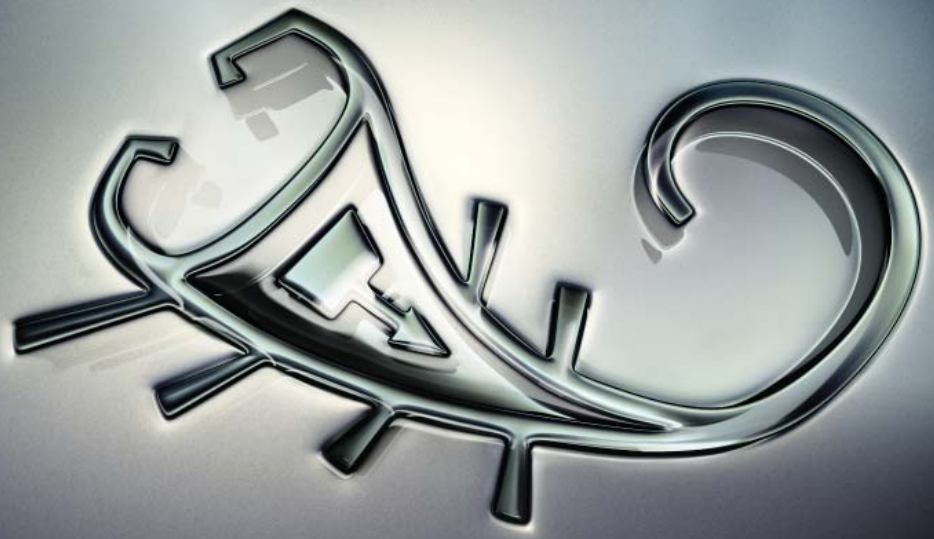
Autodesk®
Inferno® 2011
A Discreet® Systems product
with Autodesk® Flare™ 2011

New Features Guide



Autodesk®

Autodesk® Visual Effects and Finishing 2011

Trademarks

Disclaimer

Published by: Autodesk, Inc.
111 McInnis Parkway
San Rafael, CA 94903, USA

Images Courtesy of: Quietman

Title: Autodesk Inferno 2011 with Autodesk Flare 2011 New Features Guide

Document Version: 1

Date: March 09, 2010

Contents

Introduction 1

About the Documentation

Autodesk® Inferno® 2011 and Autodesk® Flare™ 2011 include documentation that helps you install, configure, and use your product.

For a list of all the documentation available to you, visit *http://www.autodesk.com/inferno-documentation-2011* and *http://www.autodesk.com/flare-documentation-2011*.

Refer to the Release Notes for all late-breaking information.

Using the New Features Guide

This New Features Guide describes the new and updated features for this release of Inferno. For a quick look at the New Features, see What's New on page 5. Some of the major features also have more information in this guide — just follow the links from the What's New chapter.

Viewing Tooltips

Your application includes tooltips that describe objects on the user interface (such as buttons and fields). The tooltips also display the hotkey for the object, if one is configured.

To view tooltips:

➤ Move the cursor over the object.

After a few seconds, the tooltip displays.

In the Preferences menu, you can turn on and off the display of tooltips. You can also change the amount of time your cursor must rest on an object before the tooltip displays.

Viewing the Help

Included with your application is a Help system that you can view in a Web browser. The Help is installed automatically and is accessible from anywhere within your application.

The Help is best viewed using Firefox® 2 or Internet Explorer 7.

To view the Help:

1. Start your application.
2. Click Preferences to open the Preferences menu and click Help.

   You can also access the Help by clicking the Help button, which appears on the bottom-right of the Desktop.

   TIP Press **Ctrl**+= to open the Help from anywhere in your application.

   A browser launches displaying the Help.

TIP To view the Help without interrupting a client session, copy the *documentation/help* folder from the product DVD to another system, such as your laptop. To view the Help, open the *help/index.html* file.

Notation Conventions

A number of style conventions are used throughout your documentation. These conventions and examples of their use are shown as follows.

Convention	Example
Text that you enter in a command line or shell appears in Courier bold. Press the Enter key after each command.	**`install rpm -qa`**
Variable names appear in Courier, enclosed in angle brackets.	`<filename>`

Convention	Example
Feedback from the command line or shell appears in Courier.	`limit coredumpsize`
Directory names, filenames, URLs, and command line utilities appear in italics.	*/usr/discreet*

Autodesk Media and Entertainment Training

There are many training options available to help you be more creative and productive with your application.

For all your training options, see *http://www.autodesk.com/inferno-training*.

Contacting Customer Support

For Autodesk Media and Entertainment Customer Support, visit *http://www.autodesk.com/support*.

Customer support is also available through your Autodesk reseller. To find a reseller near you, consult the reseller look-up database at *http://www.autodesk.com/resellers*.

What's New

2

About This Release

This release of Inferno introduces many new and updated creative tools, as well as workflow improvements.

Pixel Shader Rendering

This release introduces a new GPU-based processing pipeline in Action to replace the OpenGL fixed-pipeline. This new shading pipeline opens up creative possibilities with quality and interoperability improvements.

- A new shader node allows you to selectively apply shading per object in Action, with different shading algorithms, such as Cook-Torrance, Fresnel, and Car Paint.
- New texture map nodes are introduced to add realistic 3D details to your surfaces and geometries: Parallax, Specular, and Emissive maps.
- The Texture node is now split into two map nodes with more user settings:
  - **Diffuse Map** A new mapping type is added: Perspective, allowing for a transformation based on a camera's field of view (FOV).
  - **Reflection Map** Allows you to simulate a mirrored surface. The Reflection Map results can be combined to the Diffuse Map results.
- Surface and map menus now allow you to select advanced filtering options, such as anisotropic or EWA (Elliptical Weighted Average) filtering.

See Action: Textures on page 15

Substance Procedural Texture Tools

You can use the new Substance Texture node in Action to quickly create photorealistic procedural textures using a library of organic, fabric, and material presets. Substance Textures take full advantage of the new pixel shading pipeline to open up your creative compositing possibilities. Over 130 presets are available to choose from.

See Substance Texture Presets on page 15.

Stereoscopic 3D Tools

This release introduces new tools and workflows for 3D stereoscopic projects.

Using the Inferno stereoscopic workflow, you can edit, conform, visualize, and composite stereo material.

You can edit stereo material directly on the Desktop and by accessing a clip's stereo tracks. To do any stereo compositing work, you must go into Batch or Action.

See Stereoscopic Workflow on page 61 and Action: Stereo on page 99

The new FBX Camera is a full-featured animatable camera in Action that allows you to build 3D compositing scenes, mixing 3D objects and stereo objects.

See Action: FBX Camera on page 109.

3D Compositing

In Action, changes for this release to support shading and stereoscopic workflows have also opened up the architecture to allow the following feature enhancements.

Action Multiple Outputs

A new Output menu in Action allows you to organize, fine-tune, and process multiple outputs from the same scene. For example, you can output different matte passes from the same scene, as well as special passes such as the Z-Depth and Normal.

See Processing Multiple Outputs on page 131.

Action Light Improvements

The new shader architecture allows for better light integration — you can now have up to 64 functioning lights in your Action scene.

You can also use the new light bevel curve to add a specific light profile, and create unique lighting effects. See Light Menu Settings on page 136.

Axis Look-At Connections

In Action, you can create interesting effects by attaching the new look-at connection between an axis and another object in your scene. The axis then rotates to face the look-at object, no matter where it is positioned.

See Applying an Axis Look-at Connection on page 139.

Surface Blending Curves

With the new Blending tab in the Action Surface menu, you can adjust the blending curves of each surface separately. The Action blending curve is similar to the Keyer luminance blending curve, but you can adjust it per surface in Action, and see the results instantly.

See Applying Blending Curves per Surface on page 140.

Sorting Surfaces

A new Sort button is present for bilinear and bicubic surfaces in Action, allowing you to sort non-flat surfaces in the particular instances when overlapping transparent regions are causing artefacts.

Action Node Bin Redesign

The Action node bin is redesigned to simplify your workflow. Nodes are now displayed in alphabetical order, and divided into groups classified by tabs (Objects, Surfaces, and Maps).

Hardware Anti-Aliasing

In the setup menu in Text, Batch, Gmask, and Action, use the new hardware anti-aliasing option to accelerate edge anti-aliasing with no performance penalty. When you use hardware anti-aliasing, 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 also combine the hardware anti-aliasing option with the normal software anti-aliasing controls in Text and Action to obtain the desired level of image quality.

Clip Library

This release introduces a simplified clip import process, which gives you, among other things, RED® import directly within the clip library.

This release also introduces a new preview panel, as well as reorganized clip library functions.

New Import Codec

You can now import clips encoded with H.264 in Quicktime.

Gateway Library Import

This new feature simplifies the import process. Import clips and sequences from any connected volume using a new type of library, the Gateway library. See Accessing Gateway Libraries on page 175.

From a Gateway library, you can import almost any media by simple drag and drop. You can:

- Drag and drop to import clips and timelines; just drag a clip from any location and drop it in a local clip library.
- Define rules for each clip and timeline which note how the media is to be imported or relinked. You can manage your rules at a project or user level.
- Import RED® and Multi-Channel OpenEXR directly to the clip library.

- Use import history to update a previously imported clip.
- Leverage the new features in version 3.0 of the R3D SDK version 3.0, such as the new ISO / FLUT / Shadow / Color Space and Gamma Curve.
- Leverage a RED ROCKET™ card installed on a Mac Wiretap Gateway system to improve the speed of decoding and debayering R3D files.

See Importing Files Using A Gateway Library on page 145.

Preview Panel

Use the new preview panel in the clip library to view clips. It is essentially a mini Player. See Playing Clips from the Library on page 180. The preview panel also displays information about the clip, including any clip notes added to the clip.

The preview panel is especially useful when used in conjunction with the new Gateway library feature; it shows how a remote clip will look once imported with the selected import options. Use the preview panel to set in and out markers for clips to import.

Reorganized Library Functions

In the clip library, the library functions have been reorganized for easier access:

- The Resize menu contains all the Resize options.
- The I/O menu contains the buttons to access Input/Output clip, Import/Export image.
- A new Background Tasks button gives you direct access to the Backburner queue menu.
- The Tools, Archive, and Search buttons no longer open an independent menu, but rather a flatter structure which simplifies the navigation across the different items.

NOTE The Stonifise tool is no longer available. It has been replaced by the Store tool. See Managing Media Using the Store Tool on page 181.

Batch

Refer to the following sections for specific Batch enhancements in this release:

- Clip nodes now support dual output. See Clips on page 183.
- You can now keep nodes in sync so that changes made to one node are automatically duplicated to the other node. See Duplicating Batch Nodes on page 183.
- The interface item for enabling bypass mode has changed slightly. See Bypassing Node Inputs on page 188.
- A new Select Output box has been added to the Master Keyer, Modular Keyer, and Keyer nodes, allowing you to select an output to process. See Master Keyer Node on page 191.
- Action nodes now support multiple outputs. You can expand an Action node to see all available outputs as well as display a proxy of a specific output on the expanded node.
  You can now also set a context on an output in an Action node.
  See Output Tabs on page 191 and Expanding an Action Node on page 191.
- If a node in a group contains multiple outputs, you can now press **Ctrl+Shift+up arrow** or **Ctrl+Shift+down arrow** as you navigate the Group List to display a proxy of all the node's outputs. As well, when you select a node in the group, all associated views are displayed in the View box.
- The group icon is now dark blue.
- You can now set a context on an output in a group node.
  See Grouping Nodes on page 196.
- You can now generate an alpha from a clip inside a BFX level as well as explicitly generate a black or white alpha from a clip. See Generating an Alpha from a Clip Node on page 199.
- The Batch node bins have been improved with more functionality.
  The following nodes have been added to the Comp/FX bin:
  - 3D Blur
  - Distort
  - Substance Noise
  - Substance Splatter

The following nodes have been added to the Format bin:

- Stereo Anaglyph
- Stereo Interlace

Clips

Workflow improvements when working with clips include the following:

- You can now select a processing preference when processing from modules. See Default Processing Options on page 203.
- Clip information on Desktop proxies reflecting which track and layer the focus is on has changed slightly. See Navigating Edit Sequences on page 203.
- New hotkeys now allow you to delete a clip, reel, or an entire Desktop. To delete a clip, place the cursor over the clip on the Desktop and press **D**. To delete all clips on a reel, place the cursor over an empty part of the reel and press **D**. To delete the entire Desktop, place the cursor over an empty part of the Desktop below the reels and press **D**.

Audio

Improved support of audio media in this release includes the following enhancements:

- You can now soft-import clips with audio sampled at a rate different than 48 kHz.
- Desktop now fully supports stereo audio tracks. You can import, export, and edit stereo tracks. See Editing Stereo Audio Tracks on page 205.
- The Tone and Colour Source tools have been updated to generate stereo audio tracks.

Timeline

To make working with the timeline easier, the timeline interface has been updated with new icons and functionality. Refer to the following topics for information on the specific enhancements:

- Timeline Interface on page 211
- Adding Tracks on page 212
- Selecting Tracks on page 213
- Locking Tracks on page 215
- Naming Elements on page 216
- Adding Comments to Elements on page 216
- Deleting Elements from the Timeline on page 217
- Patching on the Timeline on page 218

NOTE The Patch Panel option is no longer available from the Menu Priority box.

- Creating Layers on page 219
- Deleting Layers on page 220
- Muting Layers on page 221
- Naming Layers on page 222

Viewing

Changes in how you view clips make it easier for you to work in Inferno:

- The viewing options in the View box have been reorganized to make it easier to scan from the options available for your viewport selection. Note that if a hotkey exists for a view, it is displayed beside the option in the View box. See Displaying Clip Views on page 225 and Multiple Viewport Categories on page 226.
- When displaying views in multiple viewports using hotkeys, you no longer have to select a viewport first. You need only place the cursor over the applicable viewport and press the hotkey.

- You can now use the broadcast monitor as another viewport. See Working with Viewports and a Broadcast Monitor on page 228.
- You can now use the **Ctrl+spacebar+up arrow** and **Ctrl+spacebar+down arrow** to zoom in and out of the image window in modules. (Press **spacebar** and drag to pan the image window).
- You can now compare images from two video tracks simultaneously by displaying both the Primary and Secondary tracks in the Player. See Comparing Tracks and Layers on page 230.
- The Compact FX option has been removed from the library and from the timeline's Edit Mode box.
- A new Image Data Type setting in the Preferences menu allows you to set the default image data type for all modules.
  There is also a local Image Data Type setting now available in the View menu of the Sparks editor (including the Desktop Sparks editor, the Batch Sparks node editor, and the Soft FX Sparks editor) allowing for greater control when dealing with various image types, including 16-bit floating point support.

  For information on the image data settings, see Image Data Type on page 232.

Wiretap Gateway

The following important improvements have been made to Wiretap Gateway in this release.

R3D SDK 3.0 Support

Wiretap Gateway leverages the new R3D SDK version 3.0, and enables the following new options for transcoding jobs.

- New ISO / FLUT / Shadow / Color Space and Gamma Curve.
- A new option to choose the R3D SDK version used when importing REDCODE RAW media (SDK 2.x or 3.x)

Support for the RED ROCKET Card

Wiretap Gateway can now use a RED ROCKET™ card installed on a Mac system to improve the speed of decoding and debayering R3D files.

The RED ROCKET card can be installed locally on a Smoke for Mac OS X workstation, or on a dedicated 32-bit Mac OS X Wiretap Gateway system connected to Visual Effects, Finishing and Grading workstations over a 10 GigE network.

Wiretap Gateway systems equipped with a RED ROCKET card are identified as such in the Network panel of Visual Effects and Finishing Applications.

NOTE If you have multiple creative applications on a Mac equipped with a RED ROCKET card, only one application will be able to use the card at a time.

Action: Textures

3

About Shading and Textures

New for this release: an all-new shading architecture in Action, coupled with new texture mapping nodes and Substance Texture presets, allow you to save time and deliver high-quality results.

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.

Substance Texture Presets

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 Substance Texture node, as well as other texture maps and shaders, as needed.

To add a Substance Texture preset:

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 Maps tab of 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: *usr/discreet/<product home>/substance/presets/TEXTURE*

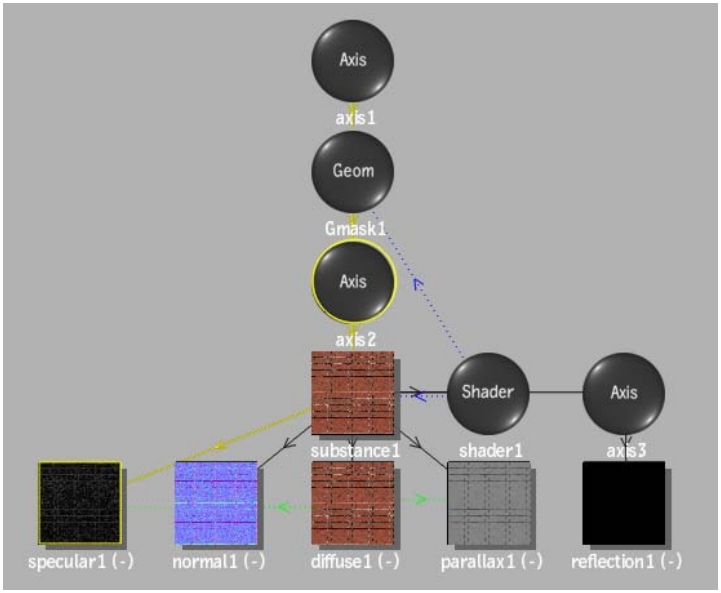
TIP Switch to Proxies view to see a visual representation of the presets.

3 Navigate through the subfolders and select the Substance Texture preset you want to load.

The preset, 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, as 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, and is the parent of some or all of the following map nodes:
  - Specular node (see Specular Mapping on page 39)
  - Normal node (see Normal Mapping on page 34)
  - Diffuse node (see Diffuse Mapping on page 47)
  - Parallax node (see Parallax Mapping on page 30)
  - Emissive node (see Emissive Mapping on page 43)

  Media is automatically applied to these mapping nodes, and can not 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 on page 52.
- 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

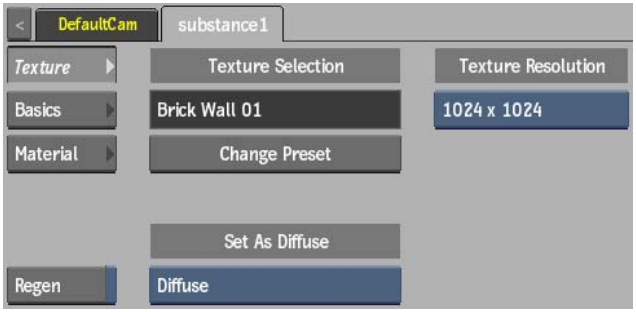
the originally selected surface or geometry. See Using the Shader Node on page 20.

- All Map parameters are loaded with default values based on the loaded Preset. 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.
- 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.
- You can not parent an object from any of the Substance objects. You can, however, parent an object to any of the Substance objects. For example, you can add a different Diffuse map and connect it to one or more maps. In this case, you may choose to hide the original Diffuse map, so that it is not applied to the overall Substance effect.
- If you select a surface or geometry before adding the Substance Texture preset, the connection to the axis 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.

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.



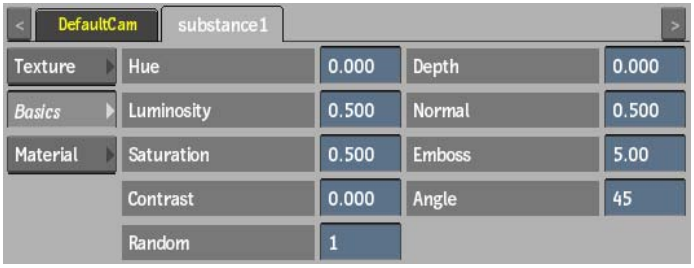
Preset Name field This locked field displays the name of the current preset.

Change Preset button Click to open the file browser to select a different preset.

Texture Resolution box Select the resolution of the pattern.

Set As Diffuse 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 preset (such as the Normal map), that might be invisible, to be seen explicitly. When used in conjunction with a source node, the newly explicit media can then be colour corrected, for example.

The Basics tab contains settings common to all Substance presets.



Hue field Displays the colour range of the texture.

Luminosity field Displays the brightness level of the texture.

Saturation field Displays the level of colour purity of the texture.

Contrast field Displays the gradations between the light and dark areas of the texture.

Random field Displays the random seed value of the generated texture.

Depth field Displays the attenuation of depth of the texture.

Normal field Displays the attenuation of the normals of the texture.

Emboss field Displays the level of enhanced details of the texture.

Angle field Displays the angle of diffuse in relation to the level of Emboss applied to the texture.

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.

NOTE A Regen button is available from all three Substance menu tabs. Enable Regen to dynamically refresh the image as changes are made to the settings. If you notice a slowdown in interactivity, disable Regen.

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.

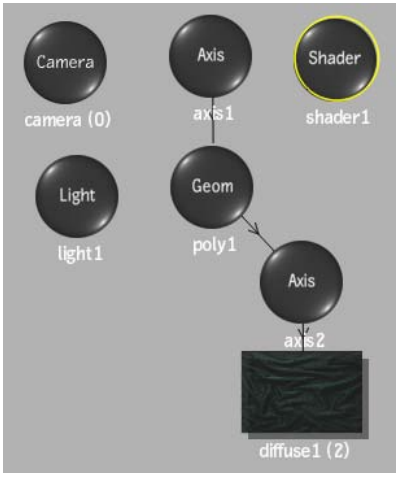
NOTE The shader node is not a typical Action object — its appearance in the schematic represents the ability to selectively include or exclude shading from objects in the scene.

To add a shader to the scene:

1 Do one of the following:

- Drag the Shader node from the Objects tab of the node bin and place it in the schematic.
- Drag the Shader node from the Objects tab of 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.

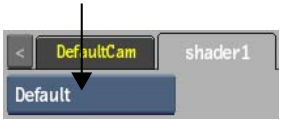
A shader is added to the scene. If you select a surface or 3D geometry in the scene before adding the Shader node, a shader inclusion link is automatically applied. You can also apply inclusion and exclusion links manually. See Applying a Selective Shader on page 24.



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.

3 From the Shader Type box, select a shading algorithm, or turn shading off.



See Shader Types on page 21 for more information on the shader algorithms, and their settings.

Shader Types

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.

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.

Y Roughness field Displays the shape of specularity of the shader along the Y axis.

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.

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.

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).

Red Mid Colour field Displays the red mid colour value.

Green Mid Colour field Displays the green mid colour value.

Blue Mid Colour field Displays the blue mid colour value.

Mid colour pot Diplays the mid colour.

Red Edge Colour field Displays the red edge colour value.

Green Edge Colour field Displays the green edge colour value.

Blue Edge Colour field Displays the blue edge colour value.

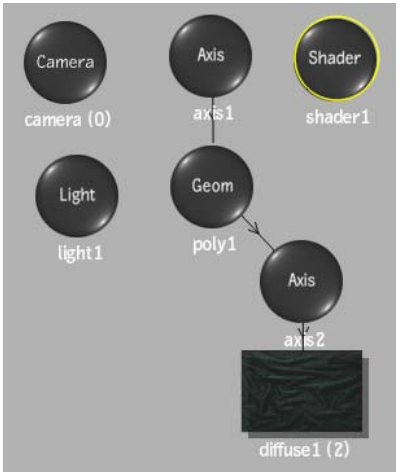
Edge colour pot Displays the edge colour.

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:

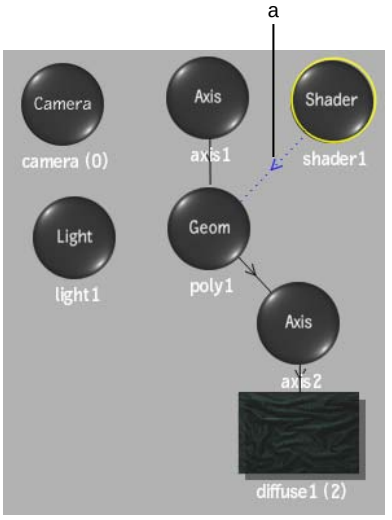
1. Add a shader to the scene.
   All objects are affected.



2 Select Lighting from the Edit Mode box (or press **l**).

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 Lighting mode).

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. When the displacement map is applied to the surface, the pixels of the surface are displaced along the positive or negative X, Y, and/or Z axes. Displacement mapping uses the media's matte clip, so you can turn the matte on or off to get the desired effect.

To add a displacement map:

1 In the schematic, select the surface or 3D geometry to which you want to apply the displacement.

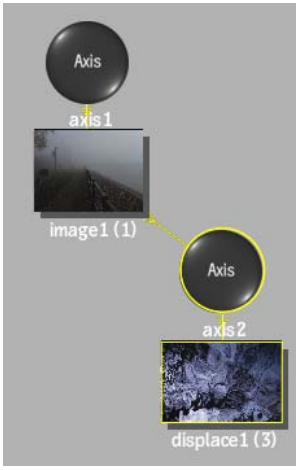
2 Click Media.

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 Maps tab of the node bin and place it in the schematic.
- Drag the Displace Map node from the Maps tab of 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.



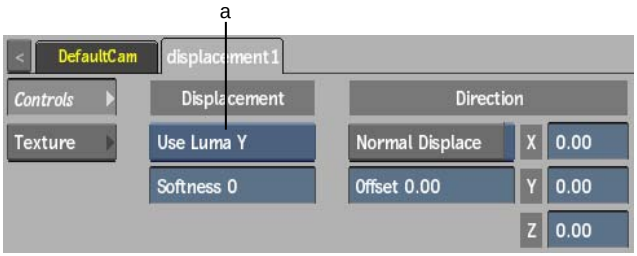
To specify different media as the displacement source, select the media in the Media menu, then click Apply.

5 Double-click the Displace node in the schematic, or follow the tab population rules for the Object menu.

The Displacement menu appears.

Displacement Menu Settings

The Displacement menu is divided into two tabbed sections: Controls and Texture.



(a) Channel box

Channel box Select a colour channel to calculate the displacement map.

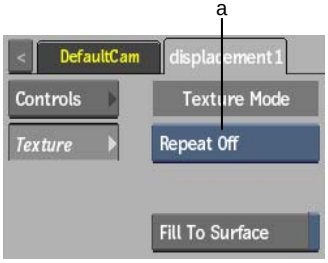
Softness field Displays the level of rounding off, or softening of the spikes that result from colour values in the image that vary from pixel to pixel in the displacement map.

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.

Normal Displace button Enable to displace bilinear and bicubic surfaces according to their normals. For flat surfaces, disable to displace in the X, Y, and Z directions.

Offset field Displays the offset to the displacement of X and Y.

Displacement axes Specifies the amount of displacement in pixel units along the X, Y, and Z axes. Use positive values for displacement on the positive axis, and negative values for displacement on the negative axis.



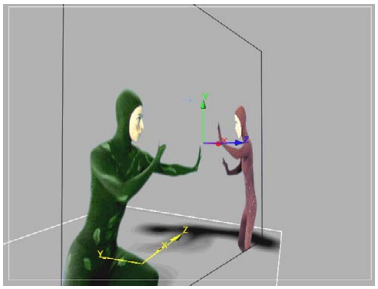
(a) Repeat Mode box

Repeat mode box Select how the displacement map pattern is repeated on the surface.

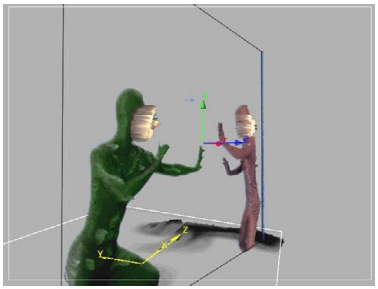
Fill to Surface button Enable to resize the displacement map to the resolution of its parent surface.

NOTE If the Displace node is attached to a 3D Geometry or 3D Text node, you must select a UV Mapping mode other than None in the Geometry menu for the displace pattern to have an effect on the geometry.

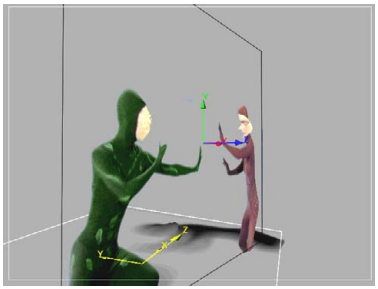
The following figures illustrate a possible use of displacement mapping and shows the difference between displacing with and without softness.



Original image



Z-axis displacement (60) using the luminance channel (Y) and a softness of 0



Z displacement (60) using the luminance channel (Y) and a softness of 12

Parallax Mapping

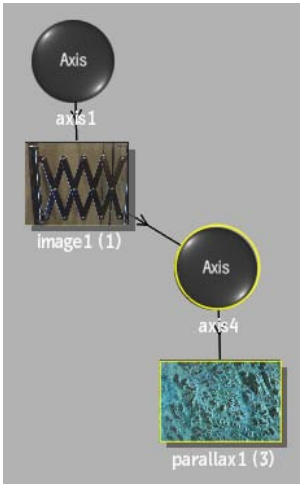
Parallax mapping can generate a visual result similar to that of a displacement map without actually displacing polygons. Parallax mapping uses the height map (similar to a displacement map) to determine how the object texture should be warped in order to simulate shading and occlusion effects, without 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 Maps tab of the node bin and place it in the schematic.
   - Drag the Parallax Map node from the Maps tab of 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.



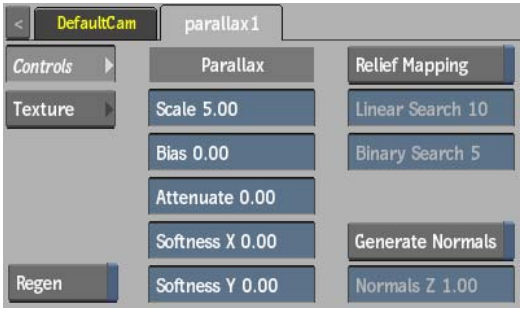
To specify different media as the parallax source, select the media in the Media menu, then click Apply.

5 Double-click the Parallax node in the schematic, or follow the tab population rules for the Object menu.

The Parallax menu appears.

Parallax Menu Settings

The Parallax menu is divided into two tabbed sections: Controls and Texture.



Scale field Displays the perceptual height of the texture driven by the parallax map.

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.

Softness X field Displays the amount of X-axis blur applied on the parallax map.

Softness Y field Displays the amount of Y-axis blur applied on the parallax map.

Relief Mapping button Enable to augment the 3D surface detail by creating occlusions where changes occur in the parallax map. Rendering is slower when Relief Mapping is 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.

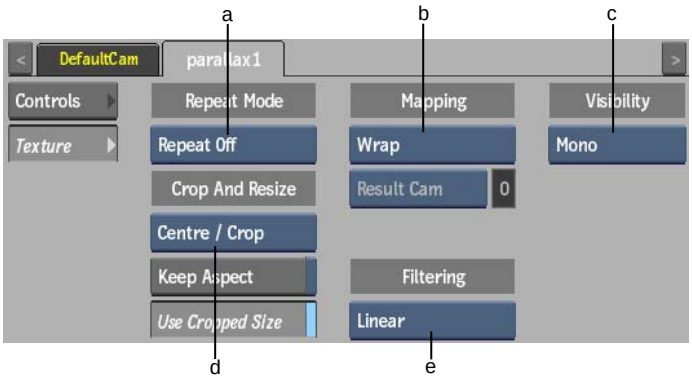
Binary Search field Displays the number of steps to refine the precision of the best depth found by the linear search.

Generate Normals button Enable to allow the parallax map to create a normal map for enhanced lighting effects. You should enable if no normal map exists for the parallax texture.

NOTE Enabling Generate Normals also affects the Normal output in Action.

Normals Z field Displays the amount of Z scale applied to the generated normals. Higher values result in attenuation of the normals.

Regen button Enable to dynamically refresh the image as changes are made to the parallax settings.



(a) Repeat Mode box (b) Mapping box (c) Fit Method box (d) Crop 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 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.

Perspective Camera box Specify which camera's FOV to take into account when using perspective mapping.

Perspective Camera field Displays the active perspective camera number.

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, as well as EWA filtering functionality in the Action Setup menu.

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.

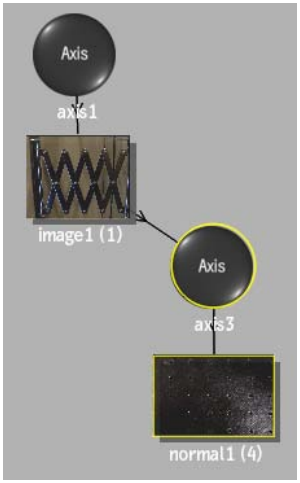
Normal Mapping

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 Maps tab of the node bin and place it in the schematic.
   - Drag the Normal Map node from the Maps tab of 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.

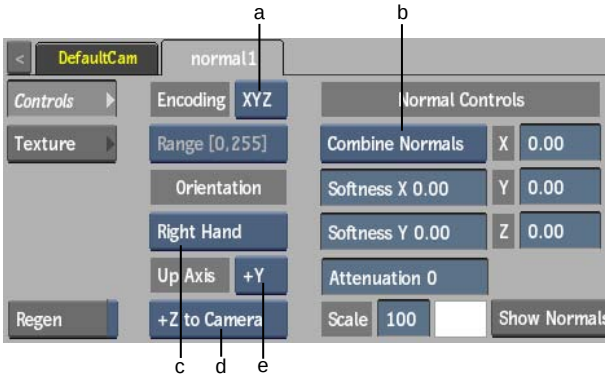


   To specify different media as the normal source, select the media in the Media menu, then click Apply.
5. Double-click the Normal node in the schematic, or follow the tab population rules for the Object menu.

   The Normal menu appears.

Normal Menu Settings

The Normal menu is divided into two tabbed sections: Controls and Texture.



(a) Encoding box (b) Normals box (c) Orientation box (d) Camera box (e) Up Axis box

Encoding box Select the encoding order of the normal map, 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 option is greyed out.

Orientation box Select whether the orientation of the coordinate system of the normal map is Left Hand or Right Hand.

Up Axis box Select which axis is the up axis of the normal map.

Camera box Select which axis of the normal map corresponds to the Z axis in Action. The selection in the Up Axis box determines the available selections in the Camera box.

Normals box Select how the normal interacts with the surface.

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 softness applied to the normal map texture.

Softness Y field Displays the amount of Y-axis softness applied to the normal map texture.

Attenuation field Displays the level of amplitude of the effect caused by the normal map texture.

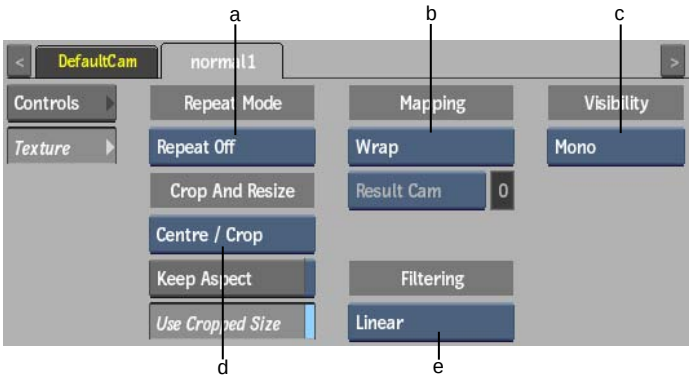
Normal axes Specifies the amount of offset in pixel units along the X, Y, and Z axes.

Show Normals button Enable to display normal vectors over the surface.

Scale field Displays the scale of the normal vectors.

Normals colour pot Select a colour for the display of normal vectors.

Regen button Enable to dynamically refresh the image as changes are made to the normal settings.



(a) Repeat Mode box (b) Mapping box (c) Fit Method box (d) Crop 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 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.

Perspective Camera box Specify which camera's FOV to take into account when using perspective mapping.

Perspective Camera field Displays the active perspective camera number.

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, as well as EWA filtering functionality in the Action Setup menu.

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 or 3D Text node, you must select a UV Mapping mode other than None in the Geometry menu for the normal pattern to have an effect on the geometry.

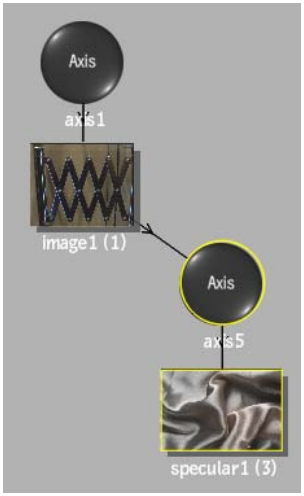
Specular Mapping

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 Maps tab of the node bin and place it in the schematic.
   - Drag the Specular Map node from the Maps tab of 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.



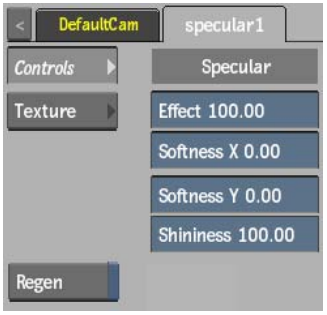
To specify different media as the specular source, select the media in the Media menu, then click Apply.

5 Double-click the Specular node in the schematic, or follow the tab population rules for the Object menu.

The Specular menu appears.

Specular Menu Settings

The Specular menu is divided into two tabbed sections: Controls and Texture.



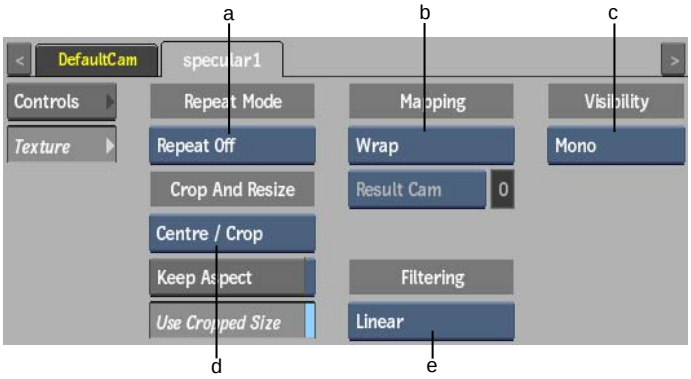
Effect field Displays the amount of specular colour.

Softness X field Displays the amount of X-axis blur applied on the specular map.

Softness Y field Displays the amount of Y-axis blur applied on the specular map.

Shininess field Displays the amount of shininess in the specular map.

Regen button Enable to dynamically refresh the image as changes are made to the specular settings.



(a) Repeat Mode box (b) Mapping box (c) Fit Method box (d) Crop 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 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.

Perspective Camera box Specify which camera's FOV to take into account when using perspective mapping.

Perspective Camera field Displays the active perspective camera number.

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).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

TIP You can set the default filtering type, as well as EWA filtering functionality in the Action Setup menu.

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.

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.

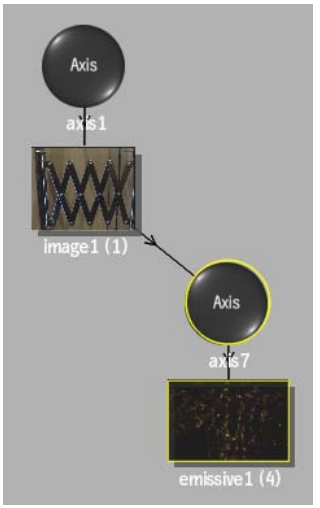
Emissive Mapping

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 Maps tab of the node bin and place it in the schematic.
   - Drag the Emissive Map node from the Maps tab of 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.



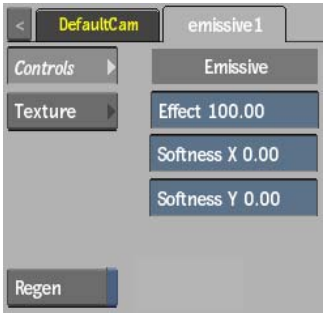
To specify different media as the emissive source, select the media in the Media menu, then click Apply.

5 Double-click the Emissive node in the schematic, or follow the tab population rules for the Object menu.

The Emissive menu appears.

Emissive Menu Settings

The Emissive menu is divided into two tabbed sections: Controls and Texture.

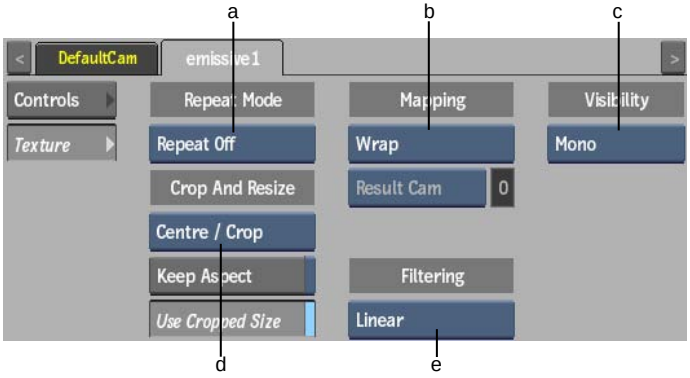


Effect field Displays the amount of emissive lighting.

Softness X field Displays the amount of X-axis blur applied on the emissive map.

Softness Y field Displays the amount of Y-axis blur applied on the emissive map.

Regen button Enable to dynamically refresh the image as changes are made to the emissive settings.



(a) Repeat Mode box (b) Mapping box (c) Fit Method box (d) Crop 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 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.

Perspective Camera box Specify which camera's FOV to take into account when using perspective mapping.

Perspective Camera field Displays the active perspective camera number.

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, as well as EWA filtering functionality in the Action Setup menu.

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.

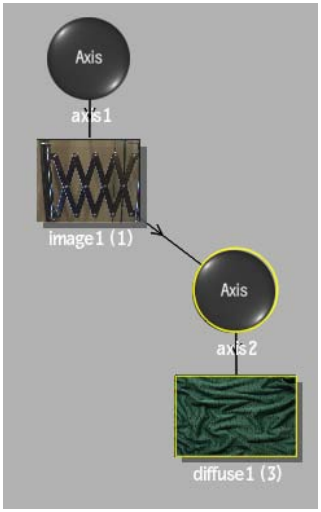
When adding a diffuse map to a shaded surface, the diffuse is used when generating the shadow. The diffuse is only used to apply the colour to the shadow, so effects such as surface displacement still reference the media associated with the surface. If a node has an applied diffuse map, it is the matte 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.

   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 Maps tab of the node bin and place it in the schematic.
   - Drag the Diffuse Map node from the Maps tab of 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 diffuse source, select the media in the Media menu, then click Apply.

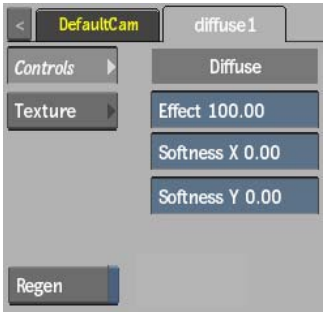
5 In the Rendering section of the Action Setup menu, enable or disable Shading depending on the method of diffuse mapping you are using. When Shading is enabled, normals are used. You must enable Shading when using Reflection mapping because it also uses normals.

6 Double-click the Diffuse node in the schematic, or follow the tab population rules for the Object menu.

The Diffuse menu appears.

Diffuse Menu Settings

The Diffuse menu is divided into two tabbed sections: Controls and Texture.

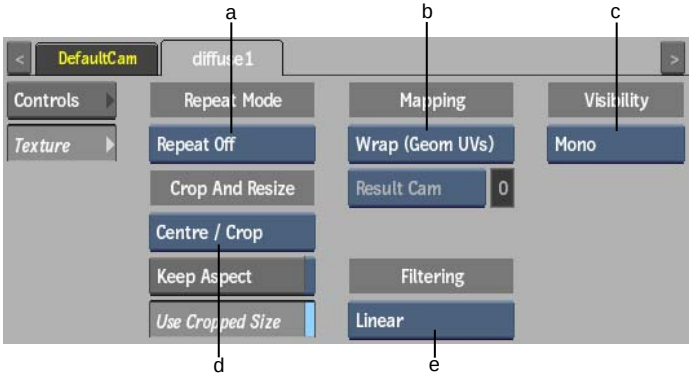


Effect field Displays the amount of diffuse colour.

Softness X field Displays the amount of X-axis blur applied to the diffuse map.

Softness Y field Displays the amount of Y-axis blur applied to the diffuse map.

Regen button Enable to dynamically refresh the image as changes are made to the diffuse settings.



(a) Repeat Mode box (b) Mapping box (c) Fit Method box (d) Crop box (e) Filter box

Repeat Mode box Select how the diffuse map pattern is repeated on the surface.

Fit Method box Select a fit method option to be applied to the diffuse map.

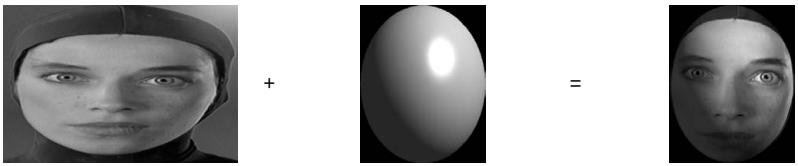
Keep Aspect button Enable to preserve the aspect ratio of non-square pixels (not available for the Fill fit method).

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.

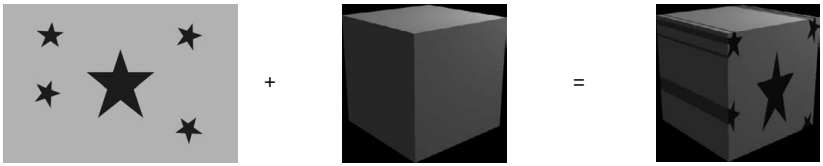
Mapping box Select the type of texture mapping. You can use the following mapping types.

Cylindrical Cylindrical mapping wraps the diffuse map around the 3D model in a cylindrical manner, starting from the lower-left corner of the diffuse map. Use cylindrical mapping for models of cans, bottles, or other circular and semicircular bodies. Cylindrical mapping is oriented counterclockwise—left-to-right—when viewed from the top of the model.

Wrap 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.



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.

Perspective Camera box Specify which camera's FOV to take into account when using perspective mapping.

Perspective Camera field Displays the active perspective camera number.

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.
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, as well as EWA filtering functionality in the Action Setup menu.

Camera Type box Select the camera type visibility for the diffuse 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.

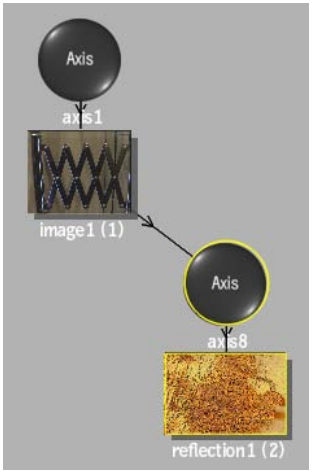
Reflection Mapping

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 Maps tab of the node bin and place it in the schematic.
   - Drag the Reflection Map node from the Maps tab of 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.



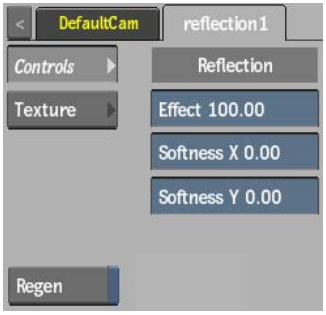
To specify different media as the reflection source, select the media in the Media menu, then click Apply.

5 Double-click the Reflection node in the schematic, or follow the tab population rules for the Object menu.

The Reflection menu appears.

Reflection Menu Settings

The Reflection menu is divided into two tabbed sections: Controls and Texture.

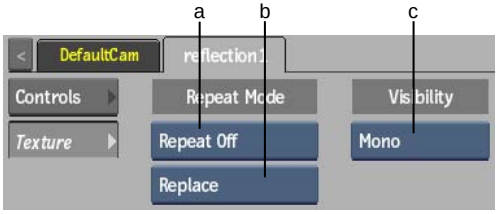


Effect field Displays the amount of reflection.

Softness X field Displays the amount of X-axis blur applied to the reflection map.

Softness Y field Displays the amount of Y-axis blur applied to the reflection map.

Regen button Enable to dynamically refresh the image as changes are made to the reflection settings.



(a) Repeat Mode box (b) Add/Replace box (c) Camera Type box

Repeat Mode box Select how the reflection map pattern is repeated on the surface.

Add/Replace box Select whether to add or replace the reflection map to the diffuse colour.

Camera Type box Select the camera type visibility for the reflection 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.

The texture projection properties are located in the Projector menu.

The Projector controls are described as follows.

MBlur Enables or disables a motion blur only for the projection.

Enable button Activates or deactivates the projected texture effect in the scene. When this button is disabled, the Projector node remains in the schematic and the projector icon appears smaller in the image area.

Position X, Y, and Z fields Position the projector in 3D space. You can also set the texture projector position by dragging the projector in the scene.

Rotation X, Y, and Z fields Rotate the projector in 3D space.

Motion Path button Enables motion path animation for the projector.

Projection Blend Mode box Displays the projected texture effect. See Blending Projections on page 57.

Transparency field Modifies the transparency of the projected texture effect. Transparency values are 0 to 100.

FOV field Adjusts the projector's field of view. FOV values are 1.00 to 180.00.

Aspect field Specifies the ratio of height to width for the projected image. Aspect values are 0.10 to 100.00.

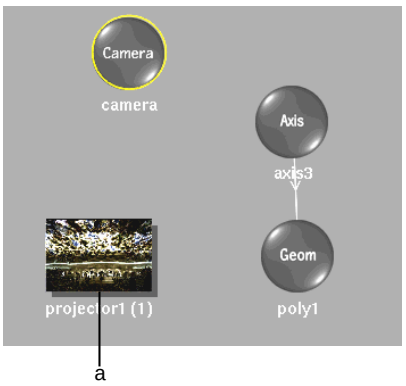
Order field Sets the order of projectors.

RGB fields and colour pot Specify RGB colour values. You can also pick a colour with the colour picker.

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.
3. Do one of the following:
   - Drag the Projector node from the Objects tab of the node bin and place it in the schematic.
   - Drag the Projector node from the Objects tab of 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.



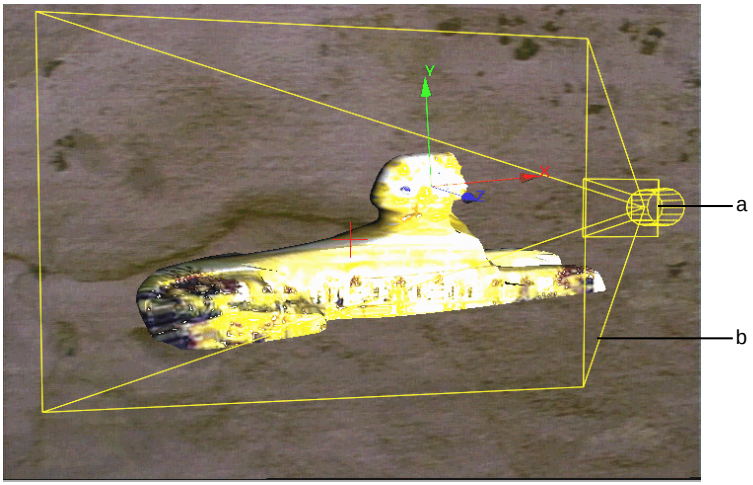
(a) Projector node

4. Double-click the Projector node to display the Projector menu, or follow the tab population rules for the Object menu.

The Projector is selected.

5 Switch to Camera view to manipulate the projector in the scene using the View box.

The following example illustrates projected media on a 3D model using Spotlight projection blend mode. See Blending Projections on page 57.



(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.

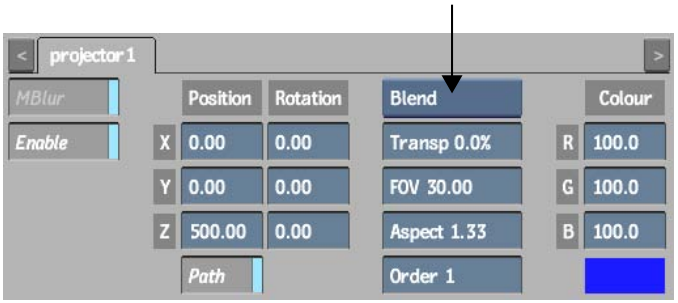
Multiple Projector nodes appear in the schematic and projector icons appear in the scene.

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.

Blending Projections

Use projection blend modes to modify how you want to blend the projected image in the scene. Select a mode from the Projection Blend Mode box.



Add Compensates for the soft or anti-aliased edge on an object in a front and matte clip media.

Blend Punches the matte through the front. This blends the edge of the front clip and adds additional softness to the media. This is the default setting. Use Blend mode when you do not want any shading cast on the geometries that are receiving the projection.

Max Compares the RGB channels of each pixel of the front clip and the back clip individually and returns the larger of the two values.

Min Compares the RGB channels of each pixel of the front clip and the back clip individually and returns the smaller of the two values.

Multiply Multiplies the RGB channel values of corresponding pixels of the front clip and the matte clip, and normalizes the result by dividing by 256 in 8-bit mode, or 4096 in 12-bit mode. The resulting RGB channel values are assigned to the corresponding pixels in the generated clip.

Hardware limitations may prevent transparency from working properly with Multiply.

Negate Creates a negative-like result. Try Negate with a soft white matte with a white front clip. This mode does not use the matte or transparency in its blend.

Screen Multiplies the inverse of the matte clip's colours with the colours of the front clip. The resulting colour is always lighter. Screen does not work with transparency.

When using a matte, set the diffuse r, g, b channels to 0, 0, 0 in the Channel Editor for the image surface you are blending.

Simple Add Punches the matte through the front using more softness than Blend mode. Simple Add mode is similar to Add but includes transparency.

Spotlight Creates a slide projector effect. This mode does not use the matte or transparency and decreases system performance.

Spotlight Blend Creates an effect where black areas of the front disappear. This mode does not use the matte or transparency; it works well with a clip on which an object is surrounded by black.

Subtract Subtracts the RGB channel values of the pixels in the matte clip from the RGB channel values of the pixels in the front clip and assigns the result to the RGB channel values of the pixel in the result.

If an RGB channel value from the matte clip is larger than the corresponding channel value in the front clip, yielding a negative result, that result is clamped at 0 (black).

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:

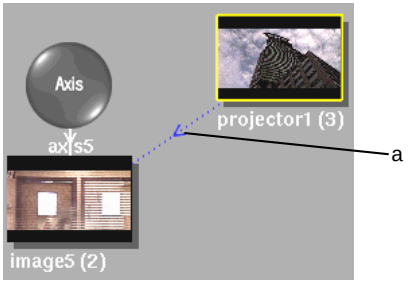
1. Select the media to use as a texture for projection.
2. Add a projector using the Node bin.

   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 Lighting from the Edit Mode box to create lighting connections.

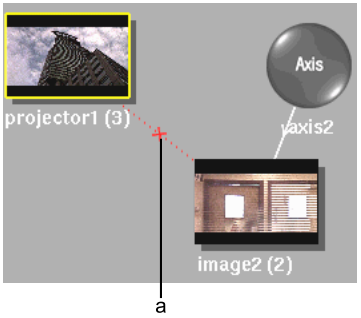
5 Do one of the following:

- 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

Stereoscopic Workflow 4

Stereoscopy in Inferno

Using the Inferno stereoscopic workflow, you can edit, conform, visualize, and composite stereo material. Clips that you use in the Inferno stereoscopic workflow can contain stereo tracks or both stereo tracks and monoscopic tracks.

A stereo track contains one left eye layer and one right eye layer. Editorial rules specific to stereo tracks help you treat the left and right eye layers as one entity. For example, the layers of a stereo track must have the same resolution and bit depth. Also, stereo tracks do not support vertical editing.

You can edit a clip containing a stereo track on the Desktop or directly through the track itself. To do any stereo compositing, you must go into Batch or Action.

This chapter provides an overview of working with stereo material in Inferno with cross-references to other sections for detailed stereo information.

Assembling and Preparing Stereo Tracks on the Desktop

You can assemble imported images into a clip containing a stereo track directly on the Desktop. Do this, for example, when your left and right eye images are imported as separate clips.

You can also preview stereo material in the Player before bringing it into Action or Batch.

Note the following when working with clips containing stereo tracks on the Desktop:

- If you create a clip containing a stereo track from two mono clips, each mono clip is placed on the corresponding left and right eye layers of the stereo track. The mono clips must share the same metadata.
  The mono clips can contain multiple layers but they must have the same number of layers.

  Before you create a clip containing a stereo track, remember that you must bring stereo tracks into Batch or Action if you want to do any compositing work.

  You can now simultaneously create clips that each contain a stereo track using two source reels: one reel for your left eye sources and one reel for your right eye sources.

  See Creating and Splitting Stereo Tracks on page 67.

- You can perform many of the same Desktop editing operations on clips containing stereo tracks as on mono clips, for example, splicing clips together or replacing frames. Note that if you are working with more than one clip, both clips must contain stereo tracks.
  To identify clips containing stereo tracks on the Desktop, see Clip Information on page 71.

- You can view the left eye and right eye layers of a stereo track on the Desktop in the same way as the layers of a mono clip—with the **up arrow** and **down arrow** keys.
  To identify how stereo layers are represented on clip proxies, see Navigating Edit Sequences on page 71.

- Inside the Player, you can select a mode, as specified for your broadcast monitor preferences, for previewing and broadcasting stereo material.
  See Comparing Tracks and Layers on page 72.

  NOTE To set broadcast monitor preferences, see Broadcast Stereo on page 75.

- If you realize that you need to go into a Desktop module other than Action or Batch, you can split the clip containing the stereo track into two mono clips. You can always recreate the stereo track after you process. You may also need to split a stereo track back into left eye and right eye images in order to export them.
  To split a clip containing a stereo track into two mono clips, see Creating and Splitting Stereo Tracks on page 67.

- You can bring a clip containing a stereo track into the Player timeline to edit the left and right eye layers.
  See Editing Stereo Tracks on page 63.

Editing Stereo Tracks

Depending on how the stereo footage was filmed, you may need to edit the left or right eye image after you create a clip containing a stereo track. For example, you may need to flip one of the images or fix brightness or colour discrepancies between the left and right eye images. You can go directly into the Player timeline from the Desktop to do this cleanup. If you already brought your clip into Batch, you can edit the stereo track from the Batch timeline.

Note the following when editing a stereo track:

- You can edit the media of the individual left and right eye layers but not the metadata. For example, you can slip one layer or replace its media, but you cannot trim only one of the layers.
- Stereo sync is on by default. With stereo sync, changes applied to one layer of a stereo track are automatically applied to the other layer.
- If you are making changes that affect the media of a stereo track such as applying soft effects, you can remove the stereo sync from the layers and apply the changes to only one of the layers.
- If you applied the same soft effects to unsynced layers and the values on each layer are different, you can resynchronize the values so that they are the same on both layers.
- You cannot do vertical editing or vertical compositing between stereo layers.
  For more information on what type of work you can do with stereo tracks and how stereo tracks are represented on the timeline, see Stereo Tracks on page 76.
- If you want to convert a video track into a stereo track, you can do so directly on the timeline as long as the video track contains two layers that share the same metadata.
  Do this, for example, with imported EDLs where the left and right eye images are multi-assembled on separate tracks.

  See Converting a Video Track to a Stereo Track on page 78.

- If you want to convert the layers of a stereo track into two mono video tracks, you can do so directly on the timeline.
  Do this, for example, if you need to change the metadata of one of the layers.

  See Converting a Stereo Track to Video Tracks on page 77.

- If you want to change the focus plane of a shot, you can adjust the convergence with an Axis soft effect.
  See Creating Axis Soft Effects on page 81.

Stereo Compositing in Batch

Bring clips containing stereo tracks into Batch to do compositing and effects work.

Note the following when working in Batch with stereo tracks:

- You load clips containing stereo tracks into Batch the same way as mono clips.
  Inside of Batch, a clip containing a stereo track is represented by a node that has one left eye output and one right eye output, which correspond to the left eye and right eye layers of its stereo track.

  For information on how clips containing stereo tracks are represented in Batch, see Clips on page 183.

- If you brought a clip containing a stereo track into Batch but decide that you need to work with its layers as individual clips, you can convert each layer of the stereo track into a separate mono clip.
  See Creating One Clip Per Stereo Track Layer on page 82.

- If you want to preview your results in a stereo mode inside of Batch, you can activate the mode directly in a viewport.
  See Displaying Stereo Modes in Multiple Viewports on page 83.

- If you want to output one RGB clip in Anaglyph or Interlace mode from a right eye mono clip and a left eye mono clip, use the Stereo Interlace or Stereo Anaglyph node.
  See Stereo Anaglyph Node on page 84 and Stereo Interlace Node on page 85.

- You can automatically set an Action node to stereo by pressing the **Shift+S** hotkey when dragging the Action node to the schematic.
  See Setting Stereo Startup Mode on page 86.

- You can easily apply most effects to the left and right eye layers simultaneously with the Duplicate feature.
  The hotkey for creating a duplicate stereo group node is **Shift+S**.
  See Duplicating Batch Nodes on page 86.

- Bring clips containing a stereo track into a BFX level to apply Batch setups to selected stereo segments. When you process back to the main timeline, the same setup is applied to both the left eye and right eye segments. You can double-click the icon on either segment and you access the same BFX setup.
  See Entering a BFX Level with Stereo Segments on page 91 and BFX Output Node on page 92.

- Use the Stereo settings of the Output node to process your stereo results.
  See Output Node on page 93 and Defining Processing Settings on page 94.

- Stereo displays are now available for Batch context views. See Viewing Nodes in Context on page 97.

Stereo Compositing in Action

Note the following when working with stereo in Action:

- You must handle the left and right eye tracks as separate media.
  See Starting a Stereoscopic Session on page 100.

- Action supports the FBX camera with stereo rigs and mono cameras for compatibility across Autodesk 3D applications. This lets you build 3D compositing scenes using a mix of 3D objects and stereo objects.
  See Action: FBX Camera on page 109.

- The stereo object allows you to composite stereo sources in a 3D compositing environment, allowing you to combine stereo footage with 3D geometries (such as 3D Text, or FBX models) within the same scene. The stereo object is available in the object bin. You can add it to your scene in the usual way, by dragging it to the schematic or scene or by double-clicking the icon.
  See Working with the Stereo Scene on page 101.

- The ability to have multiple outputs in Action lets you output different passes from the same scene, including mattes, normals, and Z-depth passes. You can also output different camera views (top, front, side, result cam)

and modes (stereo, right, left). The Output menu is where you can create, copy, delete, and rename outputs. You can also set the output and render options for each output. The Object list displays objects in the selected output.

See Processing Multiple Outputs on page 131.

Stereoscopic 3D Tools 5

Creating and Splitting Stereo Tracks

You can create one clip containing a stereo track from two mono clips directly on the Desktop. The left eye layer of the stereo track is created from one mono clip, and the right eye layer is created from the other mono clip. The mono clips can contain more than one video layer as long as the number of layers is the same. If the clips contain multiple layers, the layers will be merged when the stereo track is created.

To create a clip containing a stereo track from two mono clips, the mono clips, including all their layers, must meet the following criteria:

- They must be the same length.
- They must have the same number and position of cuts and transitions.
- They must have the same resolution, bit depth, framerate and dominance.

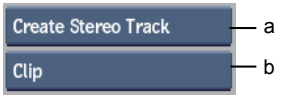
If you have multiple left and right eye mono clips from which you want to create stereo tracks, you can speed up the process and create the stereo tracks at the same time. To do this, you must place all your left eye clips on one reel and all your right eye clips on another reel. The order of the clips must be the same on both reels.

You can also split a stereo track into two mono clips. One mono clip is created from each layer of the stereo track. Split stereo tracks to export them or to bring them into a module other than Batch or Action.

To create a clip containing a stereo track:

1. Click Stereo Track in the Editing menu.

2 Select Create Stereo Track from the Stereo Track box and then select Clip from the Stereo Source box.



(a) Stereo Track box (b) Stereo Source box

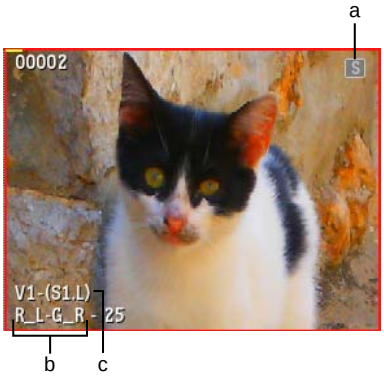
3 Select the clip that will be converted to the left eye layer with the red cursor that appears.

4 Select the clip that will be converted to the right eye layer with the blue cursor that appears.

5 Select a destination reel.

A clip with an S icon indicating that the clip contains a stereo track appears on the Desktop. The clip is renamed according to the names of the left and right eye layers. The left eye layer is the default view.

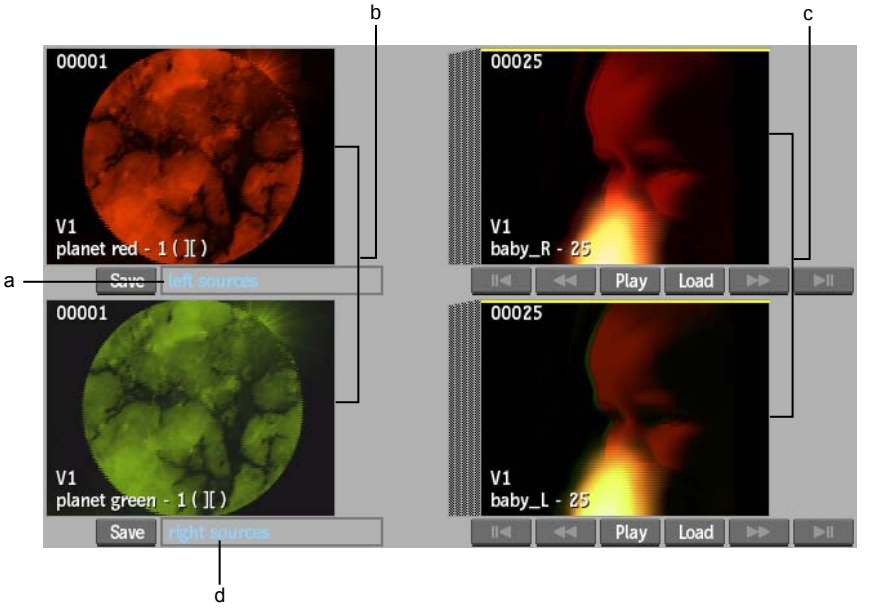
If the mono clips contained multiple layers, the layers are merged.



(a) Stereo icon (b) Renamed clip (c) Stereo track information

To create stereo tracks from multiple left and right eye sources:

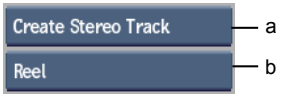
1 Place all your left eye sources are on one reel and all the corresponding right eye sources are on another reel. Make sure that the sources are placed in the same order on each reel.



(a) Reel created for left eye sources (b) Left and right eye "planet" sources placed first on each reel (c) Left and right eye "baby" sources placed second on each reel (d) Reel created for right eye sources

2 Click Stereo Track in the Editing menu.

3 Select Create Stereo Track from the Stereo Track box and then select Reel from the Stereo Source box.



(a) Stereo Track box (b) Stereo Source box

4 Select a clip on a reel containing all the left eye sources with the red cursor that appears.

5 Select a clip on a reel containing all the right eye sources with the blue cursor that appears.

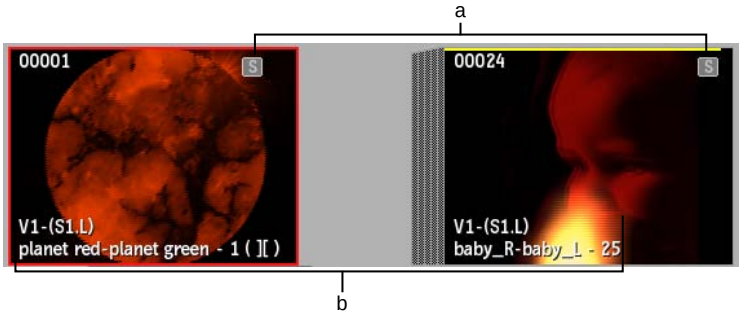
6 Select a destination reel.

A clip with an S icon for each left and right eye image that was merged into one stereo track appears on the Desktop. Each clip is renamed

according to the names of its corresponding left and right eye layers. The left eye layer of each clip is the default view.

If the mono clips contained multiple layers, the layers are merged.

A message appears in the message bar indicating how many stereo tracks were merged and if any failed or skipped.



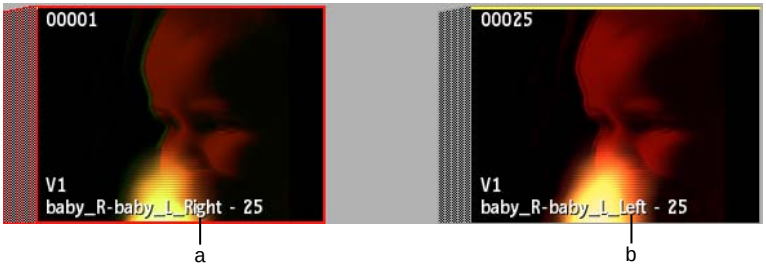
(a) Stereo icons (b) Stereo track information and renamed clips

To split a stereo track into mono clips:

1. Click Stereo in the Editing menu.
2. Select Split Stereo Track from the Stereo Track box that appears.

3. Select the clip containing the stereo track and then select a destination.

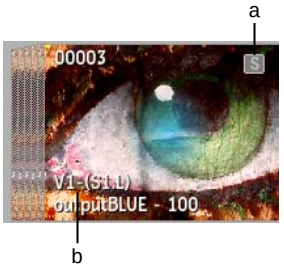
   Each stereo layer is converted into a mono clip. The two mono clips have identical record timecode.



(a) Clip converted from right eye layer (b) Clip converted from left eye layer

Clip Information

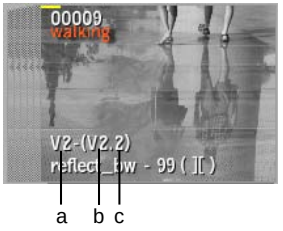
Clips whose primary track is stereo have an “S” icon in the upper-right corner. You can bring stereo clips only into Batch and Action.



(a) Stereo icon (b) Stereo track information

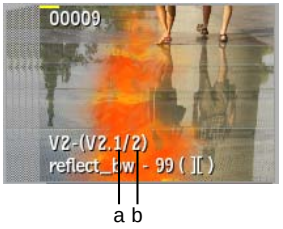
Navigating Edit Sequences

As you navigate between layers and tracks, information on the Desktop clip now updates to reflect which track and layer you are on. In the following example, there are two video tracks. The focus is on the topmost layer—L2—of track V2.



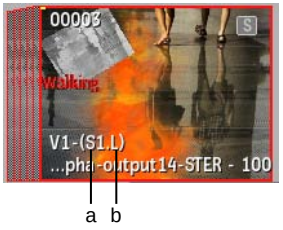
(a) Total number of video tracks (b) Focus is on track V2 (c) Focus of track V2 is on topmost layer, L2

In the next example, the focus is on the next layer down, L1, of the same track. When the focus is not on the topmost layer, the clip information reflects the total number of layers in addition to the focus layer.



(a) Focus layer (b) Total number of layers on track V2

As you navigate the layers of a stereo track, the clip information updates to reflect whether you are on the left eye or right eye layer, as shown in the following examples.



(a) Stereo track (b) Left eye layer

(a) Right eye layer

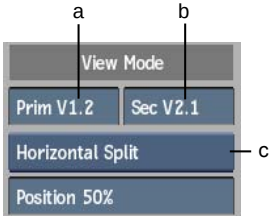
Comparing Tracks and Layers

Use the View Mode options to display images from two video tracks simultaneously. The tracks can be stereo or mono. You can choose to view the two tracks in a split screen. Alternatively, the two tracks can be viewed as a transparency blend or with a clamped or non-clamped difference. The two tracks viewed are the Primary and Secondary tracks.

If you are working with a clip containing a stereo track, you can now select how you want to preview stereo results.

To compare Primary and Secondary tracks in the Player:

1. Bring a clip into the Desktop Player or the Library Player and click the Clip & Setup tab.
2. In the View Mode controls, assign one track as the Primary track and one as the Secondary track in the respective fields. If a track has more than one layer, you can also enter the layer that you want displayed.



(a) Primary Video Track field (b) Secondary Video Track field (c) Preview Setup box

In this example, layer 2 of video track 1 is assigned as the Primary track and layer 1 of video track 2 is assigned as the Secondary track.

3. Select how you want to view the two tracks using the Preview Setup box.

If you selected:	Do this:
Horizontal or Vertical Split	Set the position of the split.
Angle Split	Set the position and angle of the split.
Blend	Set the percentage of the Secondary track to display. For example, if you specify 60%, the image is composed of a blend of 40% of the Primary track and 60% of the Secondary track. Set a value in the Blend field or press **Shift+T** and drag left or right.
Difference	Set the difference between the two tracks. You can set the minimum and maximum threshold values to be clamped or non clamped. To adjust the maximum clamp value, enter a value in the applicable field or press **Shift+U** and drag left or right. To adjust the minimum clamp value, enter a value in the applicable field or press **Shift+Y** and drag left or right.

NOTE In Realtime view, the Secondary video track appears black if it has an unrendered effect, or if both the Primary and Secondary video tracks contain unrendered dissolves.

Example: To preview stereo results in Stereo Anaglyph mode:

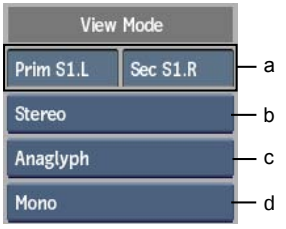
1 Bring a clip containing a stereo track into the Desktop Player or the Library Player and click the Clip & Setup tab.

2 In the View Mode controls, assign one layer (right eye or left eye) as the Primary track and the other layer as the Secondary track in the respective fields.

In the following example, the left eye layer is assigned as the Primary track and the right eye layer is assigned as the Secondary track.

Stereo is automatically selected in the Preview Setup box.

NOTE The options in the Stereo Preview box depend on your hardware configuration. As well, the option displayed is the same as the one selected in the broadcast preferences. If you change the option in one location, it is changed automatically in the other location.



(a) Primary and Secondary Video Track fields (b) Preview Setup box (c) Stereo Preview box (d) Stereo Anaglyph box

3 Select an option from the Stereo Anaglyph box.

Select:	To view:
Diff	The difference between the two images, with the minimum and maximum threshold values non clamped.
Diff Clamped	The difference between the two images, with the minimum and maximum threshold values clamped. To adjust the maximum clamp value,

Select:	To view:
	enter a value in the Max field, or press **Shift+U** and drag left or right. To adjust the minimum clamp value, enter a value in the Min field, or press **Shift+Y** and drag left or right.
Blend	A combined image of the two tracks. To adjust the Blend factor, enter a value in the Blend field, or press **Shift+T** and drag left or right.
Mono	The image with just the anaglyph effect. The RGB values are removed from the display.
Dubois	The image with reduced ghosting between the left and right eyes.

Broadcast Stereo

Use the Broadcast Stereo preferences to set how the broadcast monitor displays stereoscopic images.

NOTE Broadcast Stereo options depend on your hardware configuration. As well, the option displayed is the same as the one selected in the Player. If you change the option in one location, it is changed automatically in the other location.

Broadcast Stereo box Select the stereoscopic display option that best works for you. This option determines how the Player and the broadcast monitor viewport display stereoscopic material.

Select:	If you use:
Anaglyph	A standard broadcast monitor. (Default option)
Interlaced	A broadcast monitor capable of displaying a stereoscopic image as an interlaced image where one field is the left-eye and the other is the right eye.
Dual Output	A broadcast monitor solution which requires a dual-SDI output, such as a dual-projector setup.

NOTE If there is no broadcast monitor connected or if the Broadcast Monitor box is set to Off, the Broadcast Stereo box is set to Stereo Anaglyph.

Stereo Option box If the Broadcast Stereo box is set to Interlaced, select the field to use as the left eye. If the Broadcast Stereo box is set to Anaglyph, select the type of anaglyph setting to use.

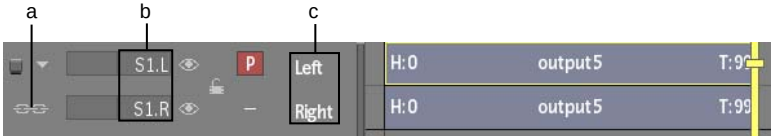
Select:	To view:
Diff	The difference between the two images, with the minimum and maximum threshold values non clamped.
Diff Clamped	The difference between the two images, with the minimum and maximum threshold values clamped.
Blend	A combined image of the two tracks.
Mono	The image with just the anaglyph effect. The RGB values are removed from the display.
Dubois	The image with reduced ghosting between the left and right eyes.

Stereo Tracks

Editorial and effects work on a stereo track is automatically synchronized between the left eye and the right eye layers. Any changes applied to one layer of a stereo track are automatically applied to the other layer.

You can remove this stereo sync between layers if you are changing the media of a clip. However, the metadata must always be identical between the left eye and right eye layers. The layers must share the same resolution, bit depth, and framerate. As well, they must be of the same duration and have the same number of edits (for example, transitions and cuts) in the same places. This means that editing operations such as trimming and sliding can only be done on both layers simultaneously. It is also impossible to do vertical editing between the layers of a stereo track or to contain its layers.

The following interface components are specific to stereo tracks.



(a) Stereo Sync icon (b) Stereo track identifiers (c) Layer names

Stereo Sync icon Changes applied to one layer of a stereo track are automatically applied to the other layer. The stereo sync is enabled by default.

Stereo track identifiers Stereo tracks have "L" and "R" as part of their identifiers to indicate the left and right eye layers. The left eye layer is always the topmost layer.

Layer names The left and right eye layers of a stereo track are named automatically. By default, the top layer is the left layer and the bottom layer is the right layer. You rename the layers in the same way as video layers. S

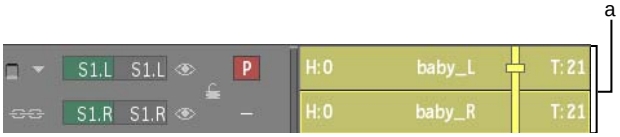
Converting a Stereo Track to Video Tracks

You can convert a stereo track to two video tracks. Do this, for example, if you need to do edits that affect the metadata of one of the stereo layers.

Note that if the two layers of a video track do not have the same metadata, you will not be able to convert them back to a stereo track.

To create a video track from a stereo track:

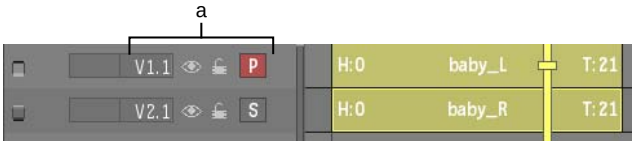
1 Select the left and right eye layers of a stereo track.



(a) Selected stereo layers

2 Select Stereo Split from the Edit Mode box.

The stereo layers are changed to video tracks, and the timeline interface is updated to reflect the components specific to video tracks.



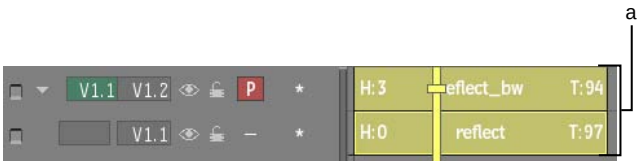
(a) Video track identifiers

Converting a Video Track to a Stereo Track

You can create a stereo track from two video layers on the timeline as long as they share the same metadata.

To create a stereo track from two video layers:

1 Select two video layers with identical metadata.

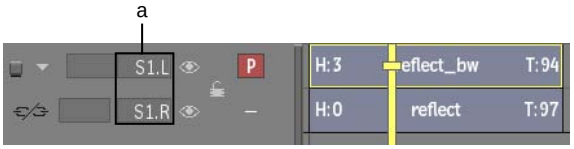


(a) Selected layers

2 Select Stereo Merge from the Edit Mode box.

NOTE If the layers do not have identical metadata, an error message appears.

The video layers are changed to stereo layers, and the track identifiers identify the left eye and right eye layers. The timeline interface is updated to reflect the components specific to a stereo track.



(a) Stereo track identifiers

If there was a soft effect on one of the two video layers, a red bar appears between the layers to indicate that the layers are not in stereo sync.

Removing Stereo Sync

If you are doing work that affects the media of a stereo layer, you can remove the stereo sync from the left and right eye layers and modify only one layer. For example, there may be instances when you need to colour correct only one eye.

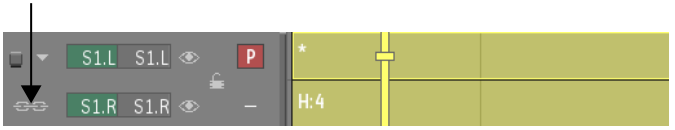
Remove the stereo sync to do any of the following to one layer of a stereo track:

- Apply and delete soft effects
- Slip a shot
- Replace a segment's source

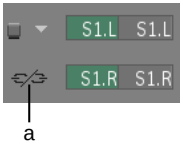
If you applied the same soft effects to unsynced layers but the soft effect settings are different, you can resynchronize the settings.

To remove the stereo sync between the layers of a stereo track:

➤ Click the Stereo Sync icon.



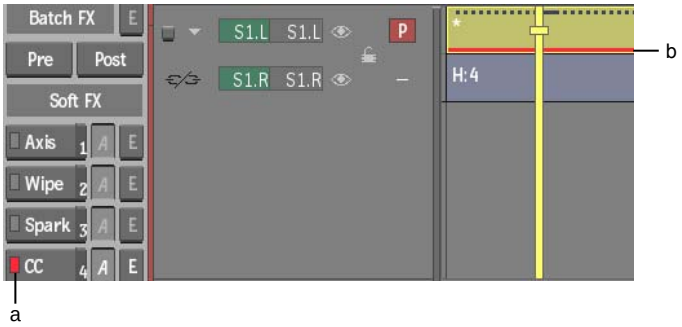
The icon turns black indicating that the stereo sync for the layers has been disabled. You can now edit the media of one stereo layer without affecting the other layer.



(a) Stereo sync disabled

NOTE If you try to perform an edit that changes the metadata of the layer, the stereo sync between the layers will be re-activated automatically and your edit will affect both layers.

If you add a soft effect to an unsynced layer, a red bar appears indicating that the layers are not in stereo sync, and the indicator on the soft effect button turns red.



(a) CC soft effect applied to only one layer of a stereo track (b) Stereo layers are not in sync

To resynchronize a soft effect between stereo layers:

1. Select the stereo layers or elements containing the soft effects that you want to synchronize.
2. If there is more than one soft effect on your timeline selection but you only want to resynchronize one type, select the applicable soft effect icon on the timeline.
3. Select Resync FX from the Commit Effect box.

NOTE This option is only available with stereo tracks.

4. From the message that appears, select whether you want to modify the soft effects on the left eye layer or the right eye layer.

   The same soft effect settings are applied to each layer based on your selection and the red bar is removed from the layer.

NOTE Resynchronizing soft effects does not re-enable the stereo sync icon.

To re-enable the stereo sync icon between the layers of a stereo track:

➤ Click the black Stereo Sync icon.

The icon turns white indicating that the stereo sync between the layers has been re-enabled. If you had applied a soft effect to one of the stereo layers when the stereo sync was removed, the red bar remains on the layer and the indicator on the soft effect button remains red.

Creating Axis Soft Effects

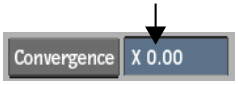
If you are working with a stereo track, you can use an Axis soft effect to adjust the convergence between the left and right eye layers.

To fix the convergence between the left and right eye layers:

1. Apply an Axis soft effect to the left and right eye layers of a stereo track.
2. Select Stereo from the Layer Selection box.

NOTE The Stereo option is available only with stereo tracks.

3. Enter a value in the Convergence field that appears.



The distance between the left and right images is updated. The layer you selected is moved by the value you specified. The other layer is moved by an equal opposite amount.

NOTE If the Convergence field is greyed out, the Axis soft effect is applied to an unsynced stereo segment.

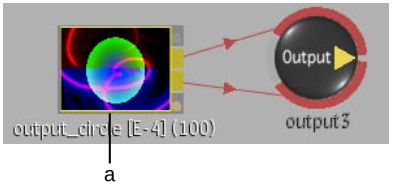
Creating One Clip Per Stereo Track Layer

When you create a clip from each layer of a stereo track, the layers are converted into mono clips.

To create one clip per stereo track layer:

1 Select a clip containing a stereo track in a Batch setup.

In the following example, a clip with Text and CC soft effects applied to its left and right eye layers is selected.

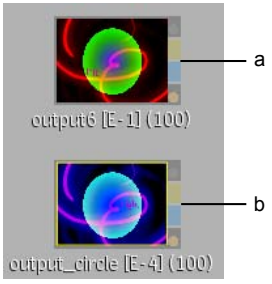


(a) Stereo track of clip contains soft effects

2 Display the clip's timeline and select Split Tracks and Layers from the Split dropdown list.

One mono clip is created from each stereo layer. In the following example, the mono clip created from the left eye layer retains the Text soft effect. The mono clip created from the right eye layer retains the Text and CC soft effects.

Note that the left and right eye outputs are converted to RGB (yellow tabs) and alpha (blue tabs) outputs.



(a) Mono clip created from left eye layer (b) Mono clip created from right eye layer

Displaying Stereo Modes in Multiple Viewports

If you bring a clip containing a stereo track into Batch or Action, you can display multiple viewports in a stereo mode. You can apply a stereo mode to the entire clip or specifically to its left or right eye output.

Example: To display a a stereo mode in Batch:

1. Select a clip containing a stereo track, or select its left or right eye output.
2. Select Left or Right in the Results category of the View box for the selected viewport.

   The Stereo Off toggle appears in the viewport.
3. Click the Stereo Off toggle and select from the stereo modes that appear.

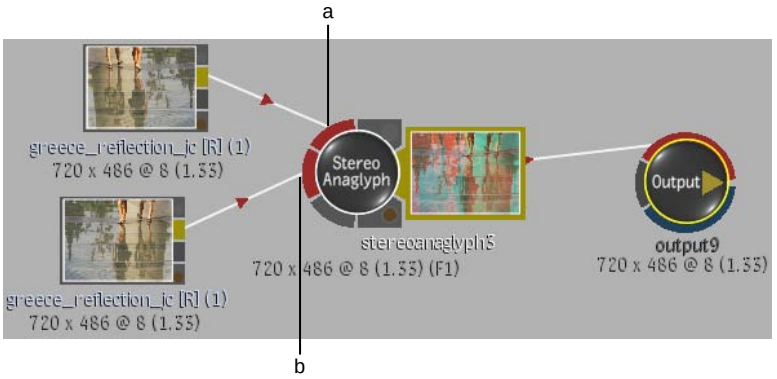
Select:	To display:
Diff	The difference between the two images, with the minimum and maximum threshold values non clamped.
Diff Clamped	The difference between the two images, with the minimum and maximum threshold values clamped. To adjust the maximum clamp value, press **Shift+U**

Select:	To display:
	and drag left or right. To adjust the minimum clamp value, press **Shift+Y** and drag left or right.
Blend	A combined image of the two tracks. To adjust the Blend factor, press **Shift+T** and drag left or right.
Anaglyph Mono	The image with just the anaglyph effect. The RGB values are removed from the display.
Anaglyph Dubois	The image with reduced ghosting between the left and right eyes.

Stereo Anaglyph Node

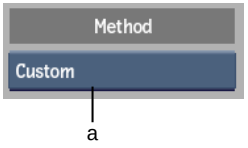
With the Stereo Anaglyph node, you can take a left eye mono clip and a right eye mono clip and output one red/cyan clip containing one video track.

The Stereo Anaglyph node has one left eye and one right eye input.



(a) Left eye input (b) Right eye input

Select an Anaglyph method from the Anaglyph Method box.



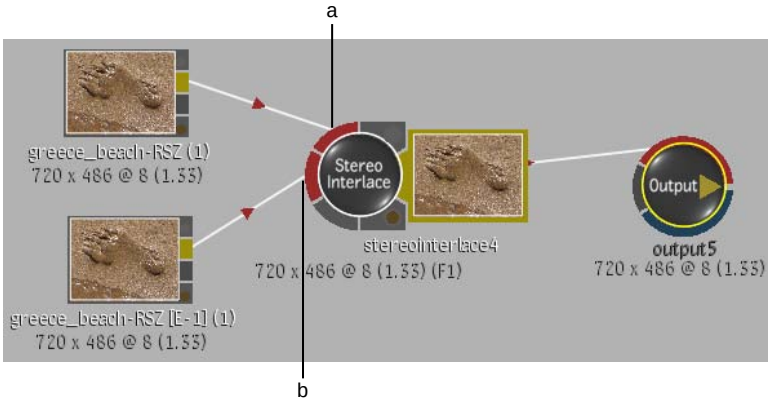
(a) Anaglyph Method box

Select:	To:
Custom	Customize the RGB left and right values. With this option, you can create anaglyph results based on the 3D lenses that will be used.
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.

Stereo Interlace Node

With the Stereo Interlace node, you can take a left eye mono clip and a right eye mono clip and output one RGB clip in Interlaced mode.

The Stereo Interlace node has one left eye and one right eye input. It outputs one interlaced RGB clip containing one track. Depending on the option you select in the Method box, the left eye input is output as field 1 or field 2.



(a) Left eye input (b) Right eye input

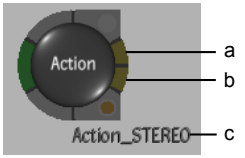
Setting Stereo Startup Mode

When dragging an Action node to the schematic, you can now 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.

To set the stereo startup mode for a new Action node:

➤ Press Shift+S and drag the Action node to the schematic.

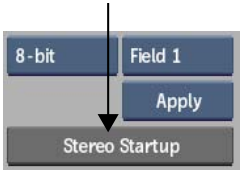
The node is renamed to Action_STEREO, the output tabs display the left eye and right eye outputs, and all stereo settings are automatically set.



(a) Left eye output tab (b) Right eye output tab (c) Stereo node name

To apply the stereo startup setting to an existing Action node:

1. Select the Action node.
2. In the NodeSetup menu, click Stereo Startup and confirm.



The current setup is replaced by the stereo setup and all media is deleted.

Duplicating Batch Nodes

New: The hotkey for creating a duplicate stereo group node when dragging a node from the node bin is now **Shift+S** instead of **Shift+W**. However, you can still use **Shift+W** to create a Duplicate link between two nodes when dragging from one node to another in Move mode.

You can now keep two nodes in the schematic in sync by creating a Duplicate link between them. When you change the settings of one node, they are automatically duplicated on the other node.

You can use Duplicate links for nodes connected to mono clips or to clips containing stereo tracks. For stereo tracks, any changes made with a linked node to the layer of one eye are automatically duplicated on the other eye. For mono clips, any changes made with a linked node to one clip are automatically duplicated on the other clips.

To duplicate 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. As well, for some nodes, the sources must share the same resolution and/or bit depth. If duplicated nodes do not support sources of differing resolutions and/or bit depth, an error message appears in the message bar.

Note that the following nodes cannot be duplicated.

■ Action	■ Modular Keyer
■ Coloured Frame	■ Paint
■ Distort	■ Substance Noise
■ Keyer	■ Text

When you create a Duplicate link between nodes, the original node becomes the master. You can modify either the master node or the duplicated node and the other node is simultaneously updated. However, only the channels of the master node are displayed in the Channel Editor since the channels of the duplicated node are identical to those of the master. If you break the Duplicate link, the channels of both nodes appear. However, the values of the duplicated node are not restored. They keep the values of the master node.

If you copy duplicate nodes, one of the following occurs depending on whether you copied the master node or the duplicated node:

- If you copy the master node, only the master node is copied.
- If you copy the duplicated node, only the duplicated node is copied, but the Duplicate link is recreated between the copy and the master.
- If you copy the master and duplicate nodes, both nodes are copied.

When using duplicate nodes with mono clips, you must create the Duplicate links manually. When using duplicate nodes with a clip containing a stereo track, you can create the Duplicate links manually or have them created automatically using a stereo group node.

To create a Duplicate link using a stereo group node:

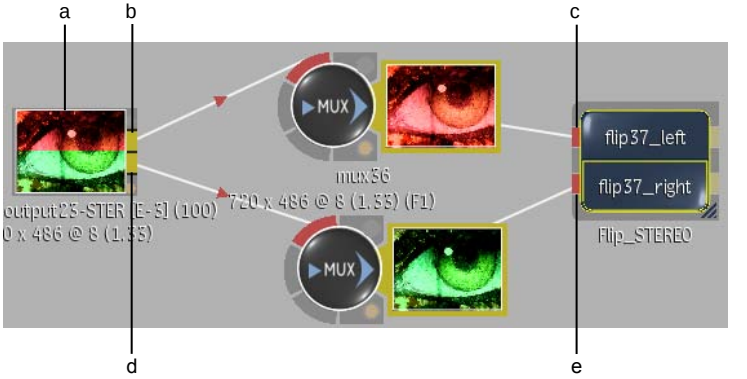
1 Press **Shift+S** and drag a node from the node bin to the schematic.

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.

TIP Use the Advanced Autolink feature to connect the input tabs.

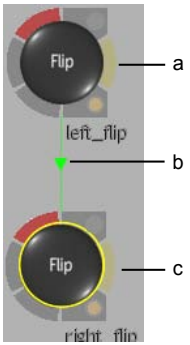


(a) Clip containing stereo track (b) Left eye output tab (c) Left eye input tab (d) Right eye input tab (e) Right eye output tab

3 Double-click the stereo group node or click Edit in the Group List menu.

NOTE Stereo group nodes have the same functionality as regular Batch group nodes. However, the Node List box only displays the option for accessing the master node's menu since the menu settings of the duplicated node are identical to those of the master node.

Another schematic opens containing 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 duplicates.



(a) Master node (b) Duplicate link (c) Duplicate node

4 Change the settings of either node.

Notice that the changes made to one node are duplicated 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 duplicated on Flip node connected to right eye

5 Click Exit Group.

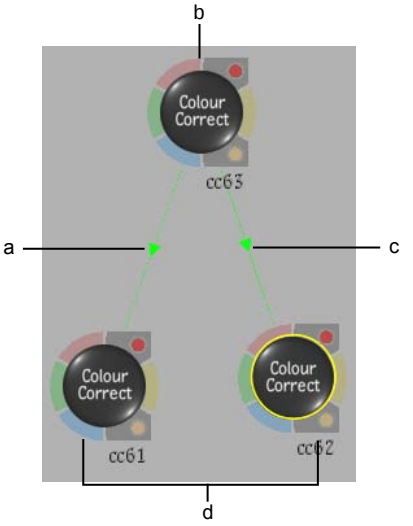
6 Select the Stereo group node and press **Shift+up or down arrow** to toggle between the left eye and right eye outputs.

To create a Duplicate link between nodes connected to mono clips:

1 From the Edit Mode box, select Duplicate (or press **W**).

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.



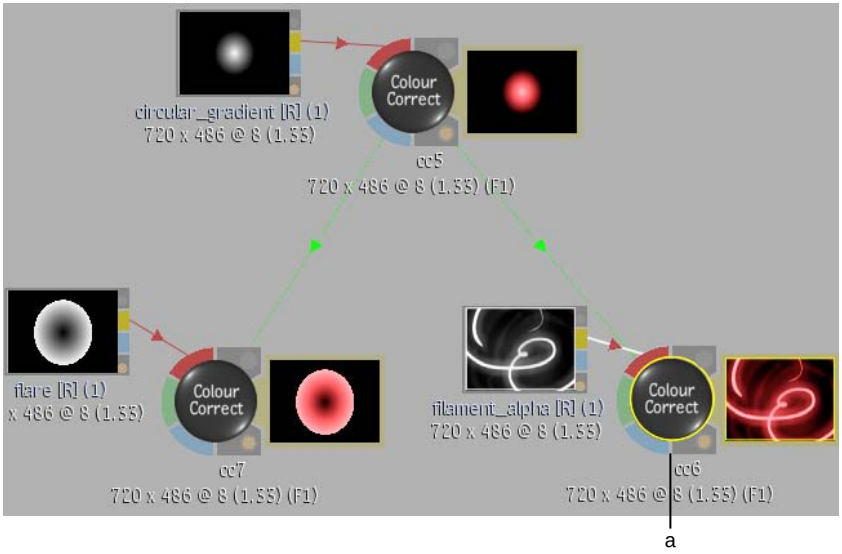
(a) Duplicate link (b) Master node (b) Duplicate link (d) Duplicated nodes

TIP While in Move mode, you can also create a Duplicate link by pressing **Shift+W** and dragging from one node to another so that the nodes touch. Press **Shift+Alt+W** to reverse the link direction.

3 Connect each node to a mono clip.

4 Change the settings of any node.

Notice that the settings made to one node are duplicated on the other nodes so that all connected mono clips are affected simultaneously. In the following example, duplicated CC6 node is edited. Master CC5 node and duplicated CC7 node are updated with the same changes.



(a) Edited node

Entering a BFX Level with Stereo Segments

You can bring stereo segments into a BFX level using the Pre or Post option. To bring in only stereo sources, use the Pre option. To bring in stereo sources and their soft effects, use the Post option.

Stereo segments are represented in a BFX level by a clip node with one left eye output and one right eye output. When you output from the BFX level, each stereo segment (left and right eye) is processed with the same setup.

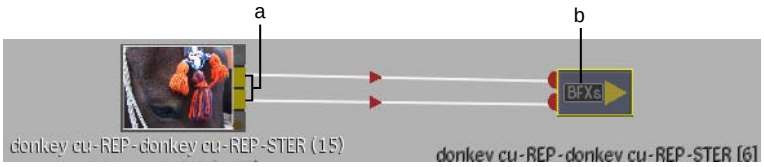
You can unsync stereo segments on the timeline and bring only one segment into a BFX level, in which case the segment is no longer considered stereo. Inside the BFX level, it is converted to a mono clip with RGB and alpha outputs. On output, only the segment that you brought into the BFX level is processed with the Batch setup.

To enter a BFX level with stereo segments:

1. Select the stereo segments that you want to bring into a BFX level or move the positioner's focus point over the applicable segments.

2 Do one of the following:

- Click Pre to bring in the left and right eye sources.
- Click Post to bring in the left and right eye sources and their soft effects.
  The segments are brought into a Batch setup as a clip with one left eye output and one right eye output. The BFXs output node indicates that a stereo track will be processed back to the main timeline level.



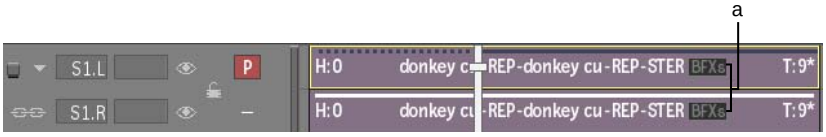
(a) Clip with left eye and right eye outputs (b) BFX stereo output node

NOTE You cannot output mattes with BFX stereo output nodes.

3 Add nodes to the clip and do compositing work.

4 Exit back to the main timeline.

Both the left and right eye segments have a BFXs icon indicating that a Batch setup is applied to the stereo segments. Because the segments (S1.L and S1.R) are considered as one entity, they share the same setup. Although there are two BFXs icons on the timeline, you enter the same setup with either icon.



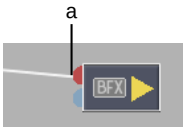
(a) Double-click either icon to access the same BFX setup

BFX Output Node

Each BFX level contains a BFX Output node. Use the BFX Output node to output a Batch setup applied to a timeline segment or to a contiguous selection of segments.

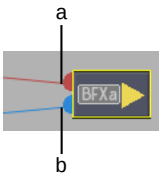
You can output an RGB-A clip or a clip containing a stereo track.

For RGB-A clips, the BFX Output node accepts a front and matte input. The front is output as the RGB result.



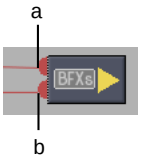
(a) RGB (front) input tab

If a matte input is attached to the node, the icon displays an ‘a’ next to the BFX symbol indicating that the matte will be output as the alpha result.



(a) RGB (front) input tab (b) Matte input tab

For clips with a stereo track, the BFX Output node accepts a left eye and right eye input. The icon displays an ‘s’ next to the BFX symbol indicating that a stereo track will be processed.

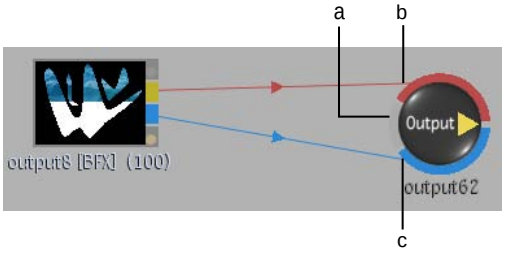


(a) Left eye input tab (b) Right eye input tab

If the BFX level was entered with the Pre option, the icon background is grey. If it was entered with the Post option, the background is black.

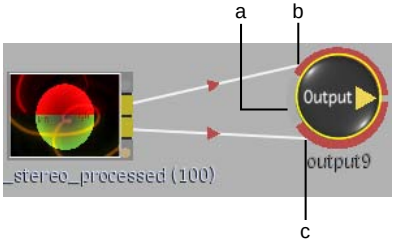
Output Node

You can now process both RGB-A clips and clips containing stereo tracks with an Output node. In RGB-A processing mode, the Output node accepts a front input and a matte input. The front is output as the RGB result and the matte is output as the alpha result.



(a) Audio input tab (b) RGB (front) input tab (c) Matte input tab

In Stereo processing mode, the Output node accepts a left eye input and a right eye input. The left eye and right eye are output on two layers of the same stereo track.



(a) Audio input tab (b) Left eye input tab (c) Right eye input tab

You select the processing mode from the Output node's menu settings.

Defining Processing Settings

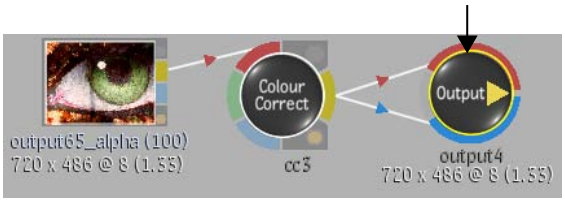
Output nodes now support Stereo processing mode.

Use node settings to define how individual Output and Export nodes are processed and to specify a destination for each processed result.

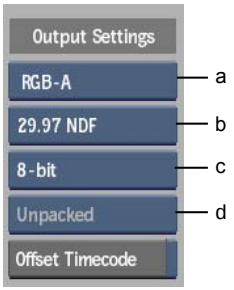
When you drag an Output node from the node bin to the schematic, the Output node that appears by default is set to RGB-A processing mode. You use the Output Settings to change it to Stereo processing mode.

To define settings for an Output node:

1 Select an Output node.



Settings for the Output node appear.

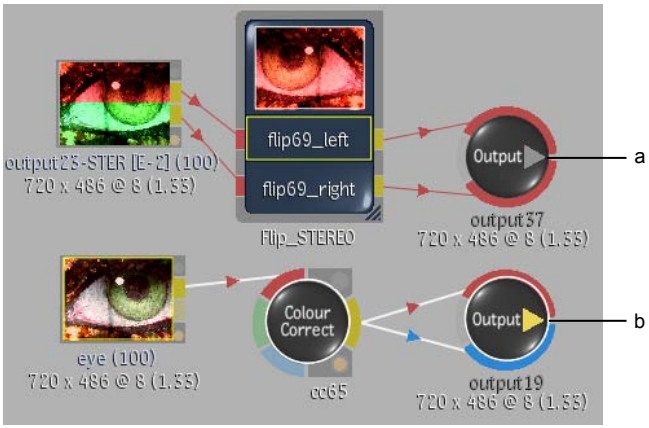


(a) Output Type box (b) Framerate box (c) Output Bit Depth box (d) Storage box

2 Select the type of clip to be processed from the Output Type box.

In Stereo processing mode, the Output node displays a red left eye input tab and a red right eye input tab. The left eye and right eye outputs will be processed on two layers of the same stereo track

In RGB-A processing mode, the Output node displays a red RGB input tab and a blue matte input tab. The front will be output as the RGB clip and the matte will be output as the alpha result.



(a) Output node with left eye and right eye inputs (b) Output node with RGB and matte inputs

3 Select the framerate for the clip to be output from the Framerate box.

4 Select a bit depth for the clip to be output from the Output Bit Depth box.

If you select 12-bit, the Packed and Unpacked options in the Storage box become active. 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.

5 Enable Offset Timecode to enable any timecode offsets you make in the Queue Manager.

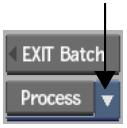
For example, with Offset Timecode enabled, if you specify a timecode of 00:00:00:00 and render from frame 10, the first frame of the render will have TC 00:00:00:09.

6 From the Output Destination box, select whether the output will be written to the reel on the Desktop or to a library. If Library is selected, you are prompted to select an existing library or create a new one.

7 (Optional) Select another Output node or an Export node and define settings for another job.

8 (Optional) Set up the processing jobs in the Queue Manager.

9 Click Process or select a processing option from the dropdown list.



All active jobs are processed to the destinations you set.

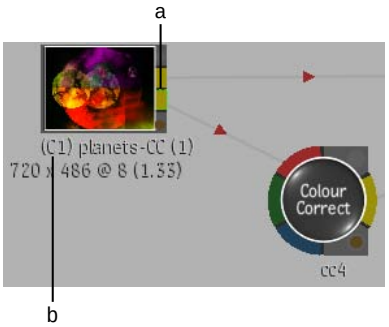
Viewing Nodes in Context

You can now set a context on a node or on any of its output tabs. You can also set a context on any node output in a group as well as on any output of an Action multiple output node.

Example: To set a context on one output of a stereo node:

1 Press = and click the tab (left or right eye output) whose context you want to set.

(C1) or (C2) is added to the output name. As well, a green dotted line appears around the tab on which you set the context. In the following example, a context is set on the right eye output.



(a) Context on right eye output (b) Context 1 identifier

2 To display the context in a viewport, select the context from the View box or press the associated hot key.

3 To display a stereo mode on the context in the viewport, click the Stereo Off toggle in the viewport and select from the stereo modes that appear. For information on stereo modes, see Displaying Stereo Modes in Multiple Viewports on page 83.

Action: Stereo

6

About 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 FBX 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 output.

When working in a stereoscopic compositing workflow in Action, there are three essential elements: a stereo camera, 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 on page 100.
2. Work with the stereo camera and stereo object to make any adjustments to the scene.	Action: FBX Camera on page 109 and Working with the Stereo Scene on page 101.
3. Output various passes of your work.	Processing Multiple Outputs on page 131.

For information on working with stereoscopic scenes in Batch, see Stereoscopic Workflow on page 61.

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 or Multitrack input, or if it is selected to replace a clip in the Media list.

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 on the Desktop. They retain the name of the original clips, and are appended with a "_Left" or "_Right" suffix. It is important to save these clips in the library, 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:

1. From the Main menu, click Effects, then 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 the Action module 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, an FBX 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 FBX 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 Stereo Startup button in the Action Setup menu. You will now have to bring in new clips.

When you exit Action, the Desktop 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:

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 Surfaces tab of the node bin and place it in the schematic.
- Drag the Stereo Object node from the Surfaces tab of 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.

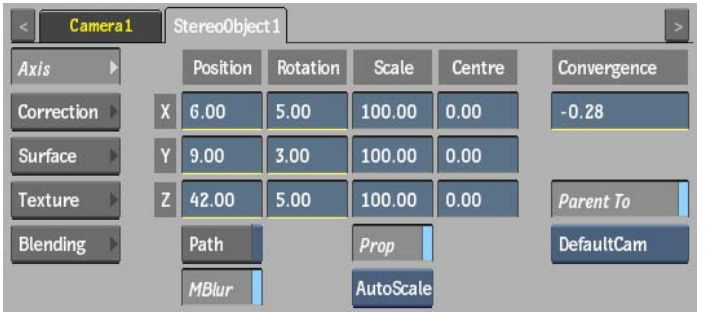
3 To display a selected viewport in any of the stereo modes (Anaglyph Mono, Anaglyph Dubois, Blend or one of the Difference modes), select one from the Stereo mode button in the lower-left corner of the viewport.

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 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 Axis controls are described as follows.

Position X, Y, and Z fields Translate the selected axis.

Rotation X, Y, and Z fields Rotate the selected axis.

Scaling X, Y, and Z fields Change the size of the axis.

Prop Scale button Enable to scale the X, Y, and Z axes proportionally.

Shear X, Y, and Z fields Shear the axis.

Centre X, Y, and Z fields Offset an axis relative to its children.

Path button Enable to animate the position of the axis using a spline drawn in the scene.

Disable Path to animate the position of the axis using explicit animation.

Autoscale box Select from among the following axis scaling options.

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.

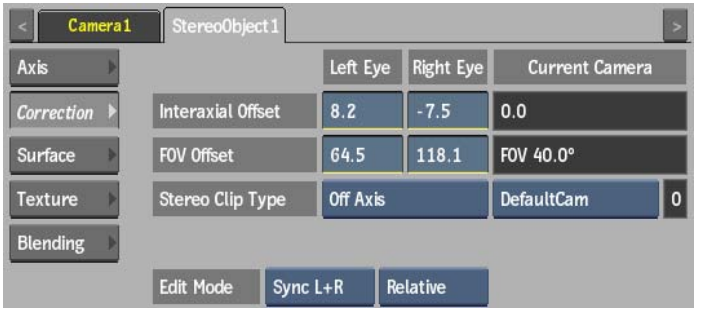
Convergence field Displays the left and right images, horizontally, an equal amount in opposite directions. Positive values make the image appear farther from the camera, and vice versa. The default is 0.

Parent To button Enable to associate a stereo object with a camera. It does not draw an explicit line in the schematic. Parenting a stereo object to a camera in the schematic takes precedence over this option. By default, this parameter

is enabled to force the stereo object to face the FBX camera at al times when the camera is moved around.

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.



The Correction controls are described as follows.

Interaxial Offset fields (Left Eye/Right Eye/Current Camera) Displays the offset distance between the left and right eye compared to the FBX camera. The Current Camera fields display the Interaxial of the selected camera.

FOV Offset fields (Left Eye/Right Eye/Current Camera) Displays the offset field of view between the left and right eye compared to the FBX camera. The Current Camera fields display the FOV of the selected camera.

Stereo Clip Type box Specify how the footage was shot: Parallel, Off-axis, or Converged.

Camera box Select the camera to which the stereo object is linked so that the correct transformations are applied when changing the Interaxial and FOV offset with regards to the FBX camera setting.

Edit Mode settings

Select:	To:
Sync L+R	Link the Left Eye and Right Eye values. 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.

Select:	To:
Absolute/Relative	Absolute: Make the values of the Left Eye and Right Eye the same. Relative: Make the value of one eye relative to the change in the value of the other eye.

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.

Stereo Object Texture Settings

Use the Texture tab of the Stereo Object menu to position, rotate, scale, shear, and centre the left and right images of a stereo object.

The Texture controls are described as follows.

Position X, Y, and Z fields Translate the selected axis.

Rotation X, Y, and Z fields Rotate the selected axis.

Scaling X, Y, and Z fields Change the size of the axis.

Shear X, Y, and Z fields Shear the axis.

Centre X, Y, and Z fields Offset an axis relative to its children.

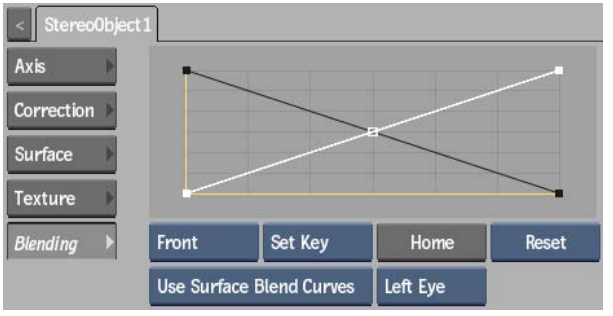
Prop Scale button Enable to scale the X, Y, and Z axes proportionally.

Edit Mode settings

Select:	To:
Sync L+R	Link the Left Eye and Right Eye values. A change to one value affects the other value.
Absolute/Relative	Absolute: Make the values of the Left Eye and Right Eye the same when adjusting the value. Relative: Make the value of one eye relative to the change in the value of the other eye.
Left Eye/Right Eye	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 on page 140. 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.

Action: FBX Camera

7

About the FBX Camera

The FBX 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 FBX camera to frame and animate the view to achieve the effect that you want. You can also animate specific camera properties.

Use the FBX 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.

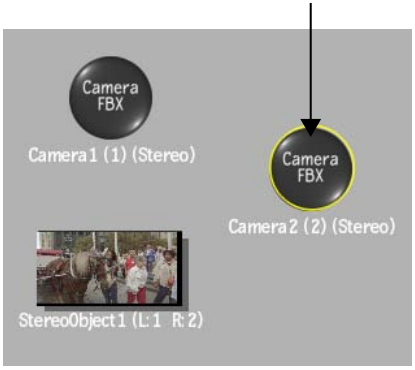
Adding an FBX Camera

By default, an FBX 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 FBX 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 FBX node from the Objects tab of the node bin and place it in the schematic.
   - Drag the Camera FBX node from the Objects tab of the node bin and place it where you want it in the Result view.
   - Double-click the Camera FBX node. You do not need to be in Schematic view to add a node in this manner.

   A new FBX camera is added to the scene. An icon representing the camera is added to the schematic.



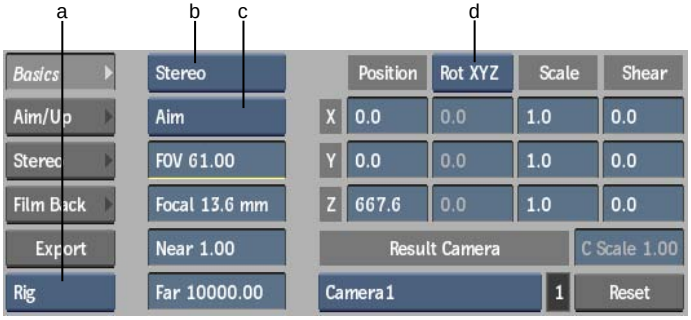
2. Select the FBX Camera node to make it the active camera in the scene.
3. In the Object menu that appears, set the Result Camera to a numbered camera representing an FBX camera.
4. In the Output menu, set the Mode to Stereo and set the Camera to Result Cam.

FBX Camera Parameters

The FBX camera contains a rich set of parameters that can be altered to set specific features, such as standard camera configurations, frustum customization, and sterescopic attributes.

Basics Tab

The Basics tab contains the options to select the camera type and to adjust camera position and scaling.



(a) Stereo Camera View Type box (b) FBX Camera Type box (c) Camera Type box (d) Rotation Order box

Stereo Camera View Type box Select Left, Right, or Rig (for Stereo Rig).

FBX Camera Type box Select whether the FBX 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 the angular field of view to adjust the width of the camera frustum.

Focal Length field Displays the focal length of the camera, measured in millimeters. Increasing the Focal Length zooms the camera in and increases the size of objects in the camera's view. Decreasing the Focal Length zooms the camera out and decreases the size of objects in the camera's view. The valid range is 1 to 1000. The default value is 40.

Near Clip Plane, and Far Clip Plane fields Display the positions of the near and far clipping planes, in pixels, which represent the distance from the camera within which image details are processed. The default setting for Near Clip Plane is 1 and for Far Clip Plane is 10000.

Set the Near Clip Plane and Far Clip Plane attributes 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.

See Moving the Near and Far Clipping Planes on page 120.

TIP The 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.

Position X, Y, and Z fields Displays the position of the camera, in pixels, on the horizontal, vertical, and perpendicular (X, Y, and Z) axes.

Rotation Order box Select the order in which the camera is rotated, on the horizontal, vertical, and perpendicular (X, Y, and Z) axes.

Rotation X, Y, and Z fields Displays the level of rotation of the camera on the horizontal, vertical, and perpendicular (X, Y, and Z) axes, in degrees. Active when Camera Type is set to Free.

Scale X, Y, and Z fields Displays the scale of the camera on the horizontal, vertical, and perpendicular (X, Y, and Z) axes, as a percentage.

Shear X, Y, and Z fields Displays the shearing of the camera (diagonal shift) on the horizontal, vertical, and perpendicular (X, Y, and Z) axes, as a percentage.

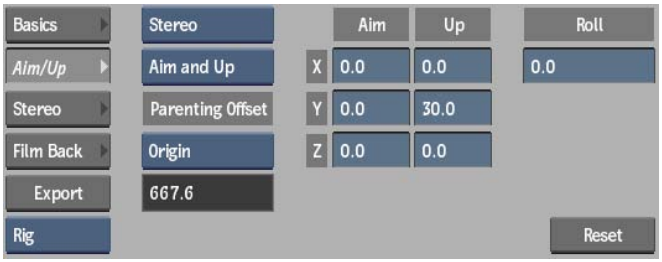
Result Camera box Specify which camera is active. The active camera is the one that will be used when processing your scene. The field displays the active camera number. A value of 0 indicates that you are viewing the scene through the default camera.

Camera Scale field Displays the size of the camera relative to the scene independantly 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.

Reset button Resets the FBX Camera menu to its default settings.

Aim/Up Tab

When Camera Type is set to Aim or Aim and Up, the available options are enabled in the Aim/Up tab.



Parenting Offset box Select Origin or Target. When parenting a camera node, the image offset gets reset to the camera origin, which is not always the desired viewing option. Setting Parenting Offset to Target restores the image to the default viewplane distance relative to the camera. The offset value is computed from the default camera field of view and the default image size, and does not change, even if other camera parameters are changed. This value is displayed in the Parenting Offset field.

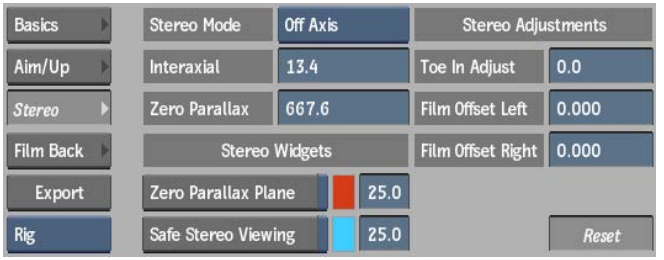
Aim X, Y, and Z fields Display the position of the aiming target of the camera on the horizontal, vertical, and perpendicular (X, Y, and Z) axes, in pixels.

Up X, Y, and Z fields Display the up direction on the horizontal, vertical, and perpendicular (X, Y, and Z) axes, in pixels.

Roll field Displays the amount of camera roll, in degrees. A positive value rolls the camera clockwise, and a negative value rolls it counter-clockwise. This field is available only with the Aim, and Aim and Up cameras.

Stereo Tab

The Stereo tab contains the options for the stereoscopic camera, including Stereo Adjustments and Stereo Widgets.



Stereo Mode box Select the method for computing the zero parallax plane from one of the following modes.

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 saccade 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.

Zero Parallax field Displays the distance on the camera view axis where the zero parallax plane occurs, in pixels. In other words, the point where objects appear off screen. If an object is in front of the zero parallax plane, it has negative parallax and if an object is behind the zero parallax plane, it has positive parallax.

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 Select the colour used for the zero parallax plane.

Zero Parallax Transparency field Displays the level of transparency for the zero parallax plane.

Safe Stereo Viewing Volume button Enable to display the safe viewing volume created by the intersection of the frustrums of the left and right cameras.

Safe Stereo Volume colour pot Select the colour used for the safe stereo viewing volume.

Safe Stereo Volume Transparency field Displays the level of transparency for the safe stereo viewing volume.

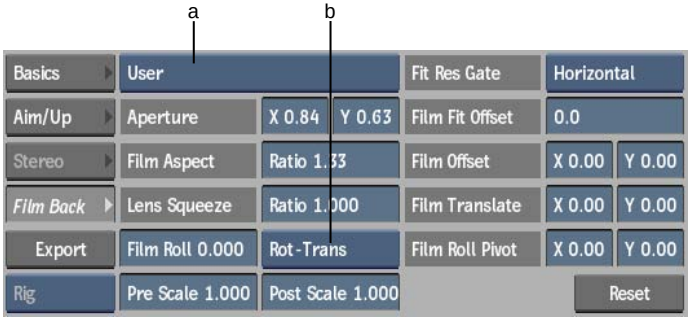
Toe In Adjust field Displays the offset, in degrees, applied to the computed toe-in effect when Stereo Mode is set to Converged.

Film Offset Right Cam field Displays the horizontal film offset for the right camera.

Film Offset Left Cam field Displays the horizontal film offset for the left camera.

Film Back Tab

The Film Back options control the basic properties of a camera (for example, the camera's film format: 16mm, 35mm, 70mm).



(a) Film Gate box (b) Film Roll Rotation Order box

Film Gate box Select a preset camera type. 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 fields Display the height and width of the camera's Film Gate setting, measured in inches. The default values are 1.417 and 0.945. This setting has a direct effect on the camera's angle of view.

Film Aspect Ratio field Displays the ratio of the camera aperture's width to its height. Maya automatically updates the Film Aspect Ratio (and vice versa). The valid range is 0.01 to 10. The default value is 1.5.

Lens Squeeze Ratio field Displays the amount that the camera's lens compresses the image horizontally. Most cameras do not compress the image they record, and their Lens Squeeze Ratio is 1. Some cameras (for example, anamorphic cameras), however, compress the image horizontally to record a large aspect ratio (wide) image onto a square area on film. The default value is 1.

Film Roll Value field Specifies, in degrees, the amount of rotation applied around the film back. The rotation occurs around the specified pivot point. This value is used to compute a film roll matrix, which is a component of the post-projection matrix.

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.
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. The Pre Scale value is used in 2D effects.

Post Scale field Displays the artificial 2D camera zoom that is applied after the film roll. The Post Scale value is used in 2D effects.

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. The default setting is Fill.

Select:	To:
Fill	Fit the resolution gate within the film gate.

Select:	To:
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, in pixels, of the resolution gate relative to the film gate either vertically (if Film Fit is Horizontal) or horizontally (if Film Fit is Vertical). Film Fit Offset has no effect if Film Fit is Fill or Overscan. The default setting is 0.

Film Offset fields Displays the vertical and horizontal offsets, in pixels, of the resolution gate and the film gate relative to the scene. Changing the Film Offset produces a two-dimensional track. The default setting is 0.

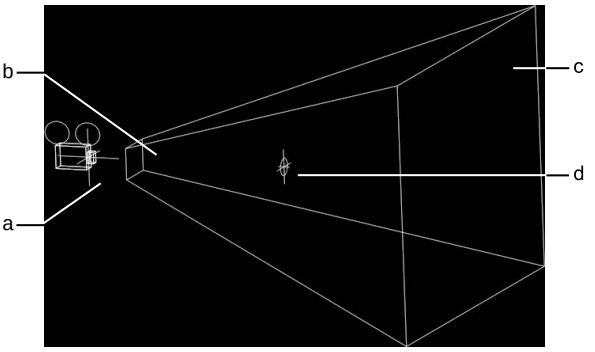
Enter:	To:
1	Have the view guide fill the view. The edges of the view guide may be exactly aligned with the edges of the view, in which case the view guide is not visible.
> 1	Increase the space outside the view guide. The higher the value, the more space is outside the view guide.

Film Translate fields Display the artificial 2D horizontal and vertical camera pans. The Film Translate values are used in 2D effects.

Film Roll Pivot fields Display the horizontal and vertical pivot points from the center of the film back, in pixels. The pivot points are used during the rotating of the film back. These double-precision parameters correspond to the normalized viewport. The Film Roll Pivot values are used to compute the film roll matrix, which is a component of the post projection matrix.

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

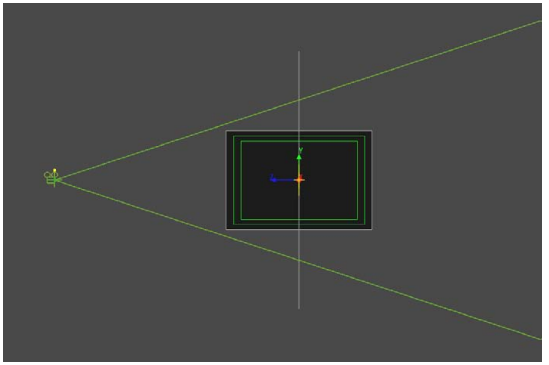
If you place a surface within the frustum, it is visible in the final animation. If the surface is located outside the scope of the frustum, it is not visible at that frame in the animation.

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.

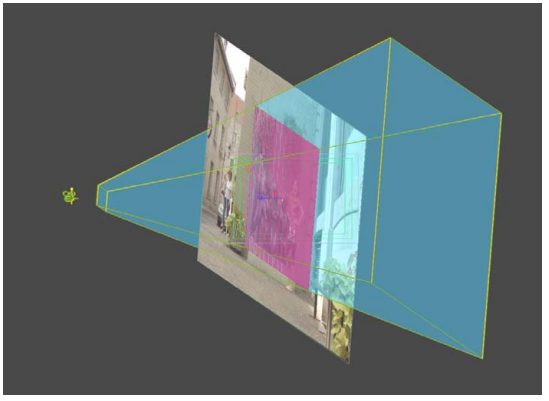
6 Drag the Roll field until you see the four sides of the frustum.

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 on page 120.

9 Change the position of the camera to alter the orientation of the frustum. See Moving the FBX Camera on page 120.



10 Change the position of the camera's aim to alter the orientation of the frustum. See Adjusting the Aim of the FBX Camera on page 121.

11 Adjust the angle of view to adjust the size of the objects as viewed by the camera. See Adjusting the Field of View on page 121.

Moving the 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.

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 FBX 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 FBX 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 FBX 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 FBX 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

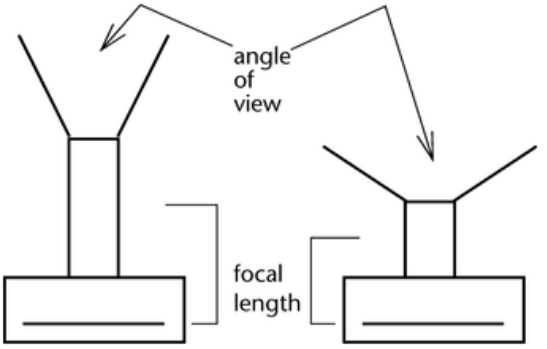
For every shot, you can decide how big an object appears in the frame, for example, whether a shot includes an entire character or just its head and shoulders. This is directly controlled by adjusting the field of view.

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 FBX Camera on page 120.

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.



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

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.

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 FBX Cameras

Inferno supports the import and export of 3D data saved in the Alias® FBX 3D format. The FBX standard format provides 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.

FBX support makes the import and export of 3ds Max, Alias Wavefront™, Maya®, Softimage® XSI™, and LightWave 3D® formats possible.

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 Inferno are not supported as animation curves in FBX. FBX Camera does not support depth of field.

Export an FMX camera from Action to FBX format, which can be used later by any other FBX-compatible application.

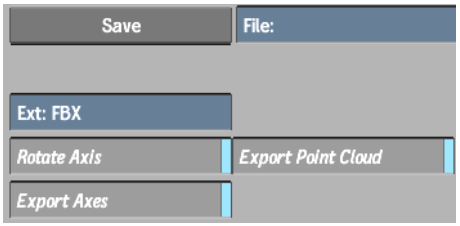
To export an FBX camera:

1 Select the camera that you want to export.

2 In the FBX Camera menu, click Export.

The Export Camera file browser appears.

3 Select which elements of the FBX files that you wish to export by clicking the corresponding filter button.



Enable:	To:
Rotate Axis	Rotate the exported camera by -90° on the X-axis so that it is compatible with the coordinate system of the 3D application.
Export Axes	Export the animated axes present in the Action scene.

Enable:	To:
Export Point Cloud	Export the 3D point cloud created by the 3D Tracker.

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.

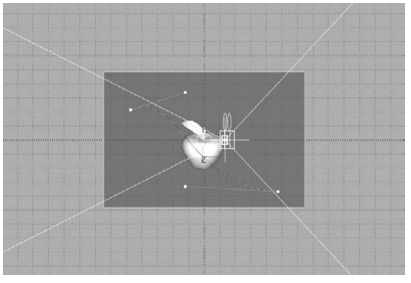
Setting Camera Views

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.

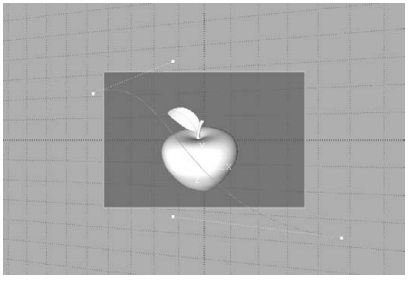
Camera, Working, and Orthographic Views

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.

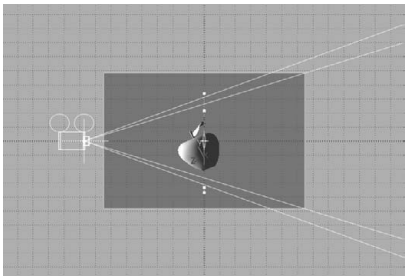
The following figures illustrate the different angles by which the scene can be viewed. The scene in this example contains a grey back clip and the 3D model of an apple.



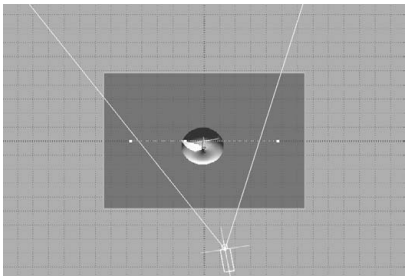
Front view



Camera view



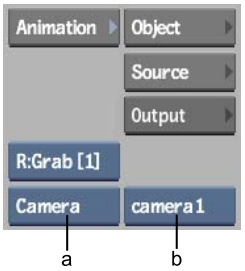
Side view



Top view

To set camera, working, and orthographic views:

1 From the View box, select Camera, Working, or an orthographic view. When in Camera view, use the Camera box that appears to select which camera is used in the image window.



(a) View box (b) Camera box

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.

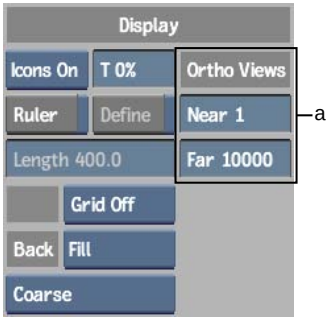
2 If you have multiple cameras in the scene, use the Camera box to define which camera is used for the Camera view in the image window. The camera selected in this box is not necessarily the camera used to process the scene.

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.

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.



(a) Ortho Views in Setup menu

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 on page 120.

In addition to Camera, Working, and the three orthographic views, you can also select Schematic view, which uses nodes to represent the objects in the scene and arrows to illustrate the relationships between objects.

Modifying the Camera

You can gesturally modify the camera directly in the scene using options in the Edit Mode box. A mode remains in effect until you select a different mode.

To gesturally modify the camera:

1 Make a selection in the Edit Mode box.

Select:	To:
Track	Move the camera lens and look-at point.
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.
FOV	Move the camera field of view.
Dolly	Move the camera lens towards (zoom in) or away from (zoom out) the look-at point.
Pan	Move the camera left and right by moving the look-at point. Also changes the camera roll, when not 0.00.

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 Edit Mode 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.

Optimizing Attributes of the FBX Camera

Here are some general guidelines for tweaking the stereo attributes of the FBX camera:

- Many parameters are relative to each other, and can be scaled and changed upon import using the FBX Unit to Pixels option.
- 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.

3D Compositing

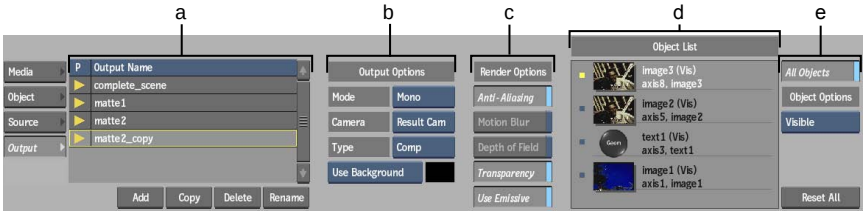
8

Processing Multiple Outputs

You can select what you want to process from Action in the Output menu, such as the normals, Z-Depth, matte, and media matte, as well as the entire composition of a scene. This gives you the ability to output different passes from the same scene, so you can experiment and fine-tune them to your liking. You can have as many outputs as you want.

There is always one default output, which cannot be deleted. By default, it is set to render the result camera and the entire scene. You can pick the camera from which to render the scene, for example, you can render through the result camera, which can be stereoscopic. When you save an Action setup, your outputs are also saved.

To access the Output menu, click the Output button.



(a) Outputs list (b) Output Options (c) Render Options (d) Object List (e) Visibility Options

To add an output:

1. From the Outputs list, click Add.
   A new, empty output is added to the Outputs list.
2. Optional: Click Rename and type in a name for the output.

To select an output:

1. From the Outputs list, select an output.
2. Set the output and render options. See Render Options on page 133 and Output Options on page 132.

To copy an output:

1. From the Outputs list, select an output.
2. Click Copy.
   A copy of the selected output is created.

To delete an output:

1. From the Outputs list, select an output.
2. Click Delete.

To select an ouput(s) to process:

➤ In the Output list, click the yellow arrow beside an output. You can click as many outputs as needed.

Output Options

Once you have determined what to output in the Output list, you can further refine the output parameters in your Action scene. The mode and camera options work together depending on what you selected.

Mode box Select a render mode: Stereo, Left, Right, or Mono.

NOTE Selecting the Stereo mode and a stereo camera results in a left and right clip on the Desktop. Selecting the 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.

Camera box Select a camera or rig to output for the selected output.

Type box Select one of the following types.

Enable:	To render:
Comp	Selected objects in the scene with their parameters.
Matte	The matte of the selected objects in the scene.
Media Matte	The matte of the media specified in the Media field.
Z-Depth	The Z depth of selected objects in the scene.
Normal	The normals of selected objects in the scene.
Emissive	The colours of selected objects in the scene.

Comp box Available when Comp is selected in the Type box. Select whether to render the scene over a background or over a colour you choose using the colour picker.

Matte Background box Available when Matte is selected in the Type box. Select whether the object's matte is rendered on top of a black (default) or white background.

Media field Available when Media Matte is selected in the Type box. Select media by entering a number that corresponds to the media in the Media list.

Render Options

Use the Render Options to further refine your output.

Anti-Aliasing button Enable to process the selected output with the Anti-Aliasing setting defined in the Action Setup menu.

Motion Blur button Enable to process the selected output with the Motion Blur setting defined in the Action Setup menu.

Depth of Field button Enable to process the selected output with the Depth of Field setting defined in the Action Setup menu.

Transparency button Enable to process the selected output taking the transparency value of the objects into account. This lets you decide if the transparency information should be included in the Comp Output or the Matte Output, for example.

Use Emissive button Enable to process a Comp output with the emissive effect.

Selecting Objects to Output

You can select objects to include in your output by using the 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 lights and 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 in the Objects list to display all objects in the scene.

(a) Visibility button

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), the visibility is indicated as Mixed.

To output all objects:

➤ In the Object List, 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 Object Options 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.
  The object is indicated by "(Occ)" in the Object List.
- **Visible** Renders the object in the final render.
  The object is indicated by "(Vis)" in the Object List.

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 Objects list:

1 From the Edit Mode box, select Edit Output or press **Alt+E**.

NOTE In Edit Output mode, All Objects in the Object List 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 grayed out (included or excluded from the output).

To set all objects to Visible mode:

➤ Click Reset All.

All objects in the selected output are set to the Visible mode.

Light Menu Settings

New for this release: Use the new lighting bevel curve to create effects that multiply the light intensity.

The Light menu settings are described as follows.

Position X, Y, and Z fields Places the light source in the scene.

Rotation X, Y, and Z fields Rotates the light source.

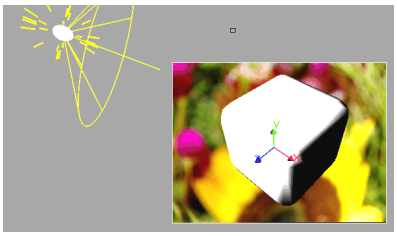
Path button Animates the position of the light using a spline drawn in the scene. Disable Path to animate the position of a light using explicit animation.

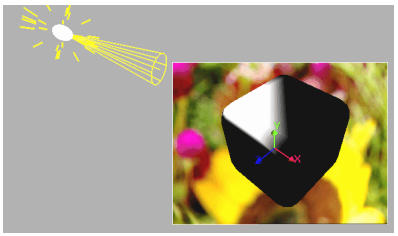
MBlur button Enables or disables motion blur only for the light.

Shading button Enable to light up the scene using added light sources. When Shading is disabled, no lighting effects appear in the scene; surface and 3D models appear flat. This same button appears in the Rendering section of the Action Setup menu.

Intensity field Adjusts the intensity of the light.

Spread field Uses a light source as either a point light or a spotlight. 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 spotlight.



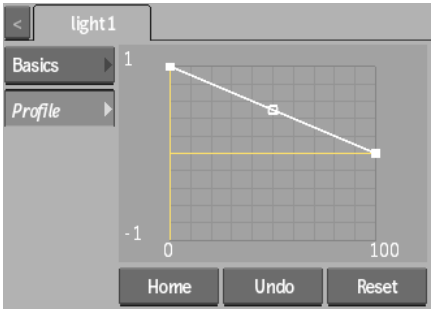


Falloff field Adjusts the amount of falloff around the edge of the light source. This value also changes the size of the specular highlight. A lower falloff value creates a larger specular highlight.

Enable button Turns the selected light source on or off.

RGB fields and Colour pot Changes the colour of the light source by entering values in the red, green, and blue channel fields or using the colour picker.

Settings for controlling the light bevel curve are located in the Light Profile tab.



Light Bevel curve Adds a specific profile to the selected light. Use the options in the Edit Mode 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. See Using the Light Bevel Curve on page 138.

Home button Resets the Bevel curve viewer to show the whole curve.

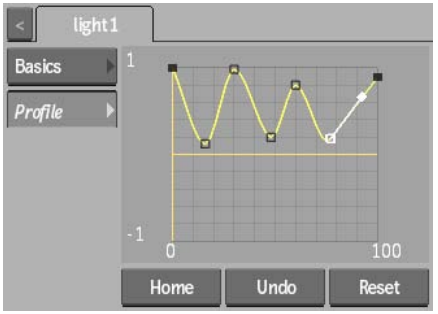
Undo button Undoes Bevel curve operations.

Reset button Resets the Bevel curve.

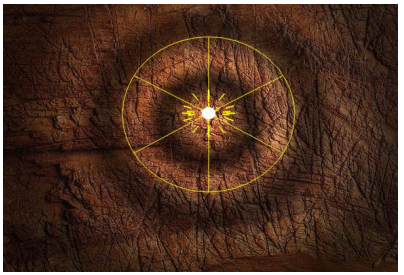
Using the Light Bevel Curve

The bevel curve allows you to create unique lighting effects in conjuntion with the Spread field in the Light Basics tab. 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.

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.

Applying an Axis Look-at Connection

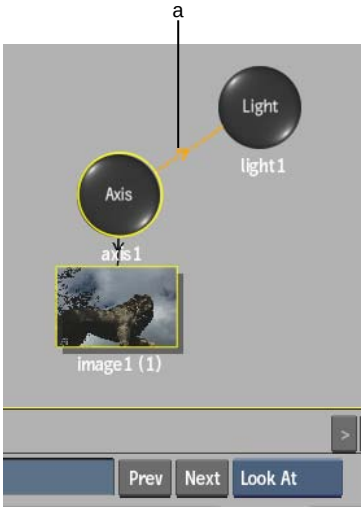
New for this release: Create interesting axis effects by using the new look-at connection.

You can attach a look-at connection between the axis and another object in your scene. The axis 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 Axis node and any object with axis characteristics (Axis, Camera, Light, Projector, Particle Animator, and DVE Object).

To apply a look-at connection:

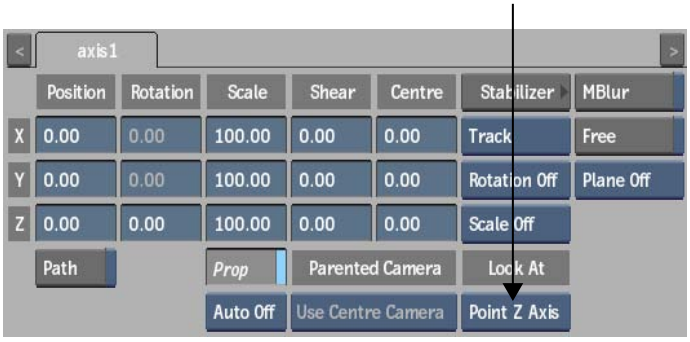
1. Select Look At in the Edit Mode box (or press **Alt+L**).
2. In the schematic, drag from the Axis node to an object with axis characteristics.

The selected object is connected to the Axis node by an orange dotted line with an arrow.



(a) Look-at connection

3 In the Axis menu, select which axis looks at the attached object in the Point Axis box.



Applying Blending Curves per Surface

New for this release: In Action, you can now modify blending curves directly in the Surface menu, saving you time by not having to switch to the Keyer or Modular Keyer menu.

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

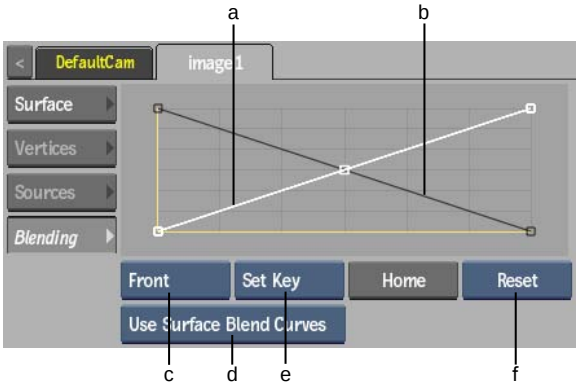
where:

- 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.

To adjust the blending curve:

1. In the Surface menu, click the Blending tab.

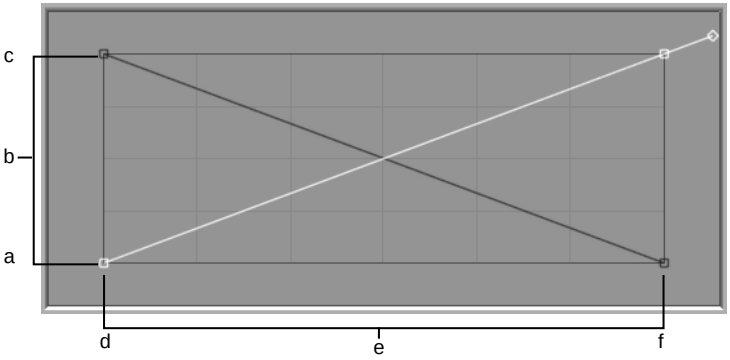
   The Blending Curve menu appears.



(a) Front matte curve (b) Back matte curve (c) Matte box (d) Blend Curves option box (e) Keyframe option box (f) Reset Selection box

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 Move edit mode, click a point to display its tangent handle and drag the handle to adjust the curve. Use other modes in the Edit Mode box (Add, Delete, or Break, 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.

Importing Files Using A Gateway Library

9

About Gateway Library Import

From the Clip Library menu, import graphic image files, film scans, video files, or AAF and FCP XML sequence files. Rules allow you to control how imports are handled.

To do so, you connect to a Gateway library which allows you to browse the file system of your workstation or even of a remote volume. See Accessing Gateway Libraries on page 175

Media Management

When you import media that resides on an external storage device, all the media is copied to Autodesk storage. Any changes that you make to this media are protected, and exclusive to this stored copy. However, it can take a long time to load large files and all of this stored media can take up a lot of space in Autodesk storage.

Clips with unmanaged media, also known as *soft-import* clips, provide a more efficient way to handle media. When you import media as a clip with unmanaged media, Inferno creates a link that references the media at its original location, so that no media is actually imported and duplicated in Autodesk storage. Later, when you move or delete clips with unmanaged media, you affect only the reference—not the actual media at the shared storage location. You can also use this method when importing files using a sequence or recapturing media.

When using clips with unmanaged media, be aware that you have no control if another user modifies the original media files. All clips with unmanaged media in Autodesk storage that refer to the modified media files are updated to reflect the change. Ensure that all users accessing the same media are aware of these implications.

You can process (render) clips with unmanaged media in any module. The resulting new clip is written to Autodesk storage. Since any links to the original media are now removed, changes to this media will no longer be updated on the shared storage. Also, if the media changes in the original external location, the changes are not reflected in your stored clip. This can be useful if you want to prevent other users with access to the shared storage from altering your clip.

Managing How Media is Imported

In the Import Settings menu of the Basic menu, use Store Local Copy to toggle between creating a clip with managed media and a clip with unmanaged media.

In both cases, the path to the original media is stored in the clips metadata, allowing the user to toggle between managed and unmanaged media, or even re-importing the media altogether.

You can change how a clip is stored after importing it using the Store tool.

To create clips with managed media:

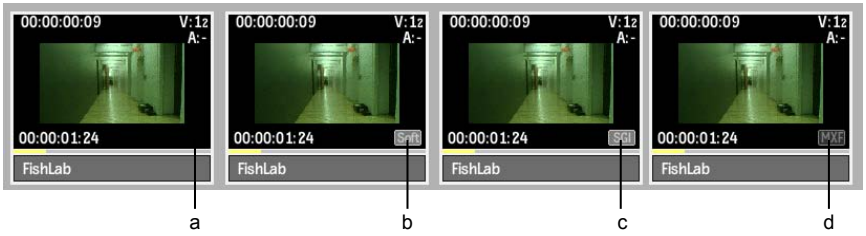
1. Open the Clip Library menu.
2. Click Basic.
3. Enable Store Local Copy.

To create clips with unmanaged media:

1. Open the Clip Library menu.
2. Click Basic.
3. Disable Store Local Copy.

Recognizing Types of Clips

Soft-imported clips on the Desktop or in the clip library display an icon in the lower-right corner of the clip. The type of indicator depends on whether the clip is a hybrid clip or a completely soft-imported clip. A hybrid clip has some frames that are soft-imported and some that are not, or it may consist of multiple soft-imported sources of different formats.



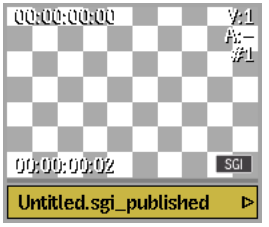
(a) Indicates a clip with managed media but without clip history (b) Indicates a clip with unmanaged media of mixed formats (c) Indicates a clip with unmanaged media (d) Indicates a clip with managed media and import history

Lower-Right indicator:	Type of clip:	Import history:
None	A clip with managed media, without clip history. The clip can be a result from a processing, a Inferno-generated clip (Create Colour Source) or imported through the Import Image menu.	No import history
SOFT	The clip contains unmanaged media from multiple sources. The source clips were imported using the Import Image menu or from a Gateway library.	Import history available if imported from a Gateway library.
Light-grey format	The clip contains unmanaged media of the type specified by the light-grey indicator. The source clip was imported through the Import Image menu or from a Gateway library.	Import history available if imported from a Gateway library.

Lower-Right indicator:	Type of clip:	Import history:
Dark-grey format	The clip contains managed media, and the original file was of the type specified by the dark-grey indicator. The source clip was was imported form a Gateway library.	Yes

Alt-clicking the clip displays an overlay that includes the soft-imported source path.

A clip with un managed media that can no longer connect to its source file appear as white and grey checkered clips.



About Proxies

A proxy is a low-resolution copy of a high-resolution image. There can be one proxy for each frame in a clip. Proxies are used to provide real-time playback of processed results and to supply low-resolution clips so that composites can be quickly created. When importing files as clips with unmanaged media, use proxies when the connection speed to your media storage location is not fast enough to support real-time streaming of the full-resolution media. When proxy generation is enabled, proxies are generated upon import of media.

You can specify how and when proxies are generated for clips in the project settings. If you enabled proxies for the project or media resolution, then only the proxies reside in Autodesk storage.

NOTE When creating DPX or Cineon images, a film scanner generates full-resolution scans that are often complemented by lower resolution proxy scans. These proxy scans can also be used in Inferno, but must be imported using the Import Image menu.

Importing Media from a Gateway Library

Import graphic image files, film scans, or video files by dragging them from a Gateway library and dropping them into a local library, or loading them directly to the Desktop. This allows you to import images from USB and Firewire® drives, NAS or SAN, or from any volume connected to a workstation.

NOTE Clips imported through a gateway are at source resolution.

Compared to Import Image, a Gateway library import is faster and more intuitive. It allows you to start working immediately with the imported media, without having to wait for the import to complete. But, there are some things that you cannot do using a Gateway library:

- Resizing on import. You should resize clips after the import.
- Applying LUT on import. You should apply 1D and 3D LUT manually after the import.
- Importing external proxies for DPX sequences. You have to use the Import Image menu.
- Importing PSD files. You have to use the Import Image menu.

To import media using drag & drop:

1. Open the Clip Library menu.

   In the Main menu, click Library.

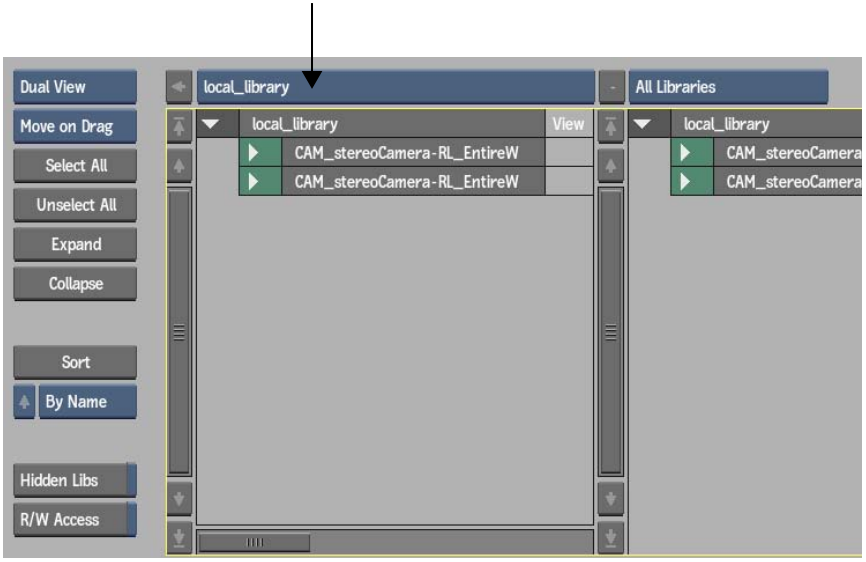
2. Set the Library Mode box to Dual View.

   The Library View Mode button is located in the top-left of the Clip Library menu.

3. Using the Network menu, connect to a Gateway library.

   Open the Network menu, and use the Gateway to select the directories which contain the clips to import.

4. In one view, open the local clip library to where you want to import the clips. Use the Clip Library box.



5 In the other view, open the Gateway library from where you want to import the clips. Use the Clip Library box.

6 In the Gateway library view, navigate to the directory that contains the clips you want to import.

7 Optional: Set the Gateway library import options. See Gateway Library Global Import Options on page 153.

NOTE If you do not edit the Gateway library import options, Inferno imports the files using the active rule. See Managing Import Settings and Rules on page 154.

8 Drag and drop the media from the Gateway library to the local library. You can also drag and drop:

- Multiple clips at the same time, mixing different resolutions, timings and even formats. The import settings used for each file depend on the active rule for each format. See Managing Import Settings and Rules on page 154.
- A directory. Enable Import Sub-Directories to also import its sub-directories. See Gateway Library Global Import Options on page 153.

TIP Use the Preview Panel to set in and out markers to import a segment of a media.

Importing Specific Tracks From Multi-Track Files

In a Gateway library, files with multiple tracks (audio and video) display the multi-track indicator and are considered container clips. OpenEXR, XDCAM, and Quicktime files with audio are examples of such files. You can expand those clips to reveal the underlying tracks to import specifc ones.

To import specific tracks from a multi-track file using drag & drop:

1. Open the Clip Library menu.

   In the Main menu, click Library.

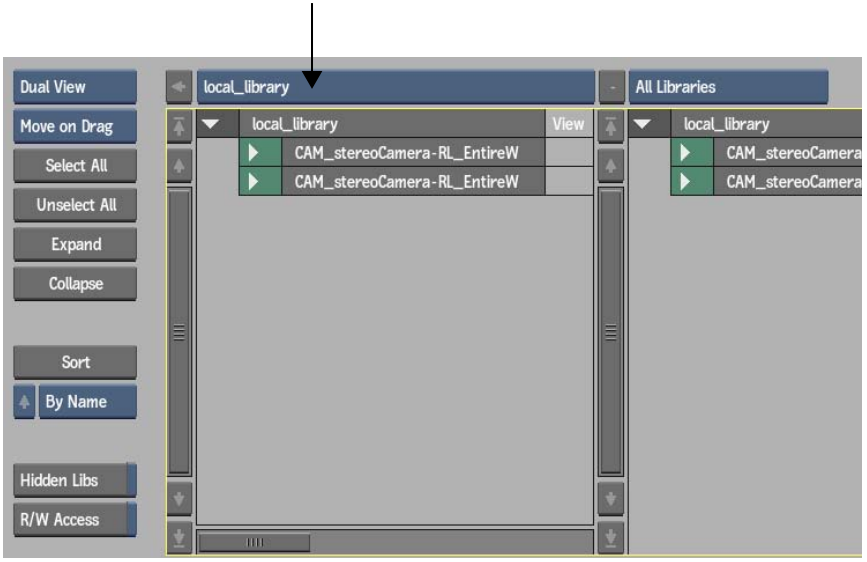
2. Set the Library Mode box to Dual View.

   The Library View Mode button is located in the top-left of the Clip Library menu.

3. Using the Network menu, connect to a Gateway library.

   Open the Network menu, and use the Gateway to select the directories which contain the clips to import.

4. In one view, open the local clip library to where you want to import the clips. Use the Clip Library box.



5 In the other view, open the Gateway library from where you want to import the clips. Use the Clip Library box.

6 In the Gateway library view, navigate to the directory that contains the clips you want to import.

7 Optional: Set the Gateway library import options. See Gateway Library Global Import Options on page 153.

NOTE If you do not edit the Gateway library import options, Inferno imports the files using the active rule. See Managing Import Settings and Rules on page 154.

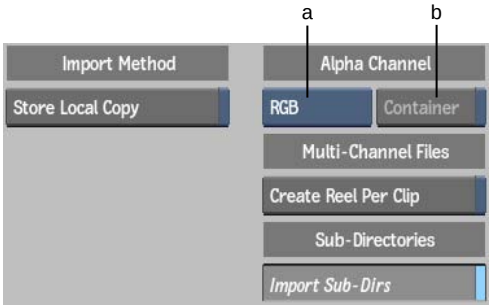
8 Double-click the multi-track indicator of the clip which contains the tracks to import; this expands the tracks cointained within the clip.

9 Drag and drop the track (or tracks) from the Gateway library to the local library. The tracks are imported using the settings which applies to the clip's format.

Gateway Library Global Import Options

Four global options manage how all clips are created when media is imported. These options are not attached to any format import options, and are either on or off for all import operations performed using a Gateway library.

You access the Gateway library import options through the Basic menu in the library menu.



(a) RGB Import box (b) Container button

Store Local Copy button Enable to copy the media to the local Inferno storage. It transcodes a clip to Inferno-native media format. This option ensures that Inferno is the sole owner of the media, preventing the media from being modified by an external source; it can also provide better playback performances, depending on your setup.

In the Clip Library menu, Inferno displays the media from the original file while waiting for the transcoding to finish. It also overlays *Pending Render* on the slate of clips with frames not yet transcoded.

NOTE Clips also display *Pending Render* if low-resolution proxies are being generated. In this case, the *Pending Render* overlay is displayed in the Player, the Clip Library, and the Desktop.

Disable Store Local Copy to create a link to the media of the imported clip; the media is not copied to the local Inferno storage. There is no transcoding, as Inferno decodes the clip as required.

Create Reel Per Clip button Enable this option to create a reel for each multi-channel clip imported (including RGB+alpha clips). The created reel is named after the multi-channel clip, and contains all the individual channels that make up the imported clip. Disable this option to import all the channels where you drag the multi-channel clip. Only applied when importing multi-channel video clips.

RGB Import box Select RGB to only import the RGB portion of RGB+alpha clips. Select RGBA to import both RGB and Alpha channels of an RGB+alpha channel clip.

Container button Enable to create a matte container when importing a clip with an alpha channel. A matte container is a multi-track clip with the RGB portion of an clip on one track and its matte on another.

Import Sub-Directories button Enable to import both the clips and the sub-directories of an imported directory. Disable to only import the clips of a directory and exclude its sub-directories.

NOTE Be careful when importing directories and enabling the Import Sub-Dirs option: Inferno imports recursively. This means that Inferno scans and imports the contents of the sub-directories, and if these sub-directories themselves contain sub-directories, it imports those sub-directories. If the directory structure is very deep and complex, the import can result in a saturation of your network and storage.

Managing Import Settings and Rules

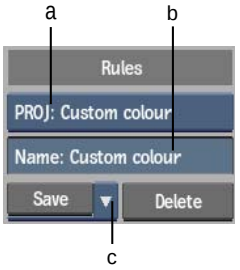
Import Settings define how files are imported into Inferno. When importing a file from a Gateway library, the current settings for that file type sets the import options used. Sets of preferences for a file format can be saved as a rule, to quickly reload or switch between preferences. Import and Import History use the same settings. See Import and Clip History Settings on page 159.

Inferno contains a default rule for each supported format. You can create as many rules as you want for each format, but only one rule can be active for each format.

Use the Rules section to:

- Create a rule.
- Edit a rule.
- Set a rule as the active import settings.

■ Delete a rule.



(a) Active Rule box
(b) Rule Name field
(c) Save dropdown list

To create a rule:

1. In the Basic menu, open the Import Settings menu.
2. Using the Format box, select the format for which you want to create a rule.
3. Enter the name of the new rule in the Rule Name field.
4. Edit the rule settings.
   An asterisk indicates the rule contains unsaved changes.
5. Click the Save dropdown list and select one of the following options.

Select:	To save the rule:
Save in Project	In the project directory. This rule becomes available to anyone who uses the current project and is identified with the PROJ prefix in the Active Rule box.
Save with User	With the user profile. This rule becomes only available to the current user and is identified with the USER prefix in the Active Rule box.

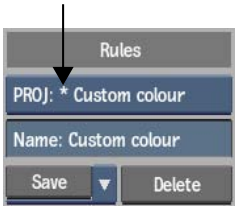
To edit a rule:

1. In the Basic menu, open the Import Settings menu.
2. Using the Format box, select the format for which you want to edit a rule.

3 From the Active Rule menu, select the rule to edit.

4 Edit the rule settings.

An asterisk indicates the rule contains unsaved changes.



5 Click Save.

To set a rule as the active import settings:

1 In the Basic menu, open the Import Settings menu.

2 Using the Format box, select the format for which you want to set the active rule.

NOTE You can also access the Import Settings of a format by double-clicking a clip in a Gateway library.

3 From the Active Rule menu, select the rule to use as the default.

Files of the selected format that you import by drag & drop are now processed using the selected rule.

To delete a rule:

1 In the Basic menu, open the Import Settings menu.

2 Using the Format box, select the format for which you want to delete a rule.

3 From the Active Rule menu, select the rule to delete.

4 Click Delete.

Managing Import History

Import History allows you to view and edit the settings used to import a file into Inferno, for a specific instance of a clip.

Only clips imported from a Gateway library have an import history; you cannot review or edit import settings of clips imported through the Import Image or Import EDL menus. Import and Import History use the same settings. See Import and Clip History Settings on page 159.

TIP To change the media status of a clip, either from unmanaged media to managed, or managed to unmanaged, use the Store Media menu located in the Tools menu. See To change the media management option of clips: on page 158.

To view the import settings used to import a clip:

1. Open the clip library containing the clip to review.
2. Open the Basic menu.
3. Double-click the clip.

   The Import History menu opens and displays the options used to import the clip.

   NOTE Only clips imported from a Gateway library have an import history.

To modify the import settings of a clip:

1. Open the clip library containing the clip to edit.
2. Double-click the clip to modify. The Import History menu displays the options used to import the clip.

   NOTE Only clips imported from a Gateway library have an import history that you can modify.

3. Set the History box to Modify Clip History.

4. Edit the settings as needed.

5 Click Apply Changes.

To modify the import settings multiple clips simultaneously:

1 Open the clip library containing the clips to edit.

2 Open the Basic menu.

3 Double-click one of the clips to modify. The Import History menu displays the options used to import the clip.

4 Select the other clips of the same format using **Ctrl**-click.

NOTE Only clips imported from a Gateway library have an import history that you can modify.

5 Set the History box to Modify Clip History.

6 Edit the settings as needed.

NOTE All the settings will be applied to the selected clips, not just the ones modified.

7 Click Apply Changes.

To change the media management option of clips:

1 Open the clip library containing the clips to edit.

2 Select one or more clips to modify. Use **Ctrl**-click to select multiple clips.

3 Open the Store Media menu, under the Tools menu.

4 Do one of the following:

- To create local, managed media for all of the selected clips, including complex clips such as sequences, click Store Local Copy.
- To have clips reference the original media files they were imported from, and delete unused managed media, click Unstore Copy.

Import and Clip History Settings

Use the Import Settings to define how files are imported in Inferno. See Managing Import Settings and Rules on page 154.

The Import History mirror the settings used to import a file from a Gateway library. See Managing Import History on page 156.

NOTE For every format, only a subset of the described options is available.

RED-Only Settings

RED media files can be processed a number of ways at the time of the import. Use the Debayering, Colour, Image, Gain and Curve settings to modify the look or the size of the imported media.

RED clips are 16-bit, but Inferno down converts them to 12-bit to optimize graphics processing.

NOTE RED files require a lot of computing resources to process. You can use the Preview panel to set In and Out points on imported clips to minimize the transcoding of extraneous material.

Debayering

Debayering Select the level of quality required from the debayering algorithm. Higher resolutions are significantly more processing intensive.

The debayering setting is the most resource-intensive setting. Try using the level of debayering the most appropriate for your work.

Detail Select the level of detail extraction required.

OLPF Compensation Select the level of Optical Low Pass Filter (OLPF) compensation to use. OLPF is a type of sharpening used to compensate for the optical anti-aliasing filter, which can induce softening of the image during recording.

Denoise Select the level of noise reduction applied to the debayered clip.

Colour

Colour Settings Select how Inferno uses the colour information stored within a R3D file.

Select:	To have:
User	Inferno import RED clips using the options you set in the Image, Gain, and Curve menus.
Camera	Inferno 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.
RSX	Inferno import RED clips using the RSX look created in RED Alert!. The RSX file of a clip must reside in the same folder 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.
RSX or Camera	Inferno import RED clips using the RSX look. If a clip has no RSX file, Inferno imports it using the camera settings. Disables the Image, Gain, and Curve menu options.
RSX or User	Inferno imports RED clips using the RSX look. If a clip has no RSX file, Inferno 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 Image, Gain, and Curve menus editable in the Clip History. 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 codec to use. Using the version 3 of the codec 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.

NOTE As a rule, always set Colour Science to Codec 3.x, unless you are working with a file shot using a RED camera with firmware 30 which, and that file was imported in Inferno prior to version 2011.

Colour Space box Set the color space of the imported clips.

Gamma Curve box Set the value of the output gamma curve that is applied to the imported clips.

Image

ISO Select the value of the linear gain operation.

Saturation Set the saturation value.

DRX Set the Dynamix Range Extension, which sets how much pixel data is copied from non-saturated channels into saturated channels.

Tint Set the tint value.

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.

Exposure Set the exposure value, an equivalent to f-stops.

Brightness Set the brightness value.

Contrast Set the contrast value.

Kelvin Set the perceptual color temperature of the image, in Kelvin.

Shadow Set the Shadow level.

Gain

Use this menu to set the RGB Gain for RED clips.

NOTE We recommend that you do not change the default settings unless you have prior experience with color management.

Curve

Use this menu to set the Colour curve for RED clips.

NOTE We recommend that you do not change the default settings unless you have prior experience with color management.

Metadata Options

The options described below are not used by every format. Only the ones relevant to the selected format are accessible.

Tape Name box Select an option to determine how the tape name is set for the imported clips.

Select:	To:
Enter Tape	Activate the Tape Name field so that you can manually enter the tape name. When selecting multiple sequences for import, this tape name is used for all imported clips.
Tape From File Name	Use the name of the imported file as the tape name.
Tape From Directory	Determine the tape name from the detected directory structure. A Level field appears below the Tape Name box. Use this to configure the relative path to the directory from which the tape name can be determined.
Tape From File Header	Read the tape name from the header of the imported file.

Level field Select an option to set the relative file path to the directory from which the tape name can be determined. Although it is available for all file types, the Level field is specifically designed to work with the directory structures that are output by film scanners. A typical file structure would look like: *./<tape>/<resolution>/clip.######.dpx*. In this case, selecting *Up 2 Levels* 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 of the imported clip. Enabled if Tape Name is set to Enter Tape.

Framerate Selection box Select how the framerate is determined. If you choose Select Edit Rate, select an option from Framerate.

Framerate box Select the frame rate of the imported clip. Enabled if Framerate Selection is set to Select Edit Rate.

Essence button Enable to browse the actual directory structure of P2 or XDCAM media. This option allows you to import specific audio or video files contained within a P2 or XDCAM directory structure.

TIP After toggling Essence, click Refresh Selected to update the P2 or XDCAM directory structure displayed in the Library.

Timecode box Select an option to specify how the timecode information for the clip is set.

Select:	To:
Enter Timecode	Manually enter a timecode in the Timecode field.
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 filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Timecode field Set the start timecode of the imported clip. Editable if the Timecode box is set to Enter Timecode.

Clip Naming Options

The options described below are not used by every format. Only the ones relevant to the selected format are accessible.

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 sequences for import, this name is used for all imported clips.

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.

Clip Names box (OpenEXR-only) Select how the channels of a multi-channel OpenEXR file are named when they are imported.

Select:	To:
File Name	Use the file name of the container for all the imported channels.
Channel	Use the channel name as the imported clip name.
File Name + Channel	Combine, in this order, the file name of the container and the channel name into the imported clip name.
Channel + File Name	Combine, in this order, the channel name and the file name of the container into the imported clip name.

Keycode Options

The KeyKode menu is available to the DPX format.

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 to the imported clip.

Select:	To:
File Header Keycode	Use the keycode information embedded in the image file header.
No Keycode	Not use keycode.

Keycode Fcm box Select the frame code mode of the tape. Set to File FCM to read from the file the frame code mode.

Film Gauge box Select a film gauge for the keycode.

Image Options

The Image menu is available to the following formats:

- P2
- XDCAM

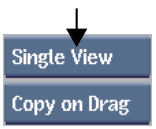
Scale to Full HD button Enable Scale to Full HD to have media with a 1280x1080 or 1440x1080 resolution appear in the Player at a standard 1920x1080 resolution. Disable Scale to Full HD to display the media at its native (1280x1080 or 1440x1080) resolution in the Player.

Importing a Sequence from a Gateway Library

Drag & drop XML and AAF files linking to file-based material to import a sequence and related sources. To import an EDL, or capture an XML or AAF containing tape-based material, see the Conform section.

To import an XML or an AAF sequence:

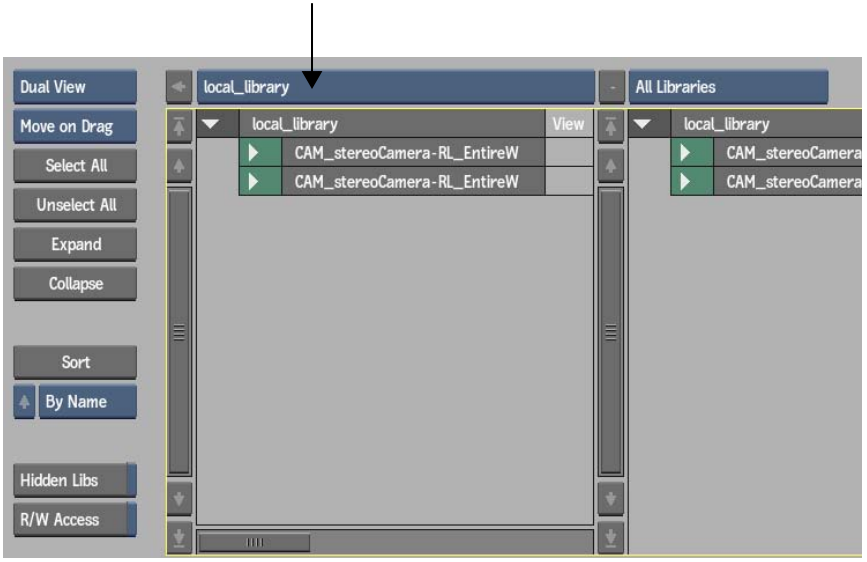
1. Open the Clip Library menu.
   In the Main menu, click Library.
2. Set the Library Mode box to Dual View.
   The Library View Mode button is located in the top-left of the Clip Library menu.



3. Using the Network menu, connect to a Gateway library.

Open the Network menu, and use the Gateway to select the directories which contain the sequence to import. A Gateway appears under a workstation name, just like a Framestore.

4 In one view, open the local clip library to where you want to import the sequence. Use the Clip Library box.



5 In the other view, open the Gateway library from where you want to import the sequence. Use the other Clip Library box.

6 In the Gateway library view, navigate to the directory that contains the sequence you want to import.

7 Drag and drop the sequence from the Gateway library to the local library.

As soon as you drop the timeline in the destination library, Inferno processes the timeline according to the import sequence settings defined in the FCP XML/AAF menu.

Relinking a Sequence to its Media

If some media files were not found according to the settings defined in the FCP XML/AAF menu, you can relink the source files to the sequence.

To relink a sequence to files located in a Gateway library:

1 Open the Clip Library menu.

In the Main menu, click Library.

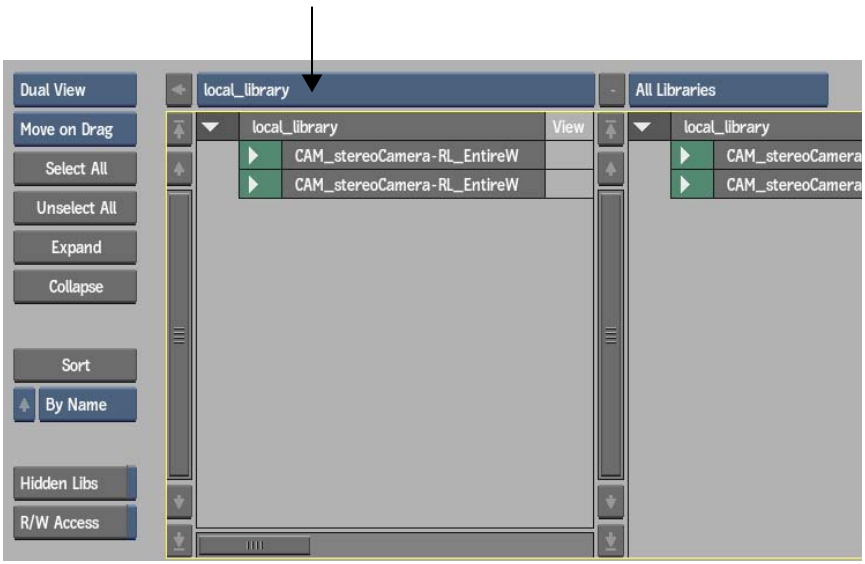
2 Set the Library Mode box to Dual View.

The Library View Mode button is located in the top-left of the Clip Library menu.

3 Using the Network menu, connect to the Gateway library which contains the sources to relink.

Open the Network menu, and use the Gateway to select the directories which contain the sequence to import. A Gateway appears under a workstation name, just like a Framestore.

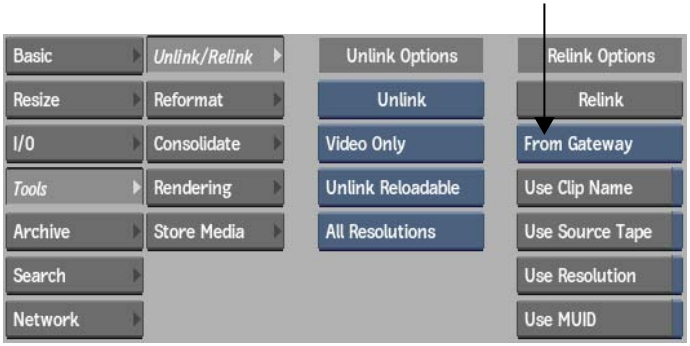
4 In one view, open the local clip library which contains the sequence to relink. Use the Clip Library box.



5 In the other view, open the Gateway library which contains the sources to relink. Use the other Clip Library box.

6 Open the Unlink/Relink menu from the Tools menu.

7 Select From Gateway from the Relink Options Source box.

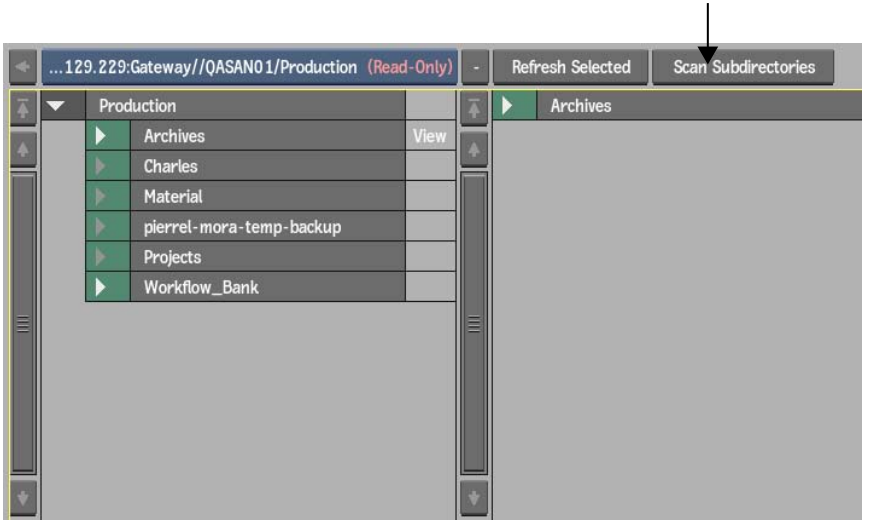


8 Optional: Enable the Relink options as required.

Enable:	To:
Use Clip Name	Use the file name sepcified in the sequence as a match criteria.
Use Source Tape	Use the tape/source name specified in the sequence as a match criteria.
Use Resolution	Use the resolution specified in the sequence as a match criteria. If this option is disabled, Inferno soft-resizes the media to the resolution specified in the imported sequence, if required.
Use MUID	Use the starting SMPTE MUID in the timeline as a match criteria. This is only used with MXF files and is ignored in any other case.

9 In the Gateway library view, navigate to the directory which contains the sources you want to relink. Make sure sources to relink are visible in the Gateway library. The Relink tool tries to relink only to displayed media files.

10 Optional: If the sources are located in sub-directories, click Scan Sub-Directories. This flattens the directory structure and makes all the sources visible to the relink tool.



11 From the clip library, select the sequence to relink.

12 Click Relink.

The application scans all the visible media and asks you to confirm the relink operation.

Defining FCP XML / AAF Settings

You define sequence import settings like you do for media files, with the exception that there is only one rule for all XML and AAF files.

To set the import settings for AAF and XML sequences:

1 Open the Clip Library menu.

In the Main menu, click Library.

2 Open the Basic Options menu.

3 Open the FCP XML / AAF menu.

4 Edit the Metadata and Media Options menus; changes are automatically saved.

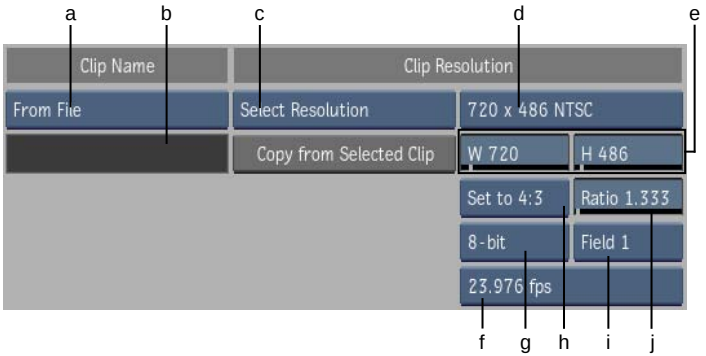
You do not set the import settings for each of the media referred to in the imported sequence. The media linked to each imported sequence is

processed and imported according to the applicable active import rule. See Importing Media from a Gateway Library on page 149.

FCP XML / AAF Import Settings

Use the FCP XML / AAF Import Settings to define how files are imported in Inferno. See Defining FCP XML / AAF Settings on page 169.

Metadata Options Menu



(a) Clip Name box (b) Clip Name field (c) Resolution box (d) Resolution Presets box (e) Width and Height fields (f) Framerate box (g) Bit Depth box (h) Aspect Ratio Presets box (i) Scan Mode box (j) Aspect Ratio field

Clip Name box Select From File to use the sequence name read from the file. Select Enter Name to rename the imported sequence using the name entered in Sequence Name field.

Clip Name field Enter the name to which the sequence is renamed when imported. Available if Timeline Name is set to Enter Name.

Resolution box Select From File to use the sequence resolution defined in the file. Select Select Resolution to override the resolution defined in the file and reformat it using the customized settings.

NOTE The Resolution setting only affects the resolution of the sequence; it does not resize the linked sources.

Copy from Selected Clip button Use to copy the formatting information of a selected clip into the Resolution parameters. Available when the Resolution box is set to Select Resolution.

Resolution Presets box Select one of many standard resolutions, as well as a Custom option you can use to specify non-standard resolutions.

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.

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.

Bit Depth box Select from one of five frame depth options: 8-bit, 10-bit, 12-bit, 12-bit u, or 16-bit fp.

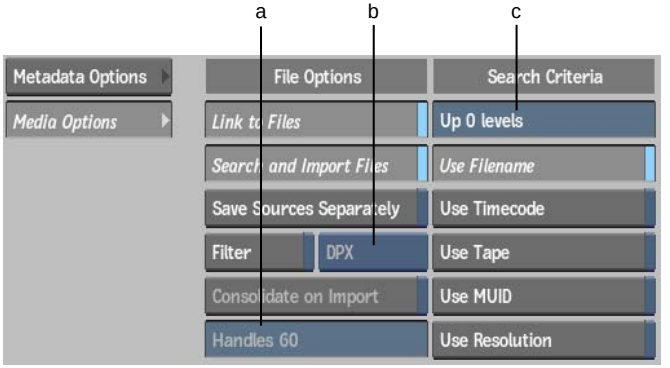
Aspect Ratio Presets box Select a standard frame aspect ratio. Select the Set to w:h option to set the clip to use square pixels. Select Custom to define a custom frame aspect ratio in the Aspect Ratio field.

Aspect Ratio field Displays the aspect ratio 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 set the order in which the fields of interlaced material is 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.

Media Options Menu



(a) Handles field (b) Filter Selection box (c) Directories Up field

Link to Files button Enable Link to Files to create a sequence with track segments that link to the original media. Disable Link to Files to create an empty shell with only the structure of the sequence, without references to files.

Search and Import Files button Enable to locate and import the media listed in the sequence, using the selected Search Criteria options. The media is imported as segments of the sequence. Enabled if Link to Files is enabled.

Save Sources Separately button Enable to create a copy of each source referred to in the sequence. The source clips are copied next to the sequence in the clip library. Enabled if Search and Import Files is enabled.

Filter button Enable to search and import only the clips of the format specified in Filter Selection. Enabled only if Search and Import Files is enabled.

Filter Selection box Select the format to filter for during a search and import operation.

Consolidate on Import button Enable to force Inferno to override the handles specified in the timeline. Disabled if Search and Import Files is enabled.

Handles field Disabled if Consolidate on Import is enabled. Set the maximum number of head and tail frames that you want to retain after consolidating the clip.

Directories Up field By default, Inferno searches for media to match, starting with the directory from where the timeline file is imported. It includes any sub-directory in this search. Use Directories Up to expand the search to parent directories. How high in the hierarchy depends on the value set.

NOTE When setting the Directories Up field, keep in mind that the Inferno will navigate the whole directory structure starting with what you specified. This means that the higher up you go in the directory structure, the longer the conform will take. And this issue is amplified in a networked environment.

Use Filename button Enable to use the filename specified in the timeline as a match criteria.

Use Timecode button Enable to use the source timecode specified in the timeline as a match criteria.

Use Tape button Enable to use the tape name specified in the timeline as a match criteria.

Use MUID button Enable to use the starting SMPTE MUID in the timeline as a match criteria. This is only used with MXF files and is ignored in any other case.

Use Resolution button Enable to use the resolution specified in the timeline as a match criteria. If this option is disabled, Inferno soft-resizes the media found to the resolution specified in the imported sequence, if required.

Clip Library

10

Accessing Gateway Libraries

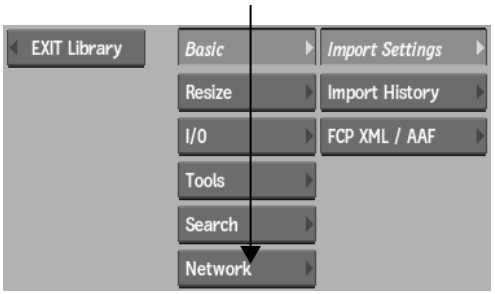
Using a Gateway library, you can access media files residing on local or remote storage. When you want to use media from a source external to an Autodesk Visual Effects, Finishing and Grading application, drag and drop it from a Gateway libray to a library belonging to the current project. See Importing Files Using A Gateway Library on page 145.

With a Gateway library, browse and import the media you need for your project. Access and view any media on any local or remote storage, from USB drives to SAN systems. All available volumes are listed in the Network panel, similar to how you view files in the operating system's file browser.

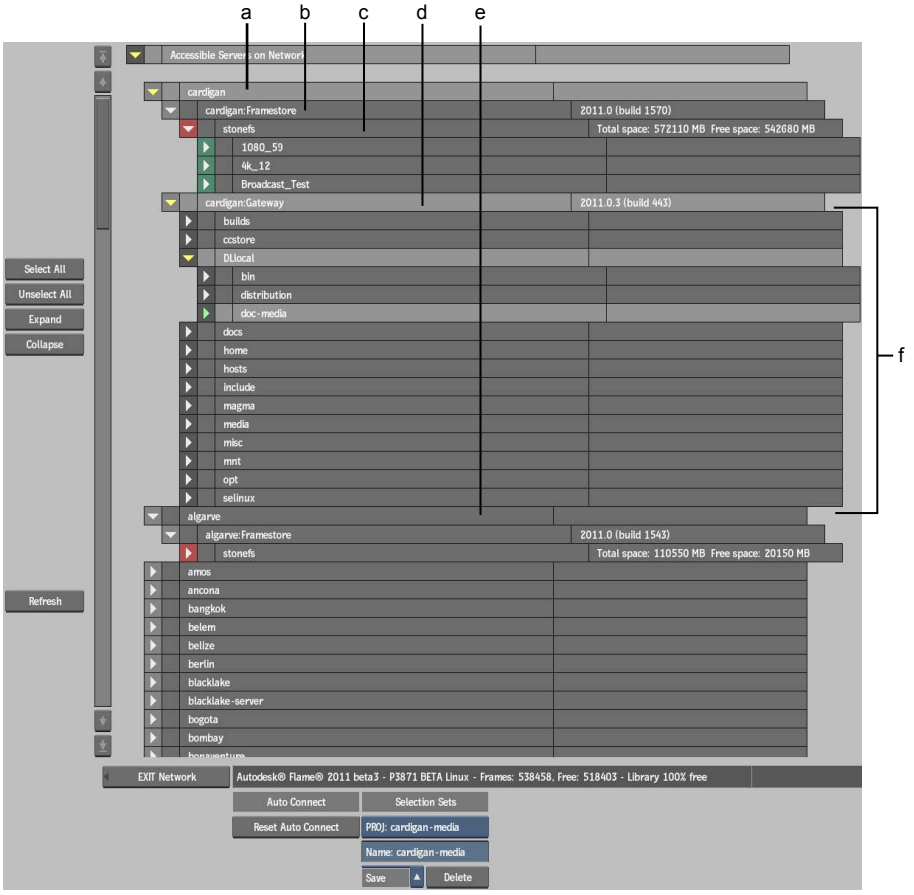
Gateway libraries differ from Network libraries in that they display the entire filesystem on each workstation, whereas Network libraries display the content of attached framestores.

To open a Gateway library:

1. In the Main menu, click Library.
2. In the Clip Library menu, click Network.



The network panel appears.



(a) Local system (b) Framestores available on the system (c) Available framestore and its projects (d) Gateway library (e) Remote system (f) Directories available through the Gateway library

The local system is listed at the top of the network library. Remote systems follow in alphabetical order.

The network panel controls are described as follows.

Select All and Unselect All buttons Click Select All to select all entries in the network library. Click Unselect All to unselect all entries.

Expand and Collapse buttons Click Expand to expand selected entries. Click Collapse to collapse selected entries.

TIP You can also click the Expand and Collapse arrows at the left of an entry to expand and collapse it.

Refresh button Updates the list of systems by probing all systems in the list to see if they are still accessible, and selected systems only to check if the directory hierarchy changed.

NOTE Systems are automatically refreshed when you expand an entry.

Reset Auto Connect button Disconnects all entries set for auto-connection.

Selection Sets Use selection sets to create and access network location bookmarks.

Gateway Library Structure

A Gateway library displays the structure of the filesystem, while only displaying the directories in the Network panel.

In the Network panel, every workstation with an enabled Wiretap Gateway server displays its filesystem under the Gateway header.

In addition, a Gateway server running on a MacOS X workstation displays *RED ROCKET* if it is equipped with a RED ROCKET™ decoding card.

Selecting Gateway Libraries in Local or Remote Workstations

To make a Gateway library in a local or remote workstation accessible, select it in the network menu. You can make local or remote gateway libraries available for the current work session only, or each time you load the application.

You can also use selection sets. See Using Selection Sets on page 179.

To select a Gateway library in a local or remote workstation:

1. Display the Network Library menu.
2. Click Refresh.

   All accessible workstations are searched for Gateway libraries.
3. Expand workstations as needed to see the Gateway libraries; there is one Gateway library per workstation.

   Each workstation displays both Framestore and Gateway entries.
4. Expand the Gateway library to display the directory structure of the selected workstation.
5. Select the directories you want to access. The procedure for selecting entries is the same as for clip libraries.

   Selected entries are light grey.
6. Optional: To automatically reconnect to a directory in a Gateway library when you restart the application, click the grey box to the left of the entry.

   The letters AC appear in the box, indicating the entry is set to be reconnected. All subentries are also set. When you restart the application and load the project, the specified entries are connected and the associated Gateway library directory is available in the Clip Library box.

   NOTE A green AC indicates that the entry and all subentries are set for auto-connection; a yellow AC indicates that some, but not all subentries, are set for auto-connection.
7. Optional: To stop automatically reconnecting to the directory associated with an entry, click the grey box next to the entry again. To disconnect all entries set for auto-connection, click Reset Auto Connect.

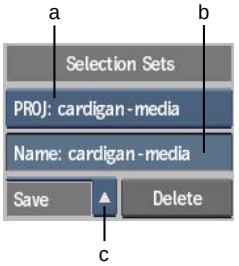
   The AC indicator disappears.
8. Exit the Network Library menu.

   All the Gateway libraries you selected in the Network menu are listed in the Clip Library box. All clip libraries set for auto-connection will be available when you restart the application.

NOTE

Using Selection Sets

Use selection sets to bookmark Gateway directories. You can then use selection sets to rapidly select locations to connect to.



(a) Selection Set box (b) Selection Set Name field (c) Save dropdown list

To use a selection set:

1. Open the Network menu.
2. Select a selection set from the Selection Set box.

   The locations bookmarked in the selection set are automatically expanded and selected.
3. Optional: Select+click additional directories to connect to them.
4. Exit the Network Library menu.

   All the Gateway libraries you selected in the Network Library menu are listed in the Clip Library box.

To create a selection set:

1. In the Network menu, navigate a Gateway library to the directory you wish to bookmark.
2. Enter the name of the selection set in the Selection Set Name field.

3 Click the Save dropdown list and select one of the following options.

Select:	To save the rule:
Save in Project	In the project directory. This selection set is available to anyone who uses the current project and is identified with the PROJ prefix in the Selection Set box.
Save with User	With the user profile. This selection set is only available to the current user and is identified with the USER prefix in the Selection Set box.

To update a selection set:

1 In the Network menu, select a selection set using the Selection Set box.

2 Navigate a Gateway library to the directory you wish to bookmark.

3 Click Save to overwrite the original selection set.

Playing Clips from the Library

You can play a selection of source clips and processed clips directly from the library. Or use the Player to view the clips at full resolution.

To view a clip in the preview panel:

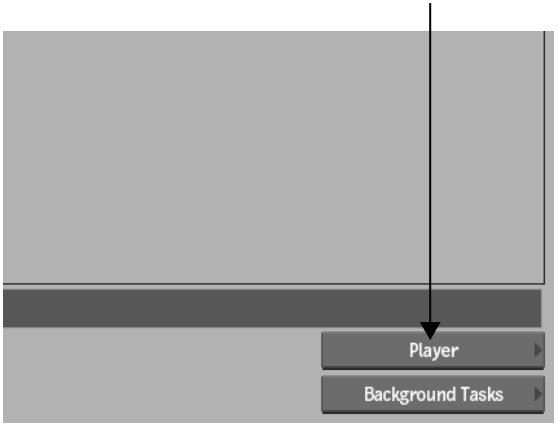
1 Select a clip in the library.

2 Swipe the cursor against the swipe bar on the right side of the library to display the clip in the preview panel.

The preview panel displays clip information. You can set in and out points in the displayed clip.

To play clips in the Player:

1 Select the clips that you want to play and then click Go To Player or press **Esc**.



All selected clips are brought into the Player.

2 To go from one clip to the next, press **Ctrl+ right or left arrow**, or select a clip from the Playback box.

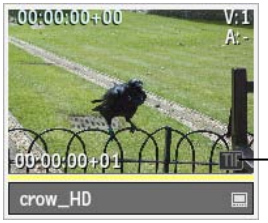
3 Click EXIT Play or press **Esc** to exit back to the library.

Managing Media Using the Store Tool

Use the Store tool to manage how media are stored by Inferno.

Store media on the framestore to ensure that Inferno is the sole owner of the media and prevents it from being modified by an outside source; it can also provide better playback performances, depending on your setup. Storing a clip converts it from a clip with unmanaged media to one with managed media.

Unstore media from the framestore to allow the substitution of sources, or to easily perform a relink operation. Unstoring a clip converts it from a clip with managed media to one with unmanaged media.



(a) Source Format indicator marking a clip with managed media and import history

To store media to the framestore:

1. In the clip library, select the clip or clips that you want to write to the framestore.
2. Click Tools and then Store Media.
3. Click Store Local Copy.

   The media of each selected clip is copied to the framestore. The clips that you selected become clips with managed media. If more than one instance of a clip with unmanaged media is present in the library, every instance of that clip changes to a clip with managed media.

   While the media is being transcoded to the framestore, Inferno marks which frames it has not yet imported using the *Pending Render* overlay. Inferno also displays the unmanaged media frames until the *Pending Render*-overlaid frames are completely transcoded.

To unstore media from the framestore:

1. In the clip library, select the clip or clips that you want to write to the framestore.
2. Click Tools and then Store Media.
3. Click Unstore Copy.

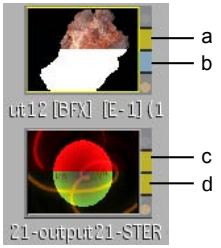
   The clip now displays the unmanaged media. If the media referred to by the clip is no longer available, the Player displays a checkered frame. But the clip proxies are still available, and can be displayed in the Player. If more than one instance of a clip with unmanaged media is present in the library, every instance of that clip changes to a clip with unmanaged media.

Batch

11

Clips

Batch clips can now contain video tracks or stereo tracks. Each type supports dual output. Clips with video tracks have RGB and matte outputs. Clips with stereo tracks have left eye and right eye outputs corresponding to the left eye and right eye layers of its stereo track.



(a) RGB output (b) Matte output (c) Left eye (layer) output (d) Right eye (layer) output

Duplicating Batch Nodes

New: The hotkey for creating a duplicate stereo group node when dragging a node from the node bin is now **Shift+S** instead of **Shift+W**. However, you can still use **Shift+W** to create a Duplicate link between two nodes when dragging from one node to another in Move mode.

You can now keep two nodes in the schematic in sync by creating a Duplicate link between them. When you change the settings of one node, they are automatically duplicated on the other node.

You can use Duplicate links for nodes connected to mono clips or to clips containing stereo tracks. For stereo tracks, any changes made with a linked node to the layer of one eye are automatically duplicated on the other eye. For mono clips, any changes made with a linked node to one clip are automatically duplicated on the other clips.

To duplicate 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. As well, for some nodes, the sources must share the same resolution and/or bit depth. If duplicated nodes do not support sources of differing resolutions and/or bit depth, an error message appears in the message bar.

Note that the following nodes cannot be duplicated.

■ Action	■ Modular Keyer
■ Coloured Frame	■ Paint
■ Distort	■ Substance Noise
■ Keyer	■ Text

When you create a Duplicate link between nodes, the original node becomes the master. You can modify either the master node or the duplicated node and the other node is simultaneously updated. However, only the channels of the master node are displayed in the Channel Editor since the channels of the duplicated node are identical to those of the master. If you break the Duplicate link, the channels of both nodes appear. However, the values of the duplicated node are not restored. They keep the values of the master node.

If you copy duplicate nodes, one of the following occurs depending on whether you copied the master node or the duplicated node:

- If you copy the master node, only the master node is copied.
- If you copy the duplicated node, only the duplicated node is copied, but the Duplicate link is recreated between the copy and the master.
- If you copy the master and duplicate nodes, both nodes are copied.

When using duplicate nodes with mono clips, you must create the Duplicate links manually. When using duplicate nodes with a clip containing a stereo

track, you can create the Duplicate links manually or have them created automatically using a stereo group node.

To create a Duplicate link using a stereo group node:

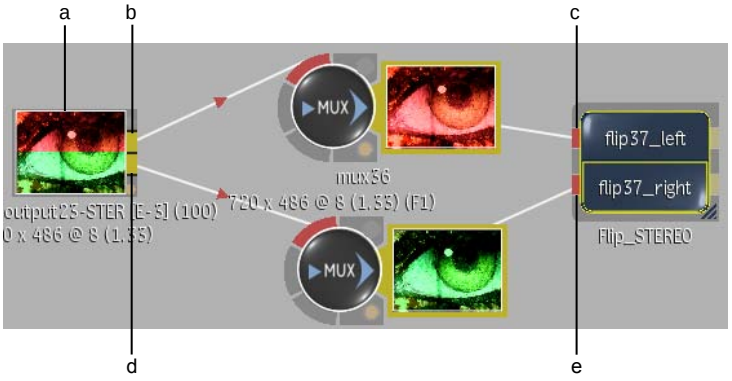
1 Press **Shift+S** and drag a node from the node bin to the schematic.

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.

TIP Use the Advanced Autolink feature to connect the input tabs.

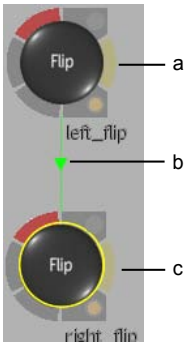


(a) Clip containing stereo track (b) Left eye output tab (c) Left eye input tab (d) Right eye input tab (e) Right eye output tab

3 Double-click the stereo group node or click Edit in the Group List menu.

NOTE Stereo group nodes have the same functionality as regular Batch group nodes. However, the Node List box only displays the option for accessing the master node's menu since the menu settings of the duplicated node are identical to those of the master node.

Another schematic opens containing 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 duplicates.



(a) Master node (b) Duplicate link (c) Duplicate node

4 Change the settings of either node.

Notice that the changes made to one node are duplicated 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 duplicated on Flip node connected to right eye

5 Click Exit Group.

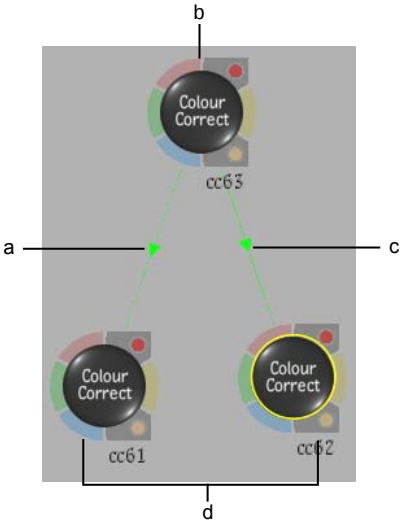
6 Select the Stereo group node and press **Shift+up or down arrow** to toggle between the left eye and right eye outputs.

To create a Duplicate link between nodes connected to mono clips:

1 From the Edit Mode box, select Duplicate (or press **W**).

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

---

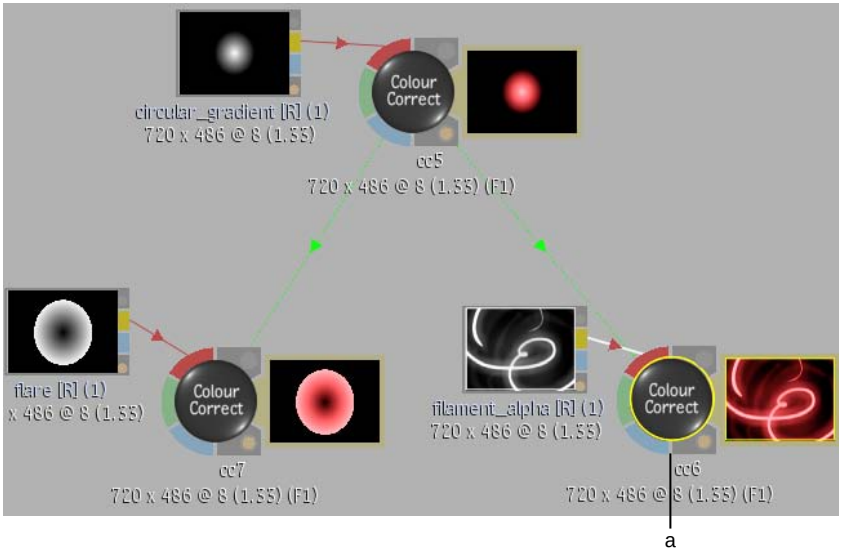
TIP While in Move mode, you can also create a Duplicate link by pressing **Shift+W** and dragging from one node to another so that the nodes touch. Press **Shift+Alt+W** to reverse the link direction.

---

3 Connect each node to a mono clip.

4 Change the settings of any node.

Notice that the settings made to one node are duplicated on the other nodes so that all connected mono clips are affected simultaneously. In the following example, duplicated CC6 node is edited. Master CC5 node and duplicated CC7 node are updated with the same changes.



(a) Edited node

Bypassing Node Inputs

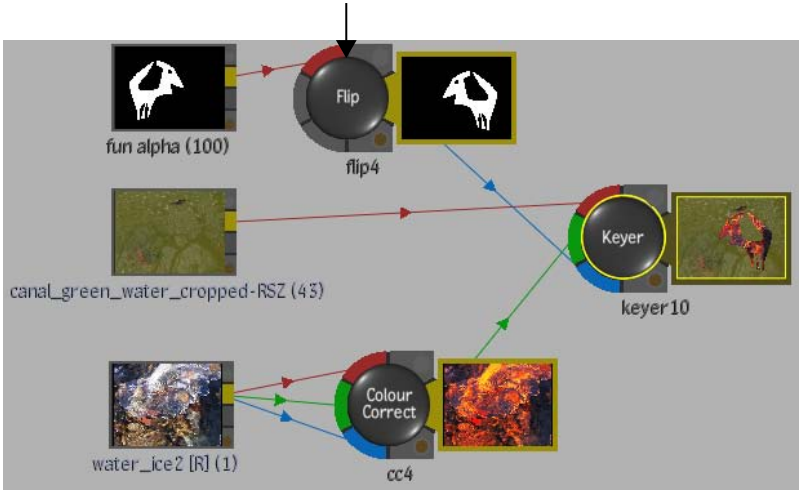
The interface items for enabling bypass mode have changed slightly.

You can improve your workflow in Batch by deactivating certain nodes and rendering only the nodes that you want to process right away.

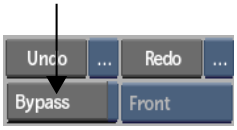
For any node, you can pass the Front, Back, Matte, or Key-in clip—depending on the type of node you select and its source tab inputs—as the input to the next node in a branch.

To bypass a node input:

1 In the schematic, select the node whose input you want to bypass.



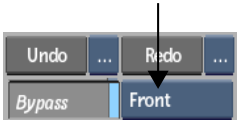
2 Enable Bypass.



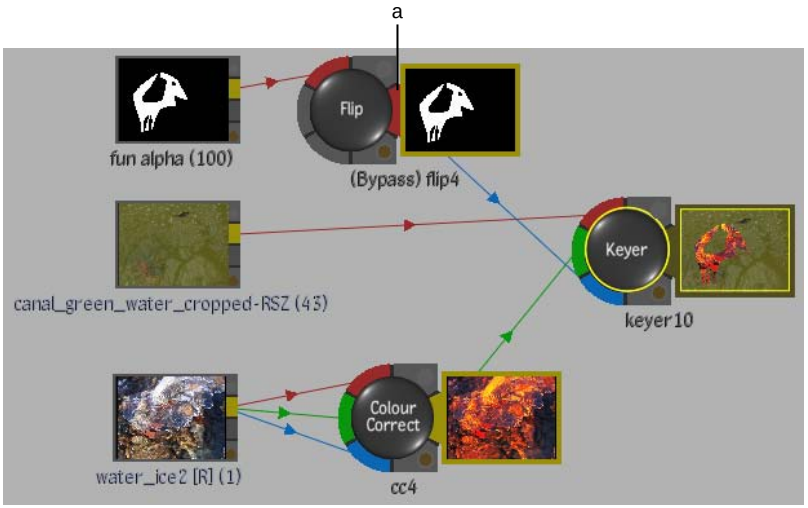
The Bypass box becomes active.

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.

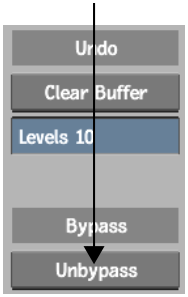


(a) Output tab of bypassed Front input

To unbypass nodes:

➤ Do any of the following:

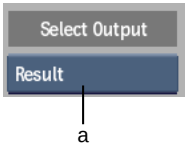
- Select the node with the bypassed input and disable the Bypass button.
- Select an option from the Selection Mode box corresponding to the part of the process tree that you want to unbypass, make a selection in the schematic, and then click Unbypass in the Setup menu.



NOTE After you click Unbypass, the Unbypass button changes to Re-bypass. Click Re-bypass to redo the bypass. You can also use the **- + click** key combination to bypass/unbypass a node.

Master Keyer Node

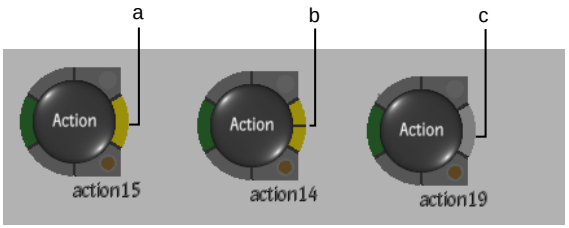
You can now select an output type from the Select Output box.



(a) Select Output box

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.



(a) One output (b) Two outputs (c) Multiple outputs (more than two)

NOTE The colours of a dual-output tab are based on the type of output (for example, yellow for composite, blue for matte).

The number of outputs is determined by the outputs set up in the Output list.

Expanding an Action Node

An Action node is collapsed by default. When you connect to its output tab, it now expands temporarily. If the node contains multiple outputs, you can then connect to any output.

You can also expand an Action node manually to connect to its outputs. Do this when you want the node to remain expanded after connecting to its outputs. You can also display a proxy of a specific output on an expanded node as well as set a context view on an output.

Outputs are set up in the Action menu's Output list.

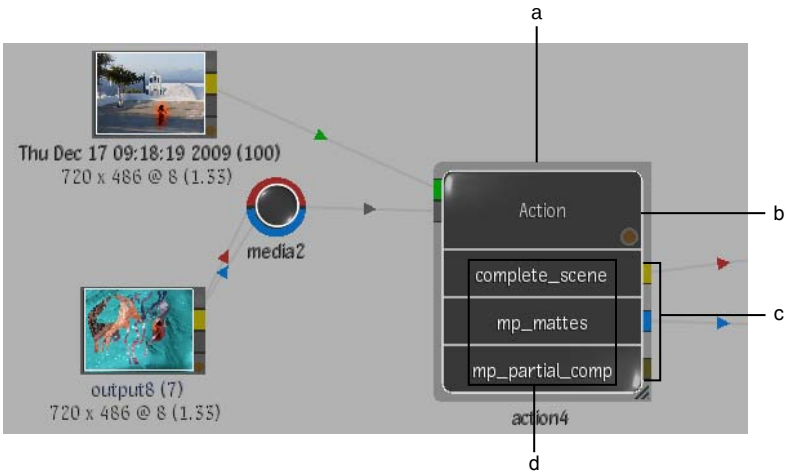
If an Action node is expanded manually, you can connect to its outputs manually, automatically, or using the Advanced Autolink feature.

If a collapsed Action node contains multiple outputs, you connect to its outputs using the Advanced Autolink feature.

To expand an Action node:

➤ Select the Action node and press **Shift+C**.

The Action node expands displaying available outputs.



(a) Expanded node (b) Proxy display window (c) Tabs of available outputs (d) Names of available outputs

NOTE Press **Shift+C** again to collapse the node.

To display a proxy of an output in an expanded Action node:

➤ Press **T** and click the tab of the output that you want to display.

The top and bottom borders of the output tab turn white indicating a proxy of its output is displayed in the display window.

NOTE When a proxy is displayed, you can also press **Ctrl+Shift+up arrow** or **Ctrl+Shift+down arrow** to navigate the outputs of the node.

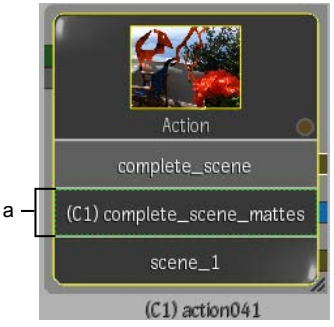
(a) Proxy of complete_scene output (b) White lines indicate proxy of output is displayed

---

NOTE The output names also appear as options in the View box. You can display any output in a viewport, including the left or right eye output of stereo footage.

To set a context view for any output in an expanded Action node:

➤ Press = and click the tab of the output whose context you want to set. (C1) or (C2) is added to the output name and the node name. As well, a green dotted line appears around the output name.



(a) Context 1 is set for complete_scene mattes

To set a context view for the current output in an expanded Action node:

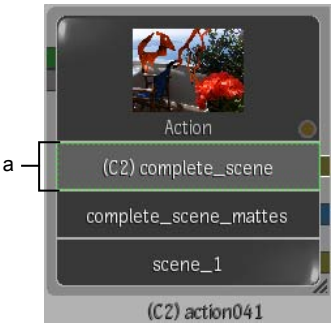
1 Select an output from the Action menu's Output list.

The selected output is the current output. Its name in the group node is highlighted.

(a) Current output

2 Press = and click anywhere in the node.

A context view is set on the current output. (C1) or (C2) is added to the output name and the node name. As well, a green dotted line appears around the output name.



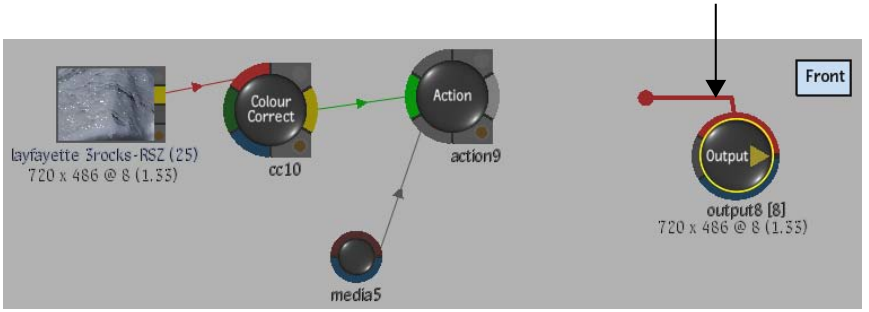
(a) Context 2 is set for current output

To change the size of an expanded Action node:

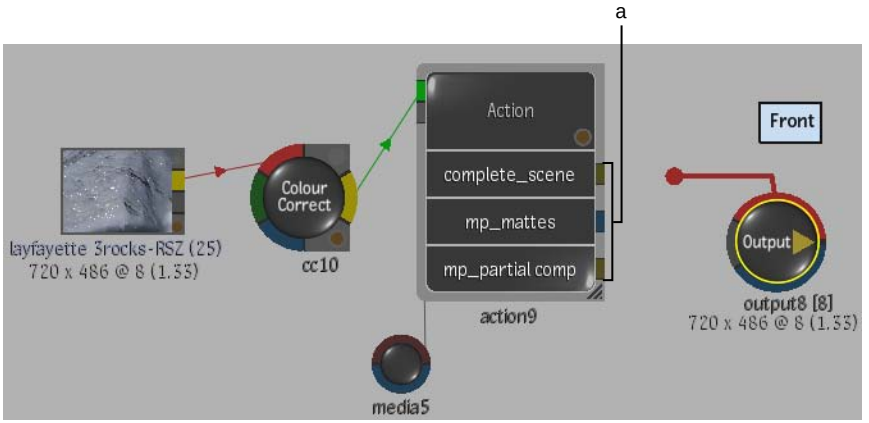
➤ Drag the lower-right corner of the icon to make it wider or narrower.

To connect to a multiple output tab of a collapsed Action node:

1 Using the Advanced Autolink feature, bring the extended arm of an input tab close to the Action output tab.



When the arm is close to the Action node, the Action node expands displaying its outputs. Three outputs have been set up in the following example.



(a) Possible outputs

NOTE If a tab is grey, its output in the Output list is muted and you cannot connect to it.

---

2 Connect the arms to the applicable output tabs.

When you are finished connecting, the Action node collapses again.

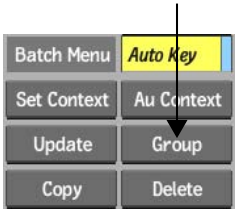
TIP If you expand the node manually, it stays expanded.

Grouping Nodes

You can now set a context on any output in a group node.

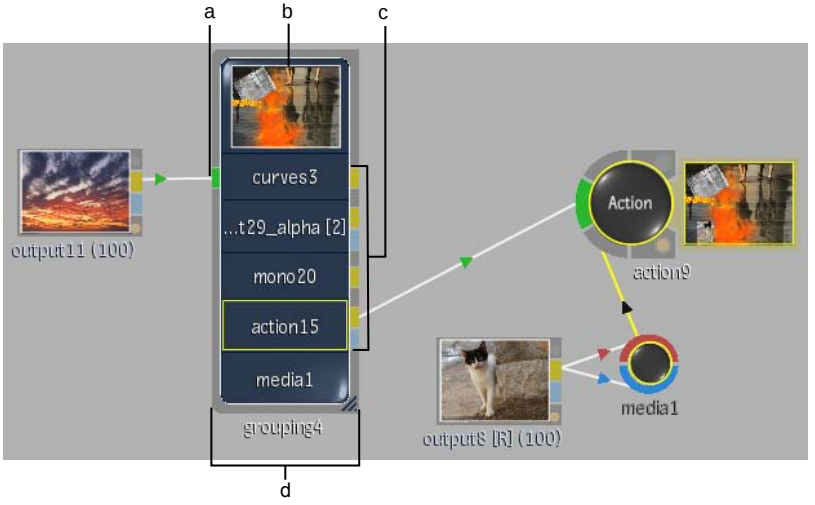
To group nodes:

1. **Ctrl**-drag to select the nodes you want to group.
   Selected nodes are outlined in white.
2. In the Process Tree controls, click 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 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.

To set a context view for any output in a group node:

1 Press = and click the tab of the output whose context you want to set.

(C1) or (C2) is added to the output name. As well, a green dotted line appears around the output name.

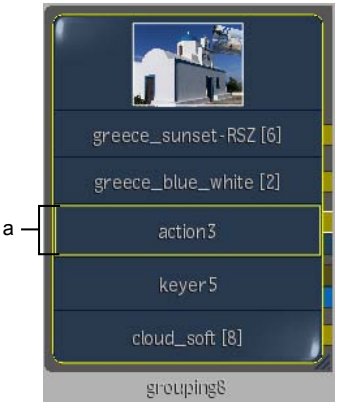
(a) Context 1 is set for keyer5

2 To display the context in a viewport, select the context from the View box.

To set a context view for the current output in a group node:

1 Select an output in the node's output list (press **Shift+up or down arrow** to navigate through the outputs).

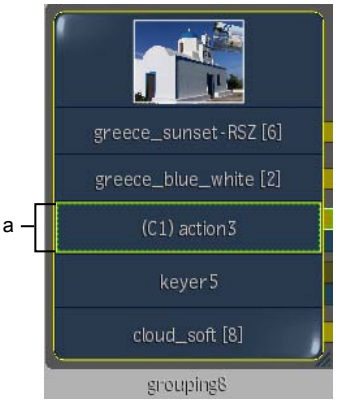
A yellow line appears around the selected output.



(a) Selected output

2 Press = and click anywhere in the node.

A context view is set on the selected output. As well, a green dotted line appears around the output name.



(a) Context 1 is set for selected output

3 To display the context in a viewport, select the context from the View box.

Generating an Alpha from a Clip Node

You can now generate an alpha from a clip inside a BFX level. When you output the alpha result, the clip node appears with an explicit alpha channel. You can then work with both the RGB and alpha outputs directly from Batch.

To generate an alpha from a clip node, the clip's primary track must contain only one layer. The layer can contain either a matte container or a BFXa segment. If the clip contains more than one layer, an Axis or Sparks soft effect will be added for vertical compositing.

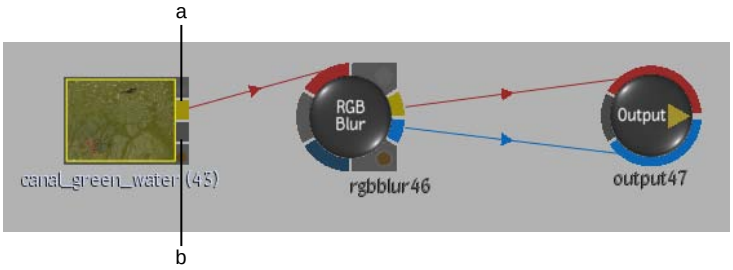
You can also explicitly generate a black or white alpha from a clip. Do this, for example, to have an alpha of the same resolution as the RGB clip.

You cannot generate an alpha from clips containing a stereo track since they do not have an alpha output.

To generate an alpha from a clip node:

1 Select a clip in the schematic.

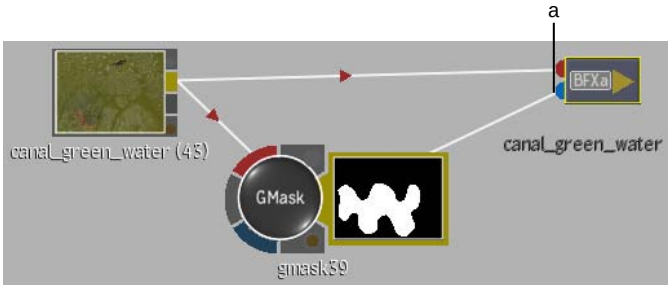
In the following example, notice that the clip's matte output tab is grey. You will not be able to connect to the matte output tab until a matte is fed to the tab. The RGB output tab outputs the RGB of the clip.



(a) RGB output tab (b) Inactive matte output tab

2 Enter a BFX level with the clip using the Pre or Post option.

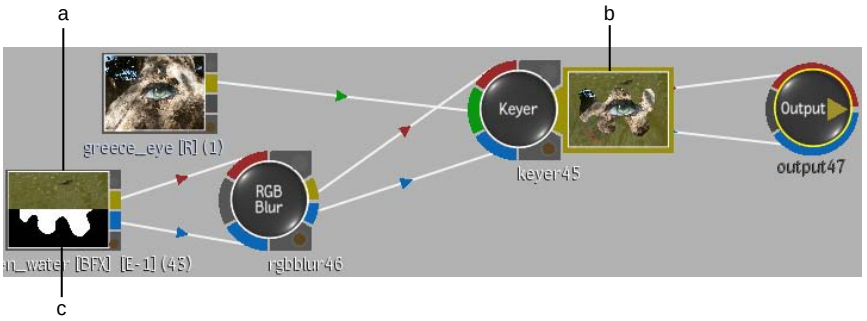
3 Create a matte and connect it to the matte input tab of the BFX Output node.



(a) Matte input tab

4 Exit to the previous level.

Notice that, in addition to its RGB result, the clip node displays its alpha output. The matte output tab is blue indicating that the clip contains an alpha output. You can use both the RGB and alpha result of the clip in your process tree.



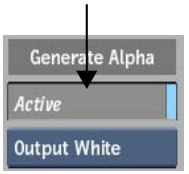
(a) RGB output (b) Alpha output (c) RGB and alpha of clip are input to the Keyer

NOTE When you process from Batch, both the RGB and alpha results are output.

To explicitly generate a black or white alpha of a clip:

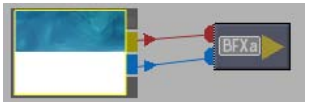
1 Select a clip in the schematic.

2 In the Basic menu, enable Active and then select to output a white or black alpha.



NOTE The Active button is greyed out if the clip contains a BFXa or a matte container. It is also greyed out for clips containing stereo tracks.

The clip's alpha is generated according to your selection.



Clips

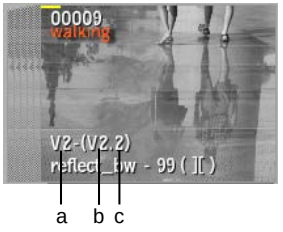
12

Default Processing Options

Use these preferences to select whether you want to include clip history when processing in a module. The default option is Process.

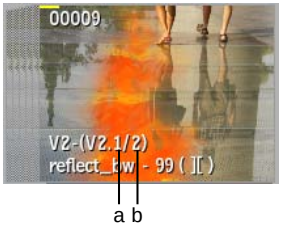
Navigating Edit Sequences

As you navigate between layers and tracks, information on the Desktop clip now updates to reflect which track and layer you are on. In the following example, there are two video tracks. The focus is on the topmost layer—L2—of track V2.



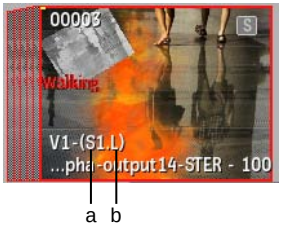
(a) Total number of video tracks (b) Focus is on track V2 (c) Focus of track V2 is on topmost layer, L2

In the next example, the focus is on the next layer down, L1, of the same track. When the focus is not on the topmost layer, the clip information reflects the total number of layers in addition to the focus layer.

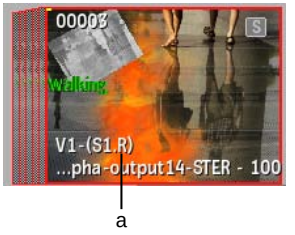


(a) Focus layer (b) Total number of layers on track V2

As you navigate the layers of a stereo track, the clip information updates to reflect whether you are on the left eye or right eye layer, as shown in the following examples.



(a) Stereo track (b) Left eye layer



(a) Right eye layer

Audio

13

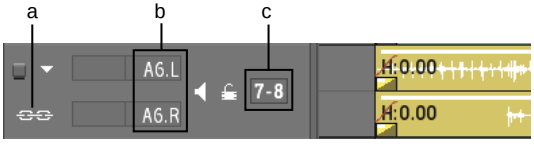
Editing Stereo Audio Tracks

You can import a stereo audio track, or create by mergin two mono tracks. A stereo audio track appears in the timeline as a single track with two layers, or channels, one for each track of stereo audio track.

Editorial and effects work on a stereo track is automatically synchronized between the left and the right channels of the track. Any changes applied to one channel of a stereo track are automatically applied to the other channel.

You can remove this stereo sync between channels if you need to perform asymmetrical edits.

The following interface components are specific to stereo audio tracks.



(a) Stereo Sync icon (b) Stereo track identifiers (c) Channel names

Stereo Sync icon Changes applied to one channel of a stereo track are automatically applied to the other channel. The stereo sync is enabled by default.

Stereo track identifiers Stereo tracks have “L” and “R” as part of their identifiers to indicate the left and right channels. The left channel is always the topmost channel.

Input Strip field Assign a stereo audio track on the timeline to a pair of input strips on the AudioDesk to adjust audio levels and audio parameters. The left and right channels are assigned a pair of consecutive input strips; the left channel

is assigned the odd input strip, while the right channel is assigned the even input strip. By default, Inferno assigns a stereo audio track to the next available pair of input strip.

Creating and Splitting Stereo Audio Tracks

You can create a stereo audio track from two audio tracks on the timeline, as long as they have the same in and out points.

To create a stereo track from two audio tracks:

1 Select two audio tracks. They must have the same in and out points.

(a) Selected tracks

2 Select Stereo Merge from the Edit Mode box.

The audio tracks are changed to stereo channels, and the track identifiers identify the left and right channels. The timeline interface is updated to reflect the components specific to a stereo track.

(a) Stereo Sync icon

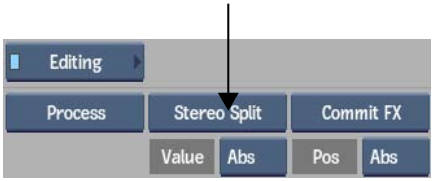
If there was a soft effect on one of the two channels, a red bar appears between the channels to indicate that they are not in stereo sync.

To split a stereo audio track in two mono tracks:

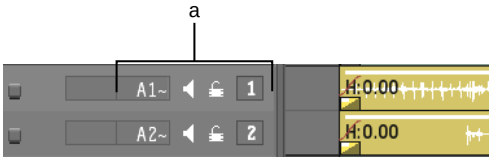
1 Select the left and right audio layers of a stereo audio track.

(a) Selected layers of a stereo track

2 Select Stereo Split from the Edit Mode box.



The stereo layers are changed to audio tracks, and the timeline interface is updated to reflect the components specific to audio tracks.



Moving Audio Channels

You can drag and drop audio channels in and out of stereo tracks.

- Select segments in a stereo track (matching segments from the other channel are automatically selected), and drop them on 2 mono tracks. The left channel is moved to the top track, the right channel to the bottom track, and all soft edits are carried over.
- Select a mono track and move it to the left channel of a stereo audio track. That segment is automatically duplicated in the right channel.
- Select two mono tracks and move them to a stereo audio track. The top track is moved to the left (top) channel, the lower track is moved to the right (bottom) channel. The two mono tracks must have the same cuts and edits or this operation is not possible.

Managing the Synchronization of Stereo Channels

If you are doing work that affects the media of a stereo track, you can disable the stereo sync from the left and right channels and modify only one channel. For example, there may be instances when you need to adjust the gain of one channel. You can remove the stereo sync to do the following to one channel of a stereo track:

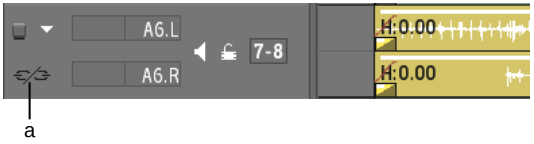
- Applying and deleting soft effects
- Slipping a channel
- Replacing a segment's source

If you applied the same soft effects to unsynced channels but the soft effect settings are different, you can resynchronize the settings.

To remove the stereo sync between the channels of a stereo track:

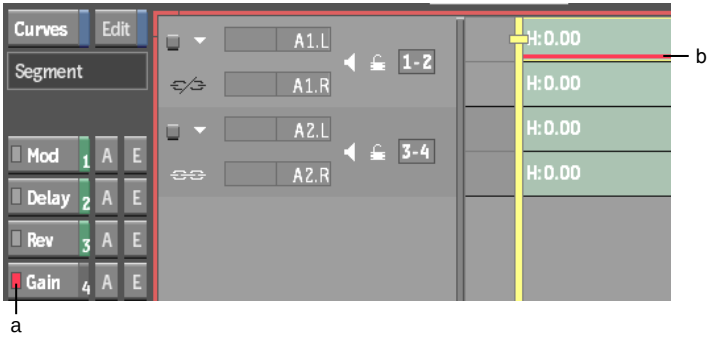
➤ Click the Stereo Sync icon.

The icon turns black indicating that the stereo sync for the channels has been disabled. You can now edit the media of one stereo channel without affecting the other channel.



(a) Stereo sync disabled

If you add a soft effect to an unsynced channel, a red bar appears indicating that the channels are not in stereo sync, and the indicator on the soft effect button turns red.



(a) Gain soft effect applied to only one channel of a stereo track (b) Stereo channels are not in sync

To resynchronize a soft effect between stereo channels:

1. Select the stereo channels or elements containing the soft effects that you want to synchronize.
2. If there is more than one soft effect on your timeline selection but you only want to resynchronize one type, select the applicable soft effect icon on the timeline.
3. Select Resync FX from the Commit Effect box.

NOTE This option is only available with stereo tracks.

4. From the message that appears, select whether you want to modify the soft effects on the left channel or the right channel.

   The same soft effect settings are applied to each channel based on your selection and the red bar is removed from the channel.

NOTE Resynchronizing soft effects does not re-enable the stereo sync icon.

To re-enable the stereo sync icon between the channels of a stereo track:

➤ Click the black Stereo Sync icon.

The icon turns white indicating that the stereo sync between the channels has been re-enabled. If you had applied a soft effect to one of the stereo channels when the stereo sync was removed, the red bar remains on the channel and the indicator on the soft effect button remains red.

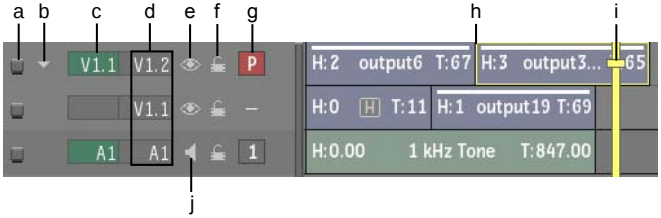
Timeline 14

Timeline Interface

The timeline area has been reorganized with new icons for manipulating the timeline.

Timelines can contain multiple video and audio tracks. An edit sequence on a track is made of segments and transitions. Video and audio segments appear as a series of rectangles on their associated tracks. Transitions appear as cuts or icons between segments.

Video tracks can be made up of multiple layers. Layers are used to stack video vertically to create composite effects and transitions. If you have more than one layer or track, the positioner's focus point indicates which one is current.



(a) Selector icon (b) Layer collapse arrow (c) Patch identifiers (d) Track identifiers (timeline contains one video track made up of two layers and one audio track) (e) Video Mute icon (f) Lock icon (g) Primary video track indicator (h) Bounding box (i) Positioner (j) Audio Mute icon

Selector icon Selects a layer or track.

Layer collapse arrow Collapses layers into one element.

Patch and track identifiers and Primary track Indicate patching for video and audio tracks.

Video Mute icon Hides the element in the vertical edit.

Audio Mute icon Mutes or solos the audio.

Lock icon Prevents editing operations from being performed on the track.

Bounding box A yellow bounding box around a timeline segment indicates there is an implicit selection by the positioner. Any editing operations you perform, for example, cuts or soft effects, will occur at the positioner location. If there is no bounding box at the positioner location and you have not explicitly selected the segment, this means there is an explicit selection elsewhere on the timeline. Any editing operations will occur at the explicit selection, not at the positioner location.

Having the visual cue of a bounding box around a segment can help you confirm that you are editing the correct segment. This is especially useful in long- form timelines where you might not see all the segment selections.

Positioner Is the "playhead" for playing the clip. The frame directly beneath the positioner is displayed in the viewport or is the current location for an edit such as a dissolve or cut.

Focus point Indicates the current track. In vertical editing, the positioner's focus point indicates the top layer (focus layer) in the vertical edit.

Track Contains the segments and transitions that you edit together.

Layer Used for vertical editing and timeline compositing.

Element Refers to the video segments, audio segments, and transitions that make up an edit sequence. Elements are colour coded to make them easier to identify, as illustrated in the following table. Coloured bars on top of the elements indicate their process and lock status.

Adding Tracks

You can add as many audio and video tracks to a timeline as you want.

To add a track to the timeline:

➤ Do one of the following:

- To add a video track above the current track, click Video+ .
- To add a video track below the current track, **Ctrl**-click Video+ .
- To add a stereo track above the current track, **Alt**-click Video+ .

- To add an audio track, click Audio+ .

To remove a track from the timeline:

➤ Click the Selector icon, drag it to the bottom of the screen and release when the cursor changes to a green recycling icon.

To move a track to another location on the timeline:

➤ Drag the Selector icon to a new location on the timeline.

Selecting Tracks

You can select one track at a time, multiple tracks, or all tracks. Selected tracks are yellow and their associated Selector icon appears recessed.

To select a track:

➤ Do one of the following:

- If the track has only one layer, click the Selector icon.
- If the track has multiple layers, click the Collapse arrow and then click the Selector icon, or **Ctrl**-click each layer's Selector icon.

(a) Selector icon (b) Collapse arrow

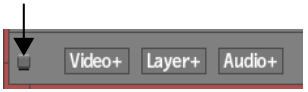
To select multiple tracks:

➤ Do one of the following:

- To select a range of tracks, **Shift**-click the Selector icon of the first and last tracks.
- To select individual tracks, **Ctrl**-click the Selector icon of each layer.

To select all tracks:

➤ Click the All Tracks Selector icon or press / on the keypad.



To deselect tracks:

➤ Do any of the following:

- To deselect all tracks, press * on the numeric keypad or click the All Tracks Selector icon.
- To deselect individual tracks, click the track's Selector icon.

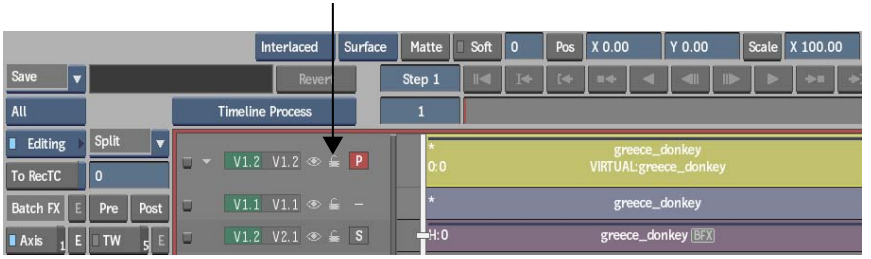
(a) Selector icon for V1.2 (b) All Tracks Selector icon

Locking Tracks

Lock tracks to prevent further editing operations from being performed on them.

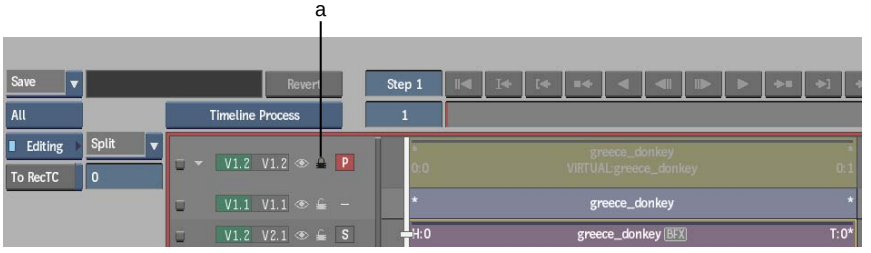
To lock a track:

➤ Click the Unlock icon. In the following example, the track has an Axis soft effect applied.



The icon turns to a black Lock icon and the locked track is greyed out. As well, soft effects, Batch FX, and Container buttons are removed from the interface.

If the locked track has any soft effects, the soft effects quick menu is also removed from the interface. In the following example, the Axis soft effect quick menu is no longer available.



(a) Lock icon

To unlock a track:

➤ Click the black Lock icon.

The icon turns to a white Unlock icon, and all timeline interface elements reappear.

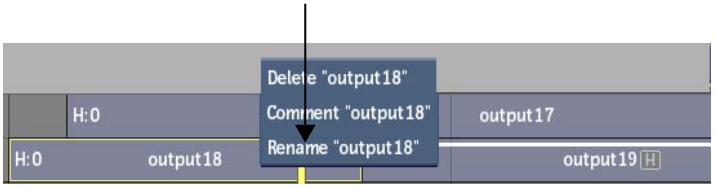
Naming Elements

You can name video elements, audio elements, as well as transitions, cue marks and track marks.

To name an element:

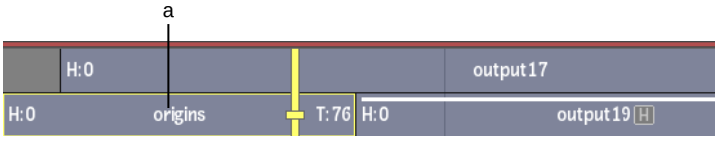
1 Select the element that you want to rename or place the cursor over the element, press the **Context** key (beside the **Ctrl** key on the right side of the keyboard), and select the Rename option from the list that appears.

NOTE If you place the cursor over an element, make sure there is no explicit selection on another element.



2 Type a name in the Name field and press **Enter**.

The element information on the timeline is updated. The new name is displayed above the original source name.



(a) Renamed element

NOTE If you do not see the new name of the element or the original source name, drag its track identifier down.

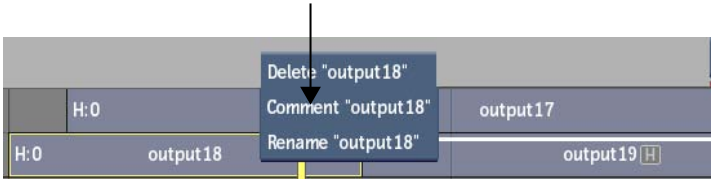
Adding Comments to Elements

You can add comments to the individual elements that make up a clip.

To add a comment to an element:

1 Select the element where you want to add a comment or place the cursor over the element, press the **Context** key (beside the **Ctrl** key on the right side of the keyboard), and select the Comment option from the list that appears.

NOTE If you place the cursor over an element, make sure there is no explicit selection on another element.



2 Enter a comment in the Name field and press **Enter**.

The element on the timeline is updated with the comment.

Deleting Elements from the Timeline

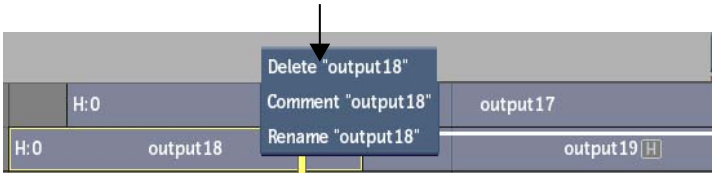
You can remove elements from the timeline without removing their track.

To delete elements from the timeline:

➤ Do one of the following:

- Select the elements, drag them to the bottom of the screen and release when the cursor changes to a green recycling icon. You can also select an element and press **Alt+D**.
- Place the cursor over the element that you want to delete, press the **Context** key (beside the **Ctrl** key on the right side of the keyboard), and select the Delete option from the list that appears.

  NOTE If you place the cursor over an element, make sure there is no explicit selection on another element.

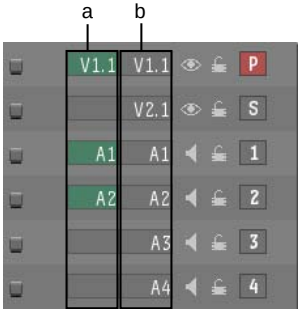


The selected elements are deleted. Unselected elements remain on the track.

Patching on the Timeline

When you set record and source clips, 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.

Assume that you have a source clip with one video channel and two audio channels. When you select the source clip, green patch identifiers indicate what channels you can record from the source to the timeline. In the following example, you are recording to a timeline that has two video tracks and 4 audio tracks. The source channels are patched to video track V1 and audio tracks A1 and A2.



(a) Patch identifiers (b) Track identifiers

You can adjust the patching to connect source channels to other record tracks. In the following example, the new destination tracks are V2, A3, and A4.

With the patch identifiers, patching information is kept with the record clip. If you patch a source clip and then add a new layer 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 layer, then setting up the patching for another source clip.

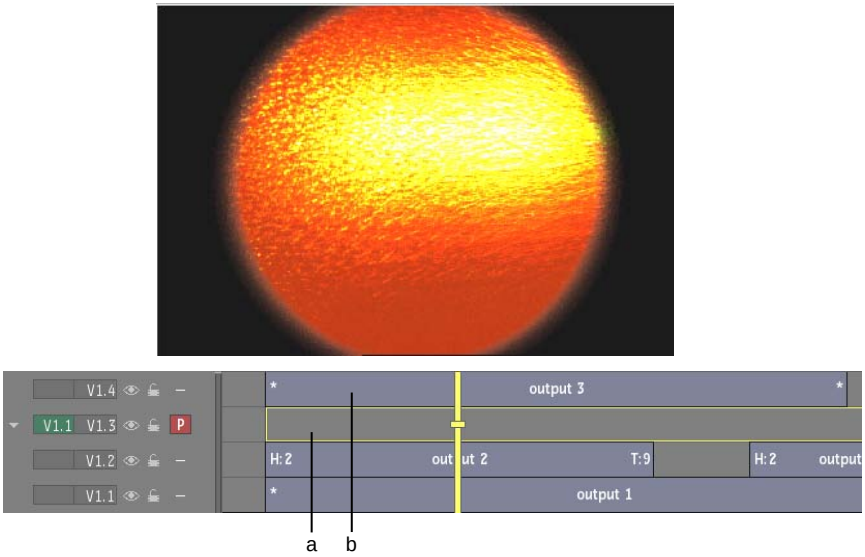
Creating Layers

Create as many layers as you need for vertical editing. Vertical editing works from the top layer to the bottom layer. Assign the topmost layer as the focus layer for it and all layers below it to be visible in the output.

To create a layer for vertical editing:

1. From a Batch timeline, do one of the following:
   - To add a layer above the current layer, click the Layer+ button.
   - To add a layer below the current layer, **Ctrl**-click the Layer+ button.

An empty layer is created and it becomes the focus layer. Only the layers below it on the same track are visible in the output. In the following example, an empty layer is added above layer 2. Only the output below the new layer, layer 3, is visible in the output. The fish layer is not visible.



(a) New layer (b) Layer 4 not visible in the output

2 Add a clip or soft effects to the empty layer.

3 Press the **up arrow** to move the focus point to the topmost layer.

The topmost layer becomes the focus layer, the channel indicator updates to the primary layer, and the image is updated.

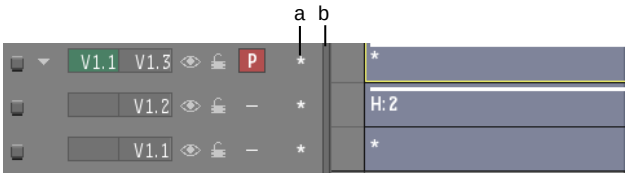
TIP To add multiple layers, type the number of layers you want to add in the numeric keypad before clicking the Layer+ button. You can add up to ten layers at a time using this method.

Deleting Layers

If you want to delete all the elements that make up a layer, you can delete the layer itself.

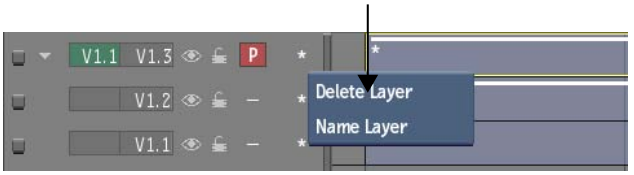
To delete a layer:

1 Drag the slider to the right until the layer name appears.



(a) Layer name (b) Slider

2 Place the cursor over the layer name, press the **Context** key (beside the **Ctrl** key on the right side of the keyboard), and select the Delete Layer option from the list that appears.



The layer is deleted from the timeline.

NOTE You can also delete a layer in the same way as a track—by dragging the Selector icon to the bottom of the screen or by selecting the layer and pressing **Alt+D**.

Muting Layers

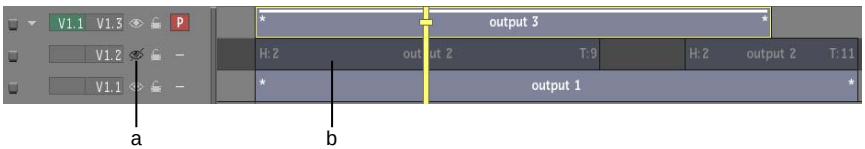
Mute layers to hide them temporarily from the vertical edit. You can mute any layer.

To mute a layer:

➤ Do one of the following:

- To mute one layer, click the Mute icon corresponding to the layer that you want to mute.
- To mute all layers in a track, **Shift**-click any Mute icon.

The icon of the muted layer turns black and the layer is removed from the output. In the following example, the output from layer 2 is muted, revealing the output from the next layer down—layer 1.



(a) Black icon for muted layer (b) Layer hidden from output

Image courtesy of Quietman

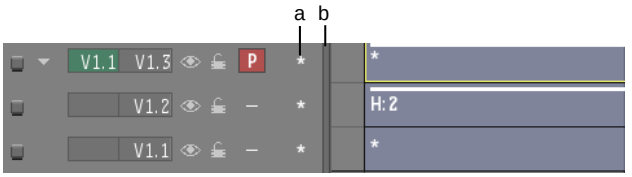
NOTE To make a layer visible, click its Mute icon. To make all muted layers visible, **Shift**-click any Mute icon.

Naming Layers

You can name any layer. Layers are named "*" by default.

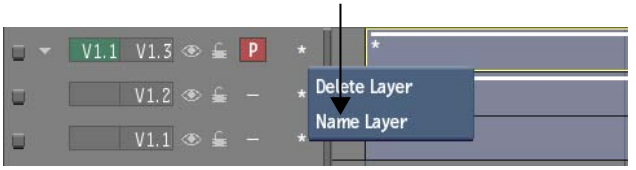
To name a layer:

1 Drag the slider to the right until the layer name appears.



(a) Default layer name (b) Slider

2 Place the cursor over the layer name, press the `Context` key (beside the `Ctrl` key on the right side of the keyboard), and select the Name Layer option from the list that appears.



3 Type a name in the Name field and press **Enter**.

The timeline menu is updated with the renamed layer.

(a) Renamed layer

Viewing 15

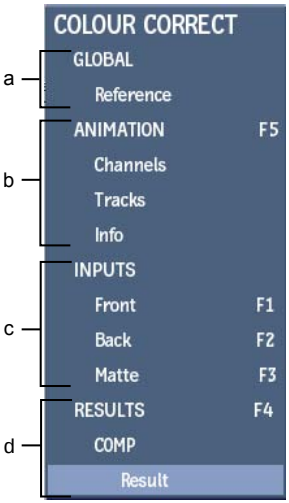
Displaying Clip Views

When you enter a module from the Desktop, the image window usually displays the result clip. In Batch and the Modular Keyer, the default view is the schematic.

You select a view option from the View box. Certain views are common to all modules. Other views depend on the module you are working from, its input clips, and what kind of results you can output.

Example: To display a clip view in the Colour Corrector:

➤ Select an option from the View box or press its associated hotkey.



(a) Global Colour Correct options (b) Animation options (c) Clip Input Type options (d) Result options

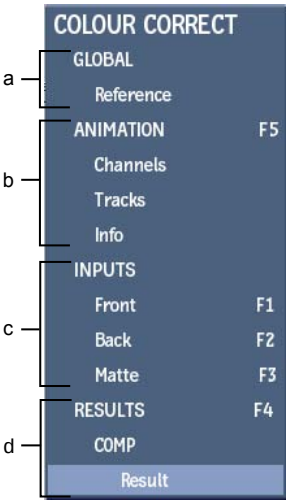
NOTE If a module supports multiple viewports, the options are now divided into categories, subcategories, and type, as in this Colour Corrector example. View options are not categorized for modules that do not support multiple viewports.

Options specific to certain modules are documented in the corresponding chapters.

Multiple Viewport Categories

The view options in modules that support multiple viewports are now divided into categories, subcategories, and type. This classification makes it easier to scan the available options for your viewport selection.

For example, the view options for the Colour Corrector are as follows.



(a) Global Colour Correct options (b) Animation options (c) Clip Input Type options (d) Result options

GLOBAL This category displays all subcategories specific to the module you are working in. When working in the Colour Corrector, for example, you can display a reference clip.

ANIMATION This category displays all subcategories available with the Channel Editor.

INPUTS This category displays the subcategories of clips with which you can enter a module or the clips that you can connect as a source to a Batch or Modular Keyer node.

RESULTS This category displays the result views. In this example, the Result type within the COMP subcategory is selected.

NOTE To cycle between views within the same category, press the **Ctrl+up arrow** or the **Ctrl+down arrow**.

The Batch and Action modules have different view options based on your node selection. As you select different nodes, the view option you select for one node may not be available for your subsequent node selection.

However, a node memorizes the type classification. Therefore, the type of view remains consistent as you select different nodes. For example, if you select Result, Result view will be displayed for all your node selections. If you set one node to display a type not supported with another node, the view will revert back to the type you selected for that node as you go back and forth

between nodes (as long as the layout and/or the category of viewport has not changed).

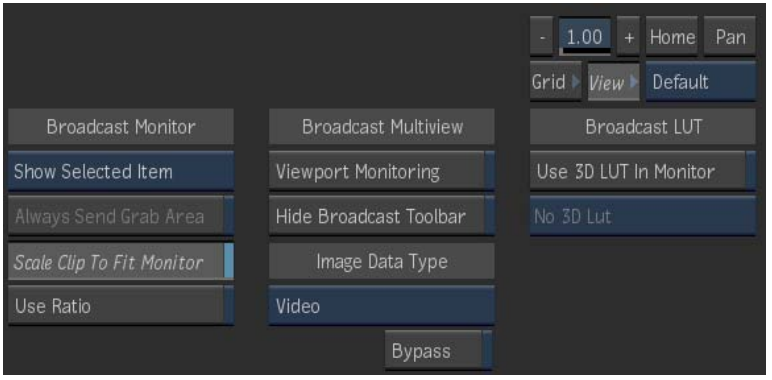
A viewport memorizes the category. For example, if you select the Channels category for a viewport, the Channels view will be displayed regardless of your node selection for that viewport.

Consider a Batch schematic containing one Gmask node and one Colour Correct node. A 2-viewport layout is selected with the Batch schematic displayed in one viewport and an Input category displayed in the other viewport. The GMask node is selected and Front is selected as the Input type. Next the Colour Correct node is selected and its Input type is set to Back. The Gmask node is then reselected. Its input type will automatically revert to Front, since this was the original selection for this node.

Working with Viewports and a Broadcast Monitor

The View menu displays options for broadcast monitor settings if your workstation includes one.

Use the Broadcast Monitor buttons to control what is sent to the broadcast monitor. The settings displayed here mirror the ones available in the Preferences menu.



Use the Broadcast Multiview buttons to control the behaviour of the broadcast monitor. The Broadcast Multiview options are available only if Show Selected Item is selected in the Broadcast Monitor option box.

Viewport Monitoring toggle button Switch between using the broadcast monitor as a mirror of one of the displayed viewports, and using the broadcast monitor as an independent viewport.

By default, the broadcast monitor mirrors the upper-right viewport. However, you can set any viewport to be sent to the broadcast monitor.

Hide Broadcast Toolbar toggle button Enable to hide the Broadcast Toolbar. Use the toolbar to control the broadcast monitor as you would any other viewport. The toolbar is displayed in the top-right corner of the graphics monitor. Even if the toolbar is disabled, you can always control the displayed image data type using the Image Data Type buttons displayed below.

TIP **Spacebar**+Click the broadcast monitor toolbar to move it.

Image Data Type option box Select the type of image you are displaying in the broadcast monitor. Your selection determines the type of transformation applied so that the broadcast monitor displays the colours accurately.

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.

Bypass button Enable to deactivate the image data type display settings in the broadcast monitor.

To send a viewport to the broadcast monitor:

1. Select Show Selected Item from the Broadcast Monitor option box.
2. Enable Viewport Monitoring.
3. Click the Monitor symbol in the viewport you want to send to the broadcast monitor.

To use the broadcast monitor as a viewport:

1. Select Show Selected Item from the Broadcast Monitor option box.
2. Disable Viewport Monitoring.
3. Use the broadcast monitor toolbar to control the broadcast monitor like any other viewport.

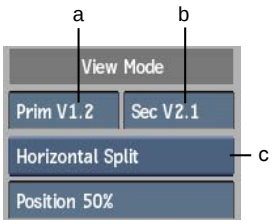
Comparing Tracks and Layers

Use the View Mode options to display images from two video tracks simultaneously. The tracks can be stereo or mono. You can choose to view the two tracks in a split screen. Alternatively, the two tracks can be viewed as a transparency blend or with a clamped or non-clamped difference. The two tracks viewed are the Primary and Secondary tracks.

If you are working with a clip containing a stereo track, you can now select how you want to preview stereo results.

To compare Primary and Secondary tracks in the Player:

1. Bring a clip into the Desktop Player or the Library Player and click the Clip & Setup tab.
2. In the View Mode controls, assign one track as the Primary track and one as the Secondary track in the respective fields. If a track has more than one layer, you can also enter the layer that you want displayed.



(a) Primary Video Track field (b) Secondary Video Track field (c) Preview Setup box

In this example, layer 2 of video track 1 is assigned as the Primary track and layer 1 of video track 2 is assigned as the Secondary track.

3 Select how you want to view the two tracks using the Preview Setup box.

If you selected:	Do this:
Horizontal or Vertical Split	Set the position of the split.
Angle Split	Set the position and angle of the split.
Blend	Set the percentage of the Secondary track to display. For example, if you specify 60%, the image is composed of a blend of 40% of the Primary track and 60% of the Secondary track. Set a value in the Blend field or press **Shift+T** and drag left or right.
Difference	Set the difference between the two tracks. You can set the minimum and maximum threshold values to be clamped or non clamped. To adjust the maximum clamp value, enter a value in the applicable field or press **Shift+U** and drag left or right. To adjust the minimum clamp value, enter a value in the applicable field or press **Shift+Y** and drag left or right.

NOTE In Realtime view, the Secondary video track appears black if it has an unrendered effect, or if both the Primary and Secondary video tracks contain unrendered dissolves.

Example: To preview stereo results in Stereo Anaglyph mode:

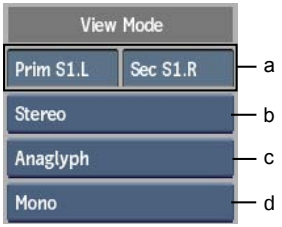
1 Bring a clip containing a stereo track into the Desktop Player or the Library Player and click the Clip & Setup tab.

2 In the View Mode controls, assign one layer (right eye or left eye) as the Primary track and the other layer as the Secondary track in the respective fields.

In the following example, the left eye layer is assigned as the Primary track and the right eye layer is assigned as the Secondary track.

Stereo is automatically selected in the Preview Setup box.

NOTE The options in the Stereo Preview box depend on your hardware configuration. As well, the option displayed is the same as the one selected in the broadcast preferences. If you change the option in one location, it is changed automatically in the other location.



(a) Primary and Secondary Video Track fields (b) Preview Setup box (c) Stereo Preview box (d) Stereo Anaglyph box

3 Select an option from the Stereo Anaglyph box.

Select:	To view:
Diff	The difference between the two images, with the minimum and maximum threshold values non clamped.
Diff Clamped	The difference between the two images, with the minimum and maximum threshold values clamped. To adjust the maximum clamp value, enter a value in the Max field, or press **Shift+U** and drag left or right. To adjust the minimum clamp value, enter a value in the Min field, or press **Shift+Y** and drag left or right.
Blend	A combined image of the two tracks. To adjust the Blend factor, enter a value in the Blend field, or press **Shift+T** and drag left or right.
Mono	The image with just the anaglyph effect. The RGB values are removed from the display.
Dubois	The image with reduced ghosting between the left and right eyes.

Image Data Type

Use these preferences to set image data type.

Image Data Type box Select the type of image data you are displaying. Your selection determines the type of transformation that is applied to the clip to modify the contrast.

Select:	To:
Logarithmic	Apply a transformation to a logarithmic film scan.
Video	Apply a transformation to a video clip.
Linear (Scene re-ferred)	Apply a transformation to a 16-bit floating point image, with a high dynamic range.