loss of precision when round-tripping.

Custom Color Transforms

Autodesk Color Management lets you create your own .ctf files to define custom color transforms.

Building Custom Transforms

In Smoke, Flame, and Lustre, you can use Custom mode to assemble a chain of transforms, either from the installed collection or by importing transforms in ASC CDL or third-party LUT formats. You can then export the whole chain as a single .ctf file for example, to use as a viewing transform or to save time with conversions that you perform frequently.

Because the .ctf file format can represent arbitrary lists of color processing operations, the exported transform is a lossless representation of the original processing. In other words, it is much more accurate than baking the original transforms into a single 3D LUT.

The .ctf file format is XML-based, so you can also use any XML or text editor to create or modify color transforms. You can either create a transform from scratch, or use <Reference> elements to build a chain from existing transform files. See Autodesk CTF File Format Version 1.3 (page 1810).

Dynamic Exposure and Contrast Controls

Some Autodesk applications, like Flame and Smoke, allow you to adjust exposure and contrast interactively for viewing if those controls have been declared as dynamic in a .ctf file. The exposure and contrast values set in the application are used for previewing, but the values set in the file are used for processing.

This means that you can create a transform that includes an ExposureContrast operator that does not affect color values (i.e., exposure of 0 and contrast of 1), and still adjust the exposure and contrast when the transform is applied to the display. This allows you to check the details in very bright or very dark areas. However when the same transform is used for processing, the output color values are unaffected.

You can take advantage of this in your own color transforms by referencing one of the exposure-contrast files in the misc/ directory. There are three versions, so that you can insert them in a scene-linear, logarithmic, or video color space. The algorithms have been adjusted so that, for example, the exposure-contrast_log transform causes the exposure and contrast sliders to behave the same in a log color space as exposure-contrast_linear behaves in a scene-linear color space.

Dynamic Look Controls

Flame and Smoke allow you to toggle a look transform on and off for previewing purposes. The Look On/Off toggle appears whenever the viewing transform includes one or more operator elements with a LOOK_SWITCH dynamic parameter. If the same operators have their bypass attribute set to "true", they will be skipped during processing. This lets you include operators that will never affect values rendered to file but that can still be toggled on and off for display.

One way to take advantage of this is to define your look transform file as the defaultLook alias. The defaultLook alias can be set in the LUT preferences, and the setting is stored in the SynColor configuration file. You can then create a transform chain that includes a reference to one of the transforms in the misc/ directory that in turn references the defaultLook alias:

- The misc/default_look transform applies the look transform directly.
- If you are using an ACES workflow and ACESproxy was used on set, you can reference the misc/default_look-ACESproxy transform instead. This transform first converts from ACES to ACESproxy, then references the defaultLook transform, and finally converts back to ACES. If your look transform is an ASC CDL operator and you use the "noClamp" option, then ACES values are not clamped to the ACESproxy range. Because internal processing is done at 32-bit floating point precision, there will be negligible quantization loss.

By default, these transforms will be applied when rendering the output. However, you can edit the XML to add `bypass="true"` to the reference element (see Common Operator Attributes (page 1819)) so that the look can be toggled for display only but not applied when rendering the output.

This gives you complete control over the color space in which the look transform is applied, and allows you to match an arbitrary on-set workflow. For example, suppose that you are provided with a 3D LUT for converting from log to video space and a set of ASC CDLs from an on-set workflow, and that you are told the CDLs should be applied before rather than after the 3D LUT. In your chain of transforms, include the default look first, followed by the 3D LUT. Apply this file as your viewer transform, and set the appropriate ASC CDL transform for the shot as your defaultLook alias. Now you are able to work on the original log media with your viewer configured to emulate what was seen on set. Furthermore, you can take advantage of the dynamic Look On/Off switch in the viewer to easily toggle the look on and off.

A Complex Color Management Example

Projects often include a variety of media from different sources, and it can be difficult to know what color management to apply. The transforms in the Autodesk Color Management collection have been designed as building blocks to provide maximum flexibility. With an understanding of the concepts presented in this guide, you can combine these building blocks to solve your color workflow challenges. With that in mind, here is an advanced workflow example.

Suppose that you have a project that was mostly shot on 35mm film, but for various reasons some specific shots used different digital cinema cameras. In addition, there are some rendered 3D CG elements as well as title cards. You need to combine all these images to produce multiple deliverables: DCDM for projection in theaters plus HD video for home Blu-ray.

Choose a Working Color Space

Your first step is to decide on a working color space, unless that decision has already been made for you.

- A scene-linear working space is a good idea, especially for compositing the 3D elements.
- Considering that the deliverables are digital cinema (which uses the P3 primaries for projection) and HD (which uses the Rec. 709 primaries), it makes sense to choose the P3 primaries. They have a larger gamut than Rec. 709, so the DCDM can use all of the colors that are available on a SMPTE reference projector. Furthermore by restricting the working space to this gamut, you reduce the possibility of out-of-gamut colors.
- Many people find the DCI calibration white point too green, and prefer to work with a different creative white point such as D60.

So, a suitable choice for a working space is a scene-linear encoding using the P3 primaries and a white point of D60.

Convert the Inputs

Next, you need to convert all the inputs to this working space:

- The film scans are most likely DPX files using ADX or another Cineon-like encoding. You can use one of the transforms in the film/ directory to convert these to scene-linear ACES on import (see Color Managing Images from Scanned Film (page 1799) for additional considerations). Looking at the transforms available in the primaries/ directory, there is no direct way to convert from ACES to the P3 primaries. However, it is possible using two steps: by applying ACES_to_CIE-XYZ followed by CIE-XYZ_to_P3-D60. You can export the whole chain of transforms as a single .ctf file for convenience when importing many shots.
- Digital cinema cameras all use different proprietary encodings. Depending on the camera make, model, and lighting settings, use the appropriate transform from the camera/ directory to convert the images to ACES (see Color Managing Digital Cinema Camera Footage (page 1799)). As with the film footage, you can

then convert the images to the P3 primaries by applying ACES_to_CIE-XYZ followed by CIE-XYZ_to_P3-D60 from the primaries/ directory.

- The rendered 3D CG elements are most likely scene-linear images already, but are probably using the sRGB primaries (which are the same as the Rec. 709 primaries). You can convert these images to your working space by applying Rec-709-sRGB_to_CIE-XYZ followed by CIE-XYZ_to_P3-D60 from the primaries/ directory (see Color Managing Rendered 3D CG Images (page 1801)).
- Suppose that the title cards are sRGB images. To convert them to scene-linear, you need to apply an inverse tone map. One way to do this is to apply inversePhotoMap_gamma_2.4 from the tone-map/ directory (see Color Managing Video Footage (page 1800)), and then convert the primaries in the same way as for the 3D elements using Rec-709-sRGB_to_CIE-XYZ followed by CIE-XYZ_to_P3-D60 from the primaries/ directory. However because you will be using the ACES RRT tone map for output (more about that coming up), use the sRGB_to_ACES transform followed by ACES_to_CIE-XYZ and CIE-XYZ_to_P3-D60 — this ensures that the original sRGB values are unchanged by the matching combination of inverse tone map for input and tone map for both display and output.

Set Up the View Transform

Now you need to set up your view transform to display these images as you work. Since the working space is scene-referred, you need to use a tone map to convert the images to output-referred values, and of course you should use the same tone map as the final deliverables. For the purpose of this example, suppose that you decide to use the ACES RRT.

To use any of the transforms in the RRT+ODT/ directory, you must first convert to ACES. You can do this using P3-D60_to_CIE-XYZ followed by CIE-XYZ_to_ACES from the primaries/ directory. Finally, you can apply ACES_to_current-monitor from the RRT+ODT/ directory. Chromatic adaptation is built into these transforms to map the D60 white point of the working space to the monitor's white point. Export this chain of transforms as a single .ctf file and set it as your viewing transform.

In addition, you may want to preview on a projector calibrated to the DCI white point. You can first convert to ACES using P3-D60_to_CIE-XYZ followed by CIE-XYZ_to_ACES from the primaries/ directory as above, and then use ACES_to_P3-DCI from the RRT+ODT/ directory. Although the calibration white is DCI, the creative white is D60, so chromatic adaptation is neither necessary nor built-in (see White Point Conversion (page 1803)).

See Color Managing Images for Display (page 1797) for more information.

Color Managing the Output

Finally, you need to apply the correct transforms for your deliverables. As for the display, you need to use a tone map to convert from scene-referred to output-referred.

Once again, you can convert from the working space to ACES using P3-D60_to_CIE-XYZ followed by CIE-XYZ_to_ACES from the primaries/ directory. After that:

- For the DCDM, apply ACES_to_DCI-60 from the RRT+ODT/ directory.
- For HD video, apply ACES_to_HD-video from the same directory.

See Color Managing Images for Output (page 1798) for more information.

Color Transform Files

The color transforms included with Autodesk Color Management are supplied as individual files in CTF format. In addition, Autodesk Color Management can import many third-party LUT and transform formats.

Autodesk Color Transform Collection

The transforms in the Autodesk Color Management collection are CTF files based on the Academy/ASC XML color transform format. They include several categories for different purposes.

The directory location depends on the Autodesk product and operating system. For Creative Finishing products:

- Linux:

```
/opt/Autodesk/Synergy/SynColor/<version>/transforms
```

- Mac OS X:

```
/Applications/Autodesk/Synergy/SynColor/<version>/transforms
```

For Maya and Maya LT:

```
<install_directory>/synColor/transforms
```

Category	Description
RRT+ODT	Various combinations of a reference rendering transform (RRT) followed by an output device transform (ODT) designed for output from the ACES color space, as well as the corresponding inverse transforms.
bit-depth	Converts between various integer and floating-point value formats.
camera	Converts from various digital-camera native formats to other color spaces such as ACES.
display	Converts colors in a connection space to and from output spaces appropriate for various display devices.
film	Converts scanned-film color spaces to and from ACES, or applies various 1D log/linear conversions.
gamma	Applies or removes various common gamma curves, including those for sRGB and Rec. 709.
interchange	Converts colors in a connection space to and from spaces that are commonly used for file input and output, such as AdobeRGB and sRGB.
levels	Various functions to clamp or adjust color values.
misc	Various transforms that may be useful for debugging or other purposes.

Category	Description
primaries	Converts between linear color spaces that use different primary colors.
tone-map	Converts from scene-linear to video color spaces, and vice versa.
whitepoint	Converts the white point of linear color spaces.

Supported Color Transform File Formats

In addition to the native CTF format, applications that use Autodesk Color Management can read several LUT and transform file formats. When imported, these formats are converted to native operators. Autodesk products that support the export of transforms will export them as CTF files.

Supported formats include:

.lut Autodesk legacy 1D LUT.

.3dl Autodesk legacy 3D LUT.

.cdl ASC color decision list.

ASC .cdl files are loaded as ASC_CDL operators. The style used by the color transform operator is determined by the SYNCOLOR_DEFAULT_CDL_STYLE environment variable. This can be one of the following values: `v1.2_Fwd`, `v1.2_Rev`, `noClampFwd`, or `noClampRev`. If this environment variable is undefined, the default is `v1.2_Fwd`. For more information about these styles, see ASC_CDL (page 1840).

When a .cdl file is applied to the display, a Look switch is added that allows you to toggle the transform on and off.

.ccc ASC color correction collection.

The first CDL defined in the file is loaded as an ASC_CDL operator. The style is also determined by the SYNCOLOR_DEFAULT_CDL_STYLE environment variable.

.clf Academy/ASC Common LUT format.

.csp Cinespace.

.icc, **.icm**, **.pf** ICC monitor profile. The PCS-to-monitor-RGB transform defined in the profile is loaded.

.cube Iridas Cube.

.itx Iridas itx.

.look Iridas Look (no mask support).

.m3d, **.mga** Pandora.

.spi1d Imageworks 1D LUT.

.spi3d Imageworks 3D LUT.

.spimtx Imageworks matrix.

.vf Nuke.

.vlt Panasonic VariCam 3D LUT.

Autodesk CTF File Format Version 1.3

There are a wide variety of colour transform (also known as "LUT") formats used in the media and entertainment industry. This creates workflow problems since you may receive a LUT which is not supported by one or more of the applications you use. Furthermore, most of these LUT formats are based on out-dated technology and lack the quality, flexibility, and metadata needed to meet today's requirements.

To address these problems, the technology committees of the Academy of Motion Picture Arts and Sciences and the American Society of Cinematographers convened a group of experts to design a common transform format that could become an industry standard.

Autodesk has chosen to adopt this format as the native format for our colour management technology and we provide a large collection of transforms in this format with our applications (e.g., to support the Academy Color Encoding System).

This guide provides information about the Academy/ASC format to help you author your own colour transforms. In addition it defines several Autodesk-specific extensions which you may use when authoring colour transforms solely for use in Autodesk applications. All Autodesk extensions are marked with the version in which they were introduced or updated. All other elements and attributes are part of the original Academy/ASC format.

Anatomy of a CTF File

The transform format allows the author to define an arbitrary chain of color operators (also called processing nodes). The set of available operator types includes 1D LUTs, 3D LUTs, matrices, and others defined below. The format is written in XML and is therefore human readable.

Here is an example:

```
<?xml version="1.0" encoding="UTF-8"?>
<ProcessList id="7401489a-9e4d-441d-a8b7-795739801f4e" name="ACES to HD"
version="1.2">
    <Info>
        <Copyright>Copyright 2013 Autodesk, Inc.  All rights
reserved.</Copyright>
        <Release>2013.3.pr27</Release>
    </Info>
   <Description>ACES Reference Rendering Transform + HD video Output Device
 Transform</Description>
    <Description>aces_dev version 0.1.1</Description>
    <InputDescriptor>Academy Color Encoding Specification
(ACES)</InputDescriptor>
    <OutputDescriptor>HD (Rec 709) video</OutputDescriptor>
    <LUT1D halfDomain="true" inBitDepth="32f" outBitDepth="32f"
rawHalfs="true">
        <Array dim="65536 1">
    0
    1
    1

    <!-- 65533 lines omitted -->

        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 0.9259999990  0.0379999988  0.0359999985
 0.1299999952  0.8000000119  0.0700000003
```

```
-0.0350000001 -0.0350000001  1.0700000525
        </Array>
    </Matrix>
    <LUT1D halfDomain="true" inBitDepth="32f" outBitDepth="32f"
rawHalfs="true">
        <Array dim="65536 1">
    0
    0
    0

    <!-- 65533 lines omitted -->

       </Array>
    </LUT1D>
    <LUT3D inBitDepth="32f" interpolation="tetrahedral" outBitDepth="32f">
        <Array dim="48 48 48 3">
 0.000669299  0.000704187  0.000766901
 0.001848371  0.001297010  0.010368987
 0.002599007  0.001144927  0.023227118

        </Array>
    </LUT3D>
    <Gamma inBitDepth="32f" outBitDepth="32f" style="moncurveFwd">
        <GammaParams gamma="2.600000000000000" offset="0.050000" />
    </Gamma>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 3.2409698963 -1.5373831987 -0.4986107647
-0.9692436457  1.8759675026  0.0415550582
 0.0556300804 -0.2039769590  1.0569715500
        </Array>
    </Matrix>
    <Range inBitDepth="32f" outBitDepth="32f">
        <minInValue>  0.0000</minInValue>
        <maxInValue>  1.0000</maxInValue>
        <minOutValue>  0.0000</minOutValue>
        <maxOutValue>  1.0000</maxOutValue>
    </Range>
    <Gamma inBitDepth="32f" outBitDepth="12i" style="basicRev">
        <GammaParams gamma="2.400000000000000" />
    </Gamma>
</ProcessList>
```

CTF Syntax Summary

Here is a brief overview of the syntax of a CTF file.

In the list below, **?** means "0 or 1 occurrence", ***** means "0 or more occurrences" , and **+** means "1 or more occurrences". If not otherwise marked, a single occurence of an element is required.

ProcessList (page 1813) Attributes: id, name Contains:

Info (page 1814) ? Contains:

Copyright (page 1815) ?

Release (page 1816) ?

Description (page 1817) *

InputDescriptor (page 1818) ?

OutputDescriptor (page 1818) ?

OperatorNode + This is a virtual element — replace it by one of the substitutions below (LUT1D, LUT3D, etc.). Attributes: id, name, inBitDepth, outBitDepth, bypass Contains:

Description (page 1817) *

DynamicParameter (page 1847) ? (up to 3 for ExposureContrast) Attributes: param

Substitute each OperatorNode by any one of the following:

LUT1D (page 1820) Inherited attributes: id, name, inBitDepth, outBitDepth, bypass Attributes: interpolation, halfDomain, rawHalfs Contains:

Array (page 1822) Attributes: dim

LUT3D (page 1823) Inherited attributes: id, name, inBitDepth, outBitDepth, bypass Attributes: interpolation Contains:

Array (page 1823) Attributes: dim

Matrix (page 1825) Inherited attributes: id, name, inBitDepth, outBitDepth, bypass Contains:

Array (page 1826) Attributes: dim

Range (page 1828) Inherited attributes: id, name, inBitDepth, outBitDepth, bypass Contains:

minInValue (page 1830) ?

maxInValue (page 1830) ?

minOutValue (page 1831) ?

maxOutValue (page 1831) ?

Gamma (page 1832) Inherited attributes: id, name, inBitDepth, outBitDepth, bypass Attributes: style Contains:

GammaParams (page 1834) Attributes: channel, gamma, offset

ExposureContrast (page 1835) Inherited attributes: id, name, inBitDepth, outBitDepth, bypass Contains:

ECParams (page 1836) Attributes: exposure, contrast, pivot

Log (page 1837) Inherited attributes: id, name, inBitDepth, outBitDepth, bypass Attributes: style Contains:

LogParams (page 1838) 1 or 3 Attributes: channel, gamma, refWhite, refBlack, highlight, shadow

ASC_CDL (page 1840) Inherited attributes: id, name, inBitDepth, outBitDepth, bypass Contains:

SOPNode (page 1841) Contains:

Slope (page 1842)

Offset (page 1843)

Power (page 1843)

SatNode (page 1844) Contains:

Saturation (page 1845)

Reference (page 1845) Inherited attributes: id, name, inBitDepth, outBitDepth, bypass attributes: alias, path, basepath

XML Declaration

The XML declaration is optional but recommended. It indicates to the XML parser the version of XML and character encoding used.

If it is present, the XML declaration must be the first line of the file.

Example

```
<?xml version="1.0" encoding="UTF-8"?>
```

ProcessList

The ProcessList element is the root element of a CTF file. It must contain at least one operator element.

Contains

In order:

- Info (page 1814). Optional, no more than one.
- Description (page 1817). Optional, any number.
- InputDescriptor (page 1818). Optional, no more than one.
- OutputDescriptor (page 1818). Optional, no more than one.
- At least one or more operator elements, in any combination in the order in which they are to be applied.

  NOTE The inBitDepth attribute of each operator must match the outBitDepth attribute of the previous operator in the chain. See Common Operator Attributes (page 1819).

Contained By

- no parent

Attributes

id A unique identifier (required).

name A nickname for the entire color transform (optional). This is used for display and selection in applications that support Autodesk Color Management.

version Indicates the presence of Autodesk extensions to the Academy/ASC XML color transform format. Files with a value higher than that supported by the application are rejected.

inverseOf The id attribute from another CTF file. It identities this transform as the inverse of the other one. This information is useful to the color management system to detect the case where a transform and then its inverse are applied, so the transform pair may be optimized out. It is an Autodesk extension to the Academy/ASC XML color transform format.

Example

```
<ProcessList id="7214fa79-6c42-48b9-b164-3d0c5e543fe0" version="1.2">
    <Description>F65 Raw integer code values to ACES (tungsten
illuminant).</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>Academy Color Encoding Specification
(ACES)</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
```

```
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.0110238791 -0.1362526119  0.1252287328
 0.1011994481  0.9562196136 -0.0574190766
 0.0600766540 -0.1010185331  1.0409418344
        </Array>
    </Matrix>
</ProcessList>
```

Metadata Elements

The metadata elements provide information about a color transformation that can be displayed in information panels in applications.

Info

The Info element is a container for the Copyright and Release elements.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.2.

Contains

In any order:

- Copyright (page 1815). Optional, no more than one.
- Release (page 1816). Optional, no more than one.

Contained By

- ProcessList (page 1813)

Attributes

None.

Example

```
<ProcessList id="f8d905db-4ab8-4af6-8465-0db81f7e993c" version="1.2">
    <Info>
        <Copyright>Copyright 2012 Autodesk, Inc.  All rights
reserved.</Copyright>
        <Release>2013.1.pr20</Release>
    </Info>
    <Description>F65 Raw integer code values to scene-linear floats with
Rec 709 primaries.</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>scene-linear Rec 709 primaries</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
```

```
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.8779151440 -0.7941687703 -0.0837463662
-0.1768069863  1.3509995937 -0.1741926372
-0.0262011271 -0.1484222561  1.1746233702
        </Array>
    </Matrix>
</ProcessList>
```

Copyright

The Copyright element is intended to contain a copyright notice covering the CTF file.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.2.

Contains

- Text string. Use `>` for >, `<` for <, and `&` for &.

Contained By

- Info (page 1814)

Attributes

None.

Example

```
<ProcessList id="f8d905db-4ab8-4af6-8465-0db81f7e993c" version="1.2">
    <Info>
        <Copyright>Copyright 2012 Autodesk, Inc.  All rights reserved.</Copyright>

        <Release>2013.1.pr20</Release>
    </Info>
    <Description>F65 Raw integer code values to scene-linear floats with
Rec 709 primaries.</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>scene-linear Rec 709 primaries</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.8779151440 -0.7941687703 -0.0837463662
-0.1768069863  1.3509995937 -0.1741926372
-0.0262011271 -0.1484222561  1.1746233702
```

```
        </Array>
    </Matrix>
</ProcessList>
```

Release

The Release element is intended to contain the version of the SynColor library corresponding to an Autodesk-supplied CTF file.

It is an Autodesk extension to the Academy/ASC XML color transform format. You should not add this element to color transforms that you author yourself.

Introduced

Version 1.2.

Contains

- Text string. Use `>` for >, `<` for <, and `&` for &.

Contained By

- Info (page 1814)

Attributes

None.

Example

```
<ProcessList id="f8d905db-4ab8-4af6-8465-0db81f7e993c" version="1.2">
    <Info>
        <Copyright>Copyright 2012 Autodesk, Inc.  All rights
reserved.</Copyright>
        <Release>2013.1.pr20</Release>
    </Info>
    <Description>F65 Raw integer code values to scene-linear floats with
Rec 709 primaries.</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>scene-linear Rec 709 primaries</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.8779151440 -0.7941687703 -0.0837463662
-0.1768069863  1.3509995937 -0.1741926372
-0.0262011271 -0.1484222561  1.1746233702
        </Array>
    </Matrix>
</ProcessList>
```

Description

The Description element allows you to add information about the function or usage of a ProcessList or operator element. It is displayed in the metadata panel of color transforms in products that support Autodesk Color Management.

Contains

- Text string. Use `>` for >, `<` for <, and `&` for &.

Contained By

- ProcessList (page 1813)
- LUT1D (page 1820)
- LUT3D (page 1823)
- Matrix (page 1825)
- Range (page 1828)
- ExposureContrast (page 1835)
- Gamma (page 1832)
- Reference (page 1845)

Attributes

None.

Example

```
<ProcessList id="f8d905db-4ab8-4af6-8465-0db81f7e993c" version="1.2">
    <Info>
        <Copyright>Copyright 2012 Autodesk, Inc.  All rights
reserved.</Copyright>
        <Release>2013.1.pr20</Release>
    </Info>
    <Description>F65 Raw integer code values to scene-linear floats with Rec 709
primaries.</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>scene-linear Rec 709 primaries</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.8779151440 -0.7941687703 -0.0837463662
-0.1768069863  1.3509995937 -0.1741926372
-0.0262011271 -0.1484222561  1.1746233702
        </Array>
    </Matrix>
</ProcessList>
```

InputDescriptor

The InputDescriptor element allows you to add a comment describing the intended input color space and encoding for a color transform. It is does not affect the processing, but is displayed in the metadata panel of color transforms in products that support Autodesk Color Management.

Contains

- Text string. Use `>` for >, `<` for <, and `&` for &.

Contained By

- ProcessList (page 1813)

Attributes

None.

Example

Example

```
<ProcessList id="f8d905db-4ab8-4af6-8465-0db81f7e993c" version="1.2">
    <Info>
        <Copyright>Copyright 2012 Autodesk, Inc.  All rights
reserved.</Copyright>
        <Release>2013.1.pr20</Release>
    </Info>
    <Description>F65 Raw integer code values to scene-linear floats with
Rec 709 primaries.</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>scene-linear Rec 709 primaries</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.8779151440 -0.7941687703 -0.0837463662
-0.1768069863  1.3509995937 -0.1741926372
-0.0262011271 -0.1484222561  1.1746233702
        </Array>
    </Matrix>
</ProcessList>
```

OutputDescriptor

The OutputDescriptor element allows you to add a comment describing the output color space and encoding for a color transform. It is does not affect the processing, but is displayed in the metadata panel of color transforms in products that support Autodesk Color Management.

Contains

- Text string. Use `>` for >, `<` for <, and `&` for &.

Contained By

- ProcessList (page 1813)

Attributes

None.

Example

Example

```
<ProcessList id="f8d905db-4ab8-4af6-8465-0db81f7e993c" version="1.2">
    <Info>
        <Copyright>Copyright 2012 Autodesk, Inc.  All rights
reserved.</Copyright>
        <Release>2013.1.pr20</Release>
    </Info>
    <Description>F65 Raw integer code values to scene-linear floats with
Rec 709 primaries.</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>scene-linear Rec 709 primaries</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.8779151440 -0.7941687703 -0.0837463662
-0.1768069863  1.3509995937 -0.1741926372
-0.0262011271 -0.1484222561  1.1746233702
        </Array>
    </Matrix>
</ProcessList>
```

Operator Elements

The operator elements specify the operations to be applied to the color data. You can have as many as you like in a ProcessList element, in the order in which they are to be applied.

NOTE The inBitDepth attribute of each operator must match the outBitDepth attribute of the previous operator in the chain. See Common Operator Attributes (page 1819).

Common Operator Attributes

Several attributes are common to all of the color transform operator elements.

id A unique identifier for the element. Optional.

name A friendly name for the element. Optional.

bypass Introduced in version 1.3. Specifies that the operator should not be applied during color processing. Optional. If this attribute exists, its value must be `"true"`. By using this attribute in combination with a DynamicParameter (page 1847) element that has its param attribute set to `"LOOK_SWITCH"`, you can define operators that will never affect values rendered to file but that can still be toggled on and off for display.

inBitDepth The input color data type expected by the operator. Required.

outBitDepth The output color data type produced by the operator. Required.

NOTE The inBitDepth attribute of each operator must match the outBitDepth attribute of the previous operator in the chain.

Supported Values for inBitDepth and outBitDepth

Value	Meaning
`"8i"`	8-bit integer
`"10i"`	10-bit integer
`"12i"`	12-bit integer
`"16i"`	16-bit integer
`"16f"`	16-bit floating point (half-float)
`"32f"`	32-bit floating point (single precision)

About Bit Depth

The values specified for inBitDepth and outBitDepth attributes do not affect the quantization of color values. All processing is performed using 32-bit floating-point values.

The inBitDepth and outBitDepth attributes affect only the format of the numbers used in the CTF file, such as the values used in an Array, Matrix, or Range element. For example, if you find it convenient to specify values as integers in the range [0, 1023] then you can use an inBitDepth of `"10i"`. You can still specify values outside this range, and those values will be used in the intermediate computations.

LUT1D

The LUT1D element specifies a 1D LUT to apply.

Contains

- Description (page 1817). Optional, any number.
- Array (LUT1D) (page 1822). Required, one only.
- DynamicParameter (page 1847). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

NOTE The IndexMap element is not supported. The most efficient way to achieve the same result is to precede the operator with another LUT1D element, or to combine both LUTs into a single one.

Contained By

- ProcessList (page 1813)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1819).

interpolation Optional. The only currently supported value is `"linear"`.

halfDomain Optional. If this attribute is present, its value must be `"true"`. In this case, the input domain is all possible 16-bit floating-point values, and there must be exactly 65536 entries in the Array element. For example, the unsigned integer 15360 has the same bit-pattern (0011110000000000) as the half-float value 1.0, so the 15360th entry (zero-indexed) in the Array element is the output value corresponding to an input value of 1.0.

rawHalfs Optional. If this attribute is present, its value must be `"true"`. In this case, the values in the array should be 16-bit floating point values expressed as unsigned 16-bit integers representing the equivalent bit pattern. For example, to represent the value 1.0, you would enter the integer `15360` in the Array element because it has the same bit-pattern. This allows you to specify the exact half-float values you want without relying on conversion from decimal text strings.

Example

1D LUT

```
<ProcessList id="8a52d5fb-a903-4805-8bae-24f7553bfb70" version="1.2">
    <LUT1D inBitDepth="10i" outBitDepth="32f">
        <Array dim="1024 1">
-0.014279292
-0.014160193
-0.014040368
<!-- 1021 lines omitted -->

        </Array>
    </LUT1D>
</ProcessList>
```

3×1D LUT

```
<ProcessList id="a76dbe2e-e610-49a6-8c3b-5962375a8b4a" version="1.2">
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
 </ProcessList>
```

Array (LUT1D)

The Array element specifies an array of values for a LUT or matrix. When it is contained by a LUT1D element, the format of its contents is different than when it is contained by LUT3D or Matrix elements.

Contains

- A series of numeric values specifying the output values of the LUT. The corresponding input values are implied by the inBitDepth (page 1819) of the containing element. For integers, the first value corresponds to an input of 0 and the last to an input of ($2^{bitdepth} - 1$). For floating-point numbers, the first value corresponds to an input of 0.0 and the last to an input of 1.0. Input values outside these domains are clamped.

  The intervals between the input values are uniform and depend on the number of entries. For example, if there are 5 floating-point entries, they correspond to inputs of 0.0, 0.25, 0.5, 0.75, and 1.0.

  For a 1D LUT, there is one value per entry which is used for all color channels. For a 3×1D LUT, there are three values per entry specifying the separate R, G, and B output values.

Contained By

- LUT1D (page 1820)

Attributes

dim Two integers representing the dimensions of the array. The first value is the number of entries in the contents of the Array element. The second value is number of values per entry: 1 for a single value applied to all channels or 3 for for separate values applied to the R, G, and B channels respectively.

The dim attribute is required, and its value must match the number of entries actually present.

Example

1D LUT

```
<ProcessList id="8a52d5fb-a903-4805-8bae-24f7553bfb70" version="1.2">
    <LUT1D inBitDepth="10i" outBitDepth="32f">
        <Array dim="1024 1">
-0.014279292
-0.014160193
-0.014040368
<!-- 1021 values omitted -->

        </Array>
    </LUT1D>
</ProcessList>
```

3×1D LUT

```
<ProcessList id="a76dbe2e-e610-49a6-8c3b-5962375a8b4a" version="1.2">
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
 </ProcessList>
```

See Also

- Array (LUT3D) (page 1823)
- Array (Matrix) (page 1826)

LUT3D

The LUT3D element specifies a 3D LUT to apply.

Contains

- Description (page 1817). Optional, any number.
- Array (LUT3D) (page 1823). Required, one only.
- DynamicParameter (page 1847). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

NOTE The IndexMap element is not supported. The most efficient way to achieve the same result is to precede the operator with a LUT1D element.

Contained By

- ProcessList (page 1813)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1819).

interpolation Supported values are `"trilinear"` and `"tetrahedral"`.

This attribute is optional. If it is not specified, the default is `"trilinear"`.

Example

```
<ProcessList id="b5b90615-573b-4959-a838-f35e0e588ac2" version="1.2">
    <LUT3D inBitDepth="32f" interpolation="tetrahedral" outBitDepth="32f">
        <Array dim="48 48 48 3">
 0.000669299  0.000704187  0.000766901
 0.001848371  0.001297010  0.010368987
 0.002599007  0.001144927  0.023227118
<!-- 48*48*48-3 lines omitted -->

        </Array>
    </LUT3D>
</ProcessList>
```

Array (LUT3D)

The Array element specifies an array of values for a LUT or matrix. When it is contained by a LUT3D element, the format of its contents is different than when it is contained by LUT1D or Matrix elements.

Contains

- A series of numeric values specifying the output values of the LUT. The corresponding input values are implied by the inBitDepth (page 1819) of the containing element and the number of entries (uniform distribution).

  For a 3D LUT, each entry has three values specifying the separate R, G, and B output values. The entries are in order from minimum to maximum with the index of the last column changing fastest and the index of the first column changing slowest. For example, the order of entries for a 2×2×2 cube is:

Entry	R	G	B
1	0	0	0
2	0	0	1
3	0	1	0
4	0	1	1
5	1	0	0
6	1	0	1
7	1	1	0
8	1	1	1

Contained By

- LUT3D (page 1823)

Attributes

dim Four integers representing the dimensions of a 3D cube followed by followed by the number of components per entry.

Only 3D LUTs with the same dimension in all three channels are supported, so the first three values must be the same. The fourth value must be 3.

The dim attribute is required, and its value must match the number of entries actually present.

Example

```
<ProcessList id="b5b90615-573b-4959-a838-f35e0e588ac2" version="1.2">
    <LUT3D inBitDepth="32f" interpolation="tetrahedral" outBitDepth="32f">
        <Array dim="48 48 48 3">
 0.000669299  0.000704187  0.000766901
 0.001848371  0.001297010  0.010368987
 0.002599007  0.001144927  0.023227118

<!-- 48*48*48-3 lines omitted -->

        </Array>
```

```
    </LUT3D>
</ProcessList>
```

See Also

- Array (LUT1D) (page 1822)
- Array (Matrix) (page 1826)

Matrix

The Matrix element specifies a matrix for multiplying color values, and optionally applying an offset.

Contains

- Description (page 1817). Optional, any number.
- Array (Matrix) (page 1826). Required, one only.
- DynamicParameter (page 1847). Optional, no more than one. The param attribute must be set to "DEFAULT_LOOK".

Contained By

- ProcessList (page 1813)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1819).

Example

3x3 Matrix

```
<ProcessList id="af6a6c73-aae9-4be6-8051-a796bc480b1c" version="1.2">
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 0.7841929793  0.0904410034  0.1253670007
 0.0445200019  1.0391299725 -0.0836500004
 0.0358299986 -0.3302420080  1.2944129705
        </Array>
    </Matrix>
</ProcessList>
```

4x4 Matrix

```
<ProcessList id="d6783d48-dd96-4c71-99af-f3be0cc9392e" version="1.2">
    <Matrix inBitDepth="32f" outBitDepth="10i">
        <Array dim="4 4 3">
499.99996948   0.00000000   0.00000000  94.99999237
  0.00000000 499.99996948   0.00000000  94.99999237
  0.00000000   0.00000000 499.99996948  94.99999237
  0.00000000   0.00000000   0.00000000   1.00000000
        </Array>
    </Matrix>
 </ProcessList>
```

Array (Matrix)

The Array element specifies an array of values for a LUT or matrix. When it is contained by a Matrix element, the format of its contents is different than when it is contained by LUT1D or LUT3D elements.

Updated

Version 1.3.

Contains

- A series of numeric values specifying the elements of the matrix.

  In a Matrix, the values specify the entries of the matrix which are serialized row by row from top to bottom and from left to right. The scaling of the array values depends on the input and output bit-depths. Color values are treated as column vectors for the purpose of multiplication.

  The scaling of the array values depends on the input and output bit-depths. For example if the input depth is 10i and the output depth is 12i, then a matrix containing entries of 4095/1023 along the main diagonal and entries of 0 everywhere else is the identity transform.

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{bmatrix} \begin{bmatrix} R_{in} \\ G_{in} \\ B_{in} \end{bmatrix} = \begin{bmatrix} R_{out} \\ G_{out} \\ B_{out} \end{bmatrix}$$

$$\begin{aligned} R_{out} &= a_{11}R_{in} + a_{12}G_{in} + a_{13}B_{in} \\ G_{out} &= a_{21}R_{in} + a_{22}G_{in} + a_{23}B_{in} \\ B_{out} &= a_{31}R_{in} + a_{32}G_{in} + a_{33}B_{in} \end{aligned}$$

3×3 Matrix Multiplication

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} & offset_R \\ a_{21} & a_{22} & a_{23} & offset_G \\ a_{31} & a_{32} & a_{33} & offset_B \end{bmatrix} \begin{bmatrix} R_{in} \\ G_{in} \\ B_{in} \\ 1.0 \end{bmatrix} = \begin{bmatrix} R_{out} \\ G_{out} \\ B_{out} \end{bmatrix}$$

$$\begin{aligned} R_{out} &= a_{11}R_{in} + a_{12}G_{in} + a_{13}B_{in} + offset_R \\ G_{out} &= a_{21}R_{in} + a_{22}G_{in} + a_{23}B_{in} + offset_G \\ B_{out} &= a_{31}R_{in} + a_{32}G_{in} + a_{33}B_{in} + offset_B \end{aligned}$$

3×4 Matrix Multiplication

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} & a_{14} \\ a_{21} & a_{22} & a_{23} & a_{24} \\ a_{31} & a_{32} & a_{33} & a_{34} \\ a_{41} & a_{42} & a_{43} & a_{44} \end{bmatrix} \begin{bmatrix} R_{in} \\ G_{in} \\ B_{in} \\ A_{in} \end{bmatrix} = \begin{bmatrix} R_{out} \\ G_{out} \\ B_{out} \\ A_{out} \end{bmatrix}$$

$$R_{out} = a_{11}R_{in} + a_{12}G_{in} + a_{13}B_{in} + a_{14}A_{in}$$
$$G_{out} = a_{21}R_{in} + a_{22}G_{in} + a_{23}B_{in} + a_{24}A_{in}$$
$$B_{out} = a_{31}R_{in} + a_{32}G_{in} + a_{33}B_{in} + a_{34}A_{in}$$
$$A_{out} = a_{41}R_{in} + a_{42}G_{in} + a_{43}B_{in} + a_{44}A_{in}$$

4×4 Matrix Multiplication

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} & a_{14} & offset_R \\ a_{21} & a_{22} & a_{23} & a_{24} & offset_G \\ a_{31} & a_{32} & a_{33} & a_{34} & offset_B \\ a_{41} & a_{42} & a_{43} & a_{44} & offset_A \end{bmatrix} \begin{bmatrix} R_{in} \\ G_{in} \\ B_{in} \\ A_{in} \\ 1.0 \end{bmatrix} = \begin{bmatrix} R_{out} \\ G_{out} \\ B_{out} \\ A_{out} \end{bmatrix}$$

$$R_{out} = a_{11}R_{in} + a_{12}G_{in} + a_{13}B_{in} + a_{14}A_{in} + offset_R$$
$$G_{out} = a_{21}R_{in} + a_{22}G_{in} + a_{23}B_{in} + a_{24}A_{in} + offset_G$$
$$B_{out} = a_{31}R_{in} + a_{32}G_{in} + a_{33}B_{in} + a_{34}A_{in} + offset_B$$
$$A_{out} = a_{41}R_{in} + a_{42}G_{in} + a_{43}B_{in} + a_{44}A_{in} + offset_A$$

4×5 Matrix Multiplication

Contained By

- Matrix (page 1825)

Attributes

dim Specifies the size of the matrix and the number of channels to operate on.

Value	Meaning
`"3 3 3"`	A 3×3 matrix operating on RGB values.
`"3 4 3"`	A 3×4 matrix operating on RGB values. Introduced in version 1.3.
`"4 4 4"`	A 4×4 matrix operating on RGBA values. Introduced in version 1.3.

Value	Meaning
"4 5 4"	A 4×5 matrix operating on RGBA values. Introduced in version 1.3.
"4 4 3"	A 4×4 matrix operating on RGB values. If you use this option, the bottom row of the matrix should be [0 0 0 1]. This option is valid only if the version attribute of the ProcessList element is "1.2". For versions 1.3 and later, use "3 4 3" instead.

Example

3x3 Matrix Operating on RGB Values

```
<ProcessList id="af6a6c73-aae9-4be6-8051-a796bc480b1c" version="1.2">
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 0.7841929793  0.0904410034  0.1253670007
 0.0445200019  1.0391299725 -0.0836500004
 0.0358299986 -0.3302420080  1.2944129705
        </Array>
    </Matrix>
</ProcessList>
```

3x4 Matrix Operating on RGB Values

```
<ProcessList id="53d366de-e200-476f-b3fd-ed1ca7044197" version="1.3">
    <Matrix inBitDepth="32f" outBitDepth="10i">
        <Array dim="3 4 3">
499.99996948   0.00000000   0.00000000  94.99999237
  0.00000000 499.99996948   0.00000000  94.99999237
  0.00000000   0.00000000 499.99996948  94.99999237
        </Array>
    </Matrix>
 </ProcessList>
```

See Also

- Array (LUT1D) (page 1822)
- Array (LUT3D) (page 1823)

Range

The Range element maps the input domain to the output range by scaling and offsetting values.

If a minInValue is present, then minOutValue must also be present and the result is clamped at the low end. Similarly, if maxInValue is present, then maxOutValue must also be present and the result is clamped at the high end. If none of minInValue, minOutValue, maxInValue, or maxOutValue are present, then the Range operator performs only bit-depth conversion.

The scaling of minInValue and maxInValue depends on the input bit-depth, and the scaling of minOutValue and maxOutValue depends the output bit-depth.

If both minimum and maximum values are specified, then the formula for Range is:

$$out = \min\left(\max\left(in\left(\frac{max_{out} - min_{out}}{max_{in} - min_{in}}\right) + min_{out} - min_{in}\left(\frac{max_{out} - min_{out}}{max_{in} - min_{in}}\right),\ min_{out}\right),\ max_{out}\right)$$

If only minimum values are specified, the formula is:

$$out = \max\left(in\left(\frac{\text{size}(outBitDepth)}{\text{size}(inBitDepth)}\right) + min_{out} - min_{in}\left(\frac{\text{size}(outBitDepth)}{\text{size}(inBitDepth)}\right),\ min_{out}\right)$$

If only maximum values are specified, the formula is:

$$out = \min\left(in\left(\frac{\text{size}(outBitDepth)}{\text{size}(inBitDepth)}\right) + max_{out} - max_{in}\left(\frac{\text{size}(outBitDepth)}{\text{size}(inBitDepth)}\right),\ max_{out}\right)$$

Where:

$$size(\text{int}) = 2^{bitDepth} - 1$$

$$size(\text{float}) = 1.0$$

Contains

- Description (page 1817). Optional, any number.
- minInValue (page 1830). Optional, no more than one.
- minOutValue (page 1831). Optional, no more than one.
- maxInValue (page 1830). Optional, no more than one.
- maxOutValue (page 1831). Optional, no more than one.
- DynamicParameter (page 1847). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

Contained By

- ProcessList (page 1813)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1819).

Example

```
<ProcessList id="7f5b1560-fec4-4603-aa8d-c55bf5e22280" version="1.2">
    <Description>10-bit full range to SMPTE (legal) range.</Description>
    <InputDescriptor>generic RGB</InputDescriptor>
    <OutputDescriptor>generic RGB</OutputDescriptor>
    <Range inBitDepth="10i" outBitDepth="10i">
        <minInValue>  0.0000</minInValue>
        <maxInValue>1023.0000</maxInValue>
        <minOutValue> 64.0000</minOutValue>
        <maxOutValue>940.0000</maxOutValue>
    </Range>
</ProcessList>
```

minInValue

The minValue element specifies the minimum input value for a Range element.

Contains

- A single numeric value used for all channels.

Contained By

- Range (page 1828)

Attributes

None.

Example

```
<ProcessList id="7f5b1560-fec4-4603-aa8d-c55bf5e22280" version="1.2">
    <Description>10-bit full range to SMPTE (legal) range.</Description>
    <InputDescriptor>generic RGB</InputDescriptor>
    <OutputDescriptor>generic RGB</OutputDescriptor>
    <Range inBitDepth="10i" outBitDepth="10i">
        <minInValue>  0.0000</minInValue>
        <maxInValue>1023.0000</maxInValue>
        <minOutValue>  64.0000</minOutValue>
        <maxOutValue>940.0000</maxOutValue>
    </Range>
</ProcessList>
```

maxInValue

The maxInValue element specifies the maximum input value for a Range element.

Contains

- A single numeric value used for all channels.

Contained By

- Range (page 1828)

Attributes

None.

Example

```
<ProcessList id="7f5b1560-fec4-4603-aa8d-c55bf5e22280" version="1.2">
    <Description>10-bit full range to SMPTE (legal) range.</Description>
    <InputDescriptor>generic RGB</InputDescriptor>
    <OutputDescriptor>generic RGB</OutputDescriptor>
    <Range inBitDepth="10i" outBitDepth="10i">
        <minInValue>  0.0000</minInValue>
```

```
        <maxInValue>1023.0000</maxInValue>
        <minOutValue> 64.0000</minOutValue>
        <maxOutValue>940.0000</maxOutValue>
    </Range>
</ProcessList>
```

minOutValue

The minOutValue element specifies the minimum output value for a Range element.

Contains

- A single numeric value used for all channels.

Contained By

- Range (page 1828)

Attributes

None.

Example

```
<ProcessList id="7f5b1560-fec4-4603-aa8d-c55bf5e22280" version="1.2">
    <Description>10-bit full range to SMPTE (legal) range.</Description>
    <InputDescriptor>generic RGB</InputDescriptor>
    <OutputDescriptor>generic RGB</OutputDescriptor>
    <Range inBitDepth="10i" outBitDepth="10i">
        <minInValue>  0.0000</minInValue>
        <maxInValue>1023.0000</maxInValue>
        <minOutValue> 64.0000</minOutValue>
        <maxOutValue>940.0000</maxOutValue>
    </Range>
</ProcessList>
```

maxOutValue

The maxOutValue element specifies the maximum output value for a Range element.

Contains

- A single numeric value used for all channels.

Contained By

- Range (page 1828)

Attributes

None.

Example

```
<ProcessList id="7f5b1560-fec4-4603-aa8d-c55bf5e22280" version="1.2">
    <Description>10-bit full range to SMPTE (legal) range.</Description>
    <InputDescriptor>generic RGB</InputDescriptor>
    <OutputDescriptor>generic RGB</OutputDescriptor>
    <Range inBitDepth="10i" outBitDepth="10i">
        <minInValue>  0.0000</minInValue>
        <maxInValue>1023.0000</maxInValue>
        <minOutValue> 64.0000</minOutValue>
        <maxOutValue>940.0000</maxOutValue>
    </Range>
</ProcessList>
```

Gamma

The Gamma element applies a gamma correction to color values.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.2.

Contains

- Description (page 1817). Optional, any number.
- GammaParams (page 1834). Required, maximum three total.
- DynamicParameter (page 1847). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

Contained By

- ProcessList (page 1813)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1819).

style Specifies the type of power law curve. Required.

NOTE The following equations assume that the input and output bit-depths are floating-point. For integers, the values are normalized to [0.0, 1.0]. In other words, the gamma and offset values stored in the CTF file do not depend on the input and output bit depths.

Value	Meaning
`"basicFwd"`	Applies the power law with the gamma value specified in the GammaParams element. The GammaParams element must not have an offset attribute. $$out = (\max(0, in))^{gamma}$$

Value	Meaning
"basicRev"	Applies the power law with the inverse of gamma value. The GammaParams element must not have an offset attribute. $$out = (\max(0, in))^{\frac{1}{gamma}}$$
"moncurve-Fwd"	Applies a function that has a linear segment near the origin (black) and a power-law segment at higher values. At the junction, the curve is continuous in both value and tangent. This type of curve is used in the Rec. 709, sRGB, and CIE L* equations. The GammaParams element must have an offset attribute. $\gamma = \text{gamma} \quad k = \text{offset} \quad x = \text{in} \quad y = \text{out}$ $x \leq \frac{k}{\gamma - 1} \qquad y = \frac{\gamma - 1}{k}\left(\frac{k\gamma}{(\gamma - 1)(1 + k)}\right)^{\gamma} x$ $x > \frac{k}{\gamma - 1} \qquad y = \left(\max\left(0, \left(\frac{x + k}{1 + k}\right)\right)\right)^{\gamma}$
"mon-curveRev"	Applies a function that has a linear segment near the origin (black) and an inverse power-law segment at higher values. The GammaParams element must have an offset attribute. $\gamma = \text{gamma} \quad k = \text{offset} \quad x = \text{in} \quad y = \text{out}$ $x \leq \left(\frac{k\gamma}{(\gamma - 1)(1 + k)}\right)^{\gamma} \qquad y = \left(\frac{\gamma - 1}{k}\right)^{\gamma - 1}\left(\frac{1 + k}{\gamma}\right)^{\gamma} x$ $x > \left(\frac{k\gamma}{(\gamma - 1)(1 + k)}\right)^{\gamma} \qquad y = \left(\max(0, x)\right)^{\frac{1}{\gamma}}(1 + k) - k$

Example

Without Offset

```
<ProcessList id="9e999646-3e76-4374-814c-e4c46c6438de" version="1.2">
    </Matrix>
    <Gamma inBitDepth="32f" outBitDepth="12i" style="basicRev">
        <GammaParams gamma="2.400000000000000" />
    </Gamma>
</ProcessList>
```

With Offset

```
<ProcessList id="1b86aac2-1fdb-4dd8-bc6b-54f3551b9bd4" version="1.2">
    <Gamma inBitDepth="32f" outBitDepth="32f" style="moncurveFwd">
        <GammaParams gamma="2.600000000000000" offset="0.050000" />
```

```
    </Gamma>
</ProcessList>
```

GammaParams

The GammaParams element defines the gamma value and optional offset for a Gamma element.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.2.

Contains

No content.

Contained By

- Gamma (page 1832)

Attributes

channel The color channel to apply gamma correction to. Possible values are "R", "G", and "B". Optional.

If this attribute is not specified, the gamma correction is applied to all channels.

gamma The power value to use. Valid values range from 0.1 to 10.0 inclusive. Required.

offset The offset to use. Valid values range from 10^{-6} to 0.9 inclusive. Optional.

If this attribute is used, the Gamma element's style attribute should be set to "moncurveFwd" or "moncurveRev". If it is not used, the style attribute should be "basicFwd" or "basicRev".

Example

Without Offset

```
<ProcessList id="9e999646-3e76-4374-814c-e4c46c6438de" version="1.2">
    </Matrix>
    <Gamma inBitDepth="32f" outBitDepth="12i" style="basicRev">
        <GammaParams gamma="2.400000000000000" />
    </Gamma>
</ProcessList>
```

With Offset

```
<ProcessList id="1b86aac2-1fdb-4dd8-bc6b-54f3551b9bd4" version="1.2">
    <Gamma inBitDepth="32f" outBitDepth="32f" style="moncurveFwd">
        <GammaParams gamma="2.600000000000000" offset="0.050000" />
    </Gamma>
</ProcessList>
```

Separate Channels

```
<ProcessList id="id">
    <Gamma inBitDepth="32f" outBitDepth="10i" style="basicRev">
```

```
        <GammaParams channel="R" gamma="2.4" />
    <GammaParams channel="G" gamma="2.35" />
    <GammaParams channel="B" gamma="2.2" />
    </Gamma>
</ProcessList>
```

ExposureContrast

The ExposureContrast element simulates adjusting the exposure of an image, and also modifies the contrast between light and dark colors.

It is an Autodesk extension to the Academy/ASC XML color transform format.

If a color transform is used for display, then depending on the application users can still adjust the exposure and contrast interactively as long as the CTF file contains at least one ExposureContrast element with DynamicParameter sub-elements. In this case, the corresponding parameter values defined in the ECParams sub-element are ignored for display only, and instead the values set in the application are applied at each point in the transform chain that an ExposureContrast element with DynamicParameter sub-elements occurs.

Introduced

Version 1.2.

Contains

- Description (page 1817). Optional, any number.
- ECParams (page 1836). Required, one only.
- DynamicParameter (page 1847). Optional, up to 3. Each param attribute must be set to a different value.

Contained By

- ProcessList (page 1813)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1819).

style Declares the input data type. Required.

NOTE The following equations assume that the input and output bit-depths are floating-point. For integers, the values are normalized to [0.0, 1.0]. In other words, the exposure offset, contrast level, and pivot values stored in the CTF file do not depend on the input and output bit depths.

Value	Meaning
`"linear"`	$out = pivot \times \max\left(0,\ \frac{2^{exposure}\, in}{pivot}\right)^{contrast}$
`"video"`	$out = pivot^{\frac{1}{1.83}} \times \left(\max\left(0,\ in \times \left(\frac{2^{exposure}}{pivot}\right)^{\frac{1}{1.83}}\right)\right)^{contrast}$

Value	Meaning
"log"	$out = \left(in + exposure \frac{0.6}{2.046}\log_{10}(2) - \left(\frac{0.6}{2.046}\log_{10}\left(\frac{pivot}{0.9}\right) + \frac{685}{1023}\right)\right)contrast + \left(\frac{0.6}{2.046}\log_{10}\left(\frac{pivot}{0.9}\right) + \frac{685}{1023}\right)$

Example

```
<ProcessList id="561c5aba-8709-477c-8211-7ec4a6fe8820">
   <ExposureContrast inBitDepth="16f" outBitDepth="32f" style="linear">
      <ECParams exposure="0.65" contrast="1.2" pivot="1" />
      <DynamicParameter param="EXPOSURE" />
      <DynamicParameter param="CONTRAST" />
   </ExposureContrast>
</ProcessList>
```

ECParams

The ECParams element specifies the parameters for an ExposureContrast element.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.2.

Contains

No content.

Contained By

- ExposureContrast (page 1835)

Attributes

exposure A single decimal value representing the exposure offset in stops. A value of 0.0 corresponds to no exposure offset. Required.

contrast A single decimal value representing the contrast level. A value of 1.0 corresponds to no contrast adjustment. Required.

pivot A single decimal value representing the color value used as a pivot for adjusting contrast. A value of 0.18 will pivot around mid-gray. Required.

Example

```
<ProcessList id="561c5aba-8709-477c-8211-7ec4a6fe8820">
   <ExposureContrast inBitDepth="16f" outBitDepth="32f" style="linear">
      <ECParams exposure="0.65" contrast="1.2" pivot="1" />
      <DynamicParameter param="EXPOSURE" />
      <DynamicParameter param="CONTRAST" />
   </ExposureContrast>
</ProcessList>
```

Log

The Log operator applies a logarithmic or anti-logarithmic function to convert between linear and Cineon-style encodings.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

- Description (page 1817). Optional, any number.
- LogParams (page 1838). Required if the style attribute is `"logToLin"` or `"linToLog"`. There should be one occurence for values applied to all RGB channels, or three occurences for different values applied to R, G, and B separately.
- DynamicParameter (page 1847). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

Contained By

ProcessList (page 1813)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1819).

style Specifies the formula to use for the conversion. Required.

NOTE The following equations assume that the input and output bit-depths are floating-point. For integers, the values are normalized to [0.0, 1.0]. In other words, the gamma, reference white, reference black, highlight, and shadow values stored in the CTF file do not depend on the input and output bit depths.

In the following equations:

$$\text{FLOAT\_MIN} = 1.1754943508222875 \times 10^{-38}$$

$$linearRefBlack = 10^{\min\left(-0.0001,\ \frac{(refBlack - refWhite) \times 0.002}{gamma}\right)}$$

$$gain = \frac{highlight - shadow}{1 - linearRefBlack}$$

Value	Meaning
`"log10"`	$out = \log_{10}(\max(in,\ \text{FLOAT\_MIN}))$
`"log2"`	$out = \log_{2}(\max(in,\ \text{FLOAT\_MIN}))$

Value	Meaning
"antiLog10"	$out = 10^{in}$
"antiLog2"	$out = 2^{in}$
"logToLin"	$relativeExposure = 10^{\frac{(1023 \times in - refWhite) \times 0.002}{gamma}}$ $out = (relativeExposure - linearRefBlack) \times gain + shadow$
"linToLog"	$relativeExposure = linearRefBlack + \frac{in - shadow}{gain}$ $out = \frac{refWhite + \log_{10}(\max(\text{FLOAT\_MIN},\ relativeExposure)) \times \frac{gamma}{0.002}}{1023}$

Example

All Channels

```
<ProcessList id="19510ea6-69a0-4d3d-bc75-d0b36d458f62" name="Log op 1"
version="1.3">
     <Log inBitDepth="16f" outBitDepth="12i" style="linToLog">
        <LogParams gamma="0.6" refWhite="685" refBlack="95" highlight="1.0"
 shadow="0.0005"/>
    </Log>
</ProcessList>
```

Separate Channels

```
<ProcessList id="19510ea6-69a0-4d3d-bc75-d0b36d458f62" name="Log op 1"
version="1.3">
     <Log inBitDepth="16f" outBitDepth="12i" style="linToLog">
        <LogParams channel="R" gamma="0.5" refWhite="685" refBlack="95"
highlight="1.0" shadow="0.0005"/>
        <LogParams channel="G" gamma="0.6" refWhite="685" refBlack="95"
highlight="1.0" shadow="0.0005"/>
        <LogParams channel="B" gamma="0.65" refWhite="685" refBlack="95"
highlight="1.0" shadow="0.0005"/>
    </Log>
</ProcessList>
```

LogParams

The LogParams element defines the parameter values used by a Log element.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

No content.

Contained By

Log (page 1837)

Attributes

channel Specifies the channel to apply the parameter values to. Possible values are `"R"`, `"G"`, and `"B"`.

Optional. If it is omitted, the values are applied to each of the RGB channels.

gamma Combined film and video gamma (contrast in log space). A typical value is 0.6.

refWhite Value in log space that maps to highlight in linear space. Specified on a 10-bit scale (e.g. 685 is typical).

refBlack Value in log space that maps to shadow in linear space. Specified on a 10-bit scale (e.g. 95 is typical).

highlight Value in linear space that maps to refWhite in log space. Specified on a floating-point scale (e.g. 1.0 is typical). Note that log values above refWhite are not clamped and will map to values above highlight in linear space.

shadow Value in linear space that maps to refBlack in log space. Specified on a floating-point scale (e.g. 0.0 is typical). Note that log values below refBlack are not clamped and will map to values below shadow in linear space (possibly even negative).

Example

All Channels

```
<ProcessList id="19510ea6-69a0-4d3d-bc75-d0b36d458f62" name="Log op 1"
version="1.3">
     <Log inBitDepth="16f" outBitDepth="12i" style="linToLog">
          <LogParams gamma="0.6" refWhite="685" refBlack="95" highlight="1.0"
shadow="0.0005"/>
     </Log>
</ProcessList>
```

Separate Channels

```
<ProcessList id="19510ea6-69a0-4d3d-bc75-d0b36d458f62" name="Log op 1"
version="1.3">
     <Log inBitDepth="16f" outBitDepth="12i" style="linToLog">
          <LogParams channel="R" gamma="0.5" refWhite="685" refBlack="95"
highlight="1.0" shadow="0.0005"/>
     <LogParams channel="G" gamma="0.6" refWhite="685" refBlack="95"
highlight="1.0" shadow="0.0005"/>
     <LogParams channel="B" gamma="0.65" refWhite="685" refBlack="95"
highlight="1.0" shadow="0.0005"/>
     </Log>
</ProcessList>
```

ASC_CDL

The ASC_CDL operator represents a color transform in the American Society of Cinematographers' Color Decision List format (*.ccc or *.cdl).

An ASC CDL transform consists of a slope-offset-power (SOP) function applied individually to each of the RGB channels, followed by a saturation function applied equally to all channels.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

- Description (page 1817). Optional, any number.
- SOPNode (page 1841). Optional, no more than one. If absent, no slope-offset-power function is applied.
- SatNode (page 1844). Optional, no more than one. If absent, no saturation function is applied.
- DynamicParameter (page 1847). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

Contained By

- ProcessList (page 1813)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1819).

style Determines the formula applied by the operator.

NOTE The equations assume that the input and output bit-depths are floating-point. For integers, the values are normalized to [0.0, 1.0]. In other words, the slope, offset, power, and saturation values stored in the CTF file do not depend on the input and output bit depths.

NOTE The luma weights assume that the image uses the primaries specified by Rec. 709 (for example, HD and sRGB).

Value	Meaning
`"v1.2_Fwd"`	$out_{SOP} = in_{SAT} = (\text{clamp}_{[0,1]}(in \times slope + offset))^{power}$ $luma = 0.2126 \times in_{SAT_R} + 0.7152 \times in_{SAT_G} + 0.0722 \times in_{SAT_B}$ $out = \text{clamp}_{[0,1]}(luma + saturation \times (in_{SAT} - luma))$
`"v1.2_Rev"`	The inverse of `"v1.2_Fwd"`.
`"noClampFwd"`	The same as `"v1.2_Fwd"`, but without the two clamp functions. If $(in \times slope + offset) < 0$

Value	Meaning
	then no power function is applied.
"noClampRev"	The inverse of "noClampFwd".

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
    <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

SOPNode

The SOPNode element defines the slope-offset-power function of an ASC CDL color transform.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

- Slope (page 1842). Optional, no more than one. If absent, the default is 1.0 for all channels, that is, no gain is applied.
- Offset (page 1843). Optional, no more than one. If absent, the default is 0.0 for all channels, that is, no offset is applied.
- Power (page 1843). Optional, no more than one. If absent, the default is 1.0 for all channels, that is, no gamma is applied.

Contained By

- ASC_CDL (page 1840)

Attributes

None.

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
    <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

Slope

The Slope element contains the RGB slope values of the slope-offset-power function of an ASC CDL color transform.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

Three decimal values representing the R, G, and B slope (gain) values.

Contained By

- SOPNode (page 1841)

Attributes

None.

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
    <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

Offset

The Offset element contains the RGB offset values of the slope-offset-power function of an ASC CDL color transform.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

Three decimal values representing the R, G, and B offset values.

Contained By

- SOPNode (page 1841)

Attributes

None.

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
   <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

Power

The Power element contains the RGB power values of the slope-offset-power function of an ASC CDL color transform.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

Three decimal values representing the R, G, and B power (gamma) values.

Contained By

- SOPNode (page 1841)

Attributes

None.

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
   <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

SatNode

The SatNode element defines the saturation function of an ASC CDL color transform.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

- Saturation (page 1845). Required, one only.

Contained By

- ASC_CDL (page 1840)

Attributes

None.

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
   <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
```

```
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

Saturation

The Saturation element defines the value used by the saturation function of an ASC CDL color transform.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

A single decimal value applied to all color channels.

Contained By

- SatNode (page 1844)

Attributes

None.

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
   <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

Reference

The Reference element refers to another CTF file by path or by alias. The external file is imported into the corresponding location of the ProcessList.

It is an Autodesk extension to the Academy/ASC XML color transform format.

NOTE When exporting a chain of transforms that includes a Reference element, the reference is replaced by the contents of the referred file.

Introduced

Version 1.2.

Contains

- Description (page 1817). Optional, any number.
- DynamicParameter (page 1847). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

Contained By

- ProcessList (page 1813)

Attributes

It is necessary to specify either a path or an alias attribute, but not both.

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1819).

Note that the values specified here override the values in the external file.

alias A short name for the path and file name of another CTF file. The alias must be defined in a Ref (page 1852) element in the SynColor configuration file (page 1848). By default, the following aliases are available, and can be changed from the user interface of some applications that support Autodesk Color Management:

- The `"graphicsMonitor"` alias should refer to the appropriate transform for display on the workstation monitor.
- The `"broadcastMonitor"` alias should refer to the appropriate transform for display on the broadcast monitor.
- The `"currentMonitor"` alias gets resolved to either the value of the `"graphicsMonitor"` alias or the value of the `"broadcastMonitor"` alias depending on where an image is displayed.
- The `"defaultLook"` alias should refer to the transform that re-creates the look used on set.

You can create additional aliases as desired.

path The path and file name of another CTF file. The path is relative to basePath if that attribute is defined; otherwise, the path is absolute.

basePath The base location for a relative path.

Value	Meaning
Autodesk	The installation location of the color transforms supplied with Autodesk applications. The exact path is resolved based on the version of Autodesk Color Management used by the running application.
Shared	The shared location for custom color transforms for any application that uses Autodesk Color Management. The location is set in the SharedHome (page 1851) of the SynColor configuration file (page 1848).

Example

```
<ProcessList id="561c5aba-8709-477c-8211-7ec4a6fe8820">
    <Reference basePath="Autodesk" inBitDepth="32f" outBitDepth="32f"
path="RRT+ODT/ACES_to_CIE-XYZ_v0.1.1.ctf" />
    <Reference alias="graphicsMonitor" inBitDepth="32f" outBitDepth="8i" />

    <Reference inBitDepth="8i" outBitDepth="8i" path="lut1d_example.xml"
basePath="Shared" />
    <Reference inBitDepth="8i" outBitDepth="8i" path="/zeus/toto/toto.xml"
 />
</ProcessList>
```

DynamicParameter

The DynamicParameter element signals to a compatible application that a parameter is dynamic. If the transform is used for display, the parameter value defined in the CTF file is ignored and the value set in the application is used instead.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Updated

Version 1.3.

Contains

No content.

Contained By

- Any operator element (LOOK_SWITCH)
- ExposureContrast (page 1835) (EXPOSURE, CONTRAST).

Attributes

param Specifies the parameter that will be overridden by the application when the CTF file is applied to the display.

Value	Meaning
"EXPOSURE"	Allows the exposure value to be overridden when the transform is used for display in applications that support this ability. This value can be used only when the DynamicParameter element is a child of an ExposureContrast (page 1835) element.
"CONTRAST"	Allows the contrast value to be overridden when the transform is used for display in applications that support this ability. This value can be used only when the DynamicParameter element is a child of an ExposureContrast (page 1835) element.

Value	Meaning
"LOOK_SWITCH"	Allows the operator's effect to be toggled on and off when the transform is used for display in applications that support this ability. This can be used in conjunction with the bypass (page 1819) attribute of the parent element to create operators that will never affect values rendered to file but that can still be toggled on and off for display. Introduced in version 1.3.

Example

```
<ProcessList id="561c5aba-8709-477c-8211-7ec4a6fe8820">
    <ExposureContrast inBitDepth="16f" outBitDepth="16f" style="linear">
        <ECParams contrast="1.0000" exposure="0.0000" pivot="0.1800" />
        <DynamicParameter param="EXPOSURE" /><DynamicParameter
param="CONTRAST" />
    </ExposureContrast>
    <Reference basePath="Autodesk" inBitDepth="16f" outBitDepth="16f"
path="misc/default_look-ACESproxy.ctf" bypass="true">
        <DynamicParameter param="LOOK_SWITCH" />
    </Reference>
    <Reference basePath="Autodesk" inBitDepth="16f" outBitDepth="16f"
path="RRT+ODT/ACES_to_CIE-XYZ_v0.1.1.ctf" />
    <Reference alias="currentMonitor" inBitDepth="16f" outBitDepth="10i" />
<</ProcessList>
```

Comments

You can include standard XML comments in CTF files.

However if you want the information to appear in the metadata panel of Autodesk applications, you can use Description (page 1817) elements instead of comments.

XML comments begin with:

```
<!--
```

and end with:

```
-->
```

Comments cannot appear inside tags (between < and >). In addition, comments cannot be nested.

SynColor Configuration File

The SynColor configuration file is an XML file that controls how Autodesk Color Management works for all applications on the same workstation.

Some applications might read this file only on start-up. You may need to restart the application after modifying this file.

The directory location depends on the operating system:

- Windows:

```
C:\ProgramData\Autodesk\Synergy\SynColor\<version>\synColorConfig.xml
```

- Linux:

```
/opt/Autodesk/Synergy/SynColor/<version>/synColorConfig.xml
```

- Mac OS X:

```
/Applications/Autodesk/Synergy/SynColor/<version>/synColorConfig.xml
```

Anatomy of the SynColor Configuration File

This section shows a sample of a SynColor configuration file.

```
<?xml (page 1812) version="1.0" encoding="UTF-8"?>
<SynColorConfig (page 1849) version="1.0">
   <SharedHome (page 1851)
dir="/opt/Autodesk/Synergy/SynColor/Shared/transforms"/>
   <ReferenceTable (page 1851)>
      <Ref (page 1852) alias="broadcastMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/broadcast/CIE-XYZ_to_HD-video.ctf"
 />
      <Ref alias="defaultLook"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/misc/identity.ctf"
/>
      <Ref alias="graphicsMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/Eizo/CIE-XYZ_to_CG245W.ctf"
 />
   </ReferenceTable>
</SynColorConfig>
```

SynColorConfig

The SynColorConfig element is the root element of the SynColor configuration file.

Contains

In any order:

- SharedHome (page 1851). Required, one only.
- ReferenceTable (page 1851). Required, one only.

Contained By

- no parent

Attributes

version The version of the SynColor configuration file format.

Example

```
<?xml version="1.0" encoding="UTF-8"?>
<SynColorConfig version="1.0">
   <SharedHome dir="/opt/Autodesk/Synergy/SynColor/Shared/transforms"/>
   <ReferenceTable>
```

```
      <Ref alias="broadcastMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/broadcast/CIE-XYZ_to_HD-video.ctf"
 />
      <Ref alias="defaultLook"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/misc/identity.ctf"
/>
      <Ref alias="graphicsMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/Eizo/CIE-XYZ_to_CG245W.ctf"
 />
  </ReferenceTable>
</SynColorConfig>
```

AutoConfigure

The AutoConfigure element determines whether certain options get automatically configured when you start an application that supports Autodesk Color Management.

Currently this works only for Autodesk Smoke on Mac OS X.

Contains

- No content.

Contained By

- SynColorConfig (page 1849)

Attributes

graphicsMonitor Uses the ICC profile set in your operating system's preferences as the graphicsMonitor alias.

Example

```
<?xml version="1.0" encoding="UTF-8"?>
<SynColorConfig version="1.0">
   <AutoConfigure graphicsMonitor="true" />
  <SharedHome dir="/Applications/Autodesk/Synergy/SynColor/Shared/transforms"
 />
   <ReferenceTable>
      <Ref alias="broadcastMonitor"
path="/Applications/Autodesk/Synergy/SynColor/2013.3/transforms/display/broadcast/CIE-XYZ_to_HD-video.ctf"
 />
      <Ref alias="defaultLook"
path="/Applications/Autodesk/Synergy/SynColor/2013.3/transforms/misc/identity.ctf"
 />
      <Ref alias="graphicsMonitor"
path="/Library/ColorSync/Profiles/Displays/Color
LCD-00000610-0000-9CCF-0000-0000042733C0.icc" />
   </ReferenceTable>
</SynColorConfig>
```

SharedHome

The SharedHome element specifies the location of the Shared directory. This is a useful location for storing your own custom transforms that can be used by other applications that use Autodesk Colour Management installed on the same computer.

Some applications allow you to set this value in their preferences.

Contains

- No content.

Contained By

- SynColorConfig (page 1849)

Attributes

dir Full path to the shared directory. This can be on the local or a remote computer.

Example

```
<?xml version="1.0" encoding="UTF-8"?>
<SynColorConfig version="1.0">
   <SharedHome dir="/opt/Autodesk/Synergy/SynColor/Shared/transforms"/>
   <ReferenceTable>
      <Ref alias="broadcastMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/broadcast/CIE-XYZ_to_HD-video.ctf"
 />
      <Ref alias="defaultLook"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/misc/identity.ctf"
/>
      <Ref alias="graphicsMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/Eizo/CIE-XYZ_to_CG245W.ctf"
 />
   </ReferenceTable>
</SynColorConfig>
```

ReferenceTable

The ReferenceTable element is a container for Ref elements defining aliases, or short names for CTF files.

Contains

- Ref (page 1852). Optional, any number.

  However, if there are no Ref elements to define the following aliases, then some Autodesk-supplied transforms will not work:

  - The `graphicsMonitor` alias should set to the appropriate transform for display on the workstation monitor.
  - The `broadcastMonitor` alias should set to the appropriate transform for display on the broadcast monitor.
  - The `defaultLook` alias should be set to the transform that you are using to re-create the look used on set.

Users can define additional aliases for other transform files as they wish.

Contained By

- SynColorConfig (page 1849)

Attributes

None.

Example

```
<?xml version="1.0" encoding="UTF-8"?>
<SynColorConfig version="1.0">
   <SharedHome dir="/opt/Autodesk/Synergy/SynColor/Shared/transforms"/>
   <ReferenceTable>
      <Ref alias="broadcastMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/broadcast/CIE-XYZ_to_HD-video.ctf"
 />
      <Ref alias="defaultLook"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/misc/identity.ctf"
/>
      <Ref alias="graphicsMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/Eizo/CIE-XYZ_to_CG245W.ctf"
 />
   </ReferenceTable>
</SynColorConfig>
```

Ref

The Ref element defines an alias for a CTF file so that it can be easily referenced in other CTF files.

Some applications allow users to set certain aliases, like graphicsMonitor, in their preferences.

When an alias is changed, references that use it might not get updated until the application is restarted.

Contains

- No content.

Contained By

- ReferenceTable (page 1851)

Attributes

alias A unique name for the alias. Required.

path The path and file name of the corresponding CTF file. Required.

basePath A base path that is prefixed to the path attribute value to fully resolve the file or location. Optional.

If this attribute is used, then the path attribute is interpreted as relative to the base path. Otherwise, the path attribute is interpreted as an absolute path.

Example

```
<?xml version="1.0" encoding="UTF-8"?>
<SynColorConfig version="1.0">
   <SharedHome dir="/opt/Autodesk/Synergy/SynColor/Shared/transforms"/>
   <ReferenceTable>
      <Ref alias="broadcastMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/broadcast/CIE-XYZ_to_HD-video.ctf"
 />
      <Ref alias="defaultLook"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/misc/identity.ctf"
/>
      <Ref alias="graphicsMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.1/transforms/display/Eizo/CIE-XYZ_to_CG245W.ctf"/>

   </ReferenceTable>
</SynColorConfig>
```

Artist Control Surfaces

30

Avid Artist Transport Control Surface Layout

The Avid Artist Series Color and Transport panels can be used with Flame on the Mac platform.

NOTE To use the Avid Artist Transport, the EUCON utility must be running before launching Flame.

Use the Avid Artist Transport to navigate the clips in the timeline, as well as to mark the clips and the timeline. Use the numeric keypad just like you would a regular keypad.

The Artist Transport has two main components, the wheel and its buttons, and the soft keys, all of which can be reassigned using the EUCON utility.

Wheel Controls

There are two sets of wheel controls. Switch between the sets using the Shift key on the Artist Transport.

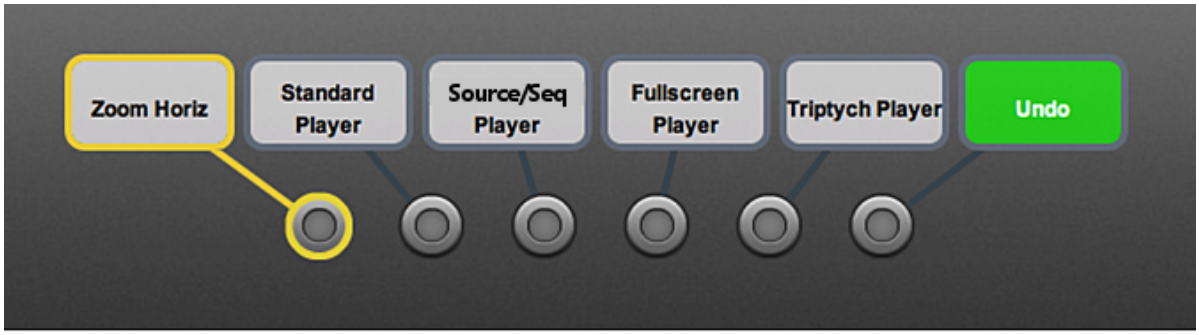
Standard mapping

Alternate mapping accessed using the Shift key.

Soft Keys Controls

There are two sets of Soft Keys. Switch between sets using the Shift key on the Artist Transport.

Standard mapping



Alternate mapping accessed using the Shift key.

Avid Artist Color Control Surface Layout

Use the Avid Artist Color to navigate or control the timeline, Timeline FX, or control editors such as the Colour Corrector or the Colour Warper.

NOTE To use the Avid Artist Color on a Mac system, the EUCON utility must be running before launching Flame.

Timeline Navigation and Timeline FX

Use the Artist Color to jog, zoom, and navigate timelines and clips.

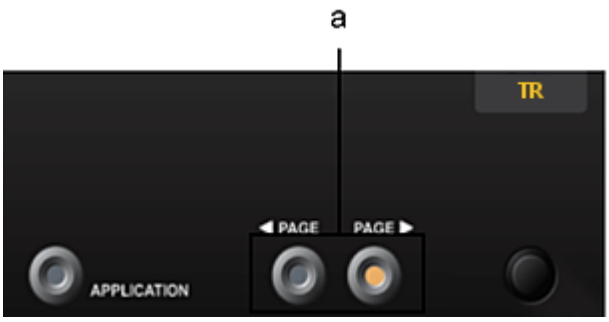
You can also access Timeline FX using the Artist Color. Video Timeline FX are available when a video segment is selected. Audio Timeline FX are available when an audio segment is selected.

The displays located at the top of the Artist Color only display six Timeline FX at a time. Use the Page buttons to display additional Timeline FX.

NOTE The Action Timeline FX editor is not supported by Artist Color.

To display additional Timeline FX:

1 Use the Page buttons and navigate the available Timeline FX.



(a) Page buttons

To enable a Timeline FX:

1. Enable the NAV and Shift keys on the Artist Color.
2. Lightly touch the Timeline FX's corresponding Soft Knob.

To delete a Timeline FX:

1. Enable the NAV and Shift keys on the Artist Color.
2. Press the Timeline FX's corresponding Soft Knob.

To edit a Timeline FX from the Quick menu:

1. Enable the NAV key on the Artist Color.
2. Lightly touch the Timeline FX's corresponding Soft Knob.
3. Disable the NAV key on the Artist Color, and then edit the Timeline FX.

To enter a Timeline FX's editor:

1. Enable the NAV key on the Artist Color.
2. Press the Timeline FX's Soft Knob.

Main Timeline navigation controls (Page 1)

Main Timeline navigation controls (Page 2)

When the NAV button is enabled, you can navigate the Timeline FX pipeline by lightly touching the Timeline FX's corresponding Soft Knob.

Alternate timeline navigation controls accessed using the Shift key.

Use the CG to cut and the associated PG buttons to paste Timeline FX form one selected segment to another selected segment. Only the active Timeline FX is copied. Shift key + CG key to clear out the stored Timeline FX.

To cut and paste Timeline FX between segments:

1. Select the timeline segment with the Timeline FX to copy.
2. Press one of the Copy Grade keys, identified as CG1 through CG4.
3. Select the timeline to which you want to apply the copied Timeline FX.
4. Press the matching Paste Grade key to paste the Timeline FX, such as PG1 for a Timeline FX stored in CG1.

Source/Sequence Player Navigation

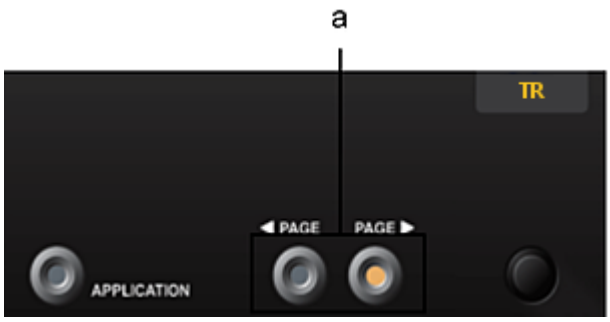
In the Source/Sequence Player, use the Artist Color to jog each Player. Use it to mark footage from the Source player, and then insert it into the Sequence.

Triptych Player Navigation

When in the Triptych Player, use the Artist Color to jog each panel using one of the assigned trackwheels.

Colour Corrector

The Colour Corrector controls are accessed over five pages that you navigate, using the Next / Previous Page buttons.



(a) Page buttons

Colour Corrector Page 1

Colour Corrector Page 2

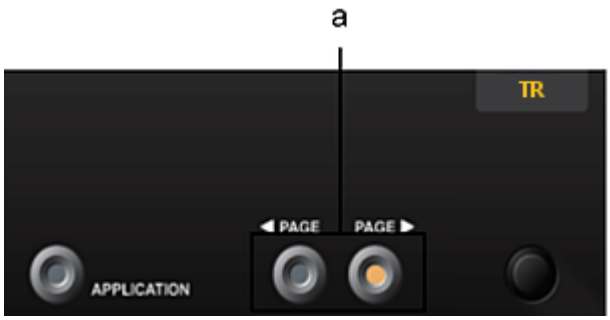
Colour Corrector Page 3

Colour Corrector Page 4

Colour Corrector Page 5 (Suppression)

Colour Warper

The Colour Warper controls are accessed over four pages that you navigate, using the Next / Previous Page buttons.



(a) Page buttons

Colour Warper Page 1

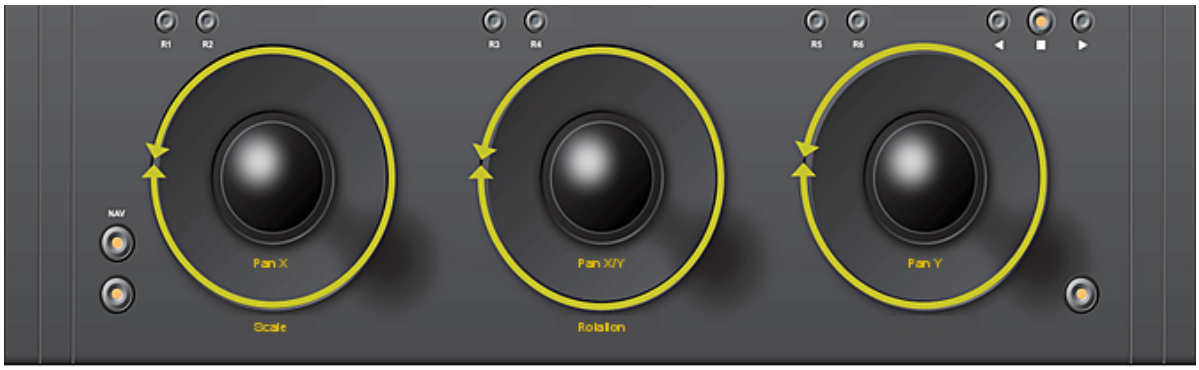
Colour Warper Page 2

Colour Warper Page 3

Colour Warper Page 4

2D Transform

The 2D Transform controls are editable using the trackballs and rings of the control surface.



Audio Effects Editor

Audio Timeline FX are handled in exactly the same way as other Timeline FX. See Timeline Navigation and Timeline FX (page 1857).

NOTE Audio effects are only available when an audio segment is selected.

EQ Timeline FX

Along with the Soft Knobs, some EQ parameters can be controlled with the trackballs and rings.

Tangent Element Control Surface

31

The Tangent Element control surface is designed to improve interactivity when working in Flame. Adding to the colour grading functions for use with Lustre, the Tangent Elements panel supports navigation and interaction with the Workspace, the thumbnails, the reels, the players and the timeline and Timeline FX, as well as the Colour Corrector and the Colour Warper in Flame.

NOTE The Tangent Element control surface has expanded functionality in Lustre, since colour grading tasks are particularly well suited for use with it. For more information, see the Tangent Element Control Surface in the Lustre user guide.

Installing the Control Surface

The first time you connect the Tangent Element Control Surface to your Flame workstation, it automatically connects to your software and populates all the menus.

If the Control Surface does not automatically connect, see Tangent Troubleshooting (page 1915).

The Modes

The basic Tangent Element workflow is mode based. The panel automatically sets itself to the appropriate mode, based on where you are in the application. Alternatively, you can manually set the panel to the mode of your choice and the application's user interface follows. For example, accessing the Colour Corrector from the user interface automatically sets the panel to the appropriate Colour Corrector mode (Master, Shadows, Midtone, Highlights). And setting the CC Master mode on the panel, opens the Colour Corrector in Master mode in the user interface.

The following modes are supported by the Tangent Element Control Surface in Flame:

Media Panel Focus Mode

This mode focuses on the Media panel and enables you to scrub the selected clip, as well as navigate the current reel within the Media panel, select, scrub and play clips in the players. Additionally, you can perform insert, overwrite and replace edits in the timeline, from the Media panel.

Thumbnail Focus Mode

This mode focuses on the Viewing panel in Thumbnails view and enables you to scrub the selected clip, as well as navigate through and select, scrub and play clips in the players. You can also perform insert, overwrite and replace edits in the timeline.

Reels Focus Mode

This mode focuses on the Viewing panel in Reels view and enables you to scrub the selected clip, as well as navigate the reels and select, scrub and play clips in the players. Additionally, you can perform insert, overwrite and replace edits in the timeline.

Timeline Focus Mode

This mode focuses on the timeline and enables you to navigate the timeline, its layers and tracks. This mode also supports panning and zooming the timeline via the trackball and wheel. In addition to the ability to perform insert, overwrite and replace edits, you can also add Timeline FX and transitions. All of the Timeline FX are supported.

NOTE The Tangent panel only supports the Colour Corrector and Colour Warper Timeline FX editors for input parameters.

Player Modes

When you bring up a player, either using the Tangent panel or from the user interface, the panel enters one of the player modes, based on the selected player. There are four player modes:

- 1Up Player
- Src-Seq Player
- Triptych Player
- Full Screen Player

From the Player modes, you can play and navigate through clips and sequences, drop markers on clips and toggle the different players.

Colour Corrector Modes

These modes focus on the Colour Corrector Timeline FX and enable you to control all of the creative settings within the Colour Corrector, including the Colour Corrector Quick Bar.

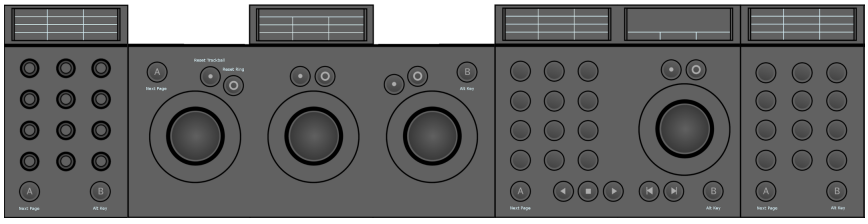
NOTE The Colour Corrector Batch node and the Colour Corrector Tool in the Tools tab are not supported.

Colour Warper Mode

This mode focuses on the Colour Warper Timeline FX and enables you to control all of the creative settings within the Colour Warper, including the Colour Warper Quick Bar.

NOTE The Colour Warper Batch node and the Colour Warper Tool in the Tools tab are not supported.

Using the Modules



The Tangent Element Control Surface is composed of four modules.

The layout of the displays of each module mirrors the functions of the knobs/buttons of each module.

KB module (Knobs)

The KB module is a knob-based interface for user input parameters.

The values of each parameter assigned to the knobs can be reset by pressing the knob.

A button: Cycle through the different pages of functions mapped to the module.

B button:

- Alt mode: Press to enable the alternate functions mapped across the entire panel.
- Precise mode: Hold to enable Precise Mode.

NOTE You can press the B button twice and lock Alt mode.

TK module (Trackballs)

The TK module is also an interface for user input parameters. The trackballs and rings provide an intuitive way of precisely modifying parameters and are especially well suited for grading, among other things.

Trackballs: You can modify parameters by spinning the trackballs left / right or up / down based on the current mode.

Rings: You can modify parameters by turning the rings left / right.

The values of the parameters assigned to the trackballs and the rings can be reset by pressing the Reset button above them.

A button: Cycle through the different pages of functions mapped to the module.

B button:

- Alt mode: Press to enable the alternate functions mapped across the entire panel.
- Precise mode: Hold to enable Precise Mode.

NOTE You can press the B button twice and lock Alt mode.

MF module (Multi-Function)

The MF module enables you to select the mode you want to work in and provides a transport section, which allows navigation via the trackball and ring as well.

Trackball: You can modify parameters by spinning the trackball left / right or up / down based on the current mode.

Ring: You can modify parameters by turning the ring left / right.

The values of the parameters assigned to the trackballs and the rings can be reset by pressing the Reset button above them.

A button: Cycle through the different pages of functions mapped to the module.

B button:

- Alt mode: Press to enable the alternate functions mapped across the entire panel.
- Precise mode: Hold to enable Precise Mode.

NOTE You can press the B button twice and lock Alt mode.

Button sequences for alternate functions on the MF module

- **Next Element Button + B Button:** Move the positioner forward to the next keyframe in your timeline .
- **Previous Element Button + B Button:** Move the positioner backward to the previous keyframe in your timeline.
- **Stop Button + Forward Play button:** Move the positioner forward by one frame in your timeline.
- **Stop Button + Backward Play button:** Move the positioner backward by one frame in your timeline.

BT module (Buttons)

The BT module is dedicated to providing quick access to the editing operations and to the Timeline FX.

A button: Cycle through the different pages of functions mapped to the module.

B button:

- Alt mode: Press to enable the alternate functions mapped across the entire panel.
- Precise mode: Hold to enable Precise Mode.

NOTE You can press the B button twice and lock Alt mode.

Control Surface Layouts

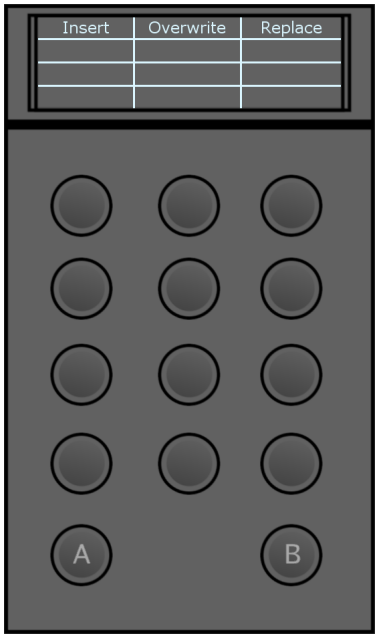
Media Panel Focus Layout

Module 3 (MF)

- **MediaPanl:** Enable Media Panel Focus mode.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.

- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Go to 1Up player mode and backward play the currently selected clip.
- **Backward Play + Stop buttons:** Move the clip displayed in the player one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the clip displayed in the player one frame forward.
- **Forward Play button:** Go to 1Up player mode and forward play the currently selected clip.
- **Previous button:** Move the selection in the current reel within the Media panel down by one increment.
- **Next button:** Move the selection in the current reel with in the Media panel up by one increment.

Module 4 (BT)



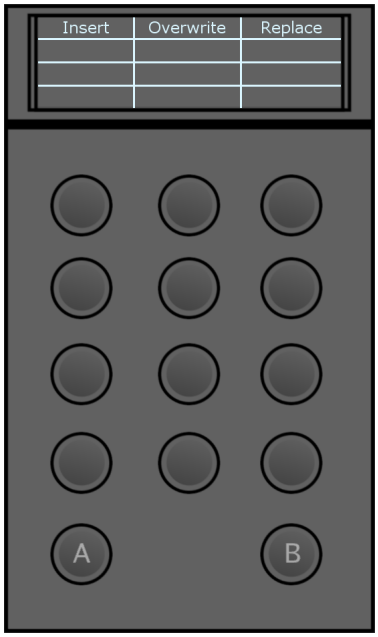
- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.

Thumbnail Focus Layout

Module 3 (MF)

- **Thumbnail:** Enable Thumbnail Focus mode.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Go to full screen player mode and backward play the currently selected clip.
- **Backward Play + Stop buttons:** Move the currently selected thumbnail one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the currently selected thumbnail one frame forward.
- **Forward Play button:** Go to full screen player mode and forward play the currently selected clip.
- **Previous button:** Select the previous thumbnail.
- **Next button:** Select the next thumbnail.

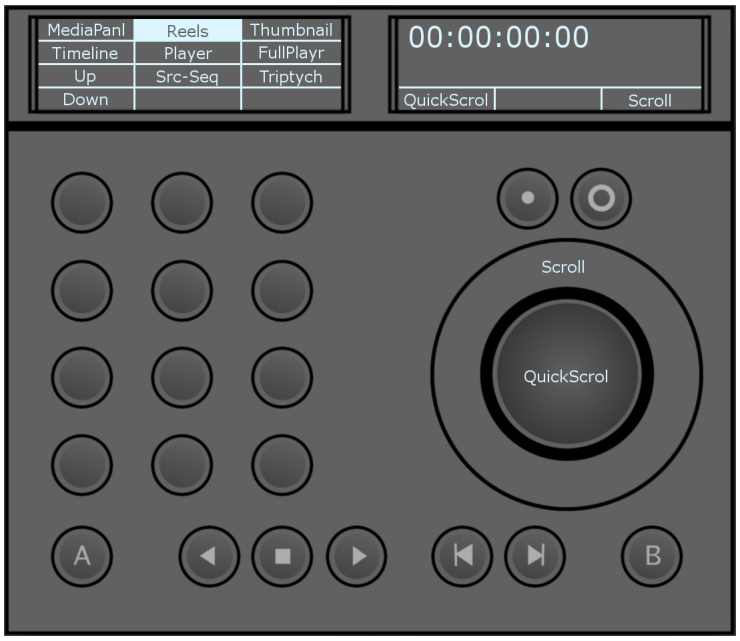
Module 4 (BT)



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.

Reels Focus Layout

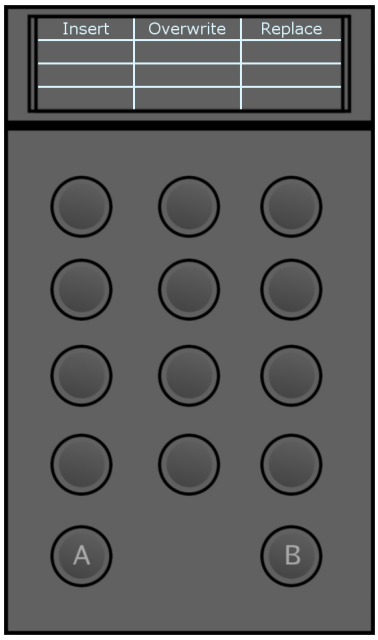
Module 3 (MF)



- **Reels:** Enable Reels Focus mode.

- **Up:**Move the selection in the Media panel up by one increment.
- **Down:** Move the selection in the Media panel down by one increment.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Go to the full screen player mode and backward play the currently selected clip.
- **Backward Play + Stop buttons:** Move the currently selected thumbnail one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the currently selected thumbnail one frame forward.
- **Forward Play button:** Go to the full screen player mode and forward play the currently selected clip.
- **Previous button:** Select the previous thumbnail.
- **Next button:** Select the next thumbnail.

Module 4 (BT)



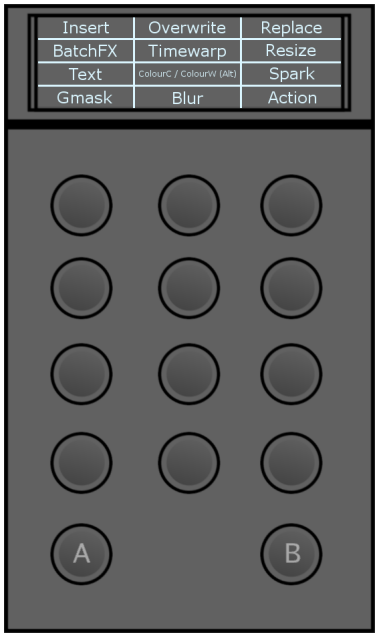
- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.

Timeline Focus Layout

Module 3 (MF)

- **Timeline:** Enable Timeline Focus mode.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip/sequence in the timeline backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip/sequence in the timeline backward or forward.
- **Backward Play button:** Go to the 1Up player mode and backward play the currently selected clip.
- **Backward Play + Stop buttons:** Move the timeline positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the timeline positioner one frame forward.
- **Forward Play button:** Go to the 1Up player mode and backward play the currently selected clip.
- **Previous button:** Move the timeline positioner to the previous transition.
- **Next button:** Move the timeline positioner to the next transition.

Module 4 (BT) - Page 1 of 2



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.
- **BatchFX:** Single press to apply a BatchFX to the currently selected timeline segment(s). You are taken into Batch.

  NOTE If the selected timeline segment already has a BatchFX applied to it, double pressing the button takes to into Batch for editing.

- **Timewarp:** Single press to apply a Timewarp Timeline FX to the currently selected timeline segment. Double press to enter the Timewarp editor.
- **Resize:** Single press to apply a Resize Timeline FX to the currently selected timeline segment. Double press to enter the Resize editor.
- **Text:** Single press to apply a Text Timeline FX to the currently selected timeline segment. Double press to enter the Text editor.
- **ColourC:** Single press to apply a Colour Corrector Timeline FX to the currently selected timeline segment. Double press to enter the Colour Corrector editor.

  NOTE The Tangent panel only supports the Colour Corrector and Colour Warper Timeline FX editors for input parameters.

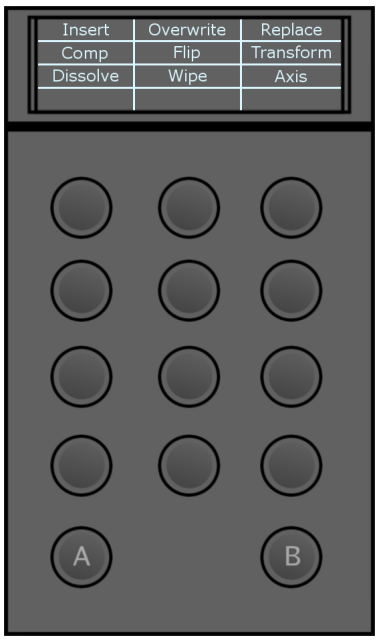
- **ColourW (Alt):** Single press (while holding Alt) to apply a Colour Warper Timeline FX to the currently selected timeline segment. Double press (while holding Alt) to enter the Colour Warper editor.

  NOTE The Tangent panel only supports the Colour Corrector and Colour Warper Timeline FX editors for input parameters.

- **Spark:** Press to apply a placeholder Spark. Click the Load Spark button (L), with your mouse or pen, to open the Sparks Browser and navigate to the appropriate Spark.
- **Gmask:** Single press to apply a Gmask Timeline FX to the currently selected timeline segment. Double press to enter the Gmask editor.

- **Blur:** Single press to apply a Blur Timeline FX to the currently selected timeline segment. Double press to enter the Blur editor.
- **Action:** Single press to apply an Action Timeline FX to the currently selected timeline segment(s). Double press to enter the Action editor.

Module 4 (BT) - Page 2 of 2



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.
- **Comp:** Single press to apply a Comp Timeline FX to the currently selected timeline segment. Double press to enter the Comp editor.
- **Flip:** Single press to apply a Flip Timeline FX to the currently selected timeline segment. Double press to enter the Flip editor.
- **Transform:** Single press to apply a Transform Timeline FX to the currently selected timeline segment. Double press to enter the Transform editor.
- **Dissolve:** Single press to apply a Dissolve video transition to the currently selected timeline segment(s). Double press to enter the Dissolve editor.
- **Wipe:** Single press to apply a Wipe video transition to the currently selected timeline segment(s). Double press to enter the Wipe editor.
- **Action:** Single press to apply an Action video transition to the currently selected timeline segment(s). Double press to enter the Action editor.

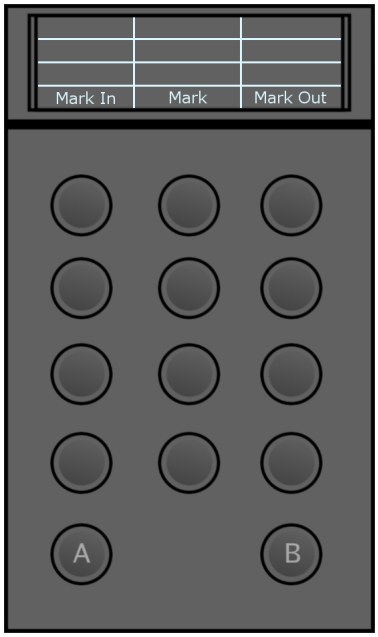
Player Focus Layouts

1Up Player Focus Layout

Module 3 (MF)

- **Player:** Enable 1Up Player Focus mode. Press again to toggle between Source Player and Sequence Player.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Play the current clip backward.
- **Backward Play + Stop buttons:** Move the player head one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the player head one frame backward.
- **Forward Play button:** Play the current clip backward.
- **Previous button:** Got to the previous transition.
- **Previous button (Alt):** Go to In mark.
- **Next button:** Go to the next transition.
- **Next button (Alt):** Got to Out mark.

Module 4 (BT)



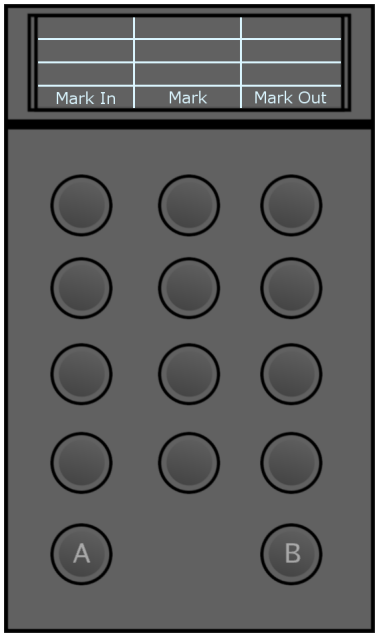
- **Mark In:** Drop an In point marker.
- **Mark:** Drop a navigation marker.
- **Mark Out:** Drop an Out point marker.

Src-Seq Player Focus Layout

Module 3 (MF)

- **Src-Seq:** Enable Src-Seq Player Focus mode. Press again to toggle between Source focus and Sequence focus.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Play the current clip backward.
- **Backward Play + Stop buttons:** Move the player head one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the player head one frame backward.
- **Forward Play button:** Play the current clip backward.
- **Previous button:** Got to the previous transition.
- **Previous button (Alt):** Go to In mark.
- **Next button:** Go to the next transition.
- **Next button (Alt):** Got to Out mark.

Module 4 (BT)



- **Mark In:** Drop an In point marker.
- **Mark:** Drop a navigation marker.
- **Mark Out:** Drop an Out point marker.

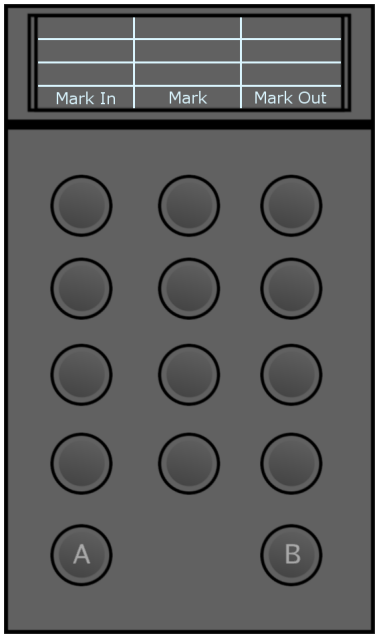
Triptych Player Focus Layout

Module 3 (MF)

- **Triptych:** Enable Triptych Player Focus mode. Press again to toggle the Player focus.

- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Play the current clip backward.
- **Backward Play + Stop buttons:** Move the player head one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the player head one frame backward.
- **Forward Play button:** Play the current clip backward.
- **Previous button:** Got to the previous transition.
- **Previous button (Alt):** Go to In mark.
- **Next button:** Go to the next transition.
- **Next button (Alt):** Got to Out mark.

Module 4 (BT)



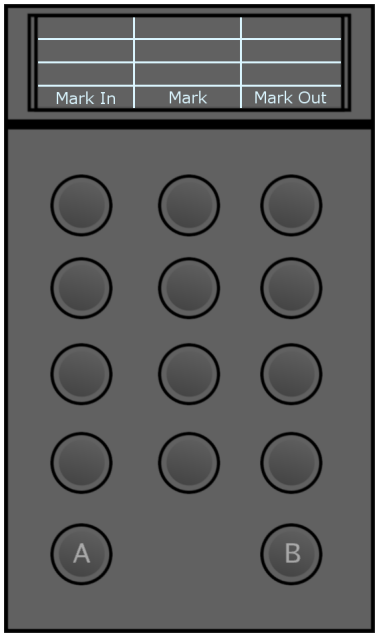
- **Mark In:** Drop an In point marker.
- **Mark:** Drop a navigation marker.
- **Mark Out:** Drop an Out point marker.

Full Screen Player Focus Layout

Module 3 (MF)

- **FullPlayr:** Enable Full Screen Player Focus mode.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Play the current clip backward.
- **Backward Play + Stop buttons:** Move the player head one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the player head one frame backward.
- **Forward Play button:** Play the current clip backward.
- **Previous button:** Got to the previous transition.
- **Previous button (Alt):** Go to In mark.
- **Next button:** Go to the next transition.
- **Next button (Alt):** Got to Out mark.

Module 4 (BT)

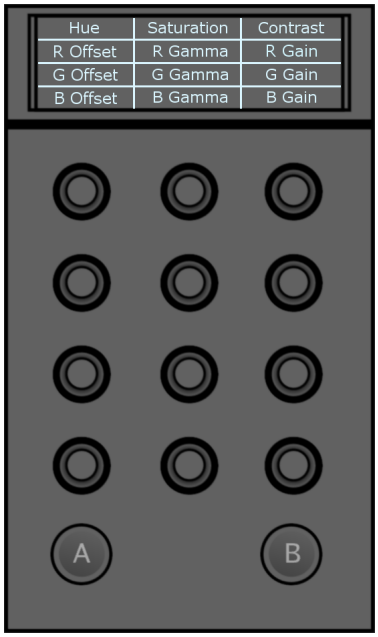


- **Mark In:** Drop an In point marker.
- **Mark:** Drop a navigation marker.
- **Mark Out:** Drop an Out point marker.

Colour Correction Focus Layouts

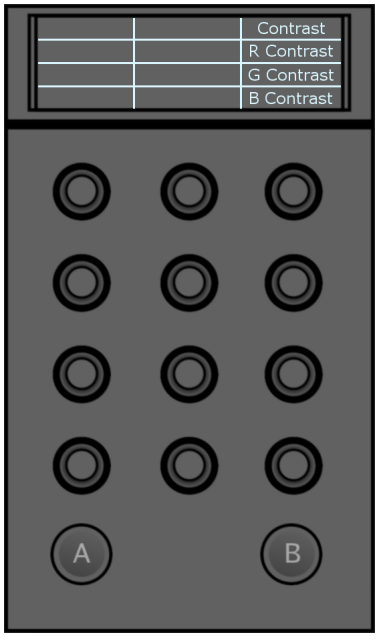
Colour Corrector Quick Bar Focus Layout

Module 1 (KB) - Page 1 of 2



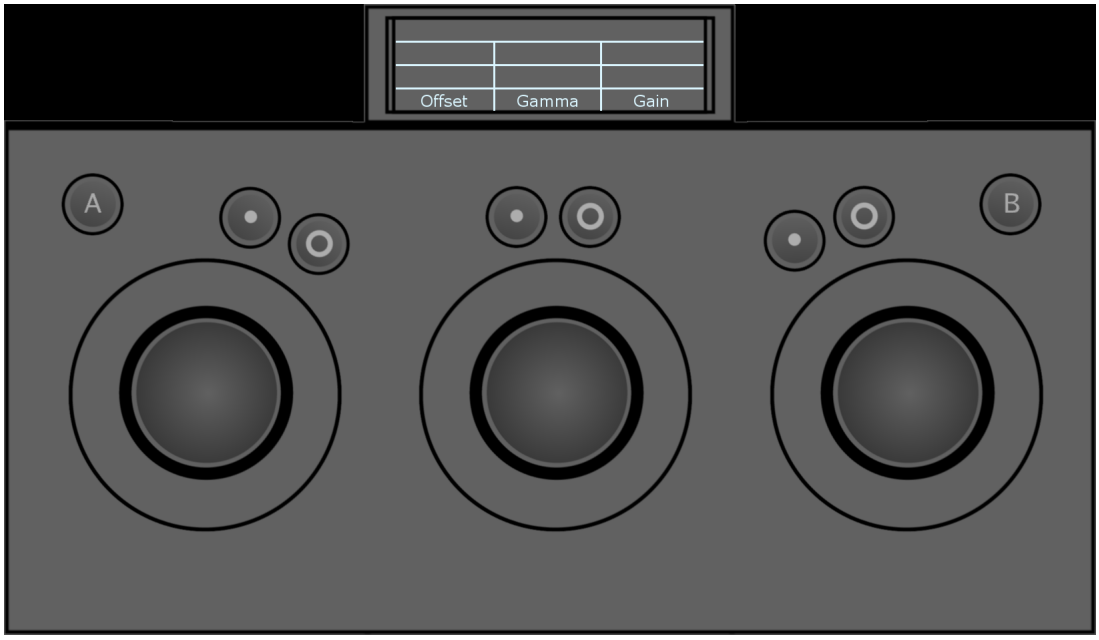
- **Hue:** Adjust the hue value across the entire spectrum.
- **Saturation:** Adjust the saturation value across the entire spectrum.
- **Contrast:** Adjust the contrast value across the entire spectrum..
- **R Offset:** Adjust the Red Offset value.
- **R Gamma:** Adjust the Red Gamma value.
- **R Gain:** Adjust the Red Gain value.
- **G Offset:** Adjust the Green Offset value.
- **G Gamma:** Adjust the Green Gamma value.
- **G Gain:** Adjust the Green Gain value.
- **B Offset:** Adjust the Blue Offset value.
- **B Gamma:** Adjust the Blue Gamma value.
- **B Gain:** Adjust the Blue Gain value.

Module 1 (KB) - Page 2 of 2



- **Contrast:** Adjust the contrast value across the entire spectrum..
- **R Contrast:** Adjust the Red Contrast value.
- **G Contrast:** Adjust the Green Contrast value.
- **B Contrast:** Adjust the Blue Contrast value.

Module 2 (TK)



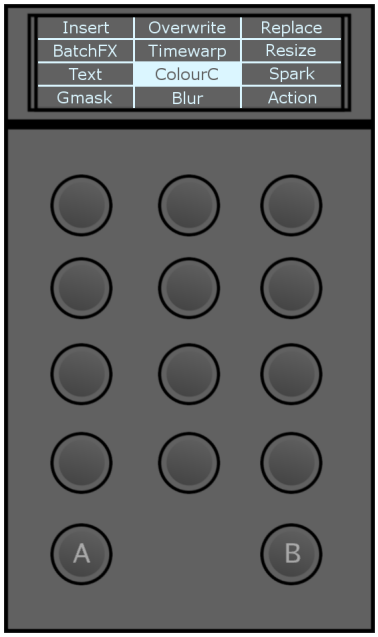
- **Offset:** Adjust the Master Offset value.
- **Gamma:** Adjust the Master Gamma value.

- **Gain:** Adjust the Master Gain value.

Module 3 (MF)

- **Timeline:** Enable Timeline Focus mode.
- **Pan Tmln (Trackball):** Spin the ball along the X axis to zoom the timeline.
- **Zoom Tmln (Trackball + Alt):** Spin the ball along the X axis to pan the the timeline left and right.
- **ScrolTmln (Ring):** Turn the ring to scroll the timeline backward or forward.
- **Backward Play button:** Go to the 1Up player mode and backward play the currently selected clip.
- **Backward Play + Stop buttons:** Move the timeline positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the timeline positioner one frame forward.
- **Forward Play button:** Go to the 1Up player mode and backward play the currently selected clip.
- **Previous button:** Move the timeline positioner to the previous transition.
- **Next button:** Move the timeline positioner to the next transition.

Module 4 (BT) - Page 1 of 2



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.
- **BatchFX:** Single press to apply a BatchFX to the currently selected timeline segment(s). You are taken into Batch.

  NOTE If the selected timeline segment already has a BatchFX applied to it, double pressing the button takes to into Batch for editing.

- **Timewarp:** Single press to apply a Timewarp Timeline FX to the currently selected timeline segment. Double press to enter the Timewarp editor.
- **Resize:** Single press to apply a Resize Timeline FX to the currently selected timeline segment. Double press to enter the Resize editor.
- **Text:** Single press to apply a Text Timeline FX to the currently selected timeline segment. Double press to enter the Text editor.
- **ColourC:** Single press to apply a Colour Corrector Timeline FX to the currently selected timeline segment. Double press to enter the Colour Corrector editor.

  NOTE The Tangent panel only supports the Colour Corrector and Colour Warper Timeline FX editors for input parameters.

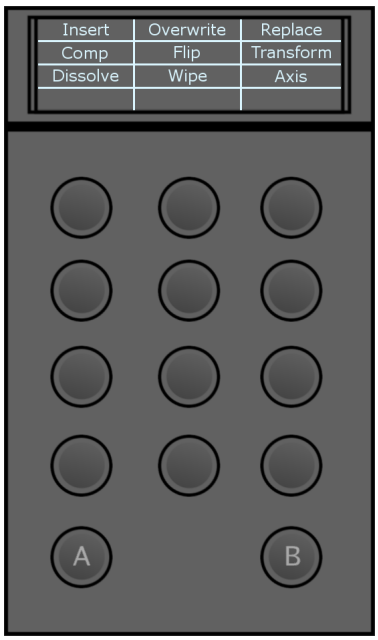
- **ColourW (Alt):** Single press (while holding Alt) to apply a Colour Warper Timeline FX to the currently selected timeline segment. Double press (while holding Alt) to enter the Colour Warper editor.

  NOTE The Tangent panel only supports the Colour Corrector and Colour Warper Timeline FX editors for input parameters.

- **Spark:** Press to apply a placeholder Spark. Click the Load Spark button (L), with your mouse or pen, to open the Sparks Browser and navigate to the appropriate Spark.
- **Gmask:** Single press to apply a Gmask Timeline FX to the currently selected timeline segment. Double press to enter the Gmask editor.

- **Blur:** Single press to apply a Blur Timeline FX to the currently selected timeline segment. Double press to enter the Blur editor.
- **Action:** Single press to apply an Action Timeline FX to the currently selected timeline segment(s). Double press to enter the Action editor.

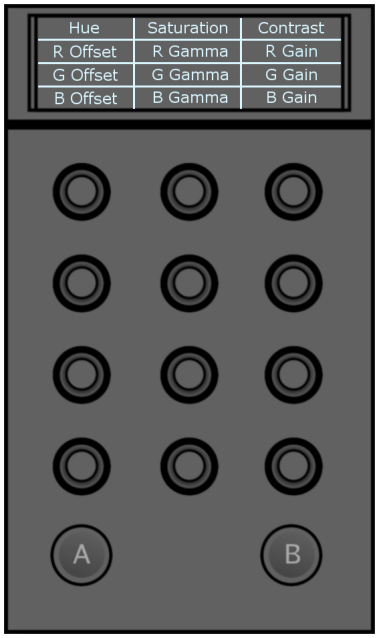
Module 4 (BT) - Page 2 of 2



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.
- **Comp:** Single press to apply a Comp Timeline FX to the currently selected timeline segment. Double press to enter the Comp editor.
- **Flip:** Single press to apply a Flip Timeline FX to the currently selected timeline segment. Double press to enter the Flip editor.
- **2D Transform:** Single press to apply a 2D Transform Timeline FX to the currently selected timeline segment. Double press to enter the Transform editor.
- **Dissolve:** Single press to apply a Dissolve video transition to the currently selected timeline segment(s). Double press to enter the Dissolve editor.
- **Wipe:** Single press to apply a Wipe video transition to the currently selected timeline segment(s). Double press to enter the Wipe editor.
- **Action:** Single press to apply an Action video transition to the currently selected timeline segment(s). Double press to enter the Action editor.

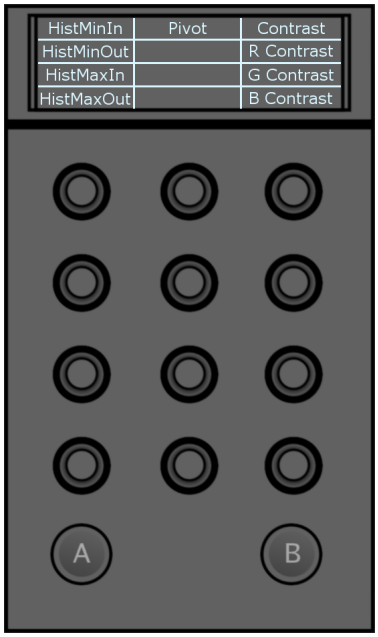
Colour Corrector Master Focus Layout

Module 1 (KB) - Page 1 of 2



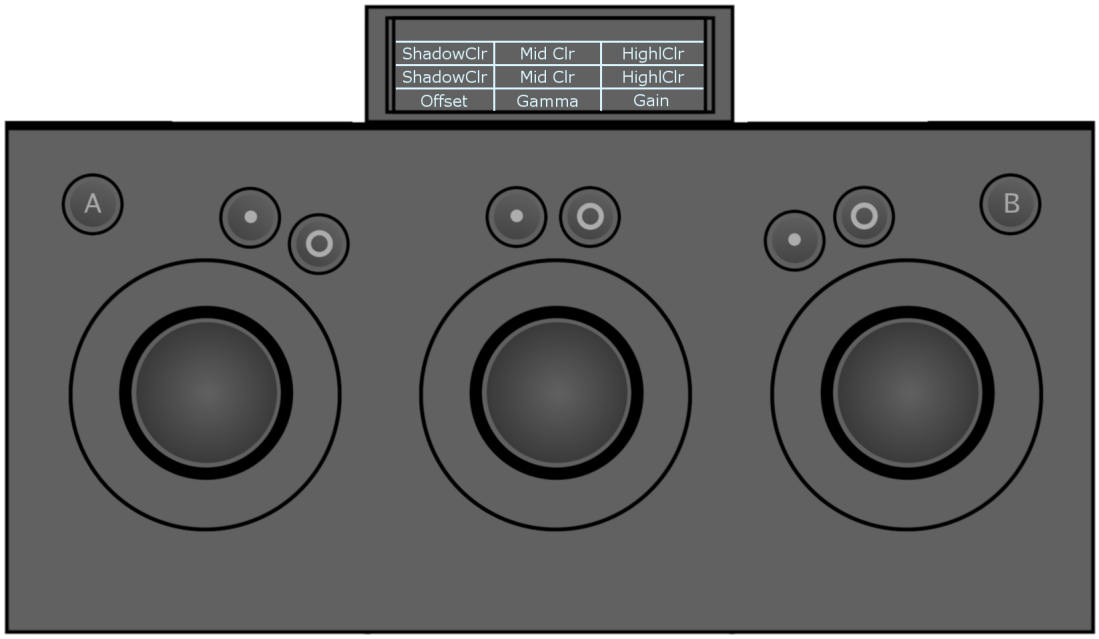
- **Hue:** Adjust the hue value across the entire spectrum.
- **Saturation:** Adjust the saturation value across the entire spectrum.
- **Contrast:** Adjust the contrast value across the entire spectrum.
- **R Offset:** Adjust the Red Offset value.
- **R Gamma:** Adjust the Red Gamma value.
- **R Gain:** Adjust the Red Gain value.
- **G Offset:** Adjust the Green Offset value.
- **G Gamma:** Adjust the Green Gamma value.
- **G Gain:** Adjust the Green Gain value.
- **B Offset:** Adjust the Blue Offset value.
- **B Gamma:** Adjust the Blue Gamma value.
- **B Gain:** Adjust the Blue Gain value.

Module 1 (KB) - Page 2 of 2



- **HistMinIn:** Adjust the minimum Black input value.
- **HistMinOut:** Adjust the minimum Black output value.
- **HistMaxIn:** Adjust the minimum White input value.
- **HistMaxOut:** Adjust the minimum White output value.
- **Pivot:** Adjust the Pivot percentage value.
- **Contrast:** Adjust the contrast value across the entire spectrum..
- **R Contrast:** Adjust the Red Contrast value.
- **G Contrast:** Adjust the Green Contrast value.
- **B Contrast:** Adjust the Blue Contrast value.

Module 2 (TK)

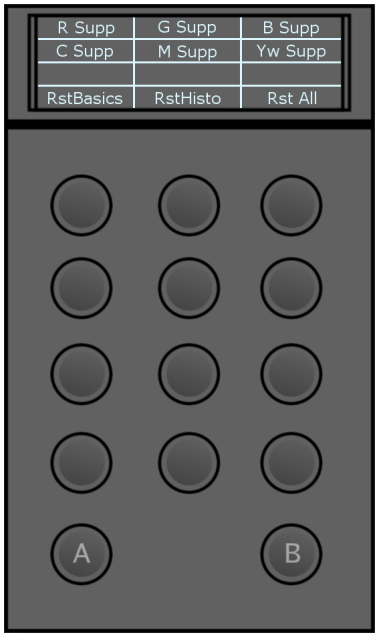


- **ShadowClr:** Move the trackball to adjust the colour of the shadows.
- **Mid Clr:** Move the trackball to adjust the colour of the midtones.
- **HighClr:** Move the trackball to adjust the colour of the highlights.
- **Offset:** Turn the ring to adjust the RGB offset value.
- **Gamma:** Turn the ring to adjust the RGB gamma value.
- **Gain:** Turn the ring to adjust the RGB gain value.

Module 3 (MF)

- **Exit:** Exit the current menu.
- **Redo:** Reapply the last operation performed after an Undo operation.
- **Undo:** Undo the last operation performed.
- **Pan (Trackball + Alt):** Spin the ball along the X axis to pan the current shot left and right.
- **Scroll (Ring):** Turn the ring to scroll the current shot backward or forward.
- **Zoom (Ring + Alt):** Turn the ring to zoom the current shot in and out
- **Backward Play button:** Play the current shot backward. Results may vary according to the number of unrendered effects on the shot.
- **Backward Play + Stop buttons:** Move the positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the positioner one frame forward.
- **Forward Play button:** Play the current shot forward. Results may vary according to the number of unrendered effects on the shot.
- **Previous button:** Go to the start of the current segment.
- **Previous button (Alt):** Go to the previous keyframe.
- **Next button:** Go to the end of the current segment.
- **Next button (Alt):** Go to the next keyframe.

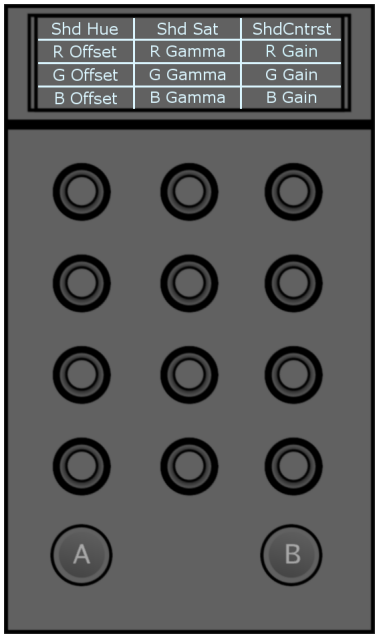
Module 4 (BT) - Page 6 of 6



- **R Supp:** Adjust the Red suppression value.
- **G Supp:** Adjust the Green suppression value.
- **B Supp:** Adjust the Blue suppression value.
- **C Supp:** Adjust the Cyan suppression value.
- **M Supp:** Adjust the Magenta suppression value.
- **Yw Supp:** Adjust the Yellow suppression value.
- **RstBasics:** Reset the basic Colour Corrector parameter values.
- **RstHisto:** Reset the Histogram values.
- **Rst All:** Reset all parameter values.

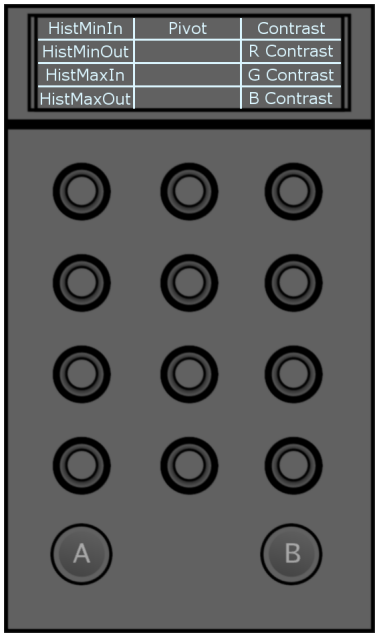
Colour Corrector Shadows Focus Layout

Module 1 (KB) - Page 1 of 2



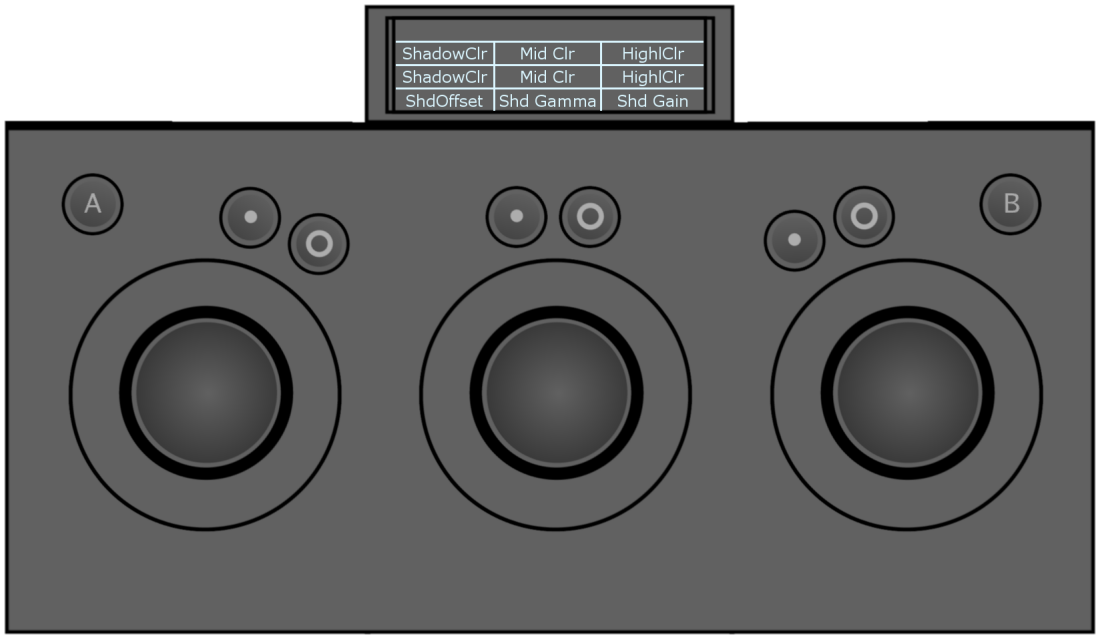
- **Shd Hue:** Adjust the hue value of the shadows.
- **Shd Sat:** Adjust the saturation value of the shadows.
- **ShdCntrst:** Adjust the contrast value of the shadows.
- **R Offset:** Adjust the Red Offset value.
- **R Gamma:** Adjust the Red Gamma value.
- **R Gain:** Adjust the Red Gain value.
- **G Offset:** Adjust the Green Offset value.
- **G Gamma:** Adjust the Green Gamma value.
- **G Gain:** Adjust the Green Gain value.
- **B Offset:** Adjust the Blue Offset value.
- **B Gamma:** Adjust the Blue Gamma value.
- **B Gain:** Adjust the Blue Gain value.

Module 1 (KB) - Page 2 of 2



- **HistMinIn:** Adjust the minimum Black input value.
- **HistMinOut:** Adjust the minimum Black output value.
- **HistMaxIn:** Adjust the minimum White input value.
- **HistMaxOut:** Adjust the minimum White output value.
- **Pivot:** Adjust the Pivot percentage value.
- **Contrast:** Adjust the contrast value across the entire spectrum.
- **R Contrast:** Adjust the Red Contrast value.
- **G Contrast:** Adjust the Green Contrast value.
- **B Contrast:** Adjust the Blue Contrast value.

Module 2 (TK)

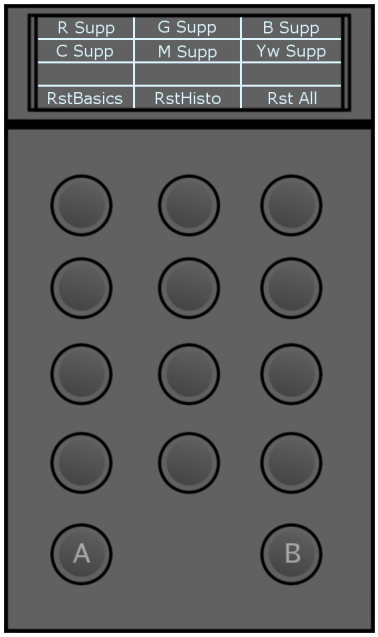


- **ShadowClr:** Move the trackball to adjust the colour of the shadows.
- **Mid Clr:** Move the trackball to adjust the colour of the midtones.
- **HighClr:** Move the trackball to adjust the colour of the highlights.
- **ShdOffset:** Turn the ring to adjust the shadows offset value.
- **Shd Gamma:** Turn the ring to adjust the shadows gamma value.
- **Shd Gain:** Turn the ring to adjust the shadows gain value.

Module 3 (MF)

- **Exit:** Exit the current menu.
- **Redo:** Reapply the last operation performed after an Undo operation.
- **Undo:** Undo the last operation performed.
- **Pan (Trackball + Alt):** Spin the ball along the X axis to pan the current shot left and right.
- **Scroll (Ring):** Turn the ring to scroll the current shot backward or forward.
- **Zoom (Ring + Alt):** Turn the ring to zoom the current shot in and out
- **Backward Play button:** Play the current shot backward. Results may vary according to the number of unrendered effects on the shot.
- **Backward Play + Stop buttons:** Move the positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the positioner one frame forward.
- **Forward Play button:** Play the current shot forward. Results may vary according to the number of unrendered effects on the shot.
- **Previous button:** Go to the start of the current segment.
- **Previous button (Alt):** Go to the previous keyframe.
- **Next button:** Go to the end of the current segment.
- **Next button (Alt):** Go to the next keyframe.

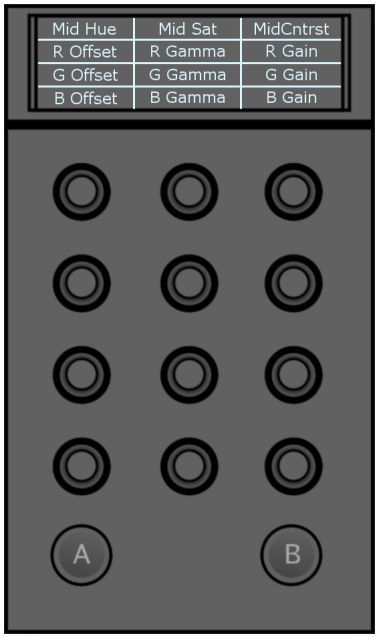
Module 4 (BT)



- **R Supp:** Adjust the Red suppression value.
- **G Supp:** Adjust the Green suppression value.
- **B Supp:** Adjust the Blue suppression value.
- **C Supp:** Adjust the Cyan suppression value.
- **M Supp:** Adjust the Magenta suppression value.
- **Yw Supp:** Adjust the Yellow suppression value.
- **RstBasics:** Reset the basic Colour Corrector parameter values.
- **RstHisto:** Reset the Histogram values.
- **Rst All:** Reset all parameter values.

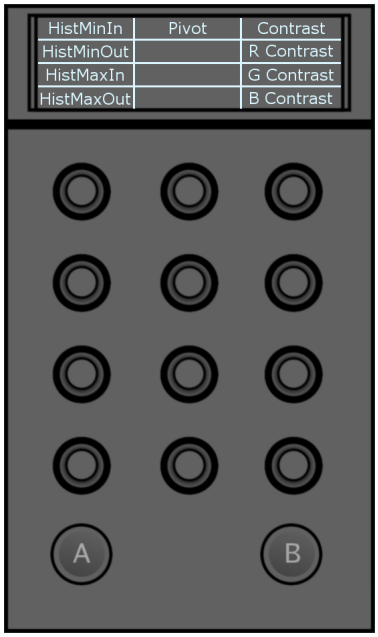
Colour Corrector Midtones Focus Layout

Module 1 (KB)



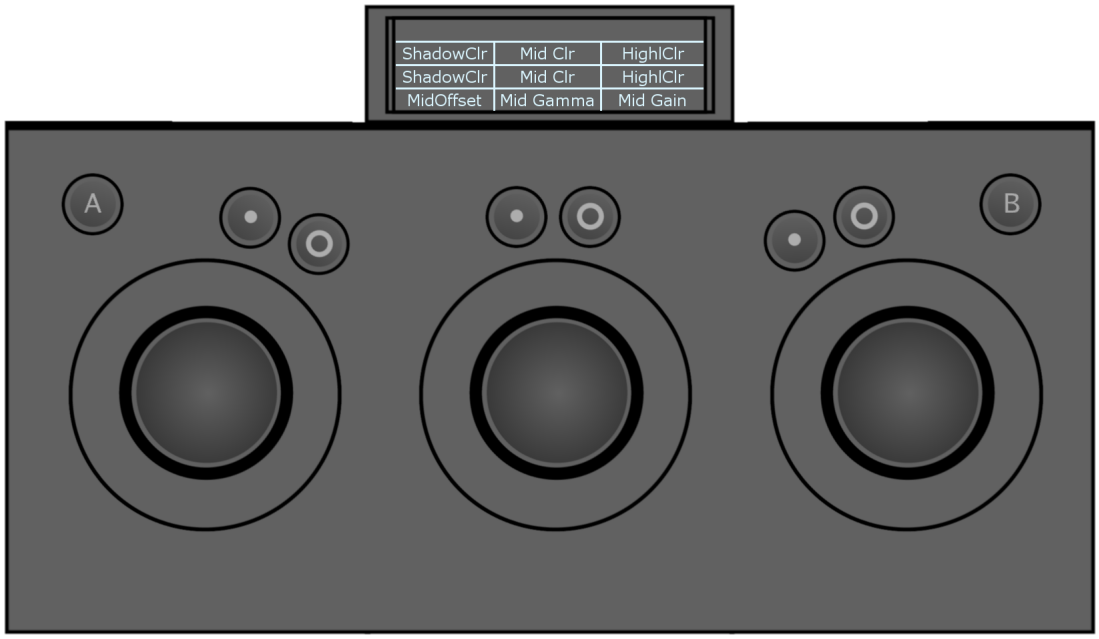
- **Mid Hue:** Adjust the hue value of the midtones.
- **Mid Sat:** Adjust the saturation value of the midtones.
- **MidCntrst:** Adjust the contrast value of the midtones.
- **R Offset:** Adjust the Red Offset value.
- **G Offset:** Adjust the Green Offset value.
- **B Offset:** Adjust the Blue Offset value.
- **R Gamma:** Adjust the Red Gamma value.
- **G Gamma:** Adjust the Green Gamma value.
- **B Gamma:** Adjust the Blue Gamma value.
- **R Gain:** Adjust the Red Gain value.
- **G Gain:** Adjust the Green Gain value.
- **B Gain:** Adjust the Blue Gain value.

Module 1 (KB) - Page 2 of 2



- **HistMinIn:** Adjust the minimum Black input value.
- **HistMinOut:** Adjust the minimum Black output value.
- **HistMaxIn:** Adjust the minimum White input value.
- **HistMaxOut:** Adjust the minimum White output value.
- **Pivot:** Adjust the Pivot percentage value.
- **Contrast:** Adjust the contrast value across the entire spectrum..
- **R Contrast:** Adjust the Red Contrast value.
- **G Contrast:** Adjust the Green Contrast value.
- **B Contrast:** Adjust the Blue Contrast value.

Module 2 (TK)

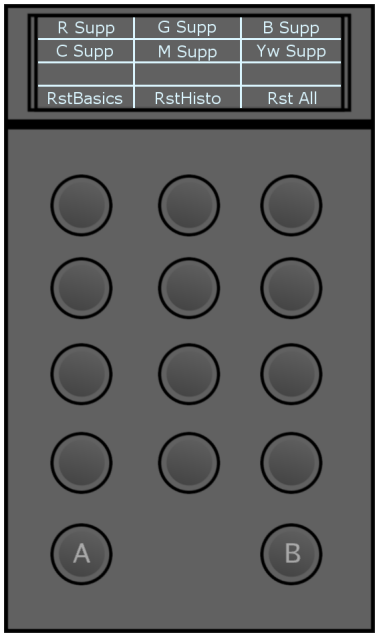


- **ShadowClr:** Move the trackball to adjust the colour of the shadows.
- **Mid Clr:** Move the trackball to adjust the colour of the midtones.
- **HighClr:** Move the trackball to adjust the colour of the highlights.
- **MidOffset:** Turn the ring to adjust the midtones offset value.
- **Mid Gamma:** Turn the ring to adjust the midtones gamma value.
- **Mid Gain:** Turn the ring to adjust the midtones gain value.

Module 3 (MF)

- **Exit:** Exit the current menu.
- **Redo:** Reapply the last operation performed after an Undo operation.
- **Undo:** Undo the last operation performed.
- **Pan (Trackball + Alt):** Spin the ball along the X axis to pan the current shot left and right.
- **Scroll (Ring):** Turn the ring to scroll the current shot backward or forward.
- **Zoom (Ring + Alt):** Turn the ring to zoom the current shot in and out
- **Backward Play button:** Play the current shot backward. Results may vary according to the number of unrendered effects on the shot.
- **Backward Play + Stop buttons:** Move the positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the positioner one frame forward.
- **Forward Play button:** Play the current shot forward. Results may vary according to the number of unrendered effects on the shot.
- **Previous button:** Go to the start of the current segment.
- **Previous button (Alt):** Go to the previous keyframe.
- **Next button:** Go to the end of the current segment.
- **Next button (Alt):** Go to the next keyframe.

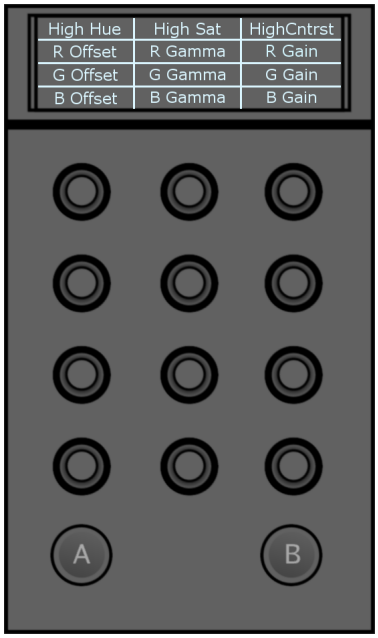
Module 4 (BT)



- **R Supp:** Adjust the Red suppression value.
- **G Supp:** Adjust the Green suppression value.
- **B Supp:** Adjust the Blue suppression value.
- **C Supp:** Adjust the Cyan suppression value.
- **M Supp:** Adjust the Magenta suppression value.
- **Yw Supp:** Adjust the Yellow suppression value.
- **RstBasics:** Reset the basic Colour Corrector parameter values.
- **RstHisto:** Reset the Histogram values.
- **Rst All:** Reset all parameter values.

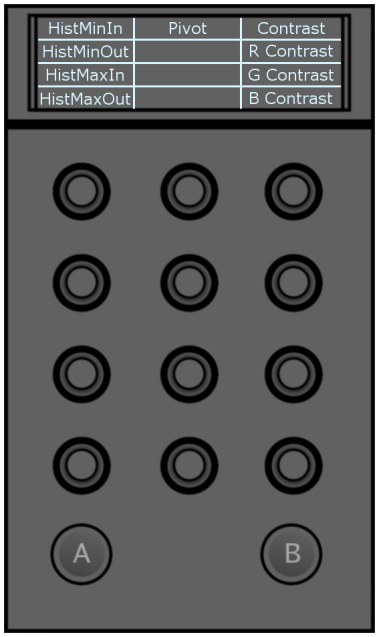
Colour Corrector Highlights Focus Layout

Module 1 (KB) - Page 1 of 2



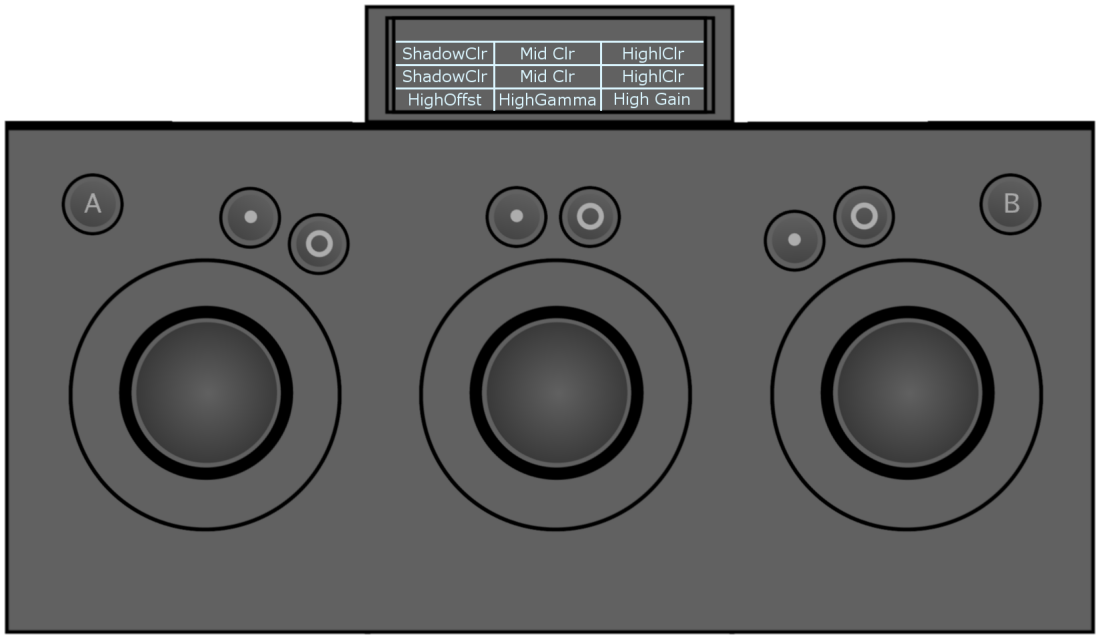
- **High Hue:** Adjust the hue value of the highlights.
- **High Sat:** Adjust the saturation value of the highlights.
- **HiCntrst:** Adjust the contrast value of the highlights.
- **R Offset:** Adjust the Red Offset value.
- **G Offset:** Adjust the Green Offset value.
- **B Offset:** Adjust the Blue Offset value.
- **R Gamma:** Adjust the Red Gamma value.
- **G Gamma:** Adjust the Green Gamma value.
- **B Gamma:** Adjust the Blue Gamma value.
- **R Gain:** Adjust the Red Gain value.
- **G Gain:** Adjust the Green Gain value.
- **B Gain:** Adjust the Blue Gain value.

Module 1 (KB) - Page 2 of 2



- **HistMinIn:** Adjust the minimum Black input value.
- **HistMinOut:** Adjust the minimum Black output value.
- **HistMaxIn:** Adjust the minimum White input value.
- **HistMaxOut:** Adjust the minimum White output value.
- **Pivot:** Adjust the Pivot percentage value.
- **Contrast:** Adjust the contrast value across the entire spectrum..
- **R Contrast:** Adjust the Red Contrast value.
- **G Contrast:** Adjust the Green Contrast value.
- **B Contrast:** Adjust the Blue Contrast value.

Module 2 (TK)

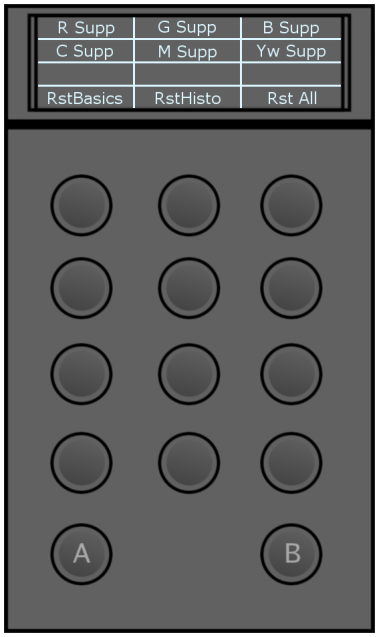


- **ShadowClr:** Move the trackball to adjust the colour of the shadows.
- **Mid Clr:** Move the trackball to adjust the colour of the midtones.
- **HighClr:** Move the trackball to adjust the colour of the highlights.
- **HighOffset:** Turn the ring to adjust the highlights offset value.
- **High Gamma:** Turn the ring to adjust the highlights gamma value.
- **High Gain:** Turn the ring to adjust the highlights gain value.

Module 3 (MF)

- **Exit:** Exit the current menu.
- **Redo:** Reapply the last operation performed after an Undo operation.
- **Undo:** Undo the last operation performed.
- **Pan (Trackball + Alt):** Spin the ball along the X axis to pan the current shot left and right.
- **Scroll (Ring):** Turn the ring to scroll the current shot backward or forward.
- **Zoom (Ring + Alt):** Turn the ring to zoom the current shot in and out
- **Backward Play button:** Play the current shot backward. Results may vary according to the number of unrendered effects on the shot.
- **Backward Play + Stop buttons:** Move the positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the positioner one frame forward.
- **Forward Play button:** Play the current shot forward. Results may vary according to the number of unrendered effects on the shot.
- **Previous button:** Go to the start of the current segment.
- **Previous button (Alt):** Go to the previous keyframe.
- **Next button:** Go to the end of the current segment.
- **Next button (Alt):** Go to the next keyframe.

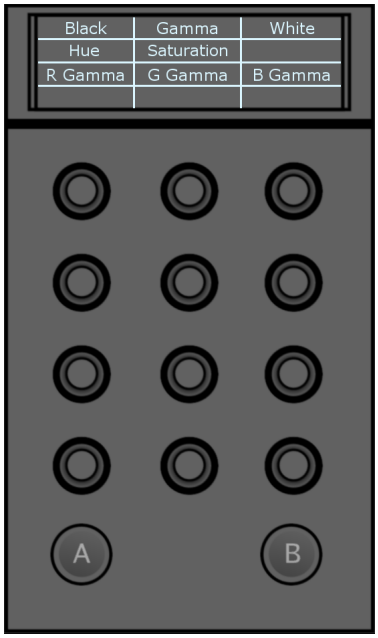
Module 4 (BT)



- **R Supp:** Adjust the Red suppression value.
- **G Supp:** Adjust the Green suppression value.
- **B Supp:** Adjust the Blue suppression value.
- **C Supp:** Adjust the Cyan suppression value.
- **M Supp:** Adjust the Magenta suppression value.
- **Yw Supp:** Adjust the Yellow suppression value.
- **RstBasics:** Reset the basic Colour Corrector parameter values.
- **RstHisto:** Reset the Histogram values.
- **Rst All:** Reset all parameter values.

Colour Warper QuickBar Focus Layout

Module 1 (KB)

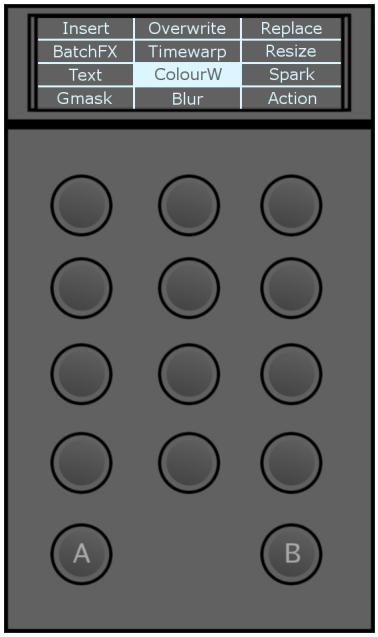


- **Black:** Adjust the black point.
- **Gamma:** Adjust the RGB gamma value.
- **White:** Adjust the white point.
- **Hue:** Adjust the RGB hue value.
- **Saturatio:** Adjust the RGB saturation value.
- **R Gamma:** Adjust the Red Gamma value.
- **G Gamma:** Adjust the Green Gamma value.
- **B Gamma:** Adjust the Blue Gamma value.

Module 3 (MF)

- **Timeline:** Enable Timeline Focus mode.
- **Pan Tmln (Trackball):** Spin the ball along the X axis to zoom the timeline.
- **Zoom Tmln (Trackball + Alt):** Spin the ball along the X axis to pan the the timeline left and right.
- **ScrolTmln (Ring):** Turn the ring to scroll the timeline backward or forward.
- **Backward Play button:** Go to the 1Up player mode and backward play the currently selected clip.
- **Backward Play + Stop buttons:** Move the timeline positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the timeline positioner one frame forward.
- **Forward Play button:** Go to the 1Up player mode and backward play the currently selected clip.
- **Previous button:** Move the timeline positioner to the previous transition.
- **Next button:** Move the timeline positioner to the next transition.

Module 4 (BT) - Page 1 of 2



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.
- **BatchFX:** Single press to apply a BatchFX to the currently selected timeline segment(s). You are taken into Batch.

  NOTE If the selected timeline segment already has a BatchFX applied to it, double pressing the button takes to into Batch for editing.

- **Timewarp:** Single press to apply a Timewarp Timeline FX to the currently selected timeline segment. Double press to enter the Timewarp editor.
- **Resize:** Single press to apply a Resize Timeline FX to the currently selected timeline segment. Double press to enter the Resize editor.
- **Text:** Single press to apply a Text Timeline FX to the currently selected timeline segment. Double press to enter the Text editor.
- **ColourC:** Single press to apply a Colour Corrector Timeline FX to the currently selected timeline segment. Double press to enter the Colour Corrector editor.

  NOTE The Tangent panel only supports the Colour Corrector and Colour Warper Timeline FX editors for input parameters.

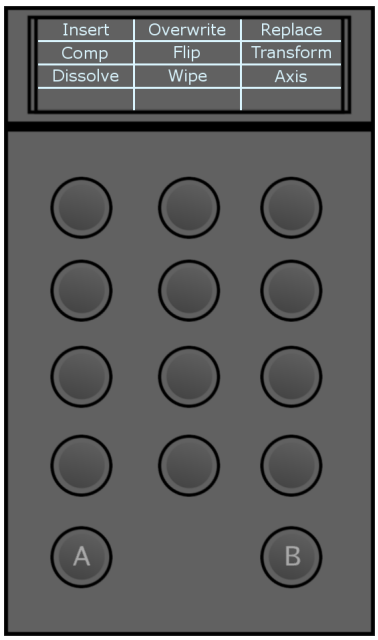
- **ColourW (Alt):** Single press (while holding Alt) to apply a Colour Warper Timeline FX to the currently selected timeline segment. Double press (while holding Alt) to enter the Colour Warper editor.

  NOTE The Tangent panel only supports the Colour Corrector and Colour Warper Timeline FX editors for input parameters.

- **Spark:** Press to apply a placeholder Spark. Click the Load Spark button (L), with your mouse or pen, to open the Sparks Browser and navigate to the appropriate Spark.
- **Gmask:** Single press to apply a Gmask Timeline FX to the currently selected timeline segment. Double press to enter the Gmask editor.

- **Blur:** Single press to apply a Blur Timeline FX to the currently selected timeline segment. Double press to enter the Blur editor.
- **Action:** Single press to apply an Action Timeline FX to the currently selected timeline segment(s). Double press to enter the Action editor.

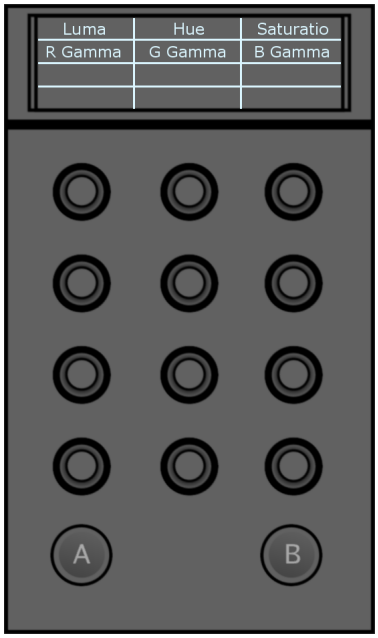
Module 4 (BT) - Page 2 of 2



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.
- **Comp:** Single press to apply a Comp Timeline FX to the currently selected timeline segment. Double press to enter the Comp editor.
- **Flip:** Single press to apply a Flip Timeline FX to the currently selected timeline segment. Double press to enter the Flip editor.
- **2D Transform:** Single press to apply a 2D Transform Timeline FX to the currently selected timeline segment. Double press to enter the Transform editor.
- **Dissolve:** Single press to apply a Dissolve video transition to the currently selected timeline segment(s). Double press to enter the Dissolve editor.
- **Wipe:** Single press to apply a Wipe video transition to the currently selected timeline segment(s). Double press to enter the Wipe editor.
- **Action:** Single press to apply an Action video transition to the currently selected timeline segment(s). Double press to enter the Action editor.

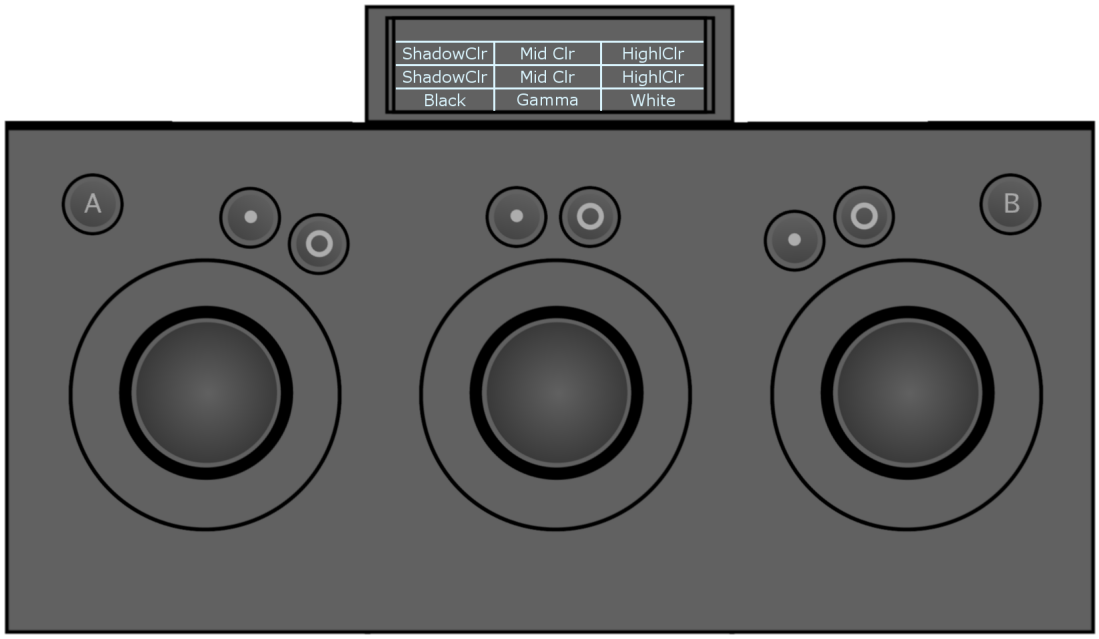
Colour Warper Master Focus Layout

Module 1 (KB)



- **Luma:** Adjust the RGB luma value.
- **Hue:** Adjust the RGB hue value.
- **Saturatio:** Adjust the RGB saturation value.
- **R Gamma:** Adjust the Red Gamma value.
- **G Gamma:** Adjust the Green Gamma value.
- **B Gamma:** Adjust the Blue Gamma value.

Module 2 (TK)

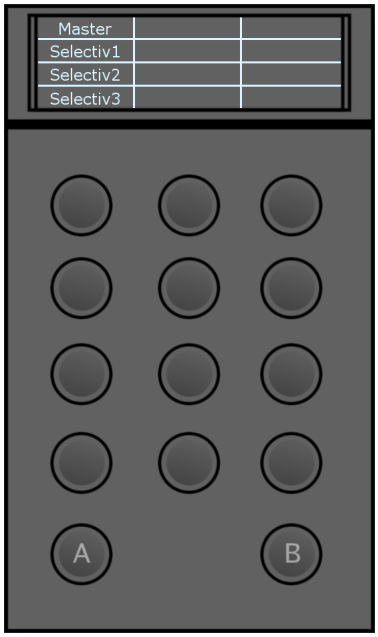


- **ShadowClr:** Move the trackball to adjust the colour of the shadows.
- **Mid Clr:** Move the trackball to adjust the colour of the midtones.
- **HighClr:** Move the trackball to adjust the colour of the highlights.
- **Black:** Adjust the black point.
- **Gamma:** Adjust the RGB gamma value.
- **White:** Adjust the white point.

Module 3 (MF)

- **Exit:** Exit the current menu.
- **Redo:** Reapply the last operation performed after an Undo operation.
- **Undo:** Undo the last operation performed.
- **Pan (Trackball + Alt):** Spin the ball along the X axis to pan the current shot left and right.
- **Scroll (Ring):** Turn the ring to scroll the current shot backward or forward.
- **Zoom (Ring + Alt):** Turn the ring to zoom the current shot in and out
- **Backward Play button:** Play the current shot backward. Results may vary according to the number of unrendered effects on the shot.
- **Backward Play + Stop buttons:** Move the positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the positioner one frame forward.
- **Forward Play button:** Play the current shot forward. Results may vary according to the number of unrendered effects on the shot.
- **Previous button:** Go to the start of the current segment.
- **Previous button (Alt):** Go to the previous keyframe.
- **Next button:** Go to the end of the current segment.
- **Next button (Alt):** Go to the next keyframe.

Module 4 (BT)



- **Master:** Adjust the Red suppression value.
- **Selectiv1:** Press to target selective 1.
- **Selectiv2:** Press to target selective 2.
- **Selectiv3:** Press to target selective 3.

Troubleshooting

In the event that one or more of your panel modules are not recognized by Flame, restarting the background service that manages the interactions between Flame and the Tangent Element Control Surface can resolve the issue.

To restart the background service:

1. Exit Flame.
2. Open a shell on your Linux machine.
3. Login as root.
4. Type: */etc/init.d/tangenthub restart*
5. Launch Flame.

All of your module panels should now be recognized.

If the above procedure does not resolve the issue, you can manually connect the Control Surface to your software. To do this, you must define each module's unique ID number in the panel-list.xml file within the Flame filesystem for the panel to be recognized.

To manually install the panel:

1. Plug and connect the four or five modules (depending on your setup) of your Tangent Element Control Panel to a power outlet and to your USB hub.

2 Once the modules are booted up and are displaying the Tangent logo, hit the A button on each module to display its unique ID number.

3 On your Linux machine, log in as root and use kEdit to edit the panel-list.xml file:
kedit /var/opt/Tangent/Hub/panel-list.xml

```
<?xml version="1.0" encoding="utf-8" standalone="yes"?>
<TangentWave fileType="PanelList" fileVersion="3.0">
    <Panel type="Element-Mf" number="1" serial="30"/>
    <Panel type="Element-Bt" number="1" serial="68"/>
    <Panel type="Element-Bt" number="2" serial="77"/>
    <Panel type="Element-Kb" number="1" serial="53"/>
    <Panel type="Element-Tk" number="1" serial="13"/>
    <Panel type="Wave"/>
    <Options>
        <DHCP enabled="true"/>
    </Options>
</TangentWave>
```

4 Add the corresponding ID numbers to the panel-list.xml file.

NOTE In a four module setup, the line <Panel type= "Element-BT" number="2" serial="00"/> should be left blank.

5 Save the file.

The Tangent Element Control Surface should be now recognized by Flame.

Flame and Smoke Classic Keyboard Shortcuts

32

Following is a selection of Flame and Smoke Classic keyboard shortcuts.

You can also find keyboard shortcuts by Searching the Keyboard Shortcut Catalogue (page 1928).

Navigation

Function	Flame Shortcut	Smoke Classic Shortcut
MediaHub	Space - F1	Shift - 1
Conform	Space - F2	Shift - 2
Timeline	Space - F3	Shift - 3
Batch	Space - F4	Shift - 4
Tools	Space - F5	Shift - 5
Tools - Composite	Space - 1	Not available.
Tools - Look	Space - 2	Not available.
Tools - Filter	Space - 3	Not available.
Tools - Plugins	Space - 4	Not available.
Tools - Clip	Space - 5	Not available.
Tools - Utilities	Space - 6	Not available.

Viewing Panel

Function	Flame Shortcut	Smoke Classic Shortcut
Desktop Reels	~(tilde)	Not available.
Thumbnail View	Ctrl - ~(tilde)	Alt - ESC
Enter Standard Player with Clip Under Cursor	ESC	Not available.
Standard Player Real-Time Deliverables	Alt - 1	Not available.
Standard Player Audio Desk	Alt - 2	Ctrl - E
Standard Player Overlays	Alt - 3	Not available.
Standard Player Vectorscope	Alt - 4	Not available.
Standard Player Viewing Settings	Alt - 5	Not available.
Player View	Ctrl - 1	ESC
Source/Sequence Player	Ctrl - 2	Shift - ESC
Triptych Player	Ctrl - 3	Space - ESC
Trim View	Ctrl - 4	Not available.
Full Screen Player	Ctrl - ESC	Ctrl - ESC
Play Next Source Clip	Shift - Right Arrow	Alt - V
Play Previous Source Clip	Shift - Left Arrow	Alt - C

Media Panel

Function	Flame Shortcut	Smoke Classic Shortcut
Tall	Shift - Tab	Not available.
Details	Shift - ~	Shift - L
Hidden	Shift - ESC / Ctrl - Swipe	Ctrl - H and Ctrl - swipe

Desktop or Thumbnail View

Function	Flame Shortcut	Smoke Classic Shortcut
Copy (Whole Clip) Gesturally	Shift - Alt - drag	F - drag
Copy (Current Frame) Gesturally	Shift - Ctrl - Alt - drag	Alt - F - drag
Copy In to Out Gesturally	Shift - Ctrl - drag	Shift - F - drag
Create Sub-Clip	Alt - C	Ctrl - F - drag
Contextual Menu (Mouse right click tablet emulation)	Win (Smoke) - Click	Menu button - Click
Frames view (Cursor Over)	C	Not available.
Storyboard view (Cursor Over)	Space - C	Not available.
Delete (Cursor Over)	D	Alt - D
Delete Selected	Not available	Shift - D
Rename (Cursor Over)	N	9
Lock (Cursor Over)	Space - L	Space - L
Go To Frame (Cursor Over)	G	Not available.
Go To Timecode (Cursor Over)	Alt - G	Not available.
Cut After Selected Frame (Cursor Over)	Space - X	Not available.
Cut At Each Splice (Cursor Over)	Alt - Space - X	Not available.
Cut At (n) Frame (Cursor Over)	Ctrl - Space - X	Not available.
Select Current Segment (Cursor Over)	Unassigned	Enter
Swap Segments (Cursor Over)	Space - S	Shift - /
Match Frame (Cursor Over)	M	Not available.

Function	Flame Shortcut	Smoke Classic Shortcut
Match Source (Cursor Over)	Alt - M	Not available.
Match Content (Cursor Over)	Space - M	Not available.
Match All Sources (Cursor Over)	Alt - Space - M	Not available.
Render Segment (Cursor Over)	Space - R	Not available.
Render Sequence (Cursor Over)	Alt - Space - R	Not available.
Hard Commit Segment (Cursor Over)	Space - H	Not available.
Hard Commit Sequence (Cursor Over)	Alt - Space - H	Not available.

Timeline Navigation

Function	Flame Shortcut	Smoke Classic Shortcut
Go to Clip Start	Ctrl - Left Arrow	Ctrl - A
Go to Clip End	Ctrl - Right Arrow	Ctrl - S
Go to In Mark	Alt - [	A
Go to Out Mark	Alt -]	S
Next Cue Mark	Ctrl - \	Alt - S
Prev Cue Mark	Alt - \	Alt - A
Zoom In	Ctrl - Space-Up Arrow	Smoke + PAD+
Zoom Out	Ctrl - Space - Down Arrow	Smoke + PAD-
Fit Timeline to Contents	Ctrl - Home	=
Fit Timeline to Selection	Shift - =	Shift - =
Next Transition (Current Track)	Down Arrow	X
Next Transition (All Versions/Tracks)	Ctrl - Down Arrow	Shift - X
Previous Transition (Current Track)	Up Arrow	Z
Previous Transition (All Versions/Tracks)	Ctrl - Up Arrow	Shift - Z
Select Outgoing Side of a Transition	Win (Smoke) - Left Arrow	Space - B
Select Incoming Side of a Transition	Win (Smoke) - Right Arrow	Space - N

Function	Flame Shortcut	Smoke Classic Shortcut
Select Both Sides of a Transition	Win (Smoke) - Down Arrow	Space - H
Next Timeline FX (Current Segment)	Shift - Ctrl - Right Arrow	Alt - Smoke - X
Previous Timeline FX (Current Segment)	Shift - Ctrl - Left Arrow	Alt - Smoke - Z
Next Instance of Current Timeline FX (Current Track)	Shift - Down Arrow	Alt - Z
Previous Instance of Current Timeline FX(Current Track)	Shift - Up Arrow	Alt - X

Editing

Function	Flame Shortcut	Smoke Classic Shortcut
Cut (Current Version/Track)	X	Delete
Cut (All Versions/Tracks)	Ctrl - Shift - X	Shift - Delete
Cut Around Selection (Current Version/Track)	Alt - X	Alt - Delete
Cut Around Selection (All Versions/Tracks)	Alt - Shift - X	Alt - Shift - Delete
Remove Cut (Match Frame Edit)	Ctrl - Alt - X	Ctrl - Delete
Add Dissolve (All Versions/Tracks)	Shift - End	Shift - End
Add Dissolve (Current Version/Track)	End	End
Mark In	[	Right Alt key
Mark Out	]	Right Ctrl key
Mark In/Out (Current Shot)	'	Shift - /
Select Between In-Out (Current Track)	Shift - '	Alt - Ctrl - /
Select Between In-Out (Current Version)	Ctrl - '	Alt - /
Select Between In-Out (All Versions)	Ctrl - Shift - '	Alt - Shift - /
Clear In	Space - [	Right Alt key - Space
Clear Out	Space -]	Right Ctrl key - Space
Clear In-Out	Space - [-]	Right Alt key - Right Ctrl key - Space

Function	Flame Shortcut	Smoke Classic Shortcut
Add Cue Mark	\	Shift - Right Alt key
Clear Cue Mark	Space - \	Shift - Right Alt key - Space
Add Segment Mark	Shift - \	Shift - Right - Ctrl key
Clear Segment Mark	Shift - Space - \	Shift - Right Ctrl key - Space
Insert	I	G
Overwrite	O	H
Replace	R	J
Ripple	Alt - R	Not available.
Ripple Replace	Ctrl - Alt - R	K
Replace Media	Ctrl - Shift - R	Shift - J
Timeline Select Mode	Alt - A	Y
Timeline Trim Mode	Alt - E	T
Trim to Positioner	Shift - P	Ctrl - P
Trim To In Mark	Shift - [	Ctrl - B
Trim To Out Mark	Shift -]	Ctrl - N
Trim 1 Frame Forward	. (period)	N
Trim 1 Frame Backward	, (comma)	B
Trim <n> Frames Forward	Shift - . (period)	Shift - N
Trim <n> Frames Backward	Shift - , (comma)	Shift - B
Slip Shot	Alt - S	, (comma)
Slide Keyframes	Shift - Alt - S	Shift - , (comma)
Slide Cuts	Shift - Alt - D	Shift - M
Slide	Alt - D	M
Snap	Alt - W	Not available.
Invert Snap during manipulation	Shift	Shift

Function	Flame Shortcut	Smoke Classic Shortcut
Invert Ripple during manipulation	Alt	Alt
Sub Clip	Alt -C	Ctrl - F
New Version	Shift - Q	Ctrl - Tab
New Video Track	Q	Shift - Ctrl - Tab
New Audio Track	Ctrl - Q	Ctrl - Q

Effects View Shortcuts

Function	Flame Shortcut	Smoke Classic Shortcut
Front View	F1	F1
Back View	F2	F2
Matte View	F3	F3
Result View	F4	F4
Batch or BFX Schematic View	Ctrl - ESC	Ctrl - ESC
Schematic View (All other tools with schematics)	ESC	ESC
1-Up View	Alt - 1	Alt - 1
2-Up View	Alt - 2	Alt - 2
3-Up View	Alt - 3	Alt - 3
4-Up View	Alt - 4	Alt - 4
Pan	Space	Space
Continuous Zoom	Ctrl - Space	Ctrl - Space
Previous Keyframe	Alt - Left	Ctrl - Z
Next Keyframe	Alt - Right	Ctrl - Y

Action Shortcuts

Function	Flame Shortcut	Smoke Classic Shortcuts
Select	M	Space - M
Add	A	Space - A

Function	Flame Shortcut	Smoke Classic Shortcuts
Delete Mode	D	Space - D
Mimic Link	W	Space - W
Schematic Autolink Mimic	Shift - W - drag	Not available.
Add Action Media Input	Ctrl - N	Ctrl - N
Reset Media Layer	Ctrl - Alt - click	Ctrl - Alt - click
"Kissing" Nodes	Shift - drag	Shift - drag
Reverse "Kiss"	Shift - Alt - drag	Shift - Alt - drag
Toggle Navigator	Ctrl - Alt - N	Ctrl - Alt - N
Move the Navigator inside the viewport	Alt - drag	Alt - drag

Animation Shortcuts

Function	Flame Shortcut	Smoke Classic Shortcut
Context Menu (field)	Win (Smoke) - click	Smoke - click
Select Channel	Shift - click	Shift - click
Add Channel to Selection	Shift - click	Shift - Ctrl - click
Reset Channel (keep current value)	Alt - click	Alt - click
Reset Channel (Default value)	Ctrl - Alt - click	Alt - Ctrl - click

Playback Controls

Function	Flame Shortcut	Smoke Classic Shortcut
Play-Stop	Enter	Not available.
Play	Not applicable.	V
Stop	Not applicable.	Spacebar
Play Forward	L	Not available.
Play Forward Increase Speed	L	Not available.
Play Forward Decrease Speed	Shift - L	Not available.
Stop Playback	K	Space

Function	Flame Shortcut	Smoke Classic Shortcut
Play Backward	J	C
Play Backward Increase Speed	J	Not available.
Play Backward Decrease Speed	Shift - J	Not available.

General Shortcuts

Function	Flame Shortcut	
Keyboard Shortcut dialog box	Ctrl - Alt - F8	Ctrl - Alt - F8
Preferences dialog box	Ctrl - Alt - F6	Ctrl - Alt - F6
Display Tooltip	Ctrl - Alt - W	Ctrl - Alt -W
Exit	Alt - F12	Alt - F12
Undo	Ctrl - Z	Backspace
Redo	Ctrl - R	Shift - Backspace
Help	Ctrl - F1	Ctrl - F1

NOTE When you press a keyboard shortcut, a white keyboard appears in the lower-right corner of the screen. If a keyboard shortcut becomes stuck at any time, the white keyboard remains until you unstick the keyboard shortcut by pressing it. Click the white keyboard to display the name of the problematic keyboard shortcut in the message bar.

Determining a Button's Current Keyboard Shortcut

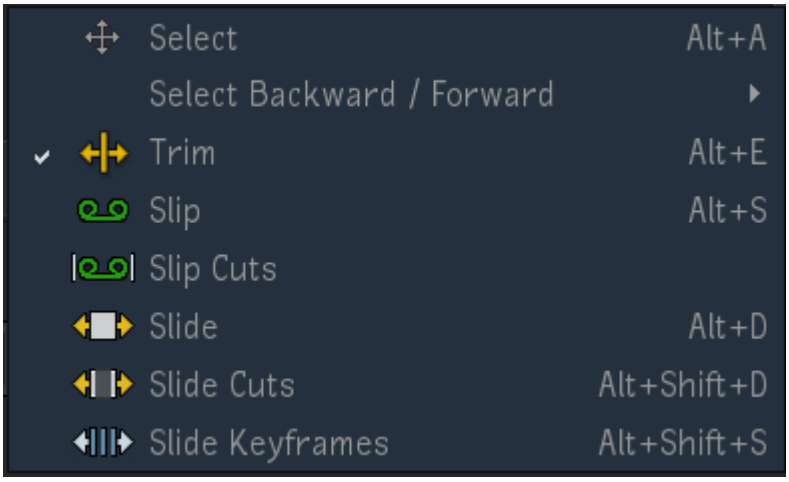
You can determine the keyboard shortcut assigned to a button from anywhere within Flame.

To determine the keyboard shortcut for a button:

1. If auto display of tooltips is enabled in the Preferences, hover over the button for which you want to determine the assigned keyboard shortcut. If you have disabled the auto display of tooltips, press and hold `Ctrl+Alt+W` and hover over the button.

   If a keyboard shortcut exists for the button, it is displayed in the tooltip.

2. In some cases where dropdown list items have separate keyboard shortcuts, the list items display the various shortcuts.

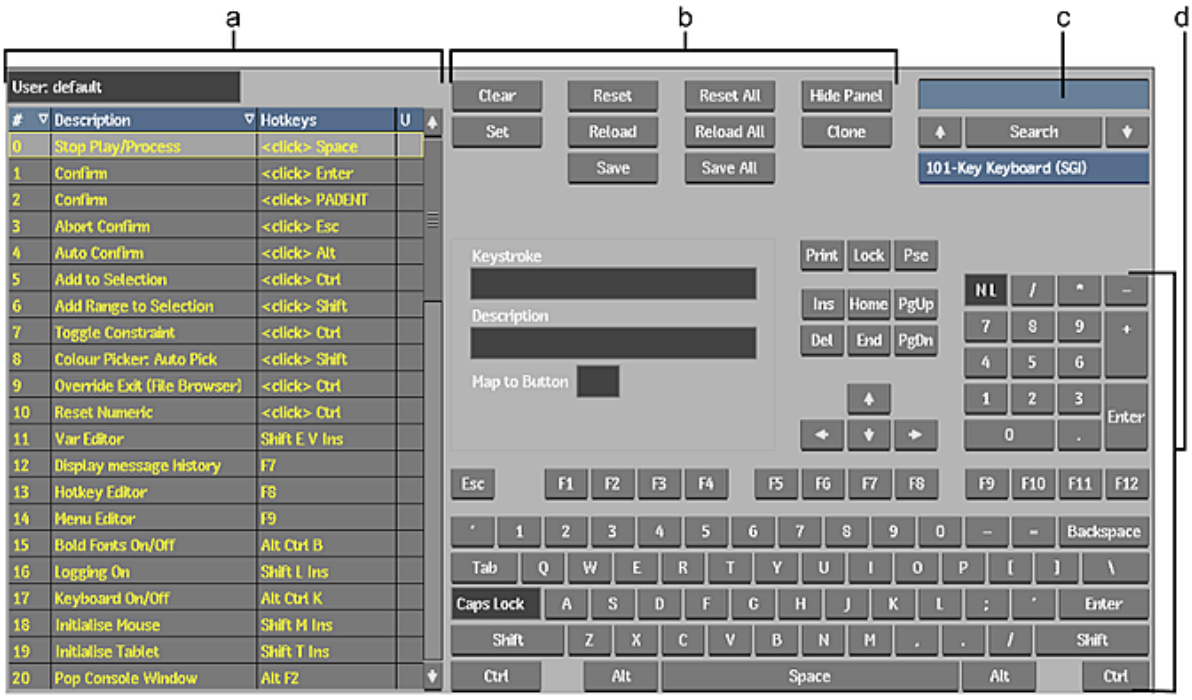


Accessing the Keyboard Shortcut Editor

Use the Keyboard Shortcut Editor to view, modify, and create keyboard shortcuts.

To access the Keyboard Shortcut Editor:

1 From the Flame menu, select Keyboard Shortcuts.



(a) Keyboard Shortcut Catalogue (b) Keyboard Shortcut Manager (c) Search Field (d) Keystroke Editor

Keyboard Shortcut Catalogue

A catalogue of keyboard shortcuts is maintained for every user. If the user profile was created in the default home directory, the catalogue is found in */opt/Autodesk/user/<user_name>/hotkey/*. If the user profile was created elsewhere, the catalogue is found in *<user home directory>/hotkey/*. The keyboard shortcuts in the keyboard shortcut catalogue are stored in the following formats:

- *default.<component>.butt.keyboard shortcut*
- *default.<component>.func.keyboard shortcut*
- *factory.<component>.butt.keyboard shortcut*
- *factory.<component>.func.keyboard shortcut*

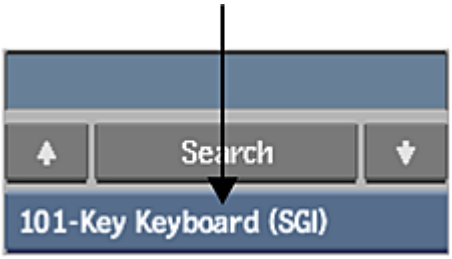
Where:	Means:
<component>	The module containing the keyboard shortcut.
default	The keystroke that is currently in use.
factory	The Autodesk factory default.
butt	A button keyboard shortcut.
func	A non-button function keyboard shortcut.

User-modified keyboard shortcuts are stored in a separate file: *default.<component>.butt.keyboard shortcut.user*. This file is loaded after the system keyboard shortcut file, *default.<component>.butt.keyboard shortcut*. Entries in the user-modified keyboard shortcut file replace the system keyboard shortcut entries, with the exception of cloned keyboard shortcuts, which are duplicated.

Selecting Your Keyboard Type

Keyboard shortcuts work best when you access the application from the dedicated account created when you installed Flame. In this case, the Autodesk *dlu* keyboard is selected by default. If you start Flame from a different account, you may notice some keyboard shortcut irregularities. On Linux systems, you can manually set the Smoke English *dlu* keyboard layout in **System Settings ➤ Regional & Language ➤ Keyboard Layout**.

In the Keyboard Shortcut Editor you can also select the type of keyboard you are using to take advantage of extra keys on certain keyboards.



When you select your keyboard, the onscreen keyboard is changed to reflect your choice, and entries in the Keyboard Shortcut Catalogue are updated accordingly.

NOTE If a keyboard shortcut is set on a key not available for the keyboard selected, the Keyboard Shortcut Catalogue displays the entry in black.

Searching the Keyboard Shortcut Catalogue

You can search the Keyboard Shortcut catalogue to find keyboard shortcuts whose description match your search criteria.

To search the Keyboard Shortcuts catalogue:

1. In the Search field, enter the characters you want to search for.
2. Click Search.

   The catalogue is searched and the keyboard shortcuts whose description match your search criteria are highlighted.
3. If more than one keyboard shortcut is highlighted, use the previous and next buttons located next to the Search field to move through the list of search results.

Changing Keyboard Shortcuts

The Keyboard Shortcut catalogue shows all keyboard shortcuts for your product. Keyboard shortcuts appear in yellow text or in white text in the list:

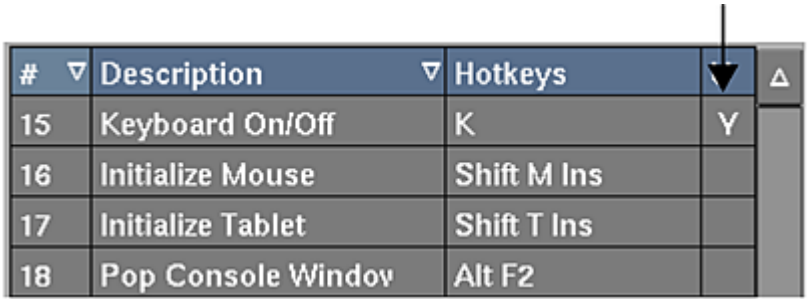
- Yellow text: identifies shortcuts that are used everywhere in your product.
- White text: identifies keyboard shortcuts that are specific to the area you are in.

To edit a keyboard shortcut:

1. From the area where you want to use the new keyboard shortcut, open the Keyboard Shortcut Editor.
2. Select the keyboard shortcut in the Keyboard Shortcut catalogue.

   The keystroke sequence and its description appear in the Keystroke Editor fields. If you select a map-to-button keyboard shortcut, the word “Yes” appears in the Map-to-Button field.
3. Click Clear in the Keyboard Shortcut Manager area to clear the existing keystroke sequence.
4. Enter the new keystroke sequence by clicking keys in the Keystroke Editor or by pressing keys on your computer's keyboard.
5. Click Set in the Keyboard Shortcut Manager area.
6. Click Save to save the changes to the current user's Keyboard Shortcut catalogue.

   The edited keyboard shortcut is marked in the catalogue with a “Y” to show that it is user-modified.



NOTE If you enter a keystroke sequence that is already in use, an error message is displayed.

Cloning Keyboard Shortcuts

You can use Clone to map multiple keystroke sequences to a single button, field, or function using the regular system keyboard. This feature does not provide macro functionality.

To clone a button, field, or function:

1. Select a button, field, or function in the Keyboard Shortcut catalogue.
2. Click Clone in the Keyboard Shortcut Manager area to create a second entry for this button, field, or function.

   A “Y” appears in the Keyboard Shortcut catalogue indicating that the cloned keyboard shortcut is user-modified.
3. Activate a control or enter a keystroke sequence on the system keyboard).

   NOTE You must enter a keystroke sequence that is not in use. Otherwise, an error message appears.
4. Click Set in the Keyboard Shortcut Manager area of the Keyboard Shortcut Editor.
5. Click Save to save the changes to the current user catalogue.

Editing Local Keyboard Shortcuts

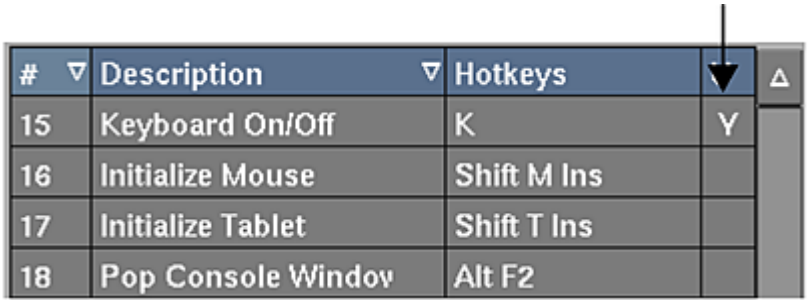
Almost every module has its own catalogue of keyboard shortcuts. The keyboard shortcuts listed in white are local keyboard shortcuts, and can be customized to suit your needs without affecting other modules. Modified keyboard shortcuts are saved as a user preference.

To edit a local keyboard shortcut:

1. In the module where you want to use the new keyboard shortcuts, access the Keyboard Shortcut Editor (`Alt+Ctrl+F8`).
2. Select the keyboard shortcut in the Keyboard Shortcut catalogue.

   The keystroke sequence and its description appear in the Keystroke Editor fields. If you select a map-to-button keyboard shortcut, the word “Yes” appears in the Map-to-Button field.
3. Click Clear in the Keyboard Shortcut Manager area to clear the existing keystroke sequence.
4. Enter the new keystroke sequence by clicking keys in the Keystroke Editor or by pressing keys on your computer's keyboard.
5. Click Set in the Keyboard Shortcut Manager area.
6. Click Save to save the changes to the current user's Keyboard Shortcut catalogue.

   The edited keyboard shortcut is marked in the catalogue with a “Y” to show that it is user-modified.



NOTE If you enter a keystroke sequence that is already in use, an error message is displayed.

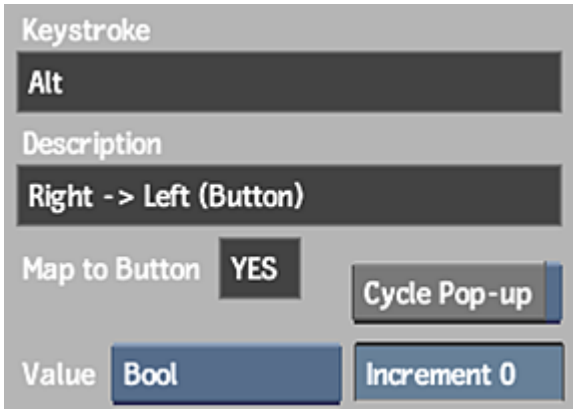
Editing Global and Shared Keyboard Shortcuts

You can edit global and shared keyboard shortcuts wherever the Keyboard Shortcut Editor is available. When you access the Keyboard Shortcut Editor through the Preferences menu, click Global or Shared to view all current global or shared keyboard shortcuts, respectively. In this menu, the names of these keys appear in yellow.

In other modules, the global and shared keyboard shortcuts appear in yellow in the Keyboard Shortcut catalogue. When changing a global or shared keyboard shortcut, you are warned that the change will affect all modules, and are asked to confirm the action.

Mapping a Button to a Keyboard Shortcut

You can create new map-to-button keyboard shortcut. New keyboard shortcuts are saved as a user preference.



To map a button to a keyboard shortcut:

1. With the Keyboard Shortcut Editor open, click the menu button for which you want to create the keyboard shortcut.

   NOTE If the button you want to click is hidden by the Keyboard Shortcut Editor, click the Hide Panel button.

   If the menu button you selected is a map-to-button keyboard shortcut, Yes appears in the Map to Button field, and the name of the menu button appears in the Description field.
2. Enter the new keystroke sequence by clicking keys in the Keystroke Editor, your computer's keyboard, or the keys on your pen tablet. The new keystroke appears in the Keystroke field.
3. If the menu button is a field, you can set a default value type and increment in the Value Type box and Increment field.
4. If the menu button is an option box, you can enable Cycle Pop-up to allow your new keyboard shortcut to cycle through the options each time it is pressed.
5. Click Set.
6. Click Save to save the changes to the current user's Keyboard Shortcut catalogue.

Keyboard Shortcut Editor Menu Settings

Keyboard Shortcut Manager Area

Clear button Clears the contents of the Keystroke field in the Keystroke Editor before you enter a new keystroke sequence.

Set button Sets the contents of the Keystroke field.

Reset button Resets the current local keyboard shortcuts to their default settings.

Reload button Reloads the current catalogue of keyboard shortcuts. This is useful if you made a change but have not yet saved it, and you wish to discard the change.

Save button Saves the current keyboard shortcuts to the Keyboard Shortcut Catalogue of the current user.

Reset All button Resets all keyboard shortcuts to their default settings.

Reload All button Reloads all keyboard shortcuts from the Keyboard Shortcut Catalogue of the current user.

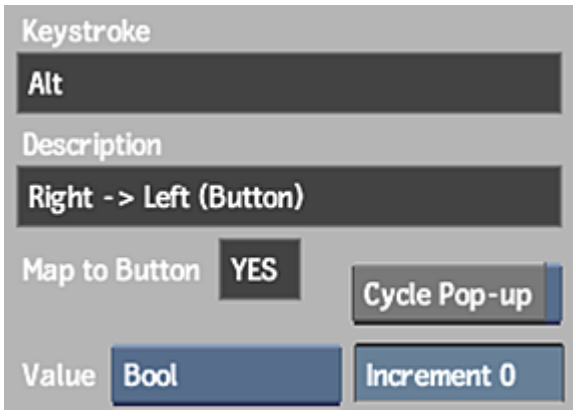
Save All button Saves all keyboard shortcuts to the Keyboard Shortcut Catalogue of the current user.

Hide Panel button Hides the Keyboard Shortcut Editor panel when you want to select a menu button hidden by the panel. The panel reappears when you select a menu button. Alternatively, click in an open area to return to the Keyboard Shortcut Editor panel without selecting anything.

NOTE This button is only available when accessing the Keyboard Shortcut Editor from a module.

Clone button Assigns a regular keyboard keystroke sequence to a single button, field, or function.

Keystroke Editor Section



Keystroke field Displays the keystroke sequence. Click keys in the Keystroke Editor or the keyboard to enter a new sequence.

Description field Displays the name of the selected menu button. Non-editable.

Map to Button field Displays whether the selected keyboard shortcut is a map-to-button hot key (a keystroke sequence that is mapped to a button or a field on the current menu). Non-editable.

Value Type box Select a value type for the selected menu button.

Integer Increment field Displays the integer increment for the selected value type. Editable.

Float Increment field Displays the float increment for the selected value type. Editable.

Cycle Pop-up button Enable to allow the new keyboard shortcut to cycle through options for the selected box.

Search Section

Search field Displays the search criteria for the Hot Key Catalogue. Editable.

Search button Performs a search based on the characters in the Search field.

Previous button Selects the previous highlighted item in the search results.

Next button Selects the next highlighted item in the search results.

Miscellaneous Settings

User field Displays the current user that hot keys are saved for. Non-editable.

Keyboard Type box Select the type of keyboard being used to take advantage of extra keys on certain keyboards. See Selecting Your Keyboard Type (page 1927).

Close button Click to close the Keyboard Shortcut Editor, and return to your previous view.

Setting Flame Preferences

33

Customize your working environment using preferences.

To open the Preferences dialog box:

1. Click **Flame ➤ Preferences**.
2. Select the category of preferences you want to modify.
   The dialog box opens to the tab with the category of preferences you selected.

Audio Preferences

Player

Enable Audio button Press to enable / disable audio in the application. It is enabled by default.

Monitor Sync box Indicates which monitor will have audio sync when you play back a clip (the broadcast monitor or the high-resolution monitor).

Broadcast Delay field Displays the number of frames (-30.00 to 30.00) to delay the audio signal when syncing to the broadcast monitor. Editable.

Hi-Res Delay field Displays the number of frames (-30.00 to 30.00) to delay the audio signal when syncing to the graphics monitor. Editable.

Auto Fade field Displays the amount of time (0 to 10 ms) that a fade is added to the start and end of each audio segment. Use this setting to avoid audible clicks between audio sources. Editable.

Meters

Meter Range field Displays the overall scale for the fader meters. Editable.

Meter Green field Displays the green meter range. Editable.

Meter Yellow field Displays the yellow meter range. Editable.

Meter Step field Displays the step value for the meters. Enter a lower value to view a more detailed meter.

Meter Units box Select whether the meters display digital or analog units.

Tone

Audio Tone box Select the default audio tone.

Reference Level field Displays the reference level for custom audio tones. Editable.

Tracks

Consider Audio Head/Tail button Enable to include audio that extends before or after video when exporting movie files.

Supported Playback Tracks field Displays the number of playback tracks supported by your sound card. Non-editable.

Supported I/O Tracks field Displays the number of I/O tracks supported by your sound card. Non-editable.

Supported Audio Monitoring Field Displays the audio monitoring option supported by your sound card. Non-editable.

Outputs

Output Source field Displays all current outputs. Non-editable.

Output Sampling Rate field Displays the current output sampling rate. Non-editable.

Monitoring Level field Displays the default volume level. Does not effect output. Editable.

Backburner Preferences

Backburner Manager box Select the hostname of the Backburner Manager system that handles background jobs submitted by the workstation.

Backburner Group box Specifies a server group (a preset group of render nodes) used to process jobs submitted by the application. By default, Backburner Manager assigns a job to all available render nodes capable of processing it. If you have a dedicated group of render nodes for processing jobs, set the value to the name of the render node group. See the Backburner User Guide for information on creating groups.

Burn Job Type box Configure according to the GPU capabilities of the nodes in your background processing network.

- Software: none of the nodes in your background processing network is equipped with a GPU. The application will not send jobs that require a GPU to the background processing network, but only jobs that can be processed in software mode (using OSMesa) by the render nodes.
- GPU Burn: all the nodes in your background processing network are GPU-enabled. The application will send all jobs to the GPU-equipped nodes in the background processing network, even if some jobs do not specifically require a GPU node. The GPU-equipped render nodes will render jobs that require a GPU, as well as OSMesa jobs. If your background processing network also contains nodes without a GPU, and this setting is used, all jobs are sent only to GPU-equipped render nodes, and the nodes without a GPU are never used.

Burn Priority field Displays the priority for renderings jobs, from 0 (highest priority) to 100 (lowest). Editable.

Wire Priority field Displays the priority for transfers from and to this workstation, from 0 (highest priority) to 100 (lowest). Editable.

Cache Priority field Displays the priority for caching imported media on this workstation, from 0 (highest priority) to 100 (lowest). Editable.

Export Priority field Displays the priority for exporting clips from this workstation, from 0 (highest priority) to 100 (lowest). Editable.

NOTE Regarding the different priorities, be careful when setting new values or you might end up with a system trying to transfer frames before they are rendered.

Hide Job Description button Enable to hide Backburner job descriptions.

Broadcast Monitor Preferences

Broadcast Monitor Preferences

Broadcast Monitor box Select the output device used by the broadcast monitor, or turn it off.

Broadcast Selection box Select what you want to preview on your broadcast monitor (can be changed on-the-fly).

Always Send Grab Area button Enable to display Player or viewport output, including times when neither is in use. Active when the Screen Grab option is selected.

Scale Clip to Fit Monitor button Enable to resize the clip to fit the broadcast monitor. Active when the Show Selected Item option is selected.

Use Ratio button Enable to maintain the original proportions of the clip. Disable to use the monitor ratio. Active when the Show Selected Item option is selected.

Show Clip On Selection button Enable to display in the broadcast monitor the selected thumbnail.

Broadcast Multiview Preferences

Viewport Monitoring button Enable to send to the broadcast monitor the content of the viewport displaying the "Monitor" symbol. Disable to use the broadcast monitor as an additional viewport; then use the Broadcast Monitor toolbar to control the broadcast monitor as you would any other viewport.

Hide Broadcast Toolbar button Enable to hide the Broadcast Monitor toolbar. By default, the toolbar is displayed in the top-right corner of the graphics monitor. Spacebar+Click to move the Broadcast Monitor toolbar.

Image Data Type Preferences

Image Data Type box Select the type of image data you are displaying in the broadcast monitor. Your selection determines the type of transformation that is applied to the clip to modify the contrast. It is disabled if the Follow button is enabled.

Bypass button Enable to deactivate the image data type display settings in the broadcast monitor.

Broadcast LUT Preferences

Use 3D LUT in Monitor button Enable to apply a 3D LUT from the 3D LUT list to the broadcast monitor.

3D LUT List box Displays the list of 3D LUTs that you imported in the LUT tab.

Broadcast Overlay Preferences

Broadcast Overlay box Select the overlays displayed on the broadcast monitor. "Off" for no letterbox or reference overlay, "Letterbox" for aspect ratio and letterbox guides, "All" for letterbox and all reference overlay elements.

Broadcast Stereo

Broadcast Stereo box Displays the currently-selected stereoscopic display option. The options are Dual Output, Interlaced, and Anaglyph.

Interlaced Option box When Interlaced is selected displays the options: Left Eye on Field 1 and Left Eye on Field 2.

Anaglyph Option box When Anaglyph is selected displays the options: Diff Clamped, Diff, Blend, Dubois, and Mono.

Broadcast Colour Space

Broadcast Colour Space box Defines the broadcast colour space.

Colour Space Option box Sets the broadcast colour space options: Normal or With Headroom.

Broadcast Timing

Broadcast Timing box Displays the timing rate options for the broadcast monitor.

Lock Timing button Locks the timing rate.

TIP For a list of supported broadcast monitor timings on Flame, see Broadcast Monitor Supported Timings on Linux Workstations (page 392).

General Preferences

Project Information

Project Frame rate field Displays the frame rate that corresponds with the current project's default resolution. Non-editable in Preferences, but can be changed in Project Settings.

Field Dominance field Displays the current project's field dominance. Non-editable in Preferences, but can be changed in Project Settings.

Incoming Sync

Incoming Sync field Displays the timing of the sync received by the workstation. Non-editable.

Timecode

Display Dual Timecodes button Enable to display both 29.97 and 23.97 fps timecode on the timeline and in the Player. This allows users to monitor which transitions will fall on jitter frames when 2:3 insertion is applied.

Dual TC: 23.97->29.97 fps NDF/DF box Select whether to view the 29.97 fps timecode in the Dual Timecode display as either drop frame or non-drop frame.

Drop Frame Reference Timecode field Displays the point at which the 29.97 fps material is in sync with the 23.97 fps material. Editable.

Drop Frame Bump Mode box Select a translation mode for invalid calculator entries.

Player

Source Time Mode box Select to display the timecode, keycode, or frame number of the source clip in the Player.

Auto-Toggle Player button Enable to open the Player automatically when you press the keyboard shortcuts for playback.

Display Sequence Viewer button Enable to display the currently selected sequence as a thumbnail with a red border in the Thumbnail view. When disabled, the selected sequence does not appear Thumbnail view.

Jog / Shuttle on Drag button Enable to shuttle a clip in the Player when jogging. The further from the center of the clip that the cursor is placed and dragged, the faster the shuttling.

Auto-Toggle Trim View button Enable to automatically display the Trim View when double-clicking a timeline cut, or when double-clicking a timeline segment with Slip or Slide mode selected.

Untitled Clip Name

Untitled Clip Name box Select an option to rename untitled clips to make it easier to distinguish multiple Untitled clips.

Rendered Clip Name

Rendered Clip Name box Select whether to add an acronym indicating the area of the software that clips were rendered from as a prefix or suffix to a rendered clip name, if a setup name does not already exist for the clip.

Stereo Naming

Left Eye field Displays the token used to identify left eye media in a stereo clip split or exported. Editable.

Right Eye field Displays the token used to identify right eye media in a stereo clip split or exported. Editable.

Animation

Default Interpolation box Select the default interpolation type for the channel editor.

Default Extrapolation box Select the default extrapolation type for the channel editor.

Audio Gain Animation

Default Audio Interpolation box Select the default audio interpolation type for the channel editor.

Autosave

Soft Save field Displays the time delay for a soft autosave. A small red icon appears indicating a two-second delay before a soft autosave occurs (allowing for a cancellation with any movement in the application). Editable.

Hard Save field Displays the time delay for a hard autosave. A hard autosave occurs at the specified time regardless of user intervention. Editable.

Tools Save field Displays the time delay for an autosave when using a tool that you accessed from the Tools tab. Editable.

Media Protection Mode button Enable to trigger an autosave after adding clips or folders to the Media Panel, or after processing clips on the Desktop. As with a regular autosave, the size of the Desktop and libraries dictates the time it takes for the save to complete.

For example, the following operations trigger a save when Media Protection is enabled.

- In the MediaHub tab, once all the clips are imported.
- In the Conform tab, after importing a sequence such as an AAF.
- After a VTR capture, once the clips are written to the Media Panel.

- After a Wire transfer, once the transfer completes.
- In remote access, whenever one of the above operation is performed.
- After an archive restore, as soon as the restore is done.
- After moving content from a reel to a library.
- After moving from a library to another library.
- After moving from a library to a reel.

Help Location

Help Location button Displays the location of the product help. By default, the product help is available online and requires an internet connection to view. Help files can be downloaded from the Autodesk website and installed locally.

If you want the local version of the Help, go to *http://www.autodesk.com/vxf* and look for the downloadable HTML version (zip) version of the help. You can unzip the file into the default Help location, or click Browse to select the folder where you unzipped the Help files.

Browse Help button Opens a browser window. Select the folder that contains the help files downloaded from the Autodesk website.

Reset Help button Resets the help location to the default.

Help Location field Displays the location of the help.

Setups

Scale Setups button Enable to scale setups when loading that were created in a resolution different from that of the current project.

Undo

Clear Buffer button Click to remove clips from the undo buffer and free up disk space on the framestore.

Undo Level field Displays how many levels of information are saved to the undo buffer. Editable.

Clip Select

Ignore Scan Format button Enable to disregard scan mode mismatches.

Background Tasks

Update field Displays a value, in seconds, for the wait time to update the Backburner Queue. Editable.

Rendering Display

Rendering Display box Select whether to display all frames as they are rendered, or to allow the display of all frames with potential tearing.

History

Keep History Mode box Select whether Sources or Intermediates and Sources appear on the workspace when working with clip history.

Default Rendering Options

Default Rendering Method box Select whether the default rendering method is Render Selection or Render.

Default Rendering Option box Select the default rendering option for the current project. Options depend on installed hardware and licenses (such as Burn or Background Reactor).

Include History button Enable to include History when rendering.

Proxy Rendering

Proxy Rendering box Select the rendering mode of your media. Options are: Full Resolution and Proxy, Full Resolution, and Proxy (Timeline Only).

Media Panel

Media Panel Position box Select the location of the Media Panel.

Thumbnail Icon Frame field Displays the frame used to create the thumbnail of a clip. If the clip is shorter than the specified value, the first frame is used instead. Editable.

Protect from Editing button Enable to protect all of the contents in the libraries from editing. In order to work on media in a protected library, it must first be moved to the Desktop area of the Media panel.

Show Batch Iterations button Enable to show the Iterations folder in Batch groups located on the Desktop and in libraries.

Duplicate Name Check box Use to select which media elements the application checks for duplicate naming in the Media panel, and prompts you to either add the item using the same name, rename the new item, or replace the existing item.

Shared Libraries

Shared Libraries Refresh Rate field Displays the frequence at which the application refreshes the contents of Shared Libraries. Editable.

Shared Libraries Lock Timeout field Displays how long an inactive shared folder remains available for writing. Editable.

Input Devices Preferences

Pointer

Tablet Model field Displays the model of the currently installed tablet. Non-editable.

Pointer Reset button Returns all pointer preferences to their default values. Available on Linux systems.

Threshold Test button Assesses the sensitivity of the pen interactively. Press the pen on the button and a sliding grey bar in the message bar displays the response to the amount of pressure applied. Available on Linux systems.

Pressure Threshold field Displays the amount of pressure that you want to apply when using the pen. Use a higher value to decrease the sensitivity of the pen (more pressure required). Available on Linux systems. Editable.

Use Mouse Cursors button When using a tablet, enable to use mouse cursors instead of tablet cursors.

Trackball Pressure button Enable to allow the trackballs to respond to changes in the pressure applied when using a stylus.

Tablet Margins (available on Linux systems).

Proportional Margins button Enable to change the four margin controls proportionally.

Top Margin Control field Displays a percentage value to adjust the top boundary of the active area on the tablet. Editable.

Bottom Margin Control field Displays a percentage value to adjust the bottom boundary of the active area on the tablet. Editable.

Left Margin Control field Displays a percentage value to adjust the left boundary of the active area on the tablet. Editable.

Right Margin Control field Displays a percentage value to adjust the right boundary of the active area on the tablet. Editable.

Tablet Stylus Buttons

Use OS Settings button Enable to use the Wacom stylus shortcuts mapped in the System Preferences of the operating system. Disable to use the buttons to access shortcuts mapped in the Keyboard Shortcut editor. Available on Mac systems.

Colour Management Preferences

Working Colour Space box Select the default Colour Space for the project. This is available as the "From Project" option in various colour space menus.

Name box Enter a name to use for your exported policy, then click Export.

Export Policy button An exported Colour Policy contains the Input and Viewing Rules, and the Working and Action default colour spaces. It also contains any user-created colour spaces. (It does not contain the Graphics/Broadcast Monitors since these are workstation specific.) The policy consists of a directory of files that is created at the Shared Path location shown below. The policies are available when creating new projects. (To change the policy of an existing project, you must edit the project).

Broadcast Colour Space box Select the colour space that best describes your Broadcast Monitor.

Graphics Colour Space box Select the colour space that best describes your Graphics Monitor.

Action Colour Space box Select the default colour space to be used in Action.

Default Look Transform field Specifies the colour transform that is used when the defaultLook alias is referenced inside the viewing transform. The ACES Output and several other view transforms are set up to work with a Look transform.

Clear button Sets the value of the defaultLook alias to the identity transform, which has no effect on colour values.

Browse button Browse to select a different colour transform for the broadcastMonitor alias.

Collaboration and Services Shared Path When collaborating with or using services from remote workstations, this is the location used to share user-created colour spaces and transforms. There will be a soft link created in this location back to the colour space data of this project. This path is set in the Flame Setup application.

Input Colour Space Rules Table The Input Rules allow you to assign a colour space to media based on its filename. The Extension field applies to the file extension (do not include the dot). The Pattern field applies to the entire path (except the file extension). The fields use Unix "Glob" style pattern matching. A "*" matches any characters. Brackets "[]" may be used to match groups of characters. Note that the matching is case-sensitive. For example, "dpx" only matches "dpx", whereas "[dD][pP][xX]" matches "dpx", "DPX", "Dpx", etc.

You can also duplicate existing rules and delete rules, using the Duplicate and Delete buttons.

Viewing Rules table The Viewing Rules allow you to define how viewports show media and how thumbnails are displayed. Start by selecting a View Transform for the rule. In a viewport, a rule will only be available if the tagged colour space of the media being viewed matches the Allowed Colour Spaces of the rule. You may make this rule only apply to a specific colour space, or use one of the "Any ..." settings to make it available to a family of colour spaces. The choices available in Allowed Colour Spaces are based on what the selected View Transform may accept (e.g. you may not allow scene-linear colour spaces into View Transforms intended for video). If the actual tagged colour space of the media does not match what the View Transform expects, auto-conversion will be applied to connect them. The Allowed Display setting controls whether a rule will be available based on your Graphics and Broadcast monitor settings. For example, if a LUT produces Rec.709 values you can choose to only show this rule when on a Rec.709 monitor. If you choose one of the "Any ..." options, auto-conversion will be applied to correct the output of the View Transform for the actual monitor. Toggling off the check mark in the left column allows you to temporarily bypass rules.

Create Rule button Use to create a new rule.

Duplicate button Use to duplicate an existing rule.

Delete button Use to delete a rule.

Priority Up button Click to raise the priority of a rule. Rules higher in the table take precedence over lower ones.

Priority Down button Click to lower the priority of a rule. Rules higher in the table take precedence over lower ones.

Import Rules button Click to browse and replace either these Input or Viewing Rules with another.

Diagnostic Shortcuts table Use this table to create shortcuts that will set specific Exposure, Contrast, and Gamma settings in a viewport. These settings are stored with your User.

Reset All button Reset all of the values set in the Diagnostic Shortcuts table.

Sync with OS button Available on Mac systems. Automatically uses the ICC profile specified in your computer's preferences as the Graphics Monitor setting. This also creates a user Display colour space from the ICC profile which is labeled with the workstation and date.

Shader Paths Preferences

Batch or Batch FX Node Bin

Default button Click to reset to the default Matchbox file path location for the dynamic Matchbox node bin.

File Path field Displays the file path location for the Batch or Batch FX dynamic Matchbox node bin. Click Browse to change the file path.

Browse button Click to open the browser to select a Matchbox file path location for the dynamic Matchbox node bin.

Action Node Bin

Lightbox

Default button Click to reset to the default Lightbox file path location for the Action dynamic Lightbox node bin.

File Path field Displays the file path location for the Action dynamic Lightbox node bin. Click Browse to change the file path.

Browse button Click to open the browser to select a Lightbox file path location for the Action dynamic Lightbox node bin.

Matchbox

Default button Click to reset to the default Matchbox file path location for the Action dynamic Matchbox node bin.

File Path field Displays the file path location for the Action dynamic Matchbox node bin. Click Browse to change the file path.

Browse button Click to open the browser to select a Matchbox file path location for the Action dynamic Matchbox node bin.

Storage Preferences

Space Calculation

Space Calculation options Choose to display all available frames in the message bar, all available and purgeable frames in the message bar, or to purge frames before making a space calculation.

Volume Statistics button Lists the number of frames, proxies, and audio sources stored in the clip libraries, and main work area of each project.

Video Framestore Estimate button Displays the estimated amount of space left on the video framestore.

Framestore

Test Disks button Displays the number of frames read per second; the number of seconds required to read a single frame; and the amount of data, in megabytes, read per second.

Local Gateway

Restart button Restarts the local Wiretap Gateway service, which must be running for you to import and export media. Use this button when troubleshooting failing import or export processes.

Temporary Libraries

Clear All button Click to delete temporary libraries from the current project. Do not use if background operations are running. Use the Background Tasks monitor to ensure all tasks are done before using.

Temporary libraries are used by background tasks, and are usually automatically deleted by Flame. But it can happen that they are not deleted correctly, build up over time and tax your storage. Clearing these libraries recovers that lost space. This is a safe operation unless background operations are running: wait for them to complete before clearing the temporary libraries.

Support Preferences

Upload Config Info Only button Click to upload only configuration information to Autodesk Customer Support. Use only if instructed by Autodesk Customer Support.

Upload Config Info and Logs button Click to upload pertinent configuration information and system logs to Autodesk Customer Support. Use only if instructed by Autodesk Customer Support.

Max Logged Days field Displays the number of logged days of information to be uploaded to Autodesk Customer Support. Editable.

Case ID field Enter the Case ID number given to you by Autodesk Customer Support so that it is included in the uploaded information.

Timeline Preferences

Editing

Default Timecode field Displays the default start timecode value for new items overwritten or inserted into a timeline sequence. Editable.

Merge Tracks Mode box Select how a timeline will be flattened when merging tracks.

Select:	To merge tracks by:
Simple Track Merge	Hard committing all soft transitions. Cut points between elements are preserved; clip handles are not.
Complex Track Merge	Hard committing soft transitions but creating separate elements with clip handles for each.
Committed Track Merge	Preserving editable soft transitions and clip handles.

Auto Timewarp button Enable when performing a four-point edit if you do not want to timewarp the frames from the source clip. When disabled, a four-point edit is treated as a three-point edit.

When you specify the number of frames in the sequence clip for the four-point edit, the same number of frames is used in the source clip. The source clip selection begins at the in point. If the source is shorter or longer than the number of frames specified in the sequence clip, a timewarp is applied to the source clip in order to match the duration of the sequence clip.

Frame Rate Converter button Enable to format the source clip to the correct destination frame rate by applying a video timewarp to the source clip.

Snap Includes Marks button Enable to snap to timeline marks.

Snap With Positioner button Enable to snap segments to the positioner.

Display Phantom Marks button Enable to turn on phantom marks on the timeline and source clip. Phantom marks indicate the result of a 4-point edit regardless of whether you have marks set.

Mark Behaviour box Select whether Out marks are inclusive or exclusive of the selected frame.

Add Track box When adding a new timeline track, select whether the track is added to the version on top of the last existing track, or to the track above the focus point.

Keep Focus After Editing button Enable to keep the timeline focus on the source after an editorial operation. Disable to switch the focus to the sequence.

Containers

Uncontain options box Select whether new tracks and versions are added or overwritten when a container segment is uncontained.

Contain Timewarped Edits button Enable to preserve the edits in a fit-to-fill four-point edit. The incoming clip is contained and a timewarp is applied to the container instead of the clip.

Segment Display

Handles box Select an option for displaying heads and tails on segments in the timeline.

Show All Metadata button Enable to display expanded clip information on timeline segments. You may need to increase the size of the track to see the metadata.

Navigation

Frame Positioner box Select whether to snap the timeline positioner frame-to-frame, or on a sub-frame basis (one-tenth of a frame).

Autoscroll field Displays the speed at which the timeline scrolls. Editable.

Scroll During Playback button Enable to scroll the timeline during playback, keeping the frame positioner visible.

Scroll Past First Frame button Enable to allow the positioner to move before frame one.

Transitions

Dissolve Duration field Displays the default duration for dissolves. Editable.

Transition Alignment box Sets the default alignment for dissolves, wipes, axis, and custom transitions.

Curve Type box Select bezier or linear as the default curve type for dissolves.

Rendering

Timeline Effects Render Mode box Select the render mode when a Timeline FX is added to a clip.

Timewarp Render Mode box Select the default rendering mode for timewarps created in the timeline.

Transitions Render Mode box Select a default rendering mode for transitions created in the timeline.

Timeline Effects Rendering box Select whether to render an entire track first, then the next track (Track Based Render), or to render all tracks frame-by-frame (Frame Based Render).

Include Handles button Enable to take the handles into account when rendering Timeline FX.

Infinity Handles slider Use to limit the handles of virtual segments. This value is set in frames.

Default Shot Name Tokens button Click to add tokens to the default shot naming convention.

Full Container Render button Enable to render all content in a container or matte container, to be used in Lustre, for example.

Timeline FX / Batch / Batch FX Preferences

Batch FX

Interactive Max field Displays the number of seconds that the system uses to attempt a render when previewing the effect. Editable.

Add Comp On Matte Output button Enable to have a Comp timeline effect automatically added to the timeline when a matte is output from the output node to the timeline.

Clear Rendered Memory button Enable to automatically clear an FX node (and all nodes in setups nested within the clip) from memory when its output is rendered.

Batch FX Media Mode

Head Media option box Select how to interpret missing information before a clip.

Gap Media option box Select how to interpret missing information during a clip.

Tail Media option box Select how to interpret missing information after a clip.

Batch FX Automatic Nodes Copy

Automatic Nodes Copy box Select whether FX nodes are never copied into memory, always copied, copied when there is an Action node in the setup, or copied in a partially rendered setup.

Timeline Timewarp

Interpolation Option box Select an interpolation option for your interlaced timewarp.

Use Last Speed button Enable to apply the last timewarp speed to future timewarps.

Timewarp Speed Display box Select whether timewarp speed is displayed as a percentage (%) or in frames per second (fps).

Timewarp Sample Option box Select whether the speed of the timewarp is set in relation to the speed at the beginning, middle, or end of the timing curve for each frame or field. You can also change the Timewarp Sample option in the Timewarp Editor to override this selection.

Timeline Resize

Fit Method box Select a fit option for clips that must be resized when they are added to the timeline because their resolution is different from the project.

Resize Quality box Select a sampling algorithm for clips that must be resized when they are added to the timeline.

Keep Aspect button Enable to maintain the aspect ratio of clips that are resized when you add the to the timeline.

Timeline Text

Text Anti-aliasing box Select an anti-aliasing rate for Text effects. Auto-Softness mode is set by default and provides a softness equal to the display quality you see in the Text effect. An anti-aliasing rate of 1 means no anti-aliasing is performed and the result is a hard edge. Other anti-aliasing rates increase the sampling rate, but also increase processing time.

Default Font field

Displays the default font for entering text. Click to open the font browser to select a different font.

Timeline Action

Use Background button Enable to connect the background automatically so that Action composites your effect. When disabled, the background is off and the matte is configured to output the result of the scene, so it can be comped by the Comp Timeline FX. You can override this preference for specific Action Timeline FX with the Use Back button in the Action quick menu.

Batch / BFX Action

Action Display Default box Select whether Action nodes in the Batch or Batch FX schematic are collapsed or expanded by default.

Timeline Comp

Default Segment Premultiplication box Select the default segment premultiplication setting for timeline comp effects.

Default Background Premultiplication box Select the default background premultiplication setting for timeline comp effects.

Default Output Premultiplication box Select the default output premultiplication setting for timeline comp effects.

Batch & BFX Defaults

Default Viewport Layout box Select the default Batch FX view. You can also change the viewport layout from within Batch FX.

Batch Start Frame

Batch Start Frame field Displays the default Batch start frame used in new Batch setups. Editable.

Ignore In Publish/Conform button Disable to have the Batch Group created by the Create Batch Group button (Connected Conform workflow) start at the source frame of the background track. Enable to start the Batch Group at frame 1.

BFX Cache Life Span

BFX Cache Life Span field Displays the number of days that Batch FX clips are cached. Editable.

BFX Setup

Batch Cache Life Span field Enable to back up your setup on exiting Batch FX. This backup is used only if something goes wrong with the Batch FX setup, and is not accessible like the regular save that you can create within the module.

Batch Import / Read File

Absolute Frame Offset button Enable to allow all clips imported with an Import node or a Read File node in Batch to automatically use the start frame of the clip.

Batch Cache Life Span

Batch Cache Life Span field Displays the number of days that Batch clips are cached. Editable.

Batch/BFX Iteration

Save Setup File button Enable to also save a setup file when saving a Batch Iteration.

Append Original Name button If enabled, when copying or moving a Batch Group Iteration to another Batch Group Iteration, this option appends the original name of a Batch Group Iteration to the new one. This enables you to determine, at a glance, where the Iteration came from. This preference is disabled by default.

Open Naming Dialog Controls the behaviour of naming fields for Batch, the Render node, and the Write File. When enabled, clicking in the naming field opens a Rename window. When disabled, clicking in the field to edit directly in the field. Shift-Click the field to invert the behaviour.

Default Paths and Names

Use this section to set default paths and names for different areas of Flame. To meet the needs of larger facilities, system administrators can use Python hooks to configure customized Setup paths. For more information, see Python Hooks Reference.

Default Batch Group / BFX Setup Path field Displays the default Setup path used for Batch Groups. This field is editable; however, removing the "~" from the path results in the setups located in the defined directory not being archived by Archive Setup or Archive Project.

Batch Group / BFX Path Token box Select a token to be included in the default Setup path for newly-created Batch Groups. Repeat to add additional tokens.

Default Batch Iteration Setup Path field Displays the default Setup path used for Batch iterations. Editable.

Batch Iteration Setup Path Token box Select a token to be included in the default Setup path for newly-created Batch Iterations. Repeat to add additional tokens.

Default Batch Iteration Name field Displays the default name used for Batch iterations. This field is editable; however <batch name> and <iteration> are mandatory tokens.

Batch Iteration Name Token box Select a token to be included in the default name for newly-created Batch schematics and iterations. Repeat to add additional tokens.

Default Render Node Name field Displays the default name used for newly-created Render nodes. Editable.

Render Node Name Token box Select a token to be included in the default name for newly-created Render nodes. Repeat to add additional tokens.

Default Write File Node Name field Displays the default name used for newly-created Write File nodes. Editable.

Write File Node Name Token box Select a token to be included in the default name for newly-created Write File nodes. Repeat to add additional tokens.

Default Action Geometry Import Path field Displays the default path used in the Action Import browser. Editable.

Action Geometry Import Token box Select a token to be included in the default path for importing Action geometries. Repeat to add additional tokens.

User Interface Preferences

Colour

Background field Displays the background brightness value. Editable

Brightness field Displays the brightness of interface elements such as buttons and fields. Editable.

Contrast field Displays the contrast of interface elements such as buttons and fields. Editable.

Colour Sampler

Show Display Colour Space Info button Enable to add a second set of RGB values in the pixel sampler showing the values after the viewport colour management has been applied. This also adds a read-out of the x/y pixel coordinates.

On-Screen Widgets

On-screen Keyboard button Enable to have the on-screen keyboard appear when you enter text in a field.

Calculator Placement box Select where the calculator appears when you click in a numeric field.

Player & Viewports

Default Viewing Zoom box Use to select whether content displayed in a viewport or player should fit the viewport by default or remain at its original size.

Fit Allows Upscale button Enable to force images smaller than the viewport to be upscaled in order to fit the viewer. When disabled the maximum zoom is locked at 100%.

Default Home Zoom field Use to set the default zoom value (percentage) of the Home function. Default is 100%.

Display

Drag Transparency field Displays the transparency level of clips while dragging. Editable.

Bold Font button Enable bold fonts on buttons. Available on Linux systems only.

Time Display Mode

Time Display Mode Box Select to display frames or timecodes in frame marking controls, the timebar, segments, and the fields below the timeline.

Force TC in Conform List button Enable to force the Conform Table to display Timecode information, even if Time Display Mode is set to Frame.

Thumbnails

Clip Name Only button Enable to only display the name of clips on thumbnails.

Secondary Information box Select the secondary clip information displayed on thumbnails. Options are:

- No Secondary Information
- Record Timecode
- Source Timecode

Resolution Information box Select the clip resolution information displayed on thumbnails. Options are:

- No Resolution Information
- Resolution
- Frame rate
- Resolution & Frame rate

Display Colour Space button Enable to display the tagged colour space on thumbnails.

Display Keycode button Enable to display keycode information on thumbnails.

Render Effects button Enable to display rendered frames in thumbnails of clips with applied but unrendered effects. Disable to display "Unrendered" in the thumbnails of such clips.

Apply Colour Management button Enable to apply the highest priority Viewing Rule to each thumbnail based on its tagged colour space and the selected graphics monitor colour space.

Auto Key Button Look

Auto Key Look box Select whether to apply a Classic (grey) or Coloured (yellow) look to the Auto Key button.

Confirm Dialog

Reset Visibility button Resets the visibility of previously hidden confirmation dialog boxes.

Freeform View

Snap to Grid button Enable to snap thumbnails to a grid. The grid becomes visible when this button is enabled.

Grid Size field Displays a value in pixels for the size of the boxes that make the grid. Editable.

Thumbnail Height field Enter a default value in pixels for the height of thumbnails displayed in the Thumbnail view. All new thumbnails are generated at this height. Existing ones are unchanged.

Desktop

Create Reel Group button Enable to automatically create a Reel group for all new or cleared Desktops, causing the Reel group to be selected and the initial Batch group to be collapsed in the Media panel. Enabled by default.

Batch Copy Mode toggle Toggle to choose whether a clip is copied or moved from / to a Batch Group. Options are: "Move From/To Batch" & "Copy From/To Batch".

Tooltips

Auto Display button Enable to automatically display tooltips. If disabled, you can still display tooltips for selected buttons using the keyboard shortcut Ctrl+Alt+W.

Hover field Displays the amount of time you must position the mouse over the object before the tooltip displays (in seconds). Editable.

Reels

Friction field Displays how much the reels scroll after you release your mouse/pen. Increase the value to limit the amount the reel scrolls after you release the mouse/pen.

Default Reels Number

Number of Batch Schematic Reels field Displays the default number of Batch Schematic reels when creating a new Batch. Editable.

Number of Batch Shelves field Displays the default number of Batch Shelves when creating a new Batch. Editable.

Number of Reels in Groups field Displays the default number of reels contained in a Reel group. Editable.

Desktop Reels

Reels Orientation box Select a horizontal or vertical orientation for the reels.

Reverse button Enable to reverse the order that clips appear on the reels.

Clip Spacing field Enter a value to control the padding between clips on the reels.

Gestural Workflow

Swipe Bars button Click to enable swipe bars that you can use to switch views in different areas of your product.

Swipe Sensitivity slider Use to set the sensitivity of the swipe gesture in the application, as a percentage value. The default value is 50%.

Layout Selection Overlay button Click to enable the menu overlay in the Timeline view. You can use the menu overlay to change what is displayed in the Timeline view.

Setting Up a Flare Workflow

34

About Flare

Flare is a fully compatible assistant to Flame, featuring the entire creative toolset in Batch. All Batch nodes found in Flame are fully supported in Flare. You can extend the capabilities of Flame by performing any Batch task on a Flare system. You can also offload time-consuming tasks such as rotoscoping and particle creation to Flare.

Because the focus of Flare is on the Batch toolset, there are some tasks performed by Flame that cannot be performed by Flare. For example, Flare does not support video I/O or conforming. In addition, Flare supports archiving, but only to or from file archives.

You can use Flare in an independent system workflow or in a remote connection workflow. In a remote connection workflow, the same storage is used by both Flare and Flame so there is no duplication of media. Multiple Flare systems can connect to the same Flame system. Each can be used to work on the same project at the same time, speeding up the production pipeline workflow. You can even perform project management tasks of Flare folders without leaving the Flame station.

If being able to work on projects collaboratively with Flame is not your main objective, you can still take advantage of the assistant capabilities of Flare in an independent system workflow. There will be duplication of media since you need Wire to transfer media to/from a Flare system. However, there are fewer workflow considerations than in a collaborative environment since Flare work is done on its own storage volume.

The rest of this chapter discusses Flare in the context of a remote connection workflow.

Flare Workflow

The following workflow provides one example of working in a remote connection collaborative environment. In the example, Flare remote connects to a Flame system and loads the media it will work on directly from a Flame project. Flame has direct access to the Flare media at all times since Flare uses the Flame storage volume.

Though you have granular control over the media you choose to share between workstations (clips, reels, Batch iterations, saved Desktops), the workflow detailed here assumes you are sharing saved Desktops.

Flare with Flame workflow:

1 Start the application and connect to a remote host computer. See Starting Flare (page 1953)

2 Restore the saved Desktop from the Flame project's shared folder by right-clicking the saved Desktop and selecting Replace Desktop from the contextual menu or by dragging it to the Flare Workspace entry in the Media Panel. You are prompted to either Replace or Cancel.

NOTE Make sure the Desktop was saved to a shared folder in Flame to make them accessible to Flare.

For more information, Shared Libraries (page 1952).

3 Work with the Batch toolset. See Using Batch and Batch FX (page 495).

4 Save your Desktop to a Flare Library. See Saving and Restoring the Desktop (page 63).

5 Copy the saved Desktop back into the shared folder, making it available to the Flame workstation.

TIP It is recommended to create a sub-folder within the Flame shared folder named From_Flare, for example, to be able to easily distinguish the initial media from the result media. In order to create a sub-folder, make sure to acquire exclusive access (lock).

Considerations for Working in a Collaborative Environment

The ability to work in a collaborative environment is integral to the Flare remote connection workflow. From Flare, you can open a Flame project on a remote storage volume and access all of the media stored in a shared folder. Conversely, Flame systems can also access Flare media.

Workspaces

When Flare connects to a remote Flame workstation, the default behavior is to automatically create its own Flare workspace, associated to its own workstation, and which has access to all the contents of the shared libraries in the Flame project. However, you can also select the Flame workspace directly, but this entails that the Flame station is not being used, and hence doing this renders simultaneous work on the Flame and Flare workstations impossible.

If you wish to work collaboratively between Flame and Flare, one of the following three scenarios applies:

- If there is only one existing workspace and that it **is** associated to the current Flare workstation, Flare selects it automatically.
- If there is only one existing workspace and that it **is not** associated to the current Flare workstation, you can either use the existing workspace or create a new one, from the Workspace drop-down menu.
- If there are many existing workspaces, you can select an existing workspace or create a new one, from the Workspace drop-down menu.

NOTE Workspaces are project specific. You can only access the workspaces of the current project.

Shared Libraries

About Shared Libraries

The Flare workflow is based on shared libraries. Any media placed in the shared libraries becomes accessible to remote workstations that connect to the project. Shared libraries enable Flame and Flare to quickly and conveniently share media, allowing you to spend more time on creative tasks and less time managing your media.

Exclusive Access

Placing media in a shared library automatically grants read access to any remote workstations connected to the project. The Flare operator can access the media and start working immediately after connecting to the Flame project. Once finished, the Flare operator needs to save the resulting media back to the shared library. To do this he must acquire exclusive access.

Exclusive access locks the library so that only the user having acquired exclusive access can modify the library contents (i.e. write to the library). Read access is still available to other remote workstations connected to the project. Once the media is saved, the user can release exclusive access, enabling other remote workstations to obtain exclusive access and modify the content.

To acquire exclusive access:

1. Select the shared library you want to acquire exclusive access to in the Media panel.
2. From the contextual menu, select Acquire Exclusive Access.

NOTE By default, after 600 seconds of inactivity, your exclusive access will be released. You can customize this setting in the user preferences (in the General tab).

To release exclusive access:

1. Select the shared library you want to acquire exclusive access to in the Media panel.
2. From the contextual menu, select Release Exclusive Access.

Open / Close

You can open and close shared libraries. Closing shared libraries turns off access to the media within the shared libraries as well as freeing up your system memory.

To open / close shared libraries:

1. Select the top level shared library you want to open or close in the Media panel.
2. From the contextual menu, select Open Library or Close Library.

Refresh

The content of shared libraries is automatically refreshed every 60 seconds. You can customize this interval in the user preferences (in the General tab). You can also manually refresh the content of shared libraries, to make sure you have access to all the shared media.

To manually refresh the content of shared libraries:

1. Select the top level shared library you want to refresh in the Media panel.
2. From the contextual menu, select Refresh.
   The content of the selected shared library is updated.

Starting Flare

To work in a remote connection environment, open a remote storage volume on start-up.

To start Flare:

1. Double-click the application icon on the desktop.

The Project Management menu appears.

2 Select a remote host from the Host Computer box.

3 Select a project from the Project box or create one. See Managing Projects and Users (page 47).

4 Select a Workspace or create a new one. See Workspaces (page 1952).

5 Select or create a user.

See Managing Projects and Users (page 47).

6 Click Start.
The creative finishing tools found in Flame Batch appear.

To exit Flare:

1 Click the Flare button at the bottom right of the user interface.

2 Select Exit Flare.

Changing Projects and Users

You can create or load other projects, as well as create, change, and edit user profiles at any time within a Flare session.

You perform these tasks from the Project and User Settings section of the Preferences menu.

You create projects and users in the same way as from the start-up menu.

To access the Project and User Settings:

1 Click the Flare button at the bottom right of the user interface.

2 Select Project and User Settings
The Project Management dialog box appears.

To change projects in a Flare session:

1 Select a project from the Project box.

2 Click Load.
The Confirm dialog box pops up.

3 Click Confirm.

4 Click Close.
The new project is loaded.

To change users in a Flare session:

1 Select a user from the User box.

2 Click Load.

NOTE You can select a local or remote user by first selecting the appropriate option from the List From box.

3 Click Close.
The new user is loaded.

To edit a user profile in a Flare session:

1. Select a user from the User box.

   NOTE When a remote user profile is selected, the profile is unavailable to other users. This avoids concurrent user profile modifications.

2. Click Edit.
3. Modify the user.
4. Click Done.

   The Confirm dialog box pops up.
5. Click Confirm.
6. Click Close.

   The modifications are applied to the user.

1956

Index

A

B

C

D

E

F

G

H

I

J

K

L

M

N

O

P

Q

R

S

T

U

V

W

X

Y

Z

1974