Autodesk®
Inferno® 2010
A Discreet® Systems product
with Autodesk® Flare™ 2010

New Features Guide

Autodesk®

Trademarks

Disclaimer

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

Images Courtesy of: Jean-Marc Belloncik

Title: Autodesk Inferno 2010 with Autodesk Flare 2010 New Features Guide

Document Version: 1

Date: March 9, 2009

Contents

Introduction

1

Topics in this chapter:

About the Documentation

Autodesk® Inferno® 2010 and Autodesk® Flare™ 2010 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* and *http://www.autodesk.com/flare-documentation*.

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

Viewing PDF Documentation

The documentation set is available in PDF for online viewing or printing. They are installed in the *documentation* directory of your application. You can view any of the PDF files in that directory from your application.

We recommend Adobe® Reader® or Xpdf for best results when viewing PDF files.

To view the PDF files from your application:

1. Click Preferences to display the Preferences menu.
2. Select a document from the PDF Documentation box.

   The document opens automatically in Xpdf on Linux® workstations.

   TIP You can access other PDF documents from your application by copying them to the directory.

Autodesk Media and Entertainment Training

There are a number of training options available to you to help you be more creative and productive with your application, including free self-paced training, and instructor-led training.

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

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`**

Convention	Example
Variable names appear in Courier, enclosed in angle brackets.	`<filename>`
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*

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

Topics in this chapter:

About This Release

This release of Inferno introduces many new and updated creative tools, as well as workflow improvements. See the themes below for a quick overview, and then follow the links for more detailed information.

Flare: Creative Assistant to Inferno

Completely new, Flare is a fully compatible assistant to Inferno.

Featuring the same creative toolset as Inferno Batch, Flare can be used for a variety of tasks from rotoscoping to particle creation. All Batch nodes found in Inferno are available in Flare.

Use Flare either in an independent system workflow or a remote connection workflow. See Setting Up a Flare Workflow on page 21.

Floating Point Workflow

With this release, Batch is almost completely floating point-compliant, opening the door for better support of OpenEXR files.

Floating Point Batch

All batch nodes support floating-point images, except the original Keyer node.

Individual Keyer Nodes in Batch and the Modular Keyer

The Keyer node is now split into individual Keyer nodes (RGB, HLS, YUV, RGBCMYL, Channel) in Batch and the Modular Keyer to add to the existing Keyer Luminance and MasterK nodes. The Modular Keyer node and new Keyer nodes support floating point images.

See Keyer-Channel Node on page 45, Keyer-HLS Node on page 46, Keyer-Luma Node on page 46, Keyer-RGB Node on page 48, Keyer-RGBCMYL Node on page 48, and Keyer-YUV Node on page 49.

Start-up modes allow the user to recreate Keyer pipelines that support floating point images in the Modular Keyer.

RGB Viewer Updated in 3D Keyer

The RGB Viewer in the 3D Keyer is updated to display floating-point colour space. The Histogram section of the Key menu includes a new Frame box that allows you to select what to frame in the new viewer area.

The 3D histogram in the RGB Viewer can store a range value that can be retrieved for later use.

See Setting the Display of the RGB Viewer on page 50.

Colour Management in 3D Histogram

The 3D histogram in the Colour Warper and the 3D Keyer RGB Viewer now updates colourspace display based on the applied exposure and contrast settings of the clip.

BFX Workflow in Floating Point

To maintain 16-bit floating point support when outputting from a BFX level, when you output a matte (BFXa) from the BFX output node back to the main timeline, a Spark Composite soft spark is added to the timeline instead of a soft Axis.

NOTE A known limitation of the floating point workflow is that soft effects in the timeline do not support floating point images.

Stabilizer Supports Floating Point

You can now track and stabilize floating point images. This allows you to access the Stabilizer from Action with a floating point image.

Support for Multichannel OpenEXR Import

Input, transcoding, and output of single OpenEXR files are now mostly handled using WiretapCentral. WiretapCentral gives you the option of saving RGB, RGBA, or all channels. Note that separate RGB clips will be generated for every EXR channel detected.

You can also access Wiretap Central directly form the Autodesk Visual Effects and Finishing application, by selecting OpenEXR from the Import Image menu.

Colour Picker Supports Floating Point

The colour picker now supports 16-bit floating point colours.

See Colour Picker on page 51.

Creative Tools

This release introduces many new and improved tools to help you save time and expand your creative results.

3D Blur Node

The new 3D Blur node offers a method of creating depth of field effects combined with motion blurring. The 3D Blur node aggregates depth information taken from a z-depth input, with motion information taken from a vector pass input, and calculates the result in a single pass. The 3D Blur node uses the GPU for faster rendering.

See 3D Blur Node on page 61.

Displacement Objects in Action

A displacement map in Action now can be applied as its own node. This new workflow extends the range of manipulations that can be applied to a displacement map. See Adding a Displacement Map on page 69.

Displace Map nodes can also be added to 3D objects. You can use the new UV Mapping tab in the Geometry menu to select how the UVs of an attached displace node are mapped to the 3D model. See Using UV Mapping on page 74.

NOTE The previous method of using displacement maps in the Surface menu has been removed. Any setups using the previous displacement method will be converted to the new displacement method upon loading. The exception to this new method is using displacement with DVE Layer Objects, which still uses the previous displacement method, but with an improved tabbed Surface menu. See Displacing DVE Layer Objects on page 75.

Normal Map Objects in Action

Normal maps are supported in Action as a new node. Normal maps allow you to simulate 3D lighting effects on flat surfaces. See Adding a Normal Map on page 76.

Normal Map nodes can also be added to 3D objects. You can use the new UV Mapping tab in the Geometry menu to select how the UVs of an attached

normal node are mapped to the 3D model. See Using UV Mapping on page 74.

Mirror Repeat in Texture Node

Mirror Repeat (smooth or fast) modes are now available as texture mapping types when using the Texture node in Action.

Batch Paint Performance Improvements

The Batch Paint node now supports floating point input and features performance optimizations.

3D Text Improvements

The 3D Text node in Action is revamped to add many new functionalities, including:

- better subdivision.
- displacement map support.
- text on path with various alignments options.
- cascading of animation on individual letters.

See Action: 3D Text on page 79.

3D Paths in Action

3D Path is a new node in the Action Node bin, adding the functionality of a 3D spline on which any Action object can slide.

See Action: 3D Paths on page 87.

Technical Tools

Like creative tools, technical tools are also necessary to help you get better results. This release introduces the following new and improved technical tools.

Pulldown Tool Improvements

Advanced and PAL pulldown are now supported in Desktop Tools and Batch nodes.

The tools and Batch nodes can also automatically detect the type of pulldown to remove.

Resize Improvements

The following features that were available only in the Real-Time Deliverables resize are now available from the Desktop, Timeline (soft-resize), and Batch:

- GPU acceleration. Your application can now use the GPU to accelerate resize processing, which results in noticeable performance increases for the more complex filtering algorithms.
- Adaptive de-interlacing. This filter, which minimizes artefacts associated when resizing interlaced material, is now available.
- Sub-pixel crop box. The crop box can now be resized on a sub-pixel level, which results in smoother animations.

Limitations of GPU-Accelerated Resizing

If your clip exceeds the maximum resolution listed in the following table, your application cannot use the GPU to accelerate resizing. When the GPU cannot accelerate resizing, adaptive de-interlacing is also not available.

NVIDIA Graphics Card	Maximum Resolution for: 8 and 16-bit Images	Maximum Resolution for: 10-bit and 12-bit (packed and unpacked) Images
Quadro FX 4500/5500	4096 x 4096	2048 x 4096
Quadro FX 5600	8192 x 8192	4096 x 8192

Batch Workflow Improvements

The Batch environment continues to improve with new functionality and workflow enhancements to help you now, and to prepare for future improvements.

Improved Node Bins in Batch and Modular Keyer

The Batch and Modular Keyer node bins are redesigned to simplify your workflow. Nodes are now displayed in alphabetical order, and divided into groups classified by tabs. You can also create and populate your own tabs with your favorite effects nodes.

In addition, you can create custom nodes in the Modular Keyer node bin and save them per user or per project.

For information on the Batch node bins, see Batch Node Bins on page 97 and Creating Custom Nodes on page 103.

For information on the Modular Keyer node bins, see Modular Keyer Node Bins on page 106, Customizing the Modular Keyer Bin on page 106, and Creating Custom Nodes on page 110.

- The new Sparks Load node allows you to populate multiple Sparks nodes at the same time into a destination bin. To reuse the populated Sparks, you just have to drag them from the bin to the schematic. You no longer have to access the Sparks browser. See Sparks Load Node on page 114.
- If you reorder nodes in any of the custom bins (not the All Nodes bin), you can now reset the nodes to alphabetical order by clicking the Sort button.
- A shadow transparency of a node now appears as you move the node in a bin.

Saving Sources and Setups

When you save a Batch setup, you can now save all sources associated with the setup to a working library with the new Snapshot feature. In addition, Inferno setups are saved such that they cannot be overridden by Flare setups.

See Saving Sources and Setups on page 114.

Load a Library Reel as a Group

You are now able to load a Library reel as a group from the Library node. This is useful in the context of multi-channel render passes, which can be more conveniently handled as one group.

See Library Node on page 119.

Group Node Improvements in Batch

The group node now displays a proxy to preview results. The group tabs user interface is improved for easier connection of group node tabs in the schematic. The group node menu has been redesigned into a more logical grouping of user interface items. See Grouping Nodes on page 120.

You can now display all the clips of a group in Timing View, and offset all or a selection of the clips. See Offsetting Clips on page 124.

Hide Input/Output Links of the MUX Node

The MUX node menu now includes toggle buttons that allow you to show or hide the input link to the MUX node or the output links from it.

See MUX Node on page 125.

Adding Media in Action

The ways in which you add media with the Action node have changed slightly. As well, you can now add media from the library or from the Desktop.

See Action Node on page 126.

Application-centric Improvements

Sometimes a small improvement to an existing feature can save you time and create a better workflow. The ultimate goal is to allow the application to help you by offering these enhancements.

Numeric Field Linking Improvements with Channel Editor

Coloured keyframe indicators are added to numeric fields, and fields are linked with their channels with hot keys to improve the workflow of the channel editor.

See Keyframe Indicators on page 135.

Contextual Menu Linked to Numeric Fields

A contextual menu can be displayed for numeric fields to provide shortcuts to channel editor-related operations. The workflow is similar to right-click menus in other applications.

See Selecting Channels Automatically on page 136.

Auto Key Button Look

A new preference available in the General section of the Preference menu allows you to set the look of the Auto Key button. A brighter option is available to help remind you that the Auto Key button is enabled.

Add Module Name as Prefix or Suffix on Rendered Clips

A new preference allows you to add a module acronym as a prefix or suffix to the name of a rendered clip.

See Rendered Clip Name on page 138.

Colour Management: Image Display Viewer Improvements

You can now gesturally bypass image data type presets in the image window. In addition to viewing the image in RGB mode, you can view it in a Matte mode with independent image display presets. See Controlling Image Display using Exposure and Image Data Type on page 139.

Broadcast Monitor: Display Grid/Guides Overlay

When the Show Selected Item option is enabled, any overlays are also displayed in the broadcast monitor. This applies to clips in the Player, modules, and clip libraries (letterbox only).

Broadcast Monitor: Support for 4:4:4 for GVO via NVidia SDI2

A new preference allows you to set 4:4:4 broadcast monitor output for extended monitor capabilities. This setting requires a broadcast monitor that supports the 4:4:4 colourspace and is connected by dual-link to the SDI card.

Running the Application from User Accounts

Users can now run the application from their own Linux user account, or use the default Inferno account. Projects now reflect correct ownership and are created with permissions set to 666.

The application no longer runs as root by default, but only when specific tasks require it.

Expected and Detected Sync

A new preference displays the detected and expected sync for the work session.

MUX Node Available in the Modular Keyer

Previously available only in Batch, the MUX node is used to create multiple outputs, typically from a group node or complex effect.

Per Surface Resolution Setting in Action

You can now set the geometry resolution per object in Action from the Surface menu, as well as a global preference in the Action Setup menu.

User Profiles

When connected to a remote framestore, you can now select a local or remote user. You can select a local or remote user from the start-up screen or from the Project Management section of the Preferences menu.

See Selecting a Project and User on Start-up on page 141.

When creating a user, you can copy a user profile as follows:

- You can copy a user profile from a local or remote system.
- You can copy all preferences from a user profile in the current version of the application
- You can copy only hot key preferences from a user profile in an older version of the application.

See Creating User Profiles on page 142.

The default user has been removed as an option from the User box.

Clip Library Improvements

The clip library has been improved with the following additions:

- An R/W button that gives you read-write access to Flare libraries. See Managing Flare Libraries from Inferno on page 32.
- A Go To Player button that allows you to go directly into the Player from the library with a selection of clips. See Playing Clips from the Library on page 144.
- A Hidden Libs button that displays libraries such as **_cache** and **._Backup**, which are by default not visible.
- A Show Libraries box that gives you the option of displaying libraries belonging to the current project (Current Project Libraries option) or all libraries belonging to projects to which you are connected through the network (All Libraries option).
- A newly improved Clip Library box that makes it is easier to distinguish between library types.

See Available Libraries on page 145.

Interoperability Workflow Improvements

As many new formats and codecs are introduced in the industry, it is important to create workflows that support them.

RED Workflow

High-quality transcoding of RED R3D media as a background task is now available via WiretapCentral, allowing for:

- import of R3D files
- batch import of R3D files referenced in an FCP XML or an EDL

You can also access WiretapCentral directly from the Autodesk Visual Effects and Finishing application, by selecting RED from the Import Image menu.

Import/Export DNxHD in QT Wrapper

DNxHD files in a QuickTime wrapper are supported for import and export. As well, for export, there are a number of new easy-to-use presets.

Recapture/Relink Improvements

The following recapture/relink improvements are introduced:

- The Recapture screen now contains updated options for media file search, import, and relink options.
- To facilitate file-based conform from EDL, XML, and AAF, the new search feature is able to intelligently and automatically find and read image sequences (DPX) or streaming media (MXF, QT), based on preset search rules and criteria.
- Relink problems are reduced with the new Copy from Selected Clip button. This button copies the formatting information of the selected clip into the Resolution parameters; in effect, providing the parameters (resolution, frame rate, bit depth, etc.) by which to “resize” a target clip, such as an FCP XML clip.
- A new Skip Recapture button appears in the Import XML and Import AAF menus. This is useful when importing timelines that point to media of a different format (such as when trying to relink to original sources).

For XML files, see Importing Final Cut Pro XML on page 149 and Relinking to File-based Media on page 152.

For AAF files, see Importing AAF Files on page 159 and Relinking to File-based Media on page 162.

P2 Support Improvements

You are now able to import P2 material shot in 24P or 24PA mode. You can also remove pulldown (regular and advanced) after import of P2 files.

Compressed Media: IMX in QT

IMX (MPEG-2) files are supported in QuickTime.

More Support for Avid AAF Transitions and Effects

There is now more support for Avid AAF transitions and effects. See Supported and Unsupported Transitions and Effects on page 169.

WiretapCentral Improvements

The following improvement have been made to WiretapCentral.

- You can now access WiretapCentral from Autodesk Visual Effects and Finishing applications. This is available in the Import Image menu when selecting OpenEXR or RED images for import.
- You can modify, re-order, add, or remove codecs listed in the export panel by editing a simple XML preset file on the server. You can also adjust the meaning of the quality settings (1-10) to suit your needs. The file resides in:
  `/var/www/html/WiretapCentral/presets/export_presets.xml`
- You can now download and install the latest version of ffmpeg alongside the Autodesk Visual Effects and Finishing version of ffmpeg (ffmpeg_flv), allowing you to leverage all the latest codecs and fixes added to ffmpeg as new builds become available.
- You can now export from WiretapCentral using the following codecs:
  - MPEG-2 in a QT wrapper
  - QT Animation

- AVI (WMV and MPEG-4)
- H264 (Main and High). Main is comparable to the quality and size of files generated in QT Pro. High is comparable to the default high quality settings shipped previously.

- The following changes have been made to performance in WiretapCentral:
  - Improved H264 quality, size, and performance.
  - Improved general encoding speed through various techniques, including multi-threading.
  - Upgraded ffmpeg support to the most current version.

- The following changes have been made to the interface in WiretapCentral:
  - The font size has been reduced to fit more on the screen.
  - The size of the clip thumbnails has been reduced by 60% to further maximize screen real-estate.
  - The playlist has been removed. Exporting will now work with clips selected from the main/centre view.
  - The Project and Clip details views have been merged.
  - There are now details on Server, Volume, and Library nodes.
  - When in fullscreen mode, you now have access to the main menu bar.
  - The choice to toggle the application of aspect ratio in the Player has been removed.

See your WiretapCentral User Guide for more information (now also included as part of the Help system installed automatically with Inferno).

Editorial and Timeline Improvements

Time-saving improvements when working in a timeline are invaluable, as they free you up to be more creative editorially.

Partial Invalidation of Segments in a Timeline

The amount of re-rendering necessary is now reduced when you modify frames that are part of a vertical composition in a timeline. When you modify frames, only the modified frames, including those overlapping in the vertical composition, are invalidated.

Navigate Through Tracks/Layers

You can now navigate multi-layer and multi-track video clips directly on the Desktop without having to go into the timeline. You can also change the focus layer directly on the Desktop.

See Navigating Edit Sequences on page 179.

Multilayer/Multitrack Gestural Editing

You can gesturally edit multilayer or multitrack clips on the Desktop except those containing dissolves.

Default Value of Editing Parameters

To conform to a typical editing workflow, the default status of the Sync, Trim, and Focus timeline options has changed to Enabled.

More EQ Bands in Audio

There are now a total of six EQ bands, or nodes, available for more precise manipulation of the audio frequencies: one Low node, four Mid nodes, and one High node. See EQ on page 183.

Muting in the Timeline

When you mute video or audio tracks, layers or soft effects, their indicators now turn black instead of yellow.

Selecting in the Timeline

A yellow bounding box around a timeline segment now 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.

New Hot Key for Deleting Layers

You can now use the **Alt+D** hot key to delete a layer from the Player timeline or Batch timeline.

Input/Output Improvements

Getting your clips in and out of the application continues to improve with support for new formats, and further enhancements to Real-Time Deliverables.

New Formats Supported Through the AJA Video Card

Embedded audio through the AJA video card now supports 16 audio tracks in the Input Clip, Output Clip, and AudioDesk menus. See Adjusting Audio Gain on Output Clip on page 185 and Using Output Strips on page 187.

Start Timecode per Real-Time Deliverable

Real-Time Deliverable can have a start timecode used when outputting the Deliverable to tape. The Player also displays the timecode of a Deliverable.

Setting Up a Flare Workflow

3

Topics in this chapter:

About Flare

Flare is a fully compatible assistant to Inferno, featuring the same creative toolset as Inferno Batch. All Batch nodes found in Inferno are fully supported in Flare. This extends the creative capabilities of Inferno by allowing any Batch task to be performed on a Flare system. It also allows Inferno to offload time-consuming tasks such as rotoscoping and particle creation.

With the focus of Flare being on the Batch toolset, there are some tasks performed by Inferno that cannot be performed by Flare. For example, Flare does not support video I/O, archiving, broadcast monitoring, or conforming.

You can use Flare in an independent system workflow or in a remote connection workflow. In a remote connection workflow, the same (Inferno) storage is used by both Flare and Inferno so there is no duplication of media. Multiple Flare systems can connect to the same Inferno system. Each can work on the same project at the same time, speeding up the production pipeline workflow. Overwriting work on a Inferno system from Flare is not a concern as special precautions have been put in place to ensure efficient collaboration between systems. As well, the Inferno artist can perform project management tasks of Flare libraries without leaving the Inferno station.

If being able to work on projects collaboratively with Inferno is not your main objective, you can still take advantage of the assistant capabilities of Flare in an independent system workflow. You need Wire to transfer media to/from a Flare system; therefore, there will be duplication of media. However, there are fewer workflow considerations than in a collaborative environment since Flare work is done on its own framestore.

The rest of this chapter discusses Flare in the context of a remote connection workflow.

Flare Workflow

The following workflow provides one example of working in a remote connection collaborative environment. In this example, Flare remote connects to a Inferno system and loads the setups and media it will work on directly from a Inferno project. Inferno has direct access to Flare setups and media at all times since Flare uses the Inferno framestore.

There may be some workflow differences, if, for example, Flare creates the initial Batch setups.

In the following workflow, steps without cross references are specific to Flare and are detailed in this chapter. Steps with cross references are covered in other chapters of the user's guide and may contain some Inferno functionality that does not apply to Flare. Any Flare-specific considerations are outside the scope of the other chapters.

Flare with Inferno workflow:

1 Start the application and connect to a remote framestore.

2 Load a Batch setup and associated clips from Inferno.

NOTE Make sure the clips were saved to a Snapshot library in Inferno before loading the Inferno setup. Because Flare does not have access to the Inferno Desktop, Inferno clips must be saved to a library in order for Flare to find them.

3 Create Snapshot libraries in which to save your clips.

4 (Optional) Add or replace clips in the current Batch setup.

5 Work with the Batch toolset. See the Batch chapters in the "Procedural Compositing with Batch" section of the user's guide.

6 Perform basic gestural editing operations such as trimming, slipping, and sliding on any source clip. See the Editing chapters in the "Editing" section of the user's guide.

NOTE Editorial operations that involve record and source clips, for example, 3- and 4-point edits, overwrites, inserts, and appends, are not supported.

7 Save your sources and Batch setups to a read-write library using the Snapshot feature.

8 Process by doing one of the following:

- Output clips to a read-write library.
- Export image sequences. See "Export Node" in the "Batch: Node Reference" chapter of the user's guide.

9 Play results processed with the Output node.

Considerations for Working in a Collaborative Environment

The ability to work in a collaborative environment is integral to the Flare remote connection workflow. From Flare, you can open a Inferno project on a remote framestore as well as access its libraries, media, and setups. Conversely, Inferno systems can access Flare projects, libraries, media, and setups.

This type of collaborative environment requires specific permissions for libraries, projects, and setups.

Library Permissions

In Flare, clips associated with Batch setups are saved in working libraries. Certain permissions are therefore required to prevent the accidental locking of libraries and overwriting of media. As such, the default read/write permissions for Flare and Inferno working libraries are set as follows:

- All Inferno-created libraries are read-only to Flare.
- All Flare-created libraries are read-only to Inferno.
- All Flare-created libraries are read-only to other Flare systems.

If you load a Batch setup from Inferno, the associated clips may be saved to a Inferno Snapshot library (read-only to Flare). The Snapshot library will be saved as a preference with the Inferno setup even if the clips have not been saved to a library. You will have to select a read-write library to be able to save the clips.

Project Permissions

Flare has read-only access to all projects created on a Inferno system. It cannot modify any Inferno project. This restriction allows multiple Flare systems to connect simultaneously to an already open Inferno project and preserve project settings created by the Inferno system.

Flare can, however, create projects on a Inferno system. A project created on a Inferno system from Flare will be owned by Inferno. Flare will not be able to modify the project after it creates it.

Setup Permissions

Flare setups are automatically saved with the **.flare** extension. They receive the **.flare** extension whether you create them locally or save a setup loaded from Inferno. This prevents the overwriting of Inferno setups since Inferno setups are never saved with the **.flare** extension. As well, it is not possible to add the **.flare** extension to a Inferno setup name manually. If a Inferno user downloads a Flare setup and then attempts to save it, the setup is converted to a Inferno setup by the automatic removal of the **.flare** extension.

Note, however, that nothing prevents you from overwriting a setup created on another Flare system or from deleting setups from another Flare system or from a Inferno system.

The first time you save a setup from Flare, the setup is automatically put in a Flare subdirectory, as follows:

~/batch/flare_<hostname>/<my_setup_name>.flare

Setups are saved in this subdirectory whether they were initially created in Flare or downloaded from Inferno. Although you can subsequently save setups in other directories, it may be easier to keep track of Flare setups (versus Inferno setups) if they are in their own subdirectory.

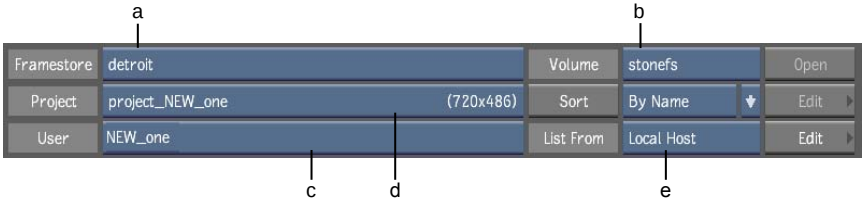
Starting Flare

To work in a remote connection environment, open a remote framestore on start-up.

To start Flare:

1. Double-click the application icon on the desktop.

   The Project Management menu appears.



(a) Framestore box (b) Volume box (c) User box (d) Project box (e) Host box

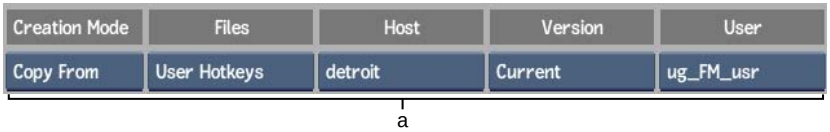
2. Select a remote framestore from the Framestore box. If the framestore has more than one volume, select a volume from the Volume box. Click Open.

3. Select a project from the Project box or create one. See "Creating a Project" in the "Managing Projects and Users" chapter of the user's guide.

NOTE You cannot modify projects created on a remote framestore. If you create a project while connected to a remote framestore, it resides on the remote framestore. You cannot modify the project after creating it.

4 Select or create a user:

- To select a user, select the user name from the User box. You can select a local or remote user by first selecting the appropriate option from the Host box.
- To create a user, select <create new user> from the User box. In the Create User menu that appears, you can choose to create a user by copying a user profile.



(a) User Profile copy options

If you copy a user profile, note the following:

- You can copy a user profile from a local or remote system.
- You can copy all preferences from a user profile in the current version of the application.
- You can copy only hotkey preferences from a user profile in an older version of the application.

NOTE If you create a user profile while connected to a remote user, the new user will reside on the remote host and not on your local machine.

See "Creating User Profiles" in the "Managing Projects and Users" chapter of the user's guide.

5 Click Start.

You enter the Batch module. You have access to the same creative toolset as is found in Inferno Batch.

To exit Flare:

1 From main level Batch, click Preferences.

2 Click EXIT Flare.

Changing Hosts, Projects, and Users

You can change hosts at any time within a Flare session. You can also create or load other projects, as well as create, change, and edit user profiles.

You do these tasks from the Project Management section of the Preferences menu.

You create projects and users in the same way as from the start-up menu. See the “Managing Projects and Users” chapter of the user’s guide.

To access the Preferences menu:

➤ In main level Batch, click Preferences.

The Project Management section is in the upper-left side of the Preferences menu.

(a) Change (Host) button (b) Project box (c) User box (d) Host box

To change projects in a Flare session:

➤ Select a project from the Project box and click Load.

To change users in a Flare session:

➤ Select a user from the User box and click Load.

NOTE You can select a user on the local host or on the remote host. Select the appropriate option from the Host box and then select the user.

To edit a user profile in a Flare session:

➤ Select a user from the User box and then click Edit.

NOTE When a remote user profile is selected, the profile is unavailable to other users. This avoids concurrent user profile modifications.

To change hosts in a Flare session:

1. Click Change.
   You are brought to the start-up screen.
2. Select another host from the Framestore box and click Open.

Working with Libraries

All clips in Flare are stored in working libraries. Use the libraries to import image sequences, play clips, load clips into Batch, and output processed clips from Batch. As well, all clips that you save as part of your Batch setups are saved to working Snapshot libraries.

You can create working libraries from the following locations:

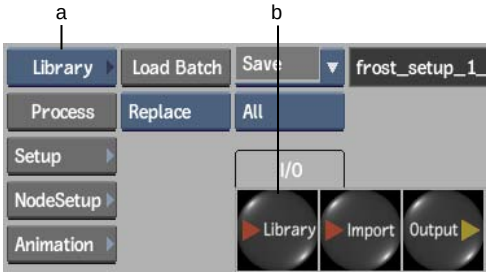
- The current clip library
- The Setup menu and the Save Setup & Snapshot browser when saving sources and setups
- The Output node menu when processing clips

To access read-only libraries and network libraries, and to manage read-write libraries, you must use the current clip library.

For information on managing clip libraries, see the “Clip Libraries” chapter of the user’s guide.

To access the clip library:

➤ Double-click the Library node, or select Library from the Library/Preference box (in main Batch only).



(a) Library/Preference box (b) Library node

To create a working library from the current clip library:

1 Select <create new library> from the Clip Library box.

2 Enter a library name in the keyboard that appears and press **Enter**.

The library is created in the panel to the right of the Clip Library box and the library name appears in the Clip Library box.

Identifying Libraries

To make it easier to identify where libraries are located and whether they belong to the current project, libraries appear in the following order in the Clip Library box:

- First, libraries that you own and that belong to the current project appear.
- Secondly, libraries that you do not own and that belong to the current project appear.
- Thirdly, libraries belonging to other projects on the system to which you are connected appear.
- Lastly, libraries belonging to other projects on other systems appear.

All read-only libraries provide more information about their status in the Clip Library box and in the Library status bar.

(a) Current project libraries owned by Flare (b) Current project libraries not owned by Flare (c) Network library of another project on the system to which Flare is connected (d) Network library of another project on another system

---

NOTE Inferno has the ability to change access permissions of Flare libraries, which are, by default, read-only to Inferno. If you are unable to modify media in a read-write library, the library may be locked by Inferno.

Displaying Libraries

If you accessed libraries through the network panel, the list of available libraries in the Clip Library box may become very long. You can select whether you want the network-accessed libraries to appear in the Clip Library box.

You can also show hidden libraries such as **_cache** and **._Backup**, which are by default not visible.

To control the working libraries displayed in the Clip Library box:

➤ Select an option from the Show Library box.

Select:	To display:
All Libraries	All libraries belonging to all projects, including those accessed through the network panel.
Current Project Libraries	All libraries belonging to the current project. Note that if the current library is accessed through the network panel, this option is greyed out.

NOTE If you are in Dual View, each view has its own Show Library box. You can select any option for either view.

To display hidden libraries in the Clip Library box:

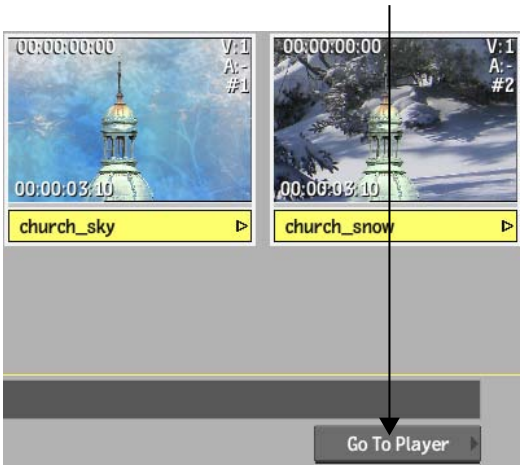
➤ Enable Hidden Libs.

Playing Clips from the Library

You can play a selection of source clips and processed clips directly from the library.

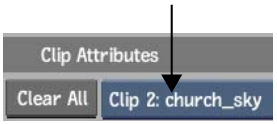
To play clips from the library:

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 Flare Libraries from Inferno

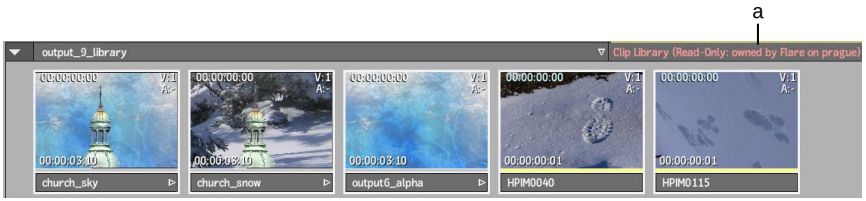
As an assistant to Inferno, Flare does not share the same permissions as Inferno. Although Flare libraries are read-only to Inferno, Inferno can enable read-write access for Flare libraries in order to perform library management tasks.

To enable read-write access of a Flare library from a Inferno system, the library must not be locked by the Flare system (it must not be the current Flare library).

To enable read/write access to Flare libraries from Inferno:

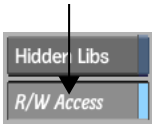
1 From Inferno, access the clip library from the Desktop or double-click the Library node in Batch.

The Library status bar indicates all Flare libraries as read-only.

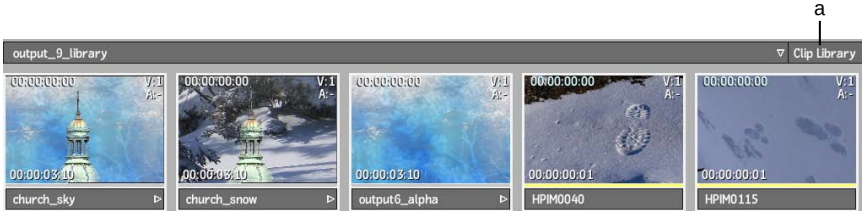


(a) Read-only library status

2 Enable R/W Access.



The Library status bar no longer indicates that the library is read-only. You have read-write access to all libraries in the current project (except the current library in Flare) until you disable the R/W button or exit the clip library.



(a) Read-write library status

NOTE If you are unable to modify media in a library in which you enabled read-write access, the library may still be locked by a Flare system.

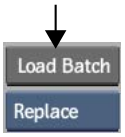
Loading Inferno Setups and Sources

When you load Batch setups, associated clips are automatically loaded with the setup. If you load a setup from Inferno, make sure the clips were saved to a Snapshot library in Inferno before you load the setup; Flare will not find clips saved to the Inferno Desktop.

To save clips associated with a Inferno setup, you must select a read-write Snapshot library.

To load a Batch setup created in Inferno:

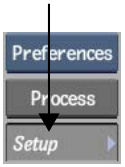
1 Click Load Batch.



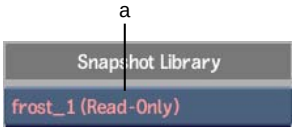
2 Select a Inferno setup from the **/batch/inferno** directory. Inferno setups do not have extensions in their name.

The setup and all associated clips are automatically loaded into Batch.

3 Click Setup.



The Snapshot library in Inferno was saved as a preference with the Inferno setup.



(a) Inferno Snapshot library

4 To be able to save the clips associated with the loaded setup, change the library to a read-write library. Select a Snapshot library from the Snapshot Library box or select <new> and create a library.

NOTE You can also select a Snapshot library from the Save Setup As & Snapshot browser when taking a snapshot of your sources.

(a) Flare Snapshot library

When you save sources and setups, the clips will be saved to this Snapshot library. The library will be saved as a preference with the Flare setup.

Adding Clips to a Setup

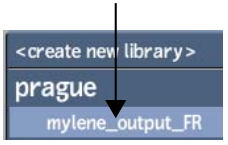
You can load clips into an empty setup as well as add clips to the current setup using the Library or Import node. To replace a clip, you select one from the library.

When adding media with the Action node, you can either add media to the Media list or to an indirect layer of the Action node.

If you are replacing clips or adding media with the Action node, you are limited to the number of clips you can select from the library. The message bar in the library indicates when you can no longer select additional clips.

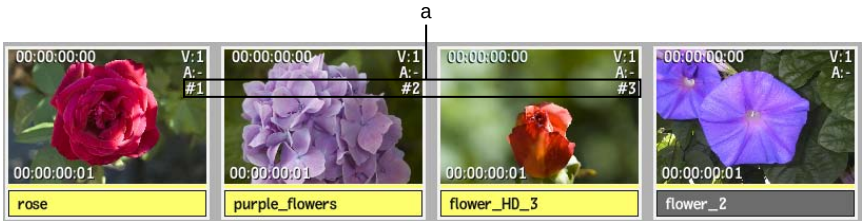
To load clips into Batch from the library:

1. Go to the library.
2. Select a library from the Clip Library box.



3. Do any of the following:
   - Click a clip to select it.
   - To select multiple clips, **Ctrl**-click each clip.
   - To select all clips in a range, select the first clip and then **Ctrl+Shift**-click the last clip in the range.

   Numbers appear in the clip indicating the order in which they will be loaded into Batch.



(a) Clip selection order

4 Click Load.

The clips are loaded in the order selected. Blue clip names indicate the clips as library clips.

To replace clips in Batch:

1 Double-click the clip in the Batch schematic.

The library appears.

2 Select the replacement clip.

NOTE You can only select one clip when replacing a clip. If you select a second clip, the first clip is deselected and the second clip becomes number 1.

---

3 Click Load.

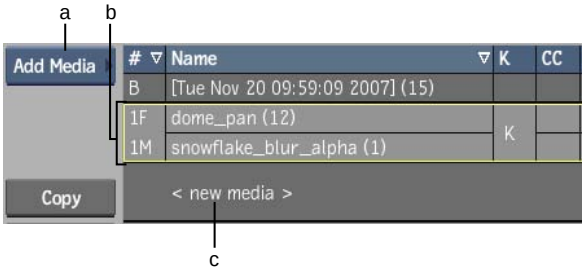
The original clip is replaced with the new selection.

To add media with the Action node:

➤ Do one of the following:

- To add media to the Media list, select Add Media from the Media List box. Select a clip in the library and then click Load. You can only select two clips (front and matte) when adding media with this option.
- To add an indirect layer to the Action node, select the <new media> line from the Media list and then select Add Input from the Media List box. Connect front and matte clips to the indirect layer.
- To replace both the front and matte media in the Media list, double-click the 1F (Front) or 1M (Matte) Media line (in the # column), and then select the front and matte clips.

- To replace either the front or matte media in the Media list, double-click the Front or Matte Media Name line (in the Name column), and then select the front or matte clip.



(a) Media List box (b) Front and Matte Media lines (c) New media line

For more information, see “Action Node” in the “Batch: Node Reference” chapter of the user’s guide.

Saving Setups and Sources

You can save setups created in Flare as well as setups loaded from Inferno.

When saving setups, you can also save the sources used by the setup. You do this by taking a snapshot of the sources. When you take a snapshot, all source clips, including all BFX clips used by the current main level setup, are saved to a working library. Sources contained in BFX clips, however, do not appear. They are part of the BFX clip.

Although sources are saved separately from setups when a snapshot is taken, they are saved concurrently. As well, the corresponding Batch setup is updated such that it points to the newly saved sources in the working library.

With the Snapshot feature, you do not have to worry about Flare not being able to find source clips associated with a setup.

To ensure that modified clips always get saved with a setup, and to prevent Inferno setups from being accidently overwritten from a Flare system, the rules for saving sources and setups are determined as follows:

- If a clip has not had any editorial operations performed on it, you can save both setups and sources, or you can save only the setups.
- If a clip has had editorial operations performed on it or has been replaced, you must save both setups and sources; you cannot save only the setups.

- If you are saving a Flare setup, you can overwrite a previous setup with the same name or save the setup under a new name. Flare setups are always saved with a **.flare** extension.
- If you are saving a Inferno setup, you can only save the setup under a new name; you cannot overwrite the Inferno setup. A **.flare** extension is always added to the name, whether you rename the setup or try to keep the same name.

The first time you save a Flare setup, the setup is automatically put in a Flare subdirectory as follows:

~/batch/flare_<hostname>/<my_setup_name>.flare

Inferno setups are never saved with an extension. Inferno setups are saved in a Inferno subdirectory as follows:

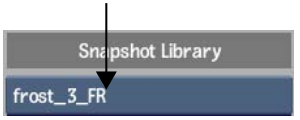
~/batch/inferno/<my_setup_name>

Although you can take snapshots explicitly, automatic snapshots are also taken when you exit Flare. Automatic snapshots are saved to a hidden library (**._Backup**), and not to a working library.

You specify the working Snapshot library to which you want sources saved. Sources are saved to a predefined reel structure named according to the setup name. When you load a setup, its corresponding sources in the Snapshot library are loaded with the setup.

To specify the library for saving sources:

1. From Batch, click Setup.
2. From the Snapshot Library box, select the library to which you want to save sources or select <new> and create a library.



NOTE If you loaded a setup from Inferno, the name in the Snapshot Library box has **(Read-Only)** beside its name.

This Snapshot library will be saved as a preference with the Batch setup. If you subsequently delete or rename the library, the name in the Snapshot Library box will have **(Non-Existent)** beside its name. You will be

prompted to have the missing library created the next time you take a snapshot.

NOTE The Snapshot Library box also appears in the Save Setup As & Snapshot browser when using the Save Setup As & Snapshot option.

To save sources and/or setups:

➤ In Batch, select one of the following options from the Save dropdown list.

Save Setup Saves a setup. The first time you save a setup, you are prompted to name it in the keyboard that appears.

It is impossible to overwrite a setup loaded from Inferno with this option since Flare setups are automatically given a **.flare** extension when saved. If you rename the Inferno setup, the **.flare** extension is automatically added to the name. If you try to overwrite the setup with the same name, a message appears offering to rename the setup by adding the **.flare** extension.

Each subsequent time you save the Flare setup with this option, you are overwriting the previous saved setup. You are prompted to confirm the overwrite. To bypass the confirm, press **Alt** as you select Save Setup.

Save Setup As Saves the setup with a new name, which you enter in the keyboard that appears. Each time you save a setup with this option, you are creating a new saved setup.

A **.flare** extension is automatically given to the setup name.

Save Setup & Snapshot Saves the setup and takes a snapshot of the sources used by the setup.

The first time you save a setup and sources, you are prompted to name the setup in the keyboard that appears. The snapshot of the sources is saved to the Snapshot library you specify.

It is impossible to overwrite a setup loaded from Inferno with this option since Flare setups are automatically given a **.flare** extension when saved. If you rename the Inferno setup, the **.flare** extension is automatically added to the name. If you try to overwrite the setup with the same name,

a message appears offering to rename the setup by adding the **.flare** extension.

Each subsequent time you save the Flare setup and sources with this option, you are overwriting the previous saved setup as well as the sources in the library. You are prompted to confirm the overwrite. To bypass the confirm, press **Alt** as you select Save Setup & Snapshot.

Save Setup As & Snapshot Saves the setup with a new name, which you enter in the keyboard that appears, and takes a snapshot of the sources used with the setup.

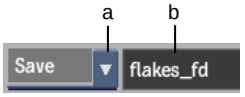
A **.flare** extension is automatically given to the setup name. The snapshot of the sources is saved to the Snapshot library you specify.

Each time you save a setup and sources with this option, you are creating a new saved setup. A new snapshot of the sources is saved to the library with the new setup name.

With this option, you can change or create a new Snapshot library directly from the Save & Snapshot browser without having to go back to the Setup menu.

Example: Saving sources and setups loaded from a Inferno system:

1 In Batch, select Save Setup As & Snapshot from the Save dropdown list.



(a) Save dropdown list (b) Loaded Inferno Batch setup

The Save Setup As & Snapshot browser appears.

NOTE Because you cannot overwrite a Inferno setup, the Save Setup As & Snapshot browser also appears if you select Save Setup & Snapshot, prompting you to rename the setup.

2 Select a read-write library in which to save the sources associated with the setup.

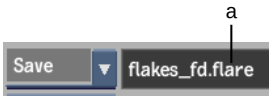
NOTE In this example, when the setup was saved in Inferno, the clips were saved to the Snapshot library (not to the Desktop). The Inferno Snapshot library was saved as a preference with the setup.

3 Rename the setup or keep the same name, and then click Save.

If you tried to keep the same name, a message appears with a prompt that the setup cannot be saved and offers to add the **.flare** extension to the setup name. Confirm.

The setup is automatically saved with the **.flare** extension (whether you renamed the setup or kept the same name).

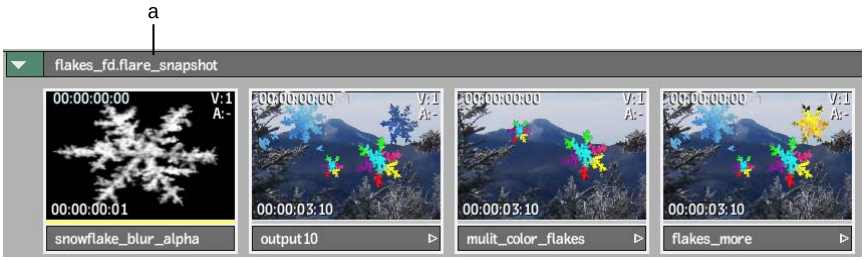
4 In Batch, the name of the Flare setup appears in the Filename field.



(a) Flare Batch setup name

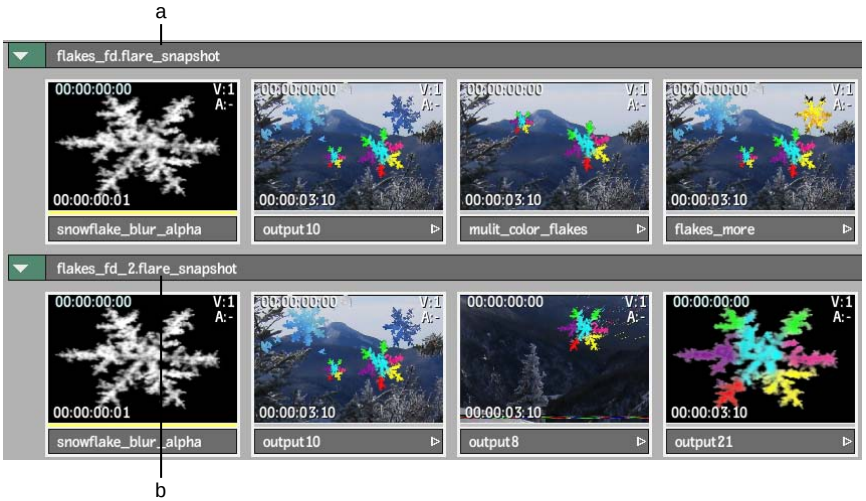
5 Go to the library.

6 The sources associated with the setup are saved in the Snapshot library in a predefined reel entry named according to the Batch setup name with _snapshot appended to the name. When the setup is loaded in a Batch session, all clips in the reel are loaded as well.



(a) Reel entry named according to Batch setup name

In the following example, the same setup is saved again with the Save Setup As & Snapshot option. A new snapshot of the sources associated with the renamed setup is taken. Another reel is created in the library with the new setup name (flakes_fd_2.flare_snapshot). The previous snapshot (and setup) is not overwritten.



(a) First snapshot taken with Save Setup As & Snapshot (b) Second snapshot taken with Save Setup As & Snapshot

NOTE To overwrite the previous snapshot (and not create a new reel), select Save Setup & Snapshot.

Each saved Batch setup points to its corresponding reel in the Snapshot library. When either setup is loaded in a Batch session, all clips in the corresponding reel are loaded as well.

Processing Clips

When you process clips, you select the destination library to which you want to output the processed clips. The Output node saves the destination library as part of the Batch setup.

If you load a setup from Inferno, its Output node will be pointing to a Inferno library (read-only to Flare). You must change the output library to a read-write library before processing.

You can play processed clips directly from the Batch Player or from the destination library.

To select the library in which to output clips:

1 Double-click the Output node.

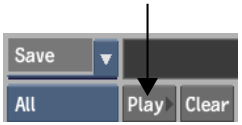
2 Select a destination library from the Library Selection box or select <new> and create one.

This preference will be saved with the Batch setup. If you subsequently delete or rename the library, the library name in the Library box will have **(Non-Existent)** beside its name. You will be prompted to have the missing library created the next time you process.

To play processed clips from the Output node:

1 Process the clips. See "Output Node" in the "Batch: Node Reference" chapter of the user's guide.

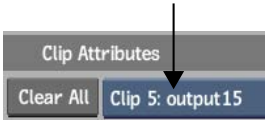
2 Click Play.



NOTE To play processed clips from the destination library, select the clips in the library and then click Go To Player.

All processed clips are brought into the Player.

3 To go from one output to the next, press **Ctrl+ right or left arrow** or select a clip from the Playback box.



4 To clear the playlist, click Clear.

NOTE Clearing the playlist does not delete the clips from the library.

Floating Point Workflow

4

Topics in this chapter:

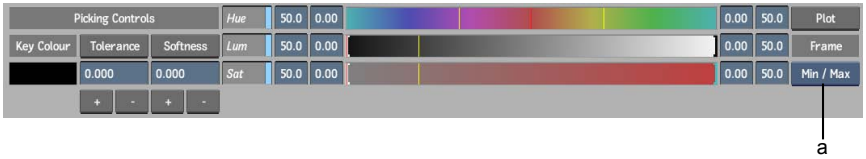
Keyer-Channel Node

New for this release: The Keyer-Channel node is new to Batch and the Modular Keyer. Unlike the Keyer node, this node supports floating-point images. Use the Keyer-Channel node to extract a key from a red, green, or blue channel, or from a custom value.

The Picking Controls are the same as those found in the Keyer module's Channel menu.

Keyer-HLS Node

New for this release: The Keyer-Channel node is new to Batch and the Modular Keyer. Unlike the Keyer node, this node supports floating-point images. Use the Keyer-HLS node to extract a key by adjusting tolerance and softness using hue, luminance, and saturation ranges.



(a) Frame box

The Picking Controls are the same as those found in the Keyer module's HLS menu. Set softness and tolerance ranges using the hue, luminance, and saturation channels.

Use the Frame box to select a colour range display option.

Select:	**To:**
Home	Reset the view.
Plot Colour	Display the range values between 0 and 1 and the plot values (display is the same as Full range for logarithmic and video input).
Full Range	Display the entire range.
Min/Max	Display the range values between the minimum and maximum slider values.

Keyer-Luma Node

New for this release: The Luminance Key node has been renamed to the Keyer-Luma node. Unlike the Keyer node, this node supports floating-point images.Use the Keyer-Luma node to extract a key from the luminance of a clip. A front clip can be connected to the Keyer-Luma node and will process a result clip that can be used as a matte.

Tolerance determines the threshold value, which is the matte minimum value output and is displayed as a white bar in the histogram. Softness determines which clip value to use as the matte maximum value. The matte maximum value is relative to the tolerance range and appears as a yellow bar in the histogram.

(a) Frame box

Use the Frame box to select a colour range display option.

Select:	To:
Home	Reset the view.
Plot Colour	Display the range values between 0 and 1 and the plot values (display is the same as Full range for logarithmic and video input).
Full Range	Display the entire range.
Min/Max	Display the range values between the minimum and maximum slider values.

Tolerance removes greys outside the key shape. Softness adjusts the softness of the edges of the matte. These settings can be animated and are available as channels.

Use the controls in the Relative To panel to indicate how tolerance and softness values are calculated.

Relative To box Select to calculate softness and tolerance ranges relative to 0 and 1 colour values, or relative to the maximum luminance.

Relative To field Set the maximum luminance. This field is active if the Maximum Luminance option is selected in the Relative To box.

Get Maximum Value button Analyse the image to determine the maximum luminance value.

Keyer-RGB Node

New for this release: The Keyer-RGB node is new to Batch and the Modular Keyer. Unlike the Keyer node, this node supports floating-point images. Use the Keyer-RGB node to extract a key by adjusting tolerance and softness using red, green, and blue ranges.

(a) Frame box

Use the Frame box to select a colour range display option.

Select:	To:
Home	Reset the view.
Plot Colour	Display the range values between 0 and 1 and the plot values (display is the same as Full range for logarithmic and video input).
Full Range	Display the entire range.
Min/Max	Display the range values between the minimum and maximum slider values.

Keyer-RGBCMYL Node

New for this release: The Keyer-RGBCMYL node is new to Batch and the Modular Keyer. Unlike the Keyer node, this node supports floating-point images. Use the Keyer-RGBCMYL node to extract a key by adjusting tolerance and softness using red, green, blue, cyan, magenta, yellow, and luminance ranges.

(a) Frame box

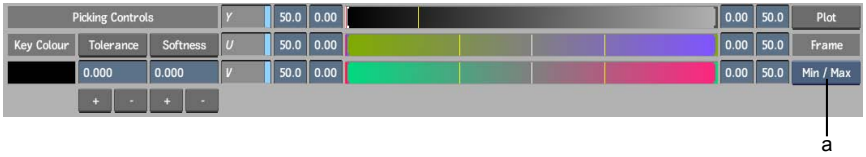
The Picking Controls are the same as those found in the Keyer module's RGBCMYL menu. Set softness and tolerance ranges using the red, green, blue, cyan, magenta, and yellow channels.

Use the Frame box to select a colour range display option.

Select:	To:
Home	Reset the view.
Plot Colour	Display the range values between 0 and 1 and the plot values (display is the same as Full range for logarithmic and video input).
Full Range	Display the entire range.
Min/Max	Display the range values between the minimum and maximum slider values.

Keyer-YUV Node

New for this release: The Keyer-YUV node is new to Batch and the Modular Keyer. Unlike the Keyer node, this node supports floating-point images. Use the Keyer-YUV node to extract a key by adjusting tolerance and softness using luma (Y) and video component (U, V) ranges.



(a) Frame box

The Picking Controls are the same as those found in the Keyer module's YUV menu. Set softness and tolerance ranges using the luma and chroma signals of YUV component video.

Use the Frame box to select a colour range display option.

Select:	To:
Home	Reset the view.
Plot Colour	Display a colour range that includes colour values and the plot values (display is the same as Full range for logarithmic and video input).
Full Range	Display the entire range.
Min/Max	Display the range values between the minimum and maximum slider values.

Setting the Display of the RGB Viewer

New for this release: The RGB Viewer in the 3D Keyer now takes colour management settings into account. The Key menu displays new framing options for floating-point colour information in the Histogram panel. When a percentage of the full range is displayed in the 3D histogram, you can also use the controls in this panel to store and retrieve the range value.

Setting the 3D Histogram Display

The 3D histogram provides a visual representation of the colours in the key-in clip. It shows how the colours in your clip are distributed in RGB colour space, with exposure and contrast settings taken into account. See Controlling Image Display using Exposure and Image Data Type on page 139.

When a floating-point image is used as a key-in clip, the RGB Viewer will support this input and also activate options to change the display of colour space in the 3D histogram. These options are available in the Histogram panel of the Key menu. When the range in the histogram exceeds 1, a cube outline indicating the colourspace from 0 to 1 is displayed in the RGB viewer.

Frame option box Select an option to display the entire histogram, or a selected range of values.

Select:	To display:
All Objects **(Shift+A)**	All objects in the 3D histogram. This is the default setting.
(0,1) **(Shift+0)**	Objects with colour values between 0 and 1.
Free **(Shift+F)**	Objects within a user-defined range. Use the Range field to navigate between views including the full range view and a percentage of the range.
Plot **(Shift+O)**	Objects with colour values between a range that comprises all values between 0 and 1, and the plot value.

Range field Displays the percentage of the full range that is displayed in the histogram. Editable when the Frame option box is set to Free.

Store button Click to store the current value in the Range field in memory for later use. Active when the Frame option box is set to Free.

Recall button Click to retrieve the last Range value that was stored in the buffer.

Colour Picker

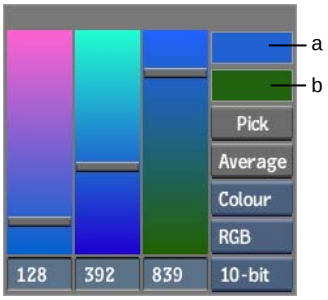
New for this release: The colour picker now supports 16-bit floating point colours.

When you click a colour pot, the colour picker appears. You can then set the colour pot to use the colour you need. Using the colour picker, you can pick colours by:

- Setting colour model channel values
- Sampling pixels in a clip
- Selecting a colour pot
- Mixing colours on a palette

The colour picker takes LUTs and the exposure and contrast settings into account. Disable LUTs and reset exposure and contrast to display actual colour values.

NOTE The colour picker used with overlays (grids, letterboxes...) does not take LUTs or the exposure and contrast settings into account.



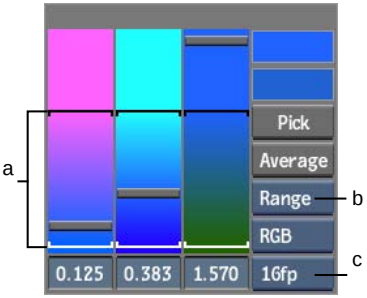
(a) Current Colour pot (b) Reference Colour pot

By default, the colour picker appears over the colour pot you clicked to open it. Once it is open, you can move it to another location by dragging the grey bar along its upper edge. You can also set colour picker preferences to open the colour picker at other locations (the colour picker inherits the same preferences as those you set for the calculator).

To cancel colour picking at any time, click anywhere outside the colour picker.

Using the Colour Picker with 16-bit Floating Point Colours

The colour picker allows you to pick colours from a 16-bit floating point media. And to accomodate the wide range of possible values in 16-bit floating point, the colour picker displays additional information.



(a) Range Brackets (b) Mode box
(c) Bit Depth box

Whenever you set the Bit Depth box to 16fp:

- Each colour slider contains a pair of brackets. The brackets define the 0-1 colour range. Only values inside this 0-1 range are valid once converted to a 8-, 10-, and 12-bit colours.
- You can set values below 0 or over 1. Such values are only possible in the 16-bit floating point colourspace and are not valid in 8-, 10-, and 12-bit colours.
- The numeric fields can use negative values.

Picking Colours by Setting Colour Model Channel Values

You can pick colours by adjusting colour model channel sliders. You can also enter channel values directly into the fields below each slider.

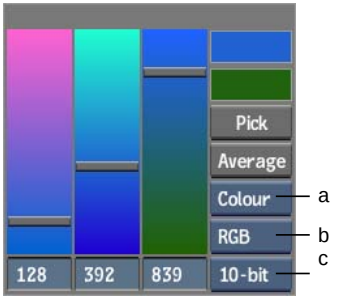
The Mode box, Colour Model box, and Bit Depth box settings, and channel value units (percentages or bit-values) are saved on a per-user basis, at the end of each session.

To pick colours by setting colour model channel values:

1 Click a colour pot to open the colour picker.

The Current Colour pot and Reference Colour pot both display the incoming colour.

2 (Optional) To pick a colour at a bit depth different from the one of the displayed image or clip, select a an option from the Bit Depth box.



(a) Mode box (b) Colour Model box (c) Bit Depth box

NOTE The option selected in the Bit Depth box only affects the representation of colours in the picker: it does not affect the bit depth of the displayed image or clip.

3 From the the Mode box, select how the numeric values represent each channel.

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

4 From the Colour Model box, select the colour model you want to work with.

Select:	To set colours using the:
RGB	Red, green, and blue channels.
HLS	Hue, luma, and saturation channels.
YUV	Luma (Y) and chroma (U, V) channels.

5 Adjust the sliders, drag the numeric fields, or enter the values in the fields for each slider.

As you adjust the sliders, the colour in the Current Colour pot changes to reflect the current colour. You can compare the current colour to the incoming colour in the Reference Colour pot.

6 To apply the selected colour, click the Current Colour pot.

Sampling Colours in a Clip

Sampling pixels in a clip is often the best way of setting the colour you need. For example, to suppress colour spill when keying a clip, the best way to set the colour suppression target is to zoom in on the result clip and then sample the colour spill directly.

You can sample single pixels, take an average along a path, or take an average from inside a selection box.

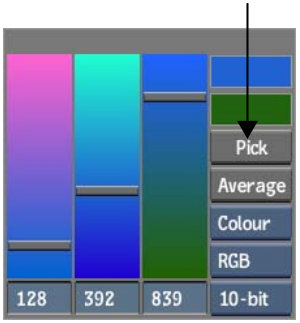
NOTE Applying a LUT or changing exposure/contrast affects the display, but colour picking is done using the original values of the media. Disabling LUTs and resetting exposure/contrast values will show the actual pixel values.

To sample a pixel:

1 Click a colour pot to open the colour picker.

The Current Colour pot and Reference Colour pot both display the incoming colour.

2 Click Pick.



3 Click in the Player or image window to sample a pixel in the clip.

The sampled colour appears in the Current Colour pot.

4 To apply the selected colour, click the Current Colour pot.

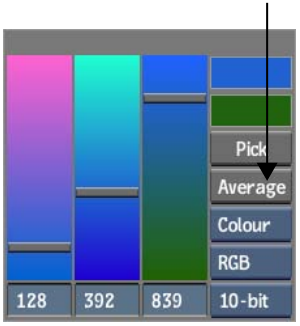
TIP From any colour pot, **Shift**-click to enter Pick mode. Clicking while dragging the Pick icon over an image will display its RGB values. Click again to transfer the colour to the colour pot.

To sample an average colour along a path:

1 Click a colour pot to open the colour picker.

The Current Colour pot and Reference Colour pot both display the incoming colour.

2 Click Avg.



3 Drag a path in the Player or image window to take an average from the clip.

The sampled colour appears in the Current Colour pot.

4 To apply the selected colour, click the Current Colour pot.

TIP To sample from paths in different parts of the clip, press **Alt** when finishing the first path-sample and then begin another path elsewhere. Repeat if necessary. Release the cursor without pressing **Alt** to apply the cumulative average to the Current Colour pot.

To sample an average inside a selection box:

1 Click a colour pot to open the colour picker.

The Current Colour pot and Reference Colour pot both display the incoming colour.

2 Click Avg.

3 **Ctrl**-drag a path in the Player or image window to take an average from the clip.

The sampled colour appears in the Current Colour pot.

4 To apply the selected colour, click the Current Colour pot.

Selecting a Colour from the Colour Pots

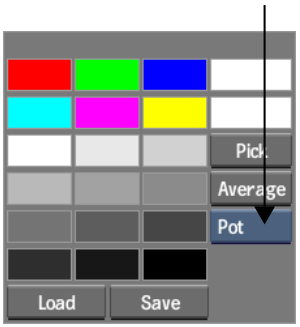
Select from 1 of 18 preset colours in the colour pots. You can also store custom colours in the colour pots.

To select a colour from the colour pots:

1 Click a colour pot to open the colour picker.

The Current Colour pot and Reference Colour pot both display the incoming colour.

2 From the Mode box, select Pot.



3 Click one of the colour pots to apply the colour to the Current Colour pot.

4 To apply the selected colour, click the Current Colour pot.

To customize the colour pots:

1 Click a colour pot to open the colour picker.

The Current Colour pot and Reference Colour pot both display the incoming colour.

2 Use the colour picker to apply the colour that you want to store to the Current Colour pot.

3 From the Mode box, select Pot.

4 Click and hold on the pot in which to store the selected colour.

When you select Pot from the Mode box, Save and Load buttons appear at the bottom of the colour picker to save custom sets of colour pots.

Mixing Colours Using the Colour Palette

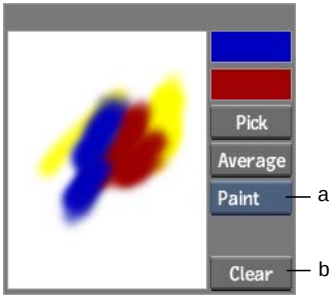
You can use the colour picker to mix colours on a palette.

To mix colours:

1 Click a colour pot to open the colour picker.

The Current Colour pot and Reference Colour pot both display the incoming colour.

2 From the Mode box, select Paint.



(a) Mode box (b) Clear button

3 Use the colour picker to apply the colour that you want to store to the Current Colour pot.

4 Drag over the mixing palette to add a swatch of the current colour.

5 Repeat the steps 3 and 4 to add other colours to the palette.

Colour mixing occurs as more colours are added and they blend.

6 To clear the mixing area at any time, click Clear.

7 To select a colour from the mixing palette, click Pick and then click in the mixing area.

The mixed colour appears in the Current Colour pot.

8 To apply the selected colour, click the Current Colour pot.

Creative Tools

5

Topics in this chapter:

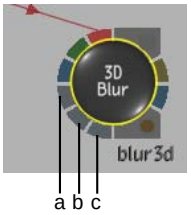
3D Blur Node

New for this release: The 3D Blur node is new to Batch. The 3D Blur node uses Z-depth map information to create a plausible blur effect. The node accepts a front, back, and matte input. Additional inputs are available for Z-depth map, motion data, and kernel media.

The 3D Blur node can be used as:

- A defocus node, that can be modulated by a Z-depth map.
- An artistic blur node, by modifying the pattern of the highlights. This can be done by editing the pattern profile curve or attaching an external clip to the appropriate node input.
- A motion blur node, by including forward motion data.

■ A node using a combination of these scenarios.



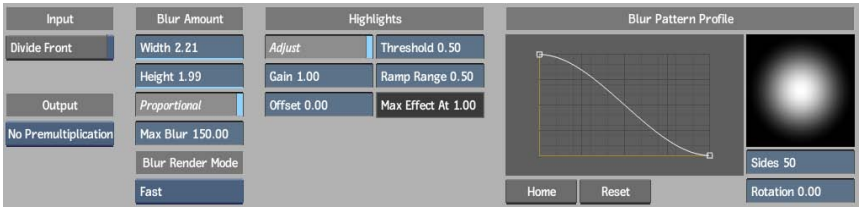
(a) Z-depth input tab (b) Forward Flow input tab (c) Kernel input tab

The following table describes a general workflow for the 3D Blur node.

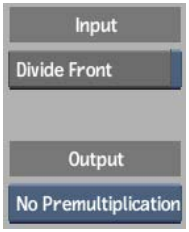
Step:	Refer to:
1. Apply a lens blur to the image.	See Creating a Lens Blur on page 62.
2. Determine the shape of the blur.	See Changing the Blur Pattern on page 64.
3. Apply depth of field effects to the image.	See Simulating a Depth of Field Effect on page 65.
4. Apply motion blur effects to the image.	See Simulating a Motion Blur Effect on page 68.

Creating a Lens Blur

Lens blur simulates the blur created by a camera lens, such as a rack defocus. Lens blur settings are displayed in the Basic menu.



The primary input of the 3D Blur node is the front clip and the matte clip. A blur effect can be created with a front clip only. However, a matte clip allows you to use the Input controls to determine how the matte is used. The Output controls determine how the input is composited in the Result view.



Divide Front Button Enable to unpremultiply the front clip using the matte clip. This button is active if you are using a matte.

Output box Select an option to determine how to composite the output in the Result view. The output can be a blurred premultiplied or unpremultiplied result using the front clip and matte clips, or a composite of the blurred front clip over a background clip. Premultiplied output is equivalent to compositing the resulting front on a black background using the resulting matte. This button is active if you are using a matte.

Use the Blur Amount controls in the Basic or Depth tab to refine and constrain the amount of blur to apply to the image and to constrain the amount of blur. Higher blur values increase processing time. You can increase processing performance by decreasing the blur quality. When you are ready to create a result clip and an output matte, you can select the highest quality blur.

Width field Enter the horizontal blur amount in pixels.

Height field Enter the vertical blur amount in pixels.

Proportional button Enable to constrain blur amount proportions.

Max Blur field Displays the maximum total horizontal and vertical blur by the entered amount. The cumulative effect of the high blur amount (Basic tab), and gamma and gain correction applied to the Depth map (Depth tab) may result in very large values, which greatly increases the processing time.

Use the Max Blur amount to clamp the value and avoid some unnecessary processing.

Blur Render Mode box Select fast processing of the blur or the highest quality blur. Quality mode provides accurate results, but requires more processing time than Fast mode.

You can offset the light and colour values in the source image. Highlights are applied to the image before the blur is applied.

Adjust button Enable to activate highlight controls.

Gain field Adjust the light values of the image.

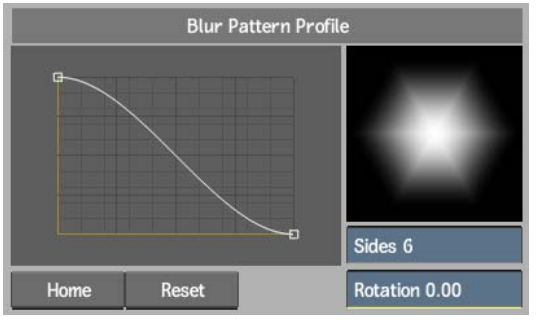
Offset field Adjust each colour value by this increment.

Threshold field Displays the minimum colour value included in highlights.

Ramp Range field Displays the difference between the Threshold value and the value at which the highlights take full effect (indicated by the Max Effect At field, which is non-editable).

Changing the Blur Pattern

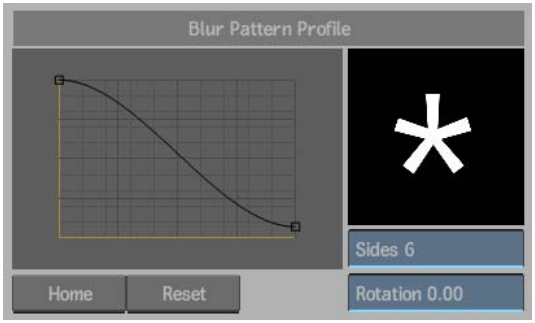
Use the Blur Pattern Profile to edit the blur kernel pattern.



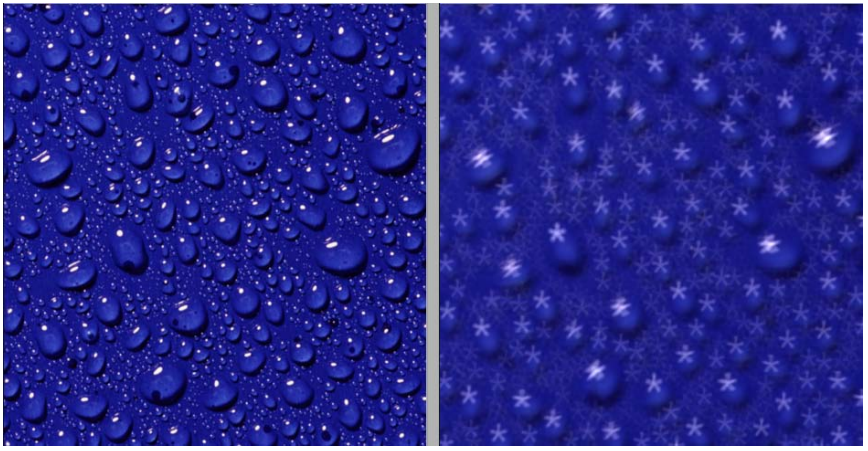
The kernel is the basic blur shape, displayed in the Blur Pattern Profile panel. The shape of the kernel is determined by its number of sides, its rotation, and the shape of its S-curve. This curve represents the shape of the pattern, from

its centre to the outside. The default S-curve defines the softness of the blur. You can change the curve by manipulating the two points that define the curve, or you can add points to the curve. Use the Edit Mode box to add and delete points on the curve.

You can define the blur shape using external kernel information by attaching an input to the Kernel tab. The input can be a different resolution than the other node inputs.

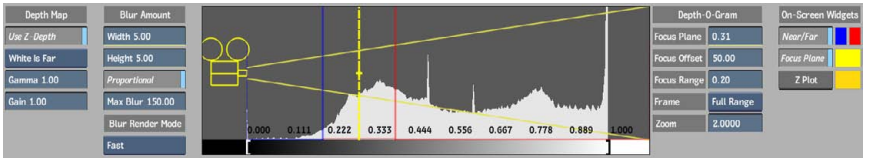


External blur information is not editable in the Blur Pattern Profile panel. Like the default blur shape selections, the shape determines the influence of the kernel's pixels on each pixel in the front input.

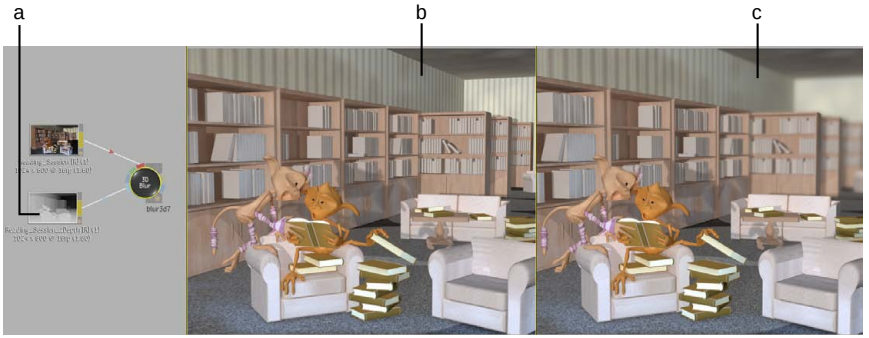


Simulating a Depth of Field Effect

Depth blur effects control the amount of blur based on a depth of field matte. Depth blur settings are displayed in the Depth menu.



A depth of field map (Z-depth map) can be connected to the node. Black portions of the map are in focus. White portions display the highest level of blur. Note that a Z-depth map imported from another application may use the opposite convention and may need to be inverted.



(a) Z-depth map (b) Front View (c) Blurred portion in Result View

Image courtesy of Jean-Marc Belloncik

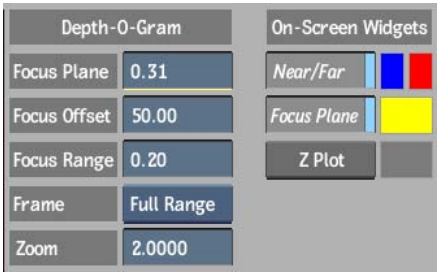
Depth Map settings can be used to invert colour values in the Z-depth map for reuse with the 3D Blur node.

Use Z-Depth button Enable depth-of-field controls and apply the effect to the image.

White Value box Select whether white pixels represent the furthest point or nearest point on the Z-axis.

Gamma field Applies a gamma curve to the Z-depth map before it is used.

Gain field Adjusts the values of the white balance. Applies a gain control to the Z-depth map before it is used.



Use the controls in the Depth-O-Gram and On-Screen Widgets panels to refine focus values, and select how and where they are represented.

Focus Plane Set the distance of your focus point, that is, the point at which there is no blur on the image.

Focus Offset Set the distance between the focus plane and the near offset represented as a percentage of the total offset range. Select 50% to make the near and far offsets equidistant from the focus point.

Focus Range Set the distance between the near and far offset.

Frame option box Choose how you want to frame the histogram.

Zoom field Select a vertical zoom value for the histogram display. You can also zoom horizontally by pressing **Ctrl+spacebar** and dragging left or right in the histogram.

Near and Far buttons Enable to display the focus offset plane in the image window.

Near and Far colour pots Select the colours that indicate the nearest and furthest points of focus.

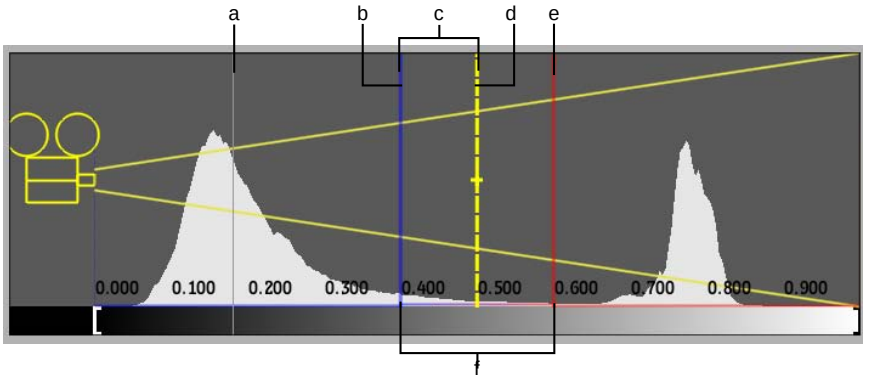
Focus Plane button Enable to display the plane on which the focus point lies in the image window.

Focus Plane colour pot Select the colour that indicates the plane on which the focus point is located.

Z Plot colour pot Select the colour that indicates the plane on which the plotted value is located.

Modifying Depth of Field Gesturally

Depth blur effects can be modified gesturally by dragging the focus, near offset, and far offset planes in the graphic representation of the depth of field. As you drag these elements, the planes are also displayed in the Result view as a preview of the areas that will be in focus. Use the depth control fields to change the gamma and gain, and to change the focus range while keeping the focus plane constant. These parameters are updated in the depth of field display automatically.



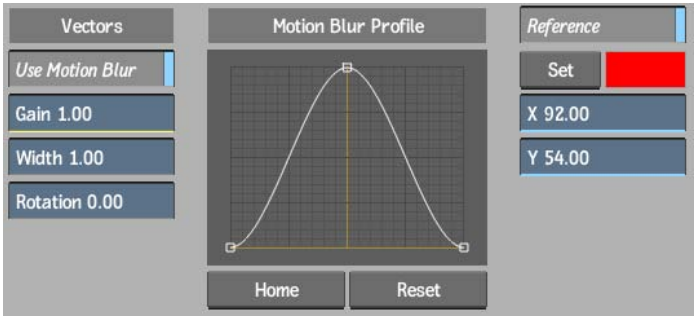
(a) Plot Value Plane (b) Near Focus Offset Plane (c) Focus Offset (d) Focus Plane (e) Far Focus Offset Plane (f) Focus Range

Simulating a Motion Blur Effect

The motion blur simulates the blur created by a fast-moving object or camera. The settings for this effect are in the Motion menu.

A blur effect can be created with front clip input and a clip with forward motion data input into the Forward Flow tab.

Use the Motion Blur Profile to define the opacity of the blur. The opacity increases at the top of the curve. By default, the blur is more transparent further away from the pixel. The midpoint represents the opacity of the blur at the position of the pixel.



Use Motion Blur button Enable the motion blur controls, which you use to apply the effect to the image.

Gain field Enter the amplitude of the motion blur.

Width field Enter the width of the blur.

Rotation field Enter an angle to apply a rotation to motion vectors connected to the node through the Forward Flow tab. Motion is rotated counterclockise.

Reference button Enable the reference controls, which you use to apply a reference point at which motion blur is negated.

Set button Subtracts the motion blur based on the motion analysis of a reference point at the selected frame. Motion blur will be negated at the selected point. Enable this button to apply the motion blur only to objects that do not have the same relative motion as the selected pixel. This button is enabled automatically when you edit values in the X and Y fields.

Set Reference colour pot Set the colour of the crosshair that marks the reference point in the image window.

X and Y fields Enter the horizontal and vertical position of the pixel to use as a reference point at the selected frame. A reference point can be selected directly in the frame using the cursor.

Adding a Displacement Map

New for this release: In Action, Displacement mapping is removed from the Surface menu, and becomes an object in the Node bin, improving the accessibility of displacement maps for surfaces and 3D geometries. The previous method of using displacement maps in the Surface menu has been removed. Any setups using the previous displacement method will be converted to the new displacement method upon loading. The exception to this new method is using displacement with DVE Layer Objects, which still uses the previous

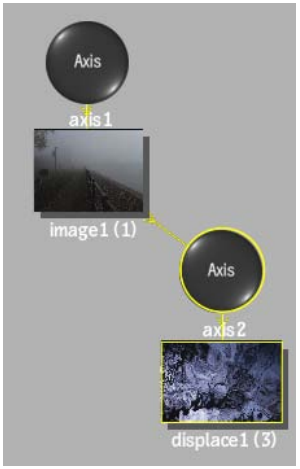
displacement method, but with an improved tabbed Surface menu. See Displacing DVE Layer Objects on page 75.

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 node bar and place it in the schematic.
   - Drag the Displace Map node from the node bar 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.



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

The Displace menu appears.

Use the following Displace menu settings (in any order) to get your desired effect.

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

Softness field Defines 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 Applies an offset to the displacement of X and Y.

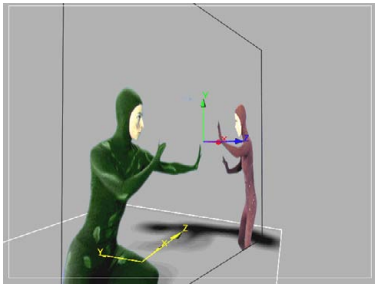
Displacement axes Specifies the amount of displacement in pixel units along the X, Y, and Z axes. Use positive values for displacement on the positive axis, and negative values for displacement on the negative axis.

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

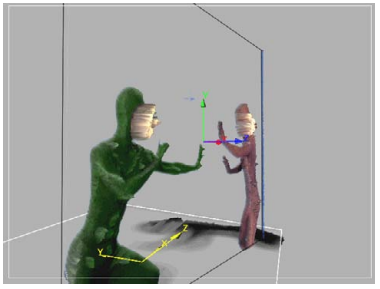
Fill to Surface button Enable to resize the displacement map to the resolution of its parent surface.

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. See Using UV Mapping on page 74.

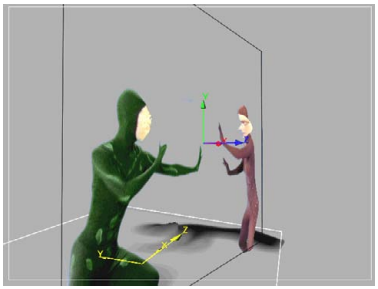
The following figure illustrates a possible use of displacement mapping and shows the difference between displacing with and without softness.



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

Using Different Media for the Displacement Source

You can specify different media as the displacement source. Do this to apply a custom matte, or alpha, as a displacement map instead of using one media for both the texture and displacement.

To use another media as the displacement source for a surface of Media1:

1. Click Media to access the Media menu.
2. Select a media from the Media list, or click <new media> and Add to return to the Desktop to add new media.

   NOTE If Auto Image or Auto DVE is selected in the Auto Image option box in the Setup menu, an image node and axis, or a DVE Layer Object node is created when adding new media. These nodes are not needed for displacement mapping, and can be hidden or deleted.

3. In the schematic, select the Displace node whose media you want to change.

   Notice that in Schematic view, the number next to the name of the selected Displace node is (1), indicating the media used for the displacement is Media1.

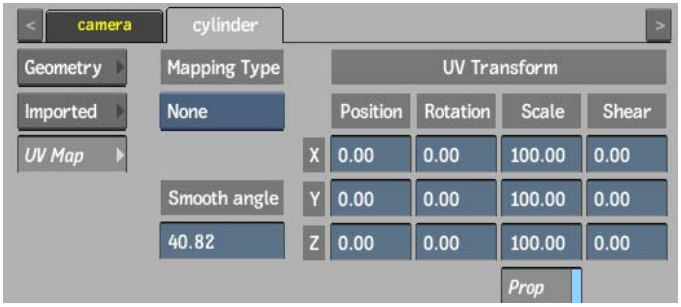
4. In the Media menu, click Apply.

   In Schematic view, the number next to the name of the selected Displace node change to (2), for example. (2) indicates Media2 is used as the displacement source.

Using UV Mapping

New for this release: Displacement, normal, and texture maps attached to 3D geometries can now be enhanced with UV mapping.

Use the UV Mapping settings to select how the UV coordinates of an attached displace, normal, or texture node are mapped to the 3D model. You can also apply axis transformations to the UV map. These transformations are different from the settings of the parent axis in that they transform the axes of the actual UV map coordinates.



UV Mapping Type box Select the type of UV mapping to apply to the attached Displace, Normal, or Texture node.

When a Displace or Normal node is attached to a geometry, a UV mapping type other than None is needed for the displace or normal pattern to have any effect on the geometry.

When a Texture node is attached to a geometry, you must select Wrap from the Mapping box in the Texture menu to be able to use the UV mapping settings.

Smooth angle field Displays the angle at which the edges of an attached Displace node become hard. Depending on the displacement map you are using, you may need to use this field to smoothen or harden the edges. Changes to this field only affect the shading of the displacement, and not the shape.

NOTE Only available when a Displace node is attached to a geometry.

Position fields Displays the position of the selected UV axis.

Rotation fields Displays the rotation of the selected UV axis.

Scale fields Displays the scale of the selected UV axis.

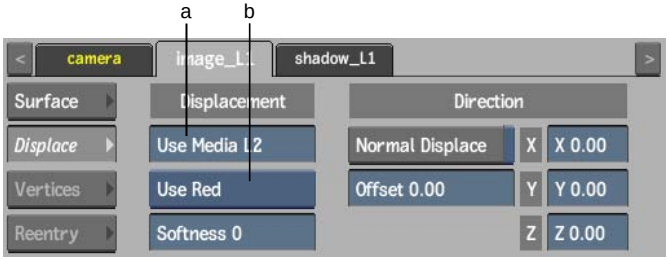
Prop Scale button Scales the X, Y, and Z UV axes proportionally.

Shear fields Displays the shear of the selected UV axis.

Displacing DVE Layer Objects

Use displacement mapping to create a 3D model from a 2D surface. Since a DVE Layer Object is comprised of a single object node, displacement mapping is applied differently than with regular objects.

You apply displacement mapping on DVE Layer Objects from the Displace tab of the Surface menu.



(a) Use Media field (b) Channel box

Use the following Displace settings to get your desired effect.

Use Media field Displays the the number of the media layer to use as the displacement source.

Notice that in Schematic view, the numbers next to the name of the selected DVE Layer Object, for example (1)(2). (1) indicates the media for the surface is Media1. (2) indicates that Media2 is the media used as the displacement source.

Channel box Select a colour channel to calculate the displacement map. Select Off to turn displacement mapping off.

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 Applies an 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.

Adding a Normal Map

New for this release: Add normal maps as objects to your Action scene to enhanve a surface or 3D geometry.

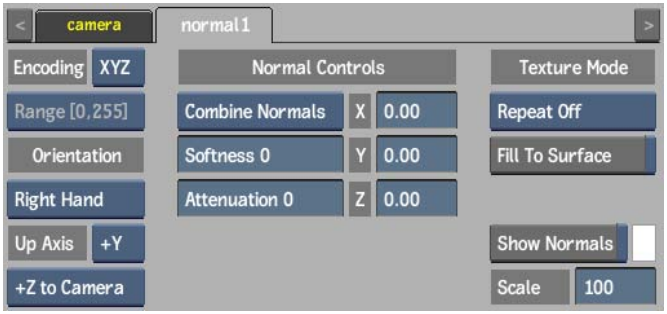
A normal map is used to change how normals are mapped on a geometry. You can apply a normal map to modify how a surface reacts to shading. You can manipulate a surface's normals based on X, Y, and Z offsets.

To add a normal map:

1. In the schematic, select the surface or 3D geomtery to which you want to apply the normal.
2. Click Media.
3. In the Media menu, select the media you want to use for the normal.
4. Do one of the following:
   - Drag the Normal Map node from the node bar and place it in the schematic.
   - Drag the Normal Map node from the node bar 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.
5. Double-click the Normal node in the schematic, or follow the tab population rules for the Object menu.

   The Normal menu appears.



Use the following Normal menu settings (in any order) to get your desired effect.

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 field Displays the amount of 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.

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

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

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

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

Scale field Displays the scale of the normal vectors.

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. See Using UV Mapping on page 74.

Using Different Media for the Normal Source

You can specify different media as the normal source.

To use another media as the normal source for a surface of Media1:

1. Click Media to access the Media menu.
2. Select a media from the Media list, or click <new media> and Add to return to the Desktop to add new media.

   NOTE If Auto Image or Auto DVE is selected in the Auto Image option box in the Setup menu, an image node and axis, or a DVE Layer Object node is created when adding new media. These nodes are not needed for normal mapping, and can be hidden or deleted.

   ---

3. In the schematic, select the Normal node whose media you want to change.

   Notice that in Schematic view, the number next to the name of the selected Normal node is (1), indicating the media used for the displacement is Media1.

4. In the Media menu, click Apply.

   In Schematic view, the number next to the name of the selected Normal node change to (2), for example. (2) indicates Media2 is used as the normal source.

Action: 3D Text

6

Topics in this chapter:

About 3D Text

You can create and manipulate 3D text strings in your Action scenes. With 3D text, you specify typical text properties such as font, font size, kerning, and italics. Since 3D text strings created in Action are also 3D geometries, you can extrude text, offset your text from a path, and apply other geometry settings.

New for this release: Improvements to 3D text allow you to change character axis properties, allowing for better manipulation of the individual text characters. You can now use cascading animation on character axis and geometry settings.

Adding a 3D Text Node

When you add a 3D Text node to your Action schematic, a special geometry node with an axis is added.

To add a 3D Text node to the scene:

1. Do one of the following:
   - Drag the 3D Text node from the node bar and place it in the schematic.
   - Drag the 3D Text node from the node bar and place it where you want it in Result view.
   - Double-click the 3D Text node. You do not need to be in Schematic view to add a node in this manner.

   A Geometry object, called Text1 by default, and parent axis appear in the schematic. In Result view, the default Text string appears.
2. To open the 3D Text menu, double-click the 3D Text node in the schematic, or follow the tab population rules for the Object menu.

Changing 3D Text Properties

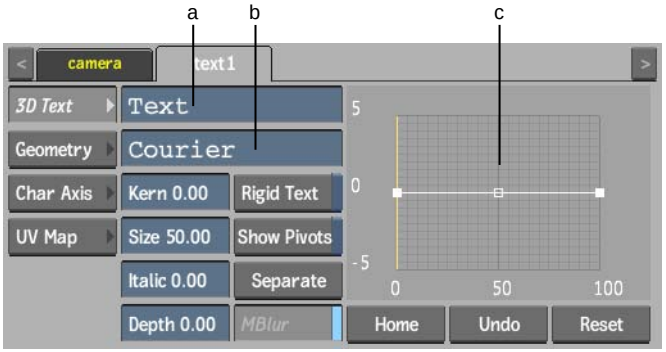
When you add a 3D Text node to your scene, the default text string "Text" appears. You can easily change this text string.

The change a text string:

1. Click the 3D Text field.

   The on-screen keyboard appears, representing the character set for the selected font. Enable Up ASCII to access the rest of the character set.
2. Type your text string or use the on-screen keyboard.
3. Click Exit Keyboard or press **Enter**. The text string is displayed in the Text field and automatically updated in the scene.

   New for this release: The Create and Replace buttons are removed from the 3D Text menu, since text string updates are now automatic.
4. Use the settings in the 3D Text tab to change the font, size, depth, and other text properties.



(a) Text field (b) Font field (c) Bevel curve

The 3D Text tab settings are described as follows.

Text field Displays the characters that make up the text string.

Font field Displays the current font. Click to open the font library, where you can select a different font for the text string.

You specify the default font using the TextDefaultFont token in the *init.cfg* configuration file. See the *Autodesk Visual Effects and Finishing Configuration File Reference Guide*.

Kern field Displays the kerning for the characters in the text string.

Size field Displays the font size for the characters in the text string.

Italic field Displays the level of italicization of the characters in the text string.

Depth field Defines the level of depth (thus extruding the selection, making it three dimensional).

Rigid Text button Enable to gang the text string characters as a single geometry. Enabling this button is particularly noticeable when attaching the 3D Text node to a 3D path.

Show Pivots button Enable to display the pivot point for each individual text character in the 3D Text string. These pivot points are displayed in the image window in red. When disabled, only the master character pivot point is displayed (in green). This setting can also be found in the Character Axis tab.

Separate button Separates text so that each letter has its own axis node. See Separating Text on page 82.

MBlur button Enable to use a motion blur effect for the selected text (can only be used if the global Motion Blur button is enabled in the Action Setup menu).

Bevel curve Applies a bevel to the depth of the text string when you manipulate the Bevel curve. You can move and add points to the curve, as well as adjust the tangent handles to produce different effects with the text string. See Creating Bevelled Text on page 82.

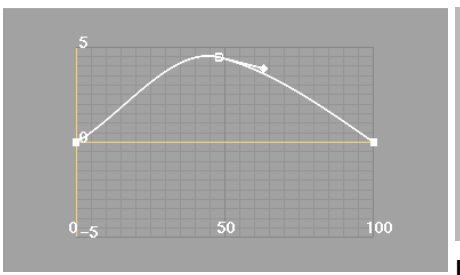
Home button Resets the Bevel curve viewer to show the whole curve.

Undo button Undoes Bevel curve operations.

Reset button Resets the Bevel curve.

Creating Bevelled Text

Use the Bevel curve to add a bevelled edge to your 3D text. 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. Experiment with different curves to create different effects.



Bevel curve

Resulting bevelled text

Separating Text

Rather than create a separate pivot point for each letter, you can separate words or sentences so that each letter can be individually manipulated by its own axis in the schematic.

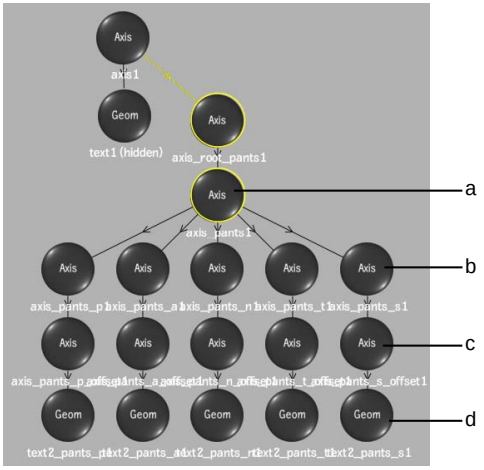
To separate text:

1 Create text as described in Changing 3D Text Properties on page 80.

2 Select the text to separate.

3 From the Text tab, click Separate.

Each letter of the text geometry is now an independent geometric object, and has its own axis and offset (to separate the letters). Each word is also given its own axis. The original 3D Text node is hidden in the schematic.



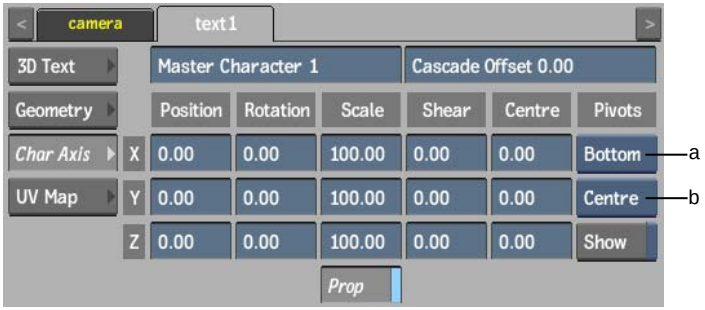
(a) Root axis for word (b) Root axis for letter (c) Offset per letter (d) Letter geometry

Changing Geometry Settings

Click the Geometry and UV Map tabs to apply any of the other geometry settings to your 3D text.

Changing Character Axis Properties

You can change the axis properties of your 3D text string characters. This can be useful in offsetting your text from a 3D path. See Action: 3D Paths on page 87.



(a) Vertical Pivot box (b) Horizontal Pivot box

The Character Axis tab settings are described as follows.

Master Character field Displays the number of the character in the text string that is considered to be the master. All other text characters follow this character in any character axis settings.

Cascade Offset field Displays the amount of time (expressed in frames) to offset the animation of other characters from the master character. The animation that is offset includes all numeric fields in the Character Axis tab, as well as the Specular, Ambient, Diffuse, Transparency, and Shine fields in the Geometry tab.

For example, if Cascade Offset is set to 0, all characters have the same animation as the master character. If Cascade Offset is set to a positive number, all characters other than the master character have their animation offset forward in time.

Position fields Displays the position of the offset along the X, Y, and Z axes.

Rotation fields Displays the rotation of the offset along the X, Y, and Z axes.

Scale fields Displays the scale of the offset along the X, Y, and Z axes.

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

Shear fields Displays the shear of the offset along the X, Y, and Z axes.

Centre fields Displays the centre of the offset along the X, Y, and Z axes.

Vertical Pivot box Select the vertical position of the pivot point for the selected text characters.

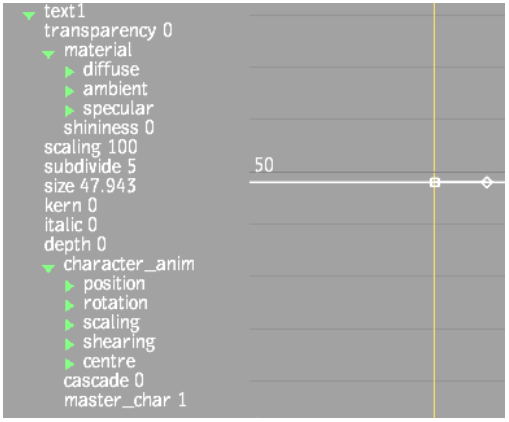
Horizontal Pivot box Select the horizontal position of the pivot point for the selected text characters.

Show Pivots button Enable to display the pivot point for each individual text character in the 3D text string. These pivot points are displayed in the

image window in red. When disabled, only the master character pivot point is displayed (in green). This setting can also be found in the 3D Text tab.

Animating 3D Text

You can animate the 3D text property and geometry channels in the Channel Editor. However, you cannot animate the text string or its bevel curve. The 3D text channels are contained in the text folder.



Action: 3D Paths

7

Topics in this chapter:

About 3D Paths

New for this release: Use your spline-creating abilities within Action to create a new type of effect.

A 3D path is an animatable 3D spline that you attach to other Action objects, such as surfaces, geometries, 3d text, cameras, or lights. The attached objects then follow the spline based on the path normals, allowing you to create effects, such as a 3D roller coaster.

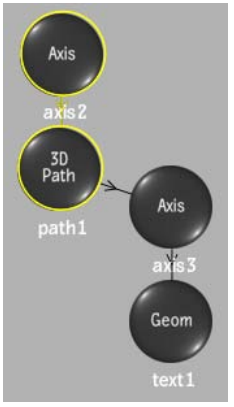
Adding a 3D Path Node

When you add a 3D Path node to your Action schematic, the node is added with an axis.

To add a 3D Path node to the scene:

1. Do one of the following:
   - Drag the 3D Path node from the node bar and place it in the schematic.
   - Drag the 3D Path node from the node bar and place it in Result view.
   - Double-click the 3D Path node. You do not need to be in Schematic view to add a node in this manner.

   A Path object (called path1, by default), with its parent axis, appears in the schematic.
2. Create mode is automatically selected in the Edit Mode box so you can create your spline. See Creating Splines on page 89.
3. Parent the 3D Path node to another object in your schematic, such as a Light node or a 3D Text node. The 3D path becomes part of the transformation hierarchy of the attached object.



3D Path node parented to a 3D Text node

4. To open the 3D Path menu, double-click the 3D Path node in the schematic, or follow the tab population rules for the Object menu.

Creating Splines

You can draw open or closed splines to use as your 3D path.

To create a spline:

1. Make sure that the Edit Mode box is in Create mode (this is the default when you first add a 3D Path node).
2. In the image window, click to add vertices.

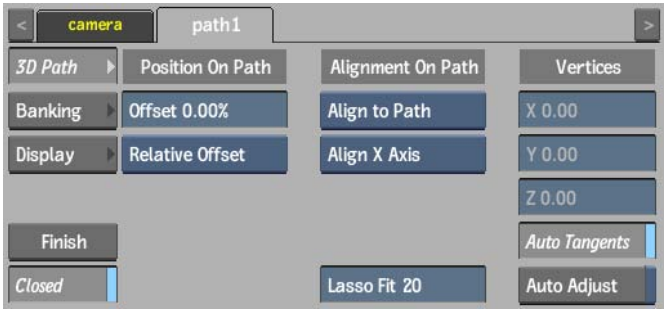
   TIP **Shift**-drag to add freehand segments to the spline. Vertices are added where you drag, and appear when you release **Shift**. After closing or finishing the spline, you can use the Lasso Fit field to increase or decrease the number of vertices that define the freehand segments of the spline.

3. To complete your spline, do one of the following:
   - Click the first vertex to close the spline.
   - Click Finish in the 3D Path tab to leave the spline open. If you decide later that you want to close the spline, enable Closed in the 3D Text tab.
4. Make sure that the Edit Mode box is in Move mode, so that you do not add more vertices by mistake.

   When the spline is closed or finished, its vertices and tangents can then be edited.

Editing Splines

Use the settings in the 3D Path tab to work with the spline you created. You edit splines in the same way as you edit garbage masks, working with their vertices and tangents.



Lasso Fit field Displays the number of points in the segments of the spline that are drawn freehand. Use a lower number to simplify the curve by removing vertices and tangents, resulting in a smoother curve.

Vertices fields Position the selected vertex or vertices.

Auto Tangents button Enable to position a tangent for each vertex set and create a smooth curve between the vertices. When enabled, it is possible to create a spline with both straight and curved segments.

When Auto Tangents is disabled, the tangents are positioned under the vertex, resulting in straight lines between vertices. When you draw freehand segments in a spline with Auto Tangents off, vertices are added with broken tangents, allowing the spline to follow your cursor movement.

Auto Adjust button Enable to automatically adjust the tangent handles of the two adjacent vertices when moving vertices and create smooth curves between the vertices.

Changing Spline Display Properties

You can change spline display properties in the Display tab. For example, you can change the colour of tangents on the splines you draw. This is useful to better contrast the spline's tangents from the clip so that they are easier to work with.

Normals button Enable to display normals along the 3D path.

Normals colour pot Select a colour for the display of normals.

Spacing field Displays the space between the displayed normals, in pixels. Spacing is also used to calculate the position of the object on the path. A lower spacing value may result in better positioning and smoother movement of the object, but rendering may be slower.

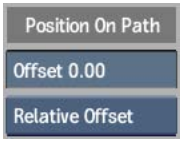
Scale field Displays the scale of the displayed normals, in pixels.

Spline colour pot Select a colour for the display of splines.

Tangents colour pot Select a colour for the display of tangents.

Positioning Objects on the Path

Use the Position On Path settings in the 3D Path or Banking tabs to offset the position of attached objects along the path and beyond. For open paths, if you offset past the first or last points on the path, the position is extrapolated accordingly. For closed paths, if you offset past the first or last points on the path, the attached object continues on the path with a tangent interpolated from the first and last normals.



Offset field Displays the amount of offset to apply to the attached object on the 3D path. Use this field to animate the attached object along the path.

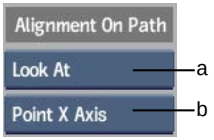
Offset box Select whether to offset the attached object from the path in a relative mode (expressed as a percentage of the path) or absolute mode (expressed in pixels).

NOTE These settings are repeated in the 3D Path and Banking tabs to make it easier for you not to have to switch tabs to change the settings. The same settings are reflected in both tabs.

Aligning Objects on the Path

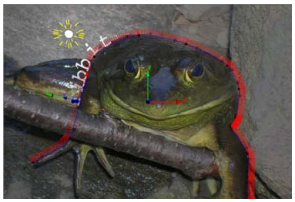
Use the Alignment to Path section of the 3D Path tab to set orientation behaviour.

(a) Alignment option box (b) Orientation Axis box



(a) Alignment option box (b) Point Axis box

Alignment option box Select how the object connected to the 3D Path node aligns to the path.

Select:	To:	Example:
Align Off	Not align the attached object to the path. In this case, the Orientation Axis box and banking controls are unavailable. This can be useful for vertical text effects.	
Align to Path	Align the attached object to the 3D path. You can then select which axis is aligned to the path in the Orientation Axis box, and use the banking controls.	
Look At	Point the attached object to a look-at object, such as a light. You can then select which axis is pointed to the look-at object in the Point Axis box, and use the banking controls. See Applying a Look-At Connection on page 93.	

Orientation Axis box Select which axis is aligned to the path. Available when Align to Path is selected in the Alignment option box.

Point Axis box Select which axis is pointed to the attached look-at object. Available when Look At is selected in the Alignment option box.

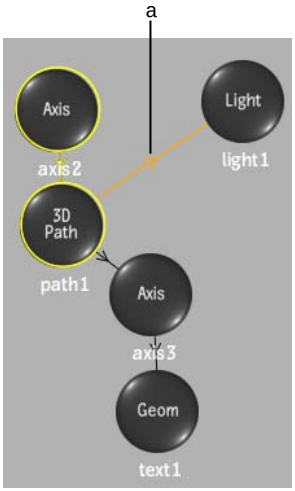
Applying a Look-At Connection

You can create interesting 3D path effects by attaching a look-at connection between the path and another object in your scene. The attached object on the path then rotates to face the look-at object, no matter where it is positioned. You attach a look-at connection in the schematic between the 3D Path node and any object with axis characteristics (Axis, Camera, Light, Projector, Particle Animator, and DVE Object).

To apply a look-at connection:

1. Do one of the following:
   - Select Look At in the Alignment option box.
   - Select Lookat in the Edit Mode box (or press **Alt+L**).
2. In the schematic, drag from the 3D Path node to an object with axis characteristics.

   The selected object is connected to the 3D Path node by an orange dotted line with an arrow.



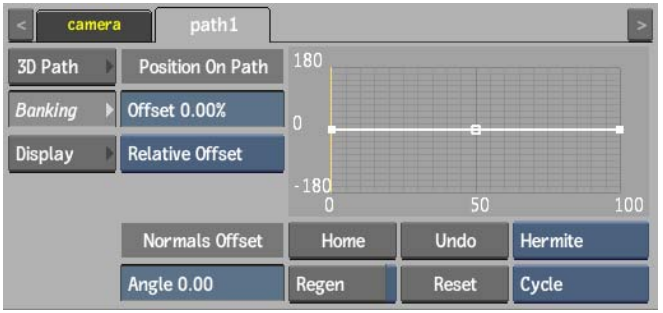
(a) Look-at connection

3 Select which axis looks at the attached object in the Point Axis box.

4 Optional: Use the banking curve to orient the attached objects.

Using the Banking Curve

When Align to Path or Look At is selected in the Alignment option box, the Banking tab becomes available. Banking uses the normals of the path to orient objects attached to it. Use the banking curve and settings to control the torsion effect of the object as it travels along the 3D path.



The Position on Path settings are the same as those in the 3D Path tab. See Positioning Objects on the Path on page 91.

Banking Curve Adds twists and torsion to the normals of the 3D path. Use the options in the Edit Mode box to add, select, delete, or move keyframes on the banking curve. The horizontal axis represents the length of the path, and the vertical axis displays the orientation, expressed in degrees.

Normals Offset Angle field Displays the angle of rotation of all normals, applied to the entire banking curve.

NOTE Changes made to the banking curve and Angle field are cumulative. The Angle field can be animated, but not the banking curve.

Regen button Enable to dynamically refresh the image as changes are made to the banking curve.

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

Undo button Undoes banking curve operations.

Reset button Resets the banking curve.

Interpolation box Select the default interpolation type for the banking curve.

Extrapolation box Select the default extrapolation type for the banking curve.

Batch Workflow Improvements

8

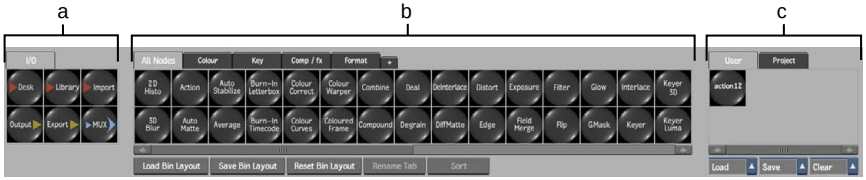
Topics in this chapter:

Batch Node Bins

New for this release: Nodes have been arranged into separate groups classified by tabs, making them easier to find.

The node bins contain all nodes required for building a process tree. The nodes are divided into the following three groups, classified by tabs:

- An I/O bin, which contains all input and output nodes
- A Tools bin, further subdivided into groups, which contains effects and formatting nodes
- A User/Project bin, which allows you to save custom setups



(a) I/O bin (b) Tools bin (c) User/Project bin

TIP If you do not see the node bins, swipe the lower left side of the screen. To return to the node bins from any module's menu, swipe again or press **Ctrl+Tab**. Use the scroll bar under the applicable bin to scroll through all available nodes.

I/O Bin

Use the nodes in the I/O bin to load clips into Batch or to output processed clips.

Use the Desk node to load clips directly from a Desktop reel and the Library node to load clips from local or remote libraries. The Import node allows you to import clips in any supported format from local or remote framestores.

Use the Output node to output clips to the Desktop, a library, or a remote framestore. Use the Export node to export image sequences in any supported format. With the Output node, metadata such as clip history and timecode is kept; with the Export node, metadata is not kept.

The I/O bin also contains a MUX node. The MUX node is a schematic tool that helps create cleaner schematics by allowing you to have multiple outputs from one input. It incorporates the hiding of connections to prevent schematic connection overlaps.

You cannot customize the I/O bin.

Tools Bin

The Tools bin contains nodes classified by tabs. The All Nodes tab contains all Batch nodes except the nodes found in the I/O bin. The other tabs in the Tools bin allow you to create and customize bins.

Some of the nodes in the All Nodes bin are duplicated in other bins according to a preset tab classification. For example, the Paint node is found in both the Colour and Key bins.

The nodes in the Tools bin are listed in alphabetical order from top to bottom of each row. You can customize any of the bins and preset tabs in the Tools bin except the All Nodes bin.

User/Project Bin

The User/Project bin contains custom nodes classified by a User tab and a Project tab. Use this bin to save custom setups per user or project. See Creating Custom Nodes on page 103.

Customizing the Tools Bin

New for this release: Create custom bins and populate them with your most commonly used nodes to optimize your workflow. As well, change the order of the tabs along the top of the bin and rename them to reflect the contents of a bin.

You can customize any bin in the Tools bin except the All Nodes bin and its tab.

To create a tab:

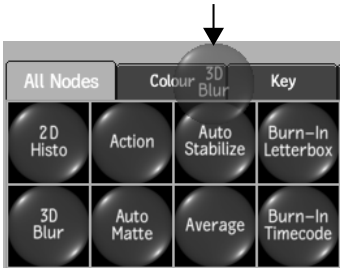
1 Click the plus sign tab.

2 Name the tab in the keyboard that appears.

NOTE You can create a maximum of 6 tabs.

To copy a node to another bin:

1 Drag the node on top of the destination tab.



2 Release the cursor when it changes to a green crosshair.

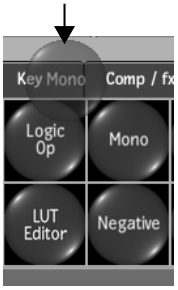
3 Click the destination tab when the standard yellow cursor reappears.

The copied node appears in the bin. Nodes are added to the end of a bin in the order copied (following the same alphabetical node order of the rows, from top to bottom of each row).

NOTE Nodes cannot be duplicated within the same bin.

To move a node to another bin:

1 Press **Ctrl+Alt** and drag the node on top of the destination tab.



2 Release the cursor when it changes to a green crosshair.

3 Click the destination tab when the standard yellow cursor reappears.

The node is moved from its original location to the destination bin. Nodes are placed at the end of a bin in the order moved (following the same alphabetical node order of the rows, from top to bottom of each row).

NOTE Nodes cannot be duplicated within the same bin.

To move a node to the schematic:

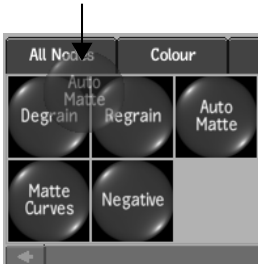
➤ Press **Ctrl+Alt** and drag a node to the schematic.

The node is moved from the bin and placed in the schematic.

To reorder a node within a bin:

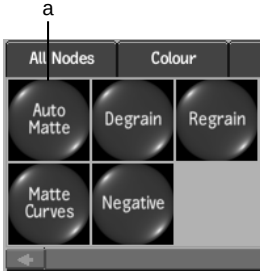
1 Press **Ctrl+Alt** and drag the node to a new location. You can move nodes from one row to another as well as reorganize nodes within a row.

In the following example, the AutoMatte node is dragged on top of the Degrain node.



2 Release the cursor when it changes to a green crosshair at the location where you want the node moved.

If you dragged the node on top of an existing node, the existing node shifts to the right and the moved node is inserted in its place. In the following example, the AutoMatte node is inserted in the place of the Degrain node, and the Degrain and Regrain node shift to the right.



(a) Reordered node

To reset a bin to alphabetical layout:

➤ With the applicable bin active, click Sort.
The nodes in the bin are reset to their alphabetical layout.

To delete a bin:

1. Press **Ctrl+Alt** and drag the tab to the bottom of the screen.
2. Release the cursor when it changes to a delete cursor.

WARNING There is no undo capability when deleting a bin.

The entire contents of the bin, including the tab, are deleted.

To delete a node from a bin:

1. Press **Ctrl+Alt** and drag the node to the bottom of the screen.
2. Release the cursor when it changes to a delete cursor.

WARNING There is no undo capability when deleting a node.

To rename a tab:

1. Click the Rename Tab button.
2. Enter a new tab name in the keyboard that appears.

To reorder a tab:

1. Press **Ctrl+Alt** and slide the tab to its new location.
2. Release the cursor when it changes to a green crosshair at the new location for the tab.
   If you dragged the tab on top of an existing tab, the existing tab shifts to the right and the moved tab is inserted in its place.

To save a bin layout:

1. Click Save Bin Layout.
2. Name the layout.

The layout of the entire Tools bin is saved, including all new and customized bins. You cannot save only select bins.

Layouts are saved per user, not by project.

To load a bin layout:

1. Click Load Bin Layout.
2. Select the layout you want to load.

   Each customized bin, including all new bins, is loaded into the Tools bin.

NOTE If you load a bin layout containing unsupported nodes, the unsupported nodes do not appear.

Creating Custom Nodes

New for this release: The User/Project bin has been redesigned to make it easier to use.

Create custom nodes of specific setups that you often use. A custom node can consist of a single node with specific settings or multiple nodes that create a particular effect.

You create custom nodes by dragging individual nodes, groups, branches, or entire trees into the User/Project bin.

To create a custom node:

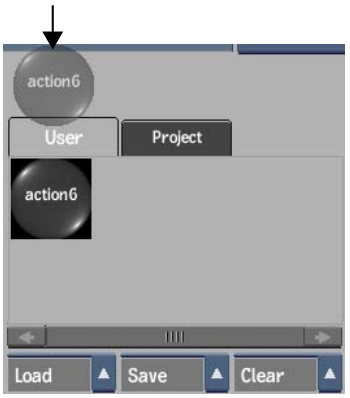
1. In the Selection Mode box, select the part of the process tree that you want to use as a custom node.
2. **Alt**-click a node that is part of your selection, and then drag the selection on top of the User or Project tab. You can also drag the selection directly into the bin if it is the active bin.

   The selection is copied to the bin. The original selection remains in the schematic.

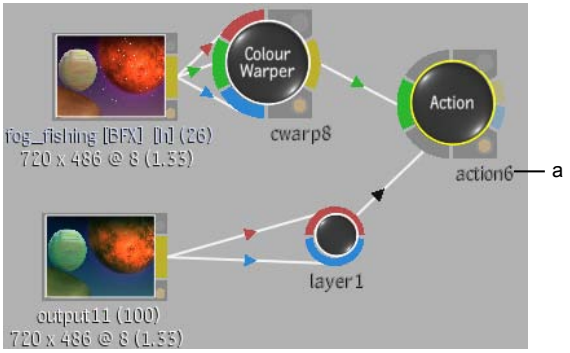
NOTE No two custom nodes can share the same name. Attempting to drop a node into a custom bin with a similarly named node is not possible.

To use a custom node:

1. Select a custom node from the User or Project bin. If necessary, scroll through the bin to find the node.
2. Drag the node to the schematic to copy it, or press **Ctrl+Alt** and drag to move it to the schematic and remove it from the bin.



The same configuration of nodes and clips that was used to create the custom node appears in the schematic.

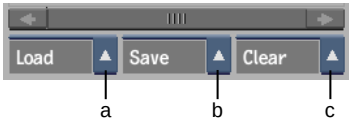


(a) Branch used to create custom node

You can use custom nodes as often as you like. Each time you drag a custom node to the schematic, a new number is appended to its name.

To manage the custom node bins:

➤ Select any of the following from the dropdown lists.



(a) Load dropdown list (b) Save dropdown list (c) Clear dropdown list

Select:	To:
Load Project Bin or Load User Bin from the Load dropdown list	Load custom nodes from another project or user. **NOTE** If you load unsupported nodes, the unsupported nodes appear greyed out when dragged to the schematic.
Save Project Bin or Save User Bin from the Save dropdown list	Save the current custom nodes so they can be loaded by another project or user.
Clear Project Bin or Clear User Bin from the Clear dropdown list	Delete all custom nodes in the Project or User bin.

To reorder a node in the User/Project bin:

1. Press **Ctrl+Alt** and drag the node to a new location. You can move nodes from one row to another as well as reorganize nodes within a row.
2. Release the cursor when it changes to a green crosshair at the location where you want the node moved.

   If you dragged the node on top of an existing node, the existing node shifts to the right and the moved node is inserted in its place.

To delete a custom node from the User/Project bin:

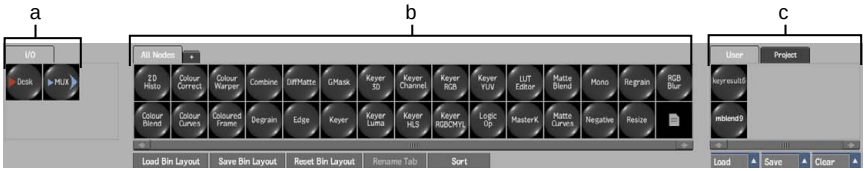
1. Press Ctrl+Alt and drag the node to the bottom of the screen.
2. Release the cursor when it changes to a delete cursor.

WARNING There is no undo capability when deleting a custom node.

Modular Keyer Node Bins

The node bins contain all the nodes needed to build a process tree. The nodes are now divided into the following three groups, classified by tabs:

- An I/O bin, which contains the Desk node and the MUX node. Use the Desk node to load clips directly from the Desktop. The MUX node allows you to have multiple outputs from one input. You cannot customize this bin.
- A Modular Keyer bin, which contains all Modular Keyer nodes classified into the All Nodes tab. The other tabs in the Modular Keyer bin allow you to create and customize bins. See Customizing the Modular Keyer Bin on page 106.
  The nodes in the All Nodes bin are listed in alphabetical order from top to bottom of each row. The All Nodes bin does not contain the nodes found in the I/O bin. You cannot customize the All Nodes bin.
- A User/Project bin, which contains custom nodes classified by a User tab and a Project tab. Use this bin to save custom setups per user or project. See Creating Custom Nodes on page 103.



(a) I/O bin (b) Modular Keyer bin (c) User/Project bin

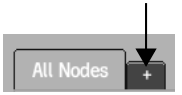
Customizing the Modular Keyer Bin

New for this release: Create custom bins and populate them with your most commonly used nodes to optimize your workflow. As well, change the order of the tabs along the top of the bin and rename them to reflect the contents of a bin.

You can customize any bin in the Modular Keyer bin except the All Nodes bin and its tab.

To create a tab:

1 Click the plus sign tab.

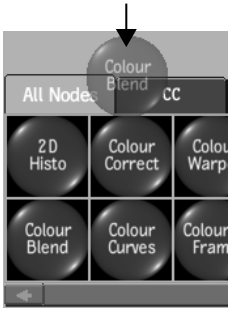


2 Name the tab in the keyboard that appears.

NOTE You can create as many tabs as fit along the top of the bin.

To copy a node to another bin:

1 Drag the node on top of the destination tab.



NOTE You must create a tab before copying a node to it. Dragging a node to the plus sign tab will not copy the node.

2 Release the cursor when it changes to a green crosshair.

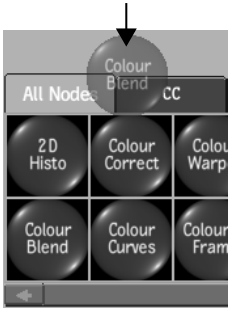
3 Click the destination tab when the standard yellow cursor reappears.

The copied node appears in the bin. Nodes are added to the end of a bin in the order copied (following the same alphabetical node order of the rows, from top to bottom of each row).

NOTE Nodes cannot be duplicated within the same bin.

To move a node to another bin:

1 Press **Ctrl+Alt** and drag the node on top of the destination tab.



2 Release the cursor when it changes to a green crosshair.

3 Click the destination tab when the standard yellow cursor reappears.

The node is moved from its original location to the destination bin. Nodes are placed at the end of a bin in the order moved (following the same alphabetical node order of the rows, from top to bottom of each row).

NOTE Nodes cannot be duplicated within the same bin.

To move a node to the schematic:

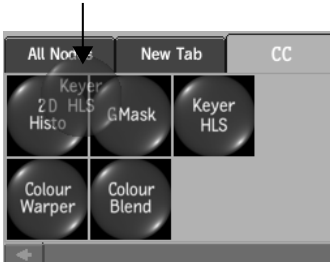
➤ Press **Ctrl+Alt** and drag a node to the schematic.

The node is moved from the bin and placed in the schematic.

To reorder a node within a bin:

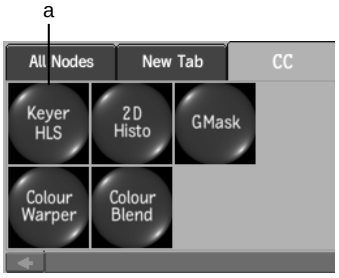
1 Press **Ctrl+Alt** and drag the node to a new location. You can move nodes from one row to another as well as reorganize nodes within a row.

In the following example, the Keyer HLS node is dragged on top of the 2D Histo node.



2 Release the cursor when it changes to a green crosshair at the location where you want the node moved.

If you dragged the node on top of an existing node, the existing node shifts to the right and the moved node is inserted in its place. In the following example, the Keyer HLS node is inserted in the place of the 2D Histo node, and the 2D Histo and GMask nodes shift to the right.



(a) Reordered node

To reset a bin to its default node layout:

➤ With the applicable bin active, click Sort.

The nodes in the bin are reset to their alphabetical layout.

To delete a bin:

1 Press **Ctrl+Alt** and drag the tab to the bottom of the screen.

2 Release the cursor when it changes to a delete cursor.

WARNING There is no undo capability when deleting a bin.

The entire contents of the bin, including the tab, are deleted.

To delete a node from a bin:

1 Press **Ctrl+Alt** and drag the node to the bottom of the screen.

2 Release the cursor when it changes to a delete cursor.

WARNING There is no undo capability when deleting a node.

To rename a tab:

1. Click the Rename Tab button.
2. Enter a new tab name in the keyboard that appears.

To reorder a tab:

1. Press **Ctrl+Alt** and slide the tab to its new location.
2. Release the cursor when it changes to a green crosshair at the new location for the tab.

   If you dragged the tab on top of an existing tab, the existing tab shifts to the right and the moved tab is inserted in its place.

To save a bin layout:

1. Click Save Bin Layout.
2. Name the layout.

   The layout of the entire Modular Keyer bin is saved, including all new and customized bins. You cannot save only select bins. Layouts are saved per user.

To load a bin layout:

1. Click Load Bin Layout.
2. Select the layout you want to load.

   Each customized bin, including all new bins, is loaded into the Modular Keyer bin.

NOTE If you load a bin layout containing unsupported nodes, the unsupported nodes do not appear.

Creating Custom Nodes

New for this release: Create custom nodes of specific setups that you often use. A custom node can consist of a single node with specific settings or multiple nodes that create a particular effect.

Modular Keyer setups and Batch setups are saved separately. You cannot load a Batch setup into the Modular Keyer. In the same vein, you cannot save a

Modular Keyer setup from Batch. If you want to save a Modular Keyer setup, you must save it from the Modular Keyer itself.

You create custom nodes by dragging individual nodes, groups, branches, or entire trees into the User/Project bin.

To create a custom node:

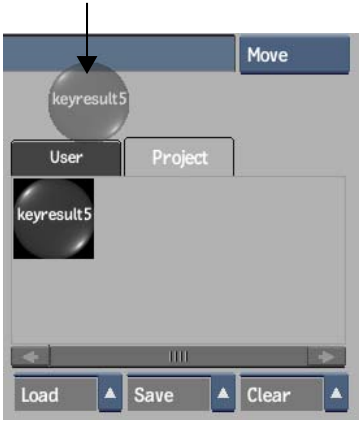
1. In the Selection Mode box, select the part of the process tree that you want to use as a custom node.
2. **Alt**-click a node that is part of your selection, and then drag the selection on top of the User or Project tab. You can also drag the selection directly into the bin if it is the active bin.

   The selection is copied to the bin. The original selection remains in the schematic.

NOTE No two custom nodes can share the same name. Attempting to drop a node into a custom bin with a similarly named node is not possible.

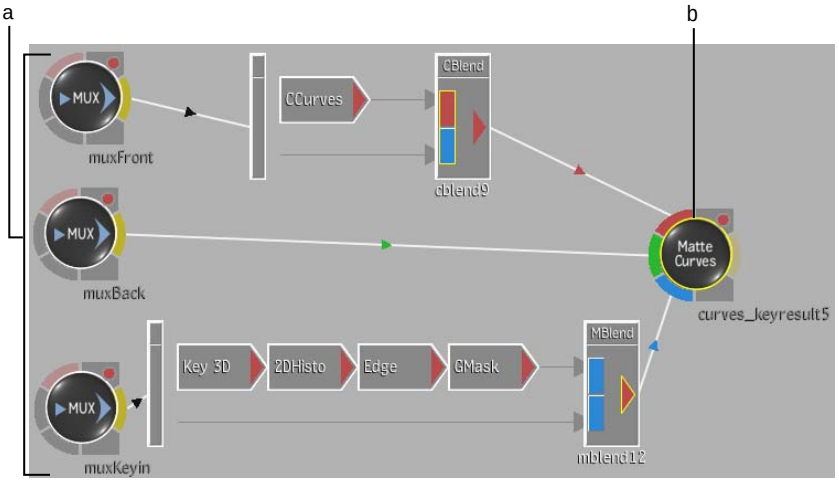
To use a custom node:

1. Select a custom node from the User or Project bin. If necessary, scroll through the bin to find the node.
2. Drag the node to the schematic to copy it, or press **Ctrl+Alt** and drag to move it to the schematic and remove it from the bin.



A similar configuration of nodes and clips that was used to create the custom node appears in the schematic. Because the Modular Keyer

schematic does not support multiple front, back, and matte clips, the front, back and matte clips are converted to MUX nodes. In the same vein, the Result node is converted to a Matte Curves node. All Matte Curves settings from the setup's Result node are saved with the Matte Curves node.



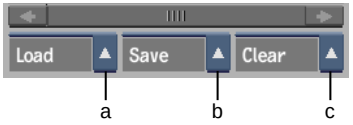
(a) MUX nodes converted from front, back, and matte clips (b) Matte Curves node converted from Result node

You can use custom nodes as often as you like. Each time you drag a custom node to the schematic, a new number is appended to its name.

3 Reconnect the front, back, and matte clips to the MUX nodes. Before outputting your results, reconnect the Result node and copy the settings from the Matte Curves node to the Result node.

To manage the custom node bins:

➤ Select any of the following from the dropdown lists.



(a) Load dropdown list (b) Save dropdown list (c) Clear dropdown list

Select:	To:
Load Project Bin or Load User Bin from the Load dropdown list	Load custom nodes from another project or user. **NOTE** If you load unsupported nodes, the unsupported nodes appear greyed out when dragged to the schematic.
Save Project Bin or Save User Bin from the Save dropdown list	Save the current custom nodes so they can be loaded by another project or user.
Clear Project Bin or Clear User Bin from the Clear dropdown list	Delete all custom nodes in the Project or User bin.

To reorder a node in the User/Project bin:

1. Press **Ctrl+Alt** and drag the node to a new location. You can move nodes from one row to another as well as reorganize nodes within a row.
2. Release the cursor when it changes to a green crosshair at the location where you want the node moved.

   If you dragged the node on top of an existing node, the existing node shifts to the right and the moved node is inserted in its place.

To delete a custom node from the User/Project bin:

1. Press Ctrl+Alt and drag the node to the bottom of the screen.
2. Release the cursor when it changes to a delete cursor.

WARNING There is no undo capability when deleting a custom node.

Sparks Load Node

New for this release: Use the new Sparks Load node to populate multiple Sparks nodes at the same time into a destination bin. Populating a bin with predefined Sparks nodes saves you time since you do not have to access the Sparks browser each time you want to use one of the preloaded Sparks.

Unlike the Sparks node, the Sparks Load node itself cannot be dragged to the schematic; only the predefined node populated into a bin can be dragged to the schematic.

To create a predefined Sparks node in a bin:

1. Drag the Sparks Load node on top of any tab in the All Tools bin except the All Nodes tab.

   The Sparks browser appears.

2. Select a Sparks. To make multiple selections, **Ctrl**-click each Sparks that you want to load.

3. Click Load.

   The selected Sparks are created as their own Sparks nodes in the destination bin. To use one of the preloaded Sparks, drag it from the destination bin to the schematic. You do not have to re-enter the Sparks browser to load a Sparks if you preloaded it into a bin.

Saving Sources and Setups

New for this release: When saving setups, you can also now save the sources used by the setup. You do this by taking a snapshot of the sources. When you take a snapshot, all source clips, including all BFX clips used by the current

main level setup, are saved to a working library. Sources contained in BFX clips, however, do not appear. They are part of the BFX clip.

Although sources are saved separately from setups when a snapshot is taken, they are saved concurrently. As well, the corresponding Batch setup is updated such that it points to the newly saved sources in the working library.

With the Snapshot feature, you do not have to worry about Inferno not being able to find source clips associated with a setup.

To prevent Inferno setups from being accidently overwritten from a Flare system, extensions in setup names are determined as follows:

- A Inferno setup never gets an extension added to its name.
- A Flare setup always gets a **.flare** extension added to its name. If you try to save a setup loaded from Flare, the **.flare** extension will be automatically removed from the name.

Inferno setups are saved in a Inferno subdirectory as follows:

~/batch/inferno/<my_setup_name>

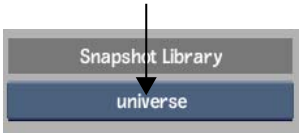
Flare setups are saved to a Flare subdirectory as follows:

~/batch/flare_<hostname>/<my_setup_name>.flare

You specify the working Snapshot library to which you want sources saved. Sources are saved to a predefined reel structure named according to the setup name. If you save setups loaded from a Flare system, you will need to select a read-write library in which to save the associated sources.

To specify the library for saved sources:

1. From Batch, click Setup.
2. From the Snapshot Library box, select the library to which you want to save sources or select <new> and create a library.



This Snapshot library will be saved as a preference with the Batch setup. If you subsequently delete or rename the library, the name in the Snapshot Library box will have **(Non-Existent)** beside its name. You will be

prompted to have the missing library created the next time you take a snapshot.

NOTE The Snapshot Library box also appears in the Save Setup As & Snapshot browser when using the Save Setup As & Snapshot option.

To save sources and/or setups:

➤ In Batch, select one of the following options from the Save dropdown list.

Save Setup Saves a setup. The first time you save a setup, you are prompted to name it in the keyboard that appears.

It is impossible to overwrite a setup loaded from Flare with this option since Inferno setups are never given an extension when saved. If you try to overwrite the setup with the same name, a message appears offering to rename the setup by removing the **.flare** extension.

Each subsequent time you save the Inferno setup with this option, you are overwriting the previous saved setup. You are prompted to confirm the overwrite. To bypass the confirm, press **Alt** as you select Save Setup.

Save Setup As Saves the setup with a new name, which you enter in the keyboard that appears. Each time you save a setup with this option, you are creating a new saved setup.

Save Setup & Snapshot Saves the setup and takes a snapshot of the sources used by the setup.

The first time you save a setup and sources, you are prompted to name the setup in the keyboard that appears. The snapshot of the sources is saved to the Snapshot library you specify.

It is impossible to overwrite a setup loaded from Flare with this option since Inferno setups are never given an extension when saved. If you try to overwrite the setup with the same name, a message appears offering to rename the setup by removing the **.flare** extension.

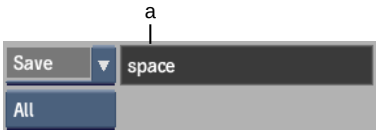
Each subsequent time you save the Inferno setup and sources with this option, you are overwriting the previous saved setup as well as the sources in the library. You are prompted to confirm the overwrite. To bypass the confirm, press **Alt** as you select Save Setup & Snapshot.

Save Setup As & Snapshot Saves the setup with a new name, which you enter in the keyboard that appears, and takes a snapshot of the sources used with the setup. The snapshot of the sources is saved to the Snapshot library you specify.

Each time you save a setup and sources with this option, you are creating a new saved setup. A new snapshot of the sources is saved to the library with the new setup name.

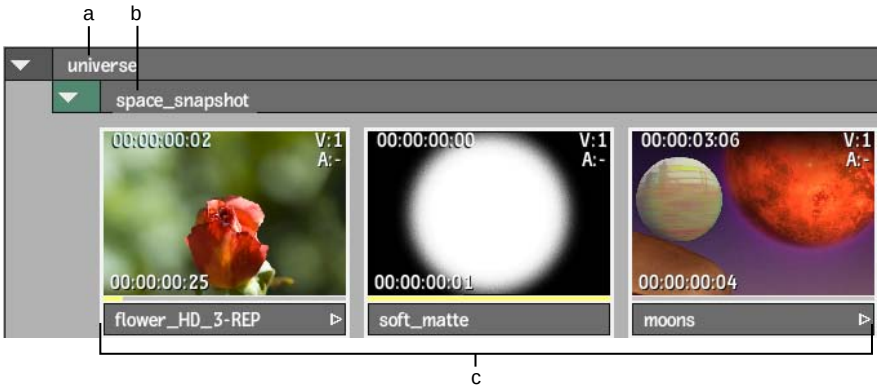
With this option, you can change or create a new Snapshot library directly from the Save & Snapshot browser without having to go back to the Setup menu.

In Batch, the name of the setup appears in the Filename field. If you saved the setup under a new name, the field is automatically updated with the new name. Note that the setup has no extension in its name.



(a) Batch setup name

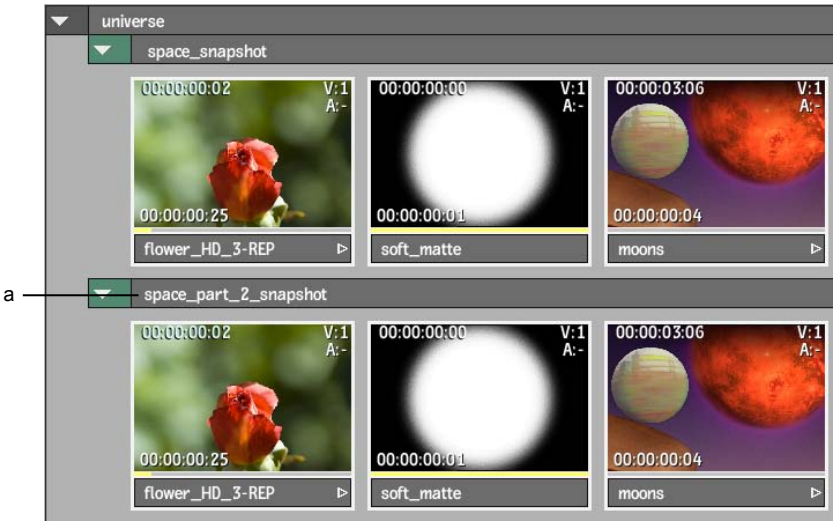
In the following example, the setup was with the Save Setup As & Snapshot option. In the working library, sources are saved in a predefined reel entry named according to the Batch setup name with _snapshot appended to the name. The setup points to this reel. When the setup is loaded in a Batch session, all clips in the reel are loaded as well.



(a) Snapshot library specified in Batch Setup menu (b) Reel entry named according to Batch setup name (c) Source clips saved with Save Setup & Snapshot operation

In the following example, the same setup is saved again with the Save Setup As & Snapshot option. A new snapshot of the sources associated with the renamed setup is taken. Another reel is created in the library with the new setup name (space_part_2_snapshot). The previous snapshot (and setup) is not overwritten.

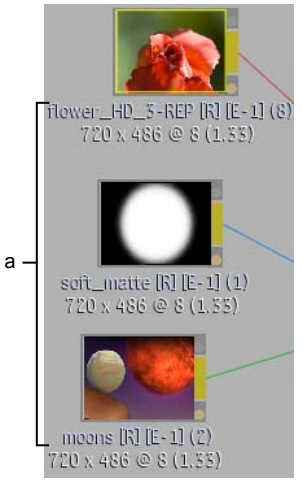
NOTE To overwrite the previous snapshot (and not create a new reel), select Save Setup & Snapshot.



(a) Reel entry named according to the new Batch setup name (specified with the Save Setup As & Snapshot option)

Each saved Batch setup points to its corresponding reel in the Snapshot library. When either setup is loaded in a Batch session, all clips in the corresponding reel are loaded as well.

In the Batch schematic, the clip names change to blue indicating they are library clips.



(a) Clips become library clips when a snapshot is taken

Library Node

New for this release: Load clips or reels as groups into Batch.

Use the Library node to load clips into Batch from the clip library.

To load clips from the clip library into Batch:

1. Drag the Library node to the schematic.

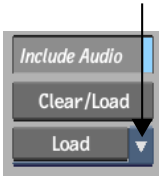
   The clip library appears.

2. Select the library you want to browse. You can browse both local and remote libraries.

3. Do one of the following:

   - To load clips directly into Batch, select the clips and click Load.
   - To load clips into one group, select the clips and then select Load Selection as Group from the Load dropdown list. Note that you can select a combination of clips and reels to load into a group.
   - To load reels into corresponding groups, select the reels and then select Load Reels as Groups from the Load dropdown list. Note that

you must select the actual reel in the libary; selecting all the clips in a reel will not load the reel into a group.



Depending on your selection, the clips appear as individual clips or as group nodes in the Batch schematic. If you loaded reels into groups, the name of the groups take on the reel names.

Grouping Nodes

New for this release: The group node, including its icon and display settings, has been redesigned to make it easier to use.

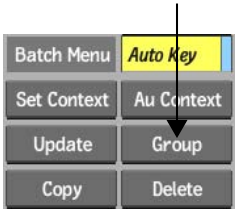
You can group nodes to simplify cluttered schematics. A group is represented by a single dynamic node that displays the group's contents, inputs, and outputs. You can use a group node as a parent to, or child of, other clips and nodes.

You can create several groups and work on each group separately. For example, group a Keying and Colour Correction branch separately from an Action and a filtering branch, and then work on each branch independently. If you need to edit the nodes in a group, you can expand the group and make the necessary modifications.

To group nodes:

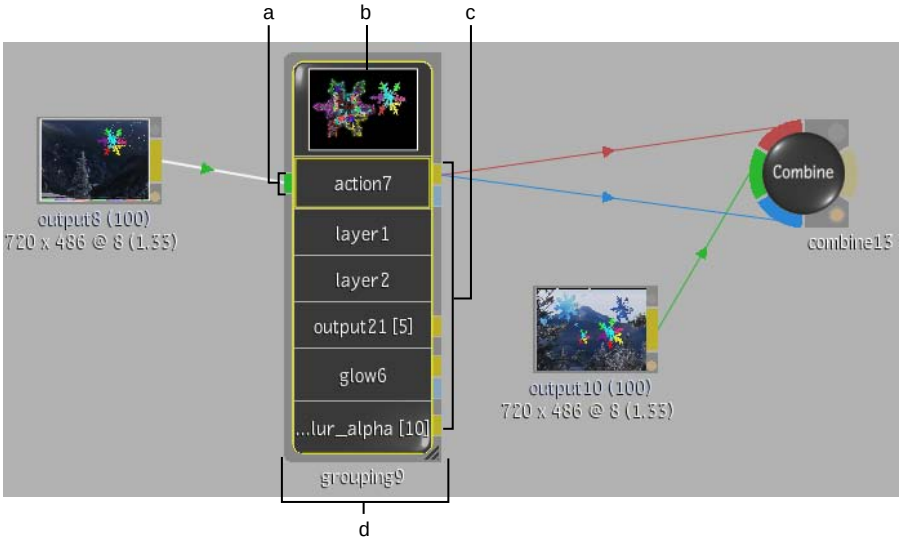
1 **Ctrl**-drag to select the nodes you want to group.
Selected nodes are outlined in white.

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 node (c) Output connections (d) Group icon

3 To rename the group, enter a name in the Name field.

Defining Group Display Settings

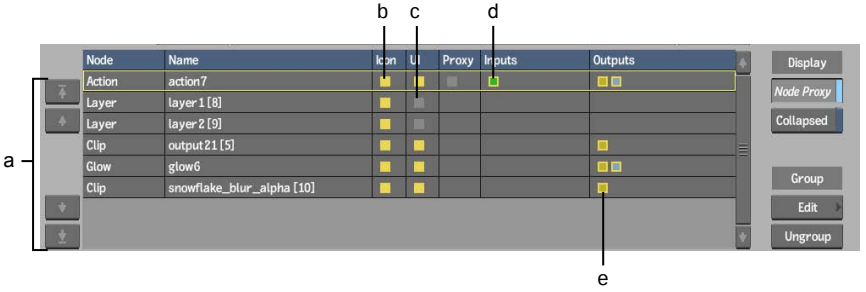
Customize the group node using the Group List menu. The Group List menu lists all the nodes and connections contained in a group. You can rename and hide the contents of the group, as well as define which input and output connection sockets are visible and available for connection in the schematic.

You can select a node in the Group List menu to display the group at the selected node's stage. The View box must be set to Result, Front, or Back to use this display option. You can also display a preview proxy of a node in the group.

To define a group's display settings:

1 Select a group in the schematic.

The Group List menu appears.



(a) Sort Order buttons (b) Enabled icon visibility (c) Disabled UI visibility (d) Enabled input visibility (e) Enabled output visibility

2 Change the group's interface by doing any of the following.

Click:	To:
Sort Order buttons	Move the selected node one position up or down (click the single-arrow buttons). To move the selected node to the first or last position in a group, click the arrow-line buttons.
Name	Change the node's name.
Icon	Toggle the node's visibility in the group. Icons in the Group List are yellow when the node is visible and grey when hidden.
UI	Toggle the node's availability in the Node List box. Icons in the Group List are yellow when the node is displayed in the box and grey when hidden.
Proxy	Lock the proxy displayed in the group icon so that it does not change as you navigate the Group List. Icons in the Proxy list are white when locked and grey when unlocked.
Inputs	Hide or unhide the selected node's input sockets. Sockets are colour-coded with the same scheme as Batch nodes that are not part of a group. Icons in the Group List have a yellow border when the input socket is visible.

Click:	To:
Outputs	Hide or unhide the selected node's output sockets. Sockets are colour-coded with the same scheme as Batch nodes that are not part of a group. Icons in the Group List have a yellow border when the output socket is visible.
Node Proxy	Display a preview proxy of a node in the group. To change the proxy that is displayed, press the **Shift** + **up or down arrow** as you navigate through the Group List.
Collapsed	Collapse the group icon so that only the preview proxy is visible.

To change the size of the group icon:

➤ Drag the lower-right corner of the icon to make it wider.

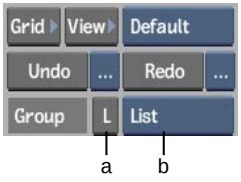
Editing a Group

Use the Edit Group controls to edit the contents of a group.

To edit a group:

1. Select the group you want to edit.
2. Click Edit.
   The nodes in the group appear in their own schematic.

   TIP Instead of editing the group, you can access a single node's menu by selecting the node's name from the Node List box. Click the L (List) button to return to the Group List menu.



(a) List button (b) Node List box

3. Modify the nodes as required.

NOTE Selecting All from the Selection Mode box while editing a group selects all nodes within the current group; any nodes outside the group in the current Batch pipeline remain unselected.

4 Click Exit Group to return to the previous schematic.

NOTE Click Ungroup to expand the group of nodes to their pre-grouped positions in the schematic.

Resolution Issues

A node's bit depth and resolution are not always compatible. It is not always possible to use clips of different resolution or bit depth. For example, it is not possible to have one custom Colour Corrector node with an 8-bit clip and another with a 12-bit clip. Garbage masks, on the other hand, work correctly with clips of different bit depth, but not of different resolutions.

Offsetting Clips

New for this release: Timing View now displays all clips in a group. You can offset all the clips in the group or a selection of the clips.

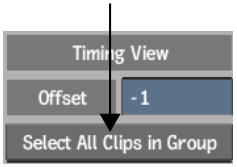
Adjust the timing of clips by offsetting them in the Timing View. Timing View displays the timing of all clips in the current Batch schematic.

When you offset a clip, you simultaneously offset all its segments on each layer. You can also offset multiple clips simultaneously, including all clips in a group.

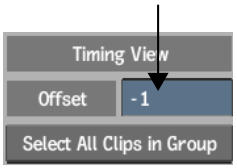
You can display or hide negative frames as you offset a clip. You can also offset clips brought into Batch to their record timecode.

To offset Batch clips:

1 Select the clips and/or segments to offset. If you are offsetting all clips in a group, select one clip or segment in the group and then click Select All Clips in Group.



2 Enter the number of frames by which to offset the clip in the Offset field.



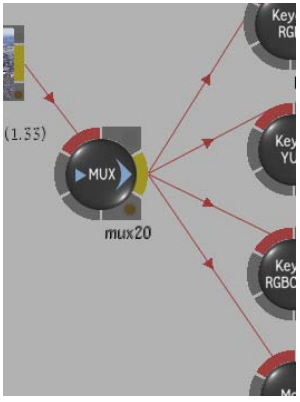
All selections are offset by the same amount.

3 **NOTE** If you offset multiple clips simultaneously, the value in the Offset field resets to 0 when the offset is complete.

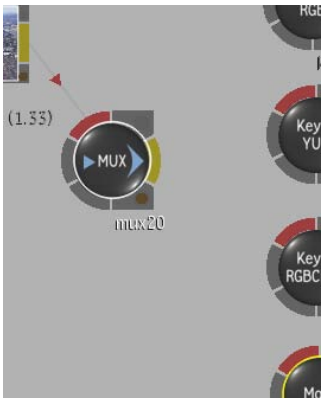
MUX Node

New for this release: The MUX node allows you to toggle the display of incoming and and outgoing links. In addition to Batch, the MUX node is now available in the Modular Keyer.

The MUX (multiplexer) node allows you to make multiple output sockets. Use this node to propagate an input to multiple other nodes through the schematic and to clarify its graphical representation. Use the MUX node to clarify the connection scheme of nodes within a group.



Default display of MUX node input and output links



MUX node with hidden output links

In the MUX Node menu, two toggle buttons control the display of connections to and from the node.

Input button Enable to hide the input link to the MUX node.

Ouput button Enable to hide all output links from the MUX node.

Action Node

New for this release: Improvements to the Action node allow you to add direct media from the library or the Desktop. As well, you can now add media by selecting an option from a new option box or by using hotkeys.

Action nodes in Batch provide in-context access to a fully functional Action module.

You use either indirect media or direct media with an Action node. Indirect media is connected directly to an Action Media node and appears in the schematic. With indirect media, you can connect any source (a clip, node, or group output, for example) to an Action node.

Direct media appears directly inside an Action node. Although direct media does not appear in the schematic, media and all settings pertaining to it are saved with the Batch setup.

Indirect media is available only with Action nodes in Batch. The Desktop Action module uses only direct media.

Adding Indirect Media to an Action Node

Media nodes contain indirect media and are connected to an Action node. Media nodes have the following properties:

- A Media node is permanently parented to the Action node (you cannot sever these process lines).
- Media nodes have red and blue input tabs for the front and matte inputs, respectively.
- Media nodes support floating-point input.

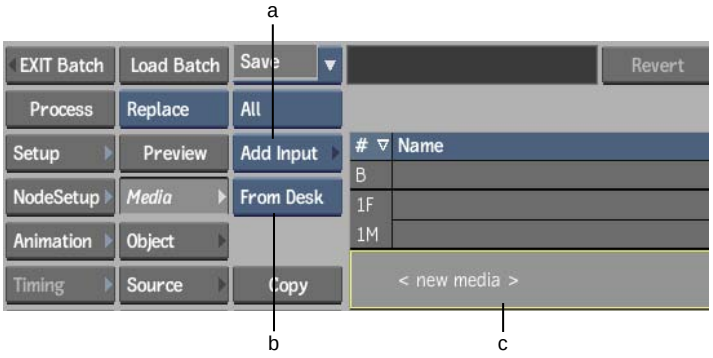
Setting up indirect media is a good way to use pre-processes (a precomposite or the result from multiple nodes) that you want to input into media.

To change the input clips of indirect media, you must either parent new front and matte clips to the Media node or add a new Media node. You cannot change the input clips from the Media menu. If you attempt to do so, a message appears indicating that the selected media is indirect media.

To add indirect media to an Action node:

1 Double-click the Action node and then click Media.

The Media menu appears.



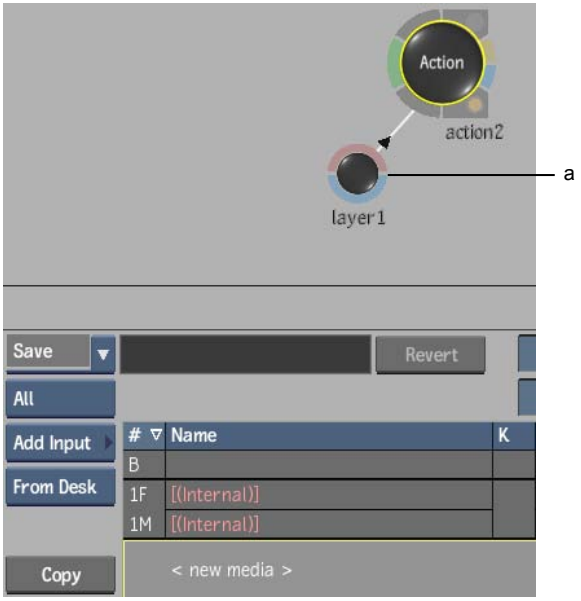
(a) Media List box (b) Media From box (c) New media line

2 Select the <new media> line in the Media list and then do one of the following:

- Click Add Input in the Media List box.

■ **Ctrl**-click Add Media in the Media List box.

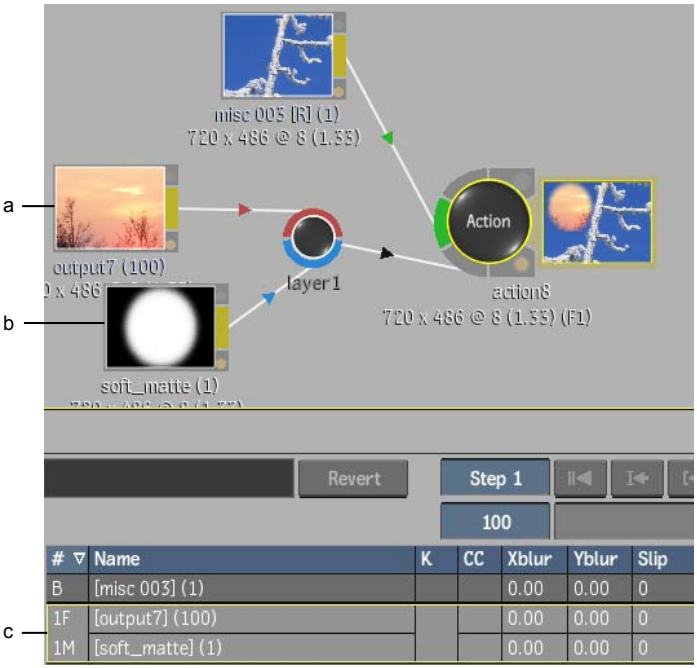
The Media node is added to the Action node.



(a) Media node

3 Connect front and matte clips to the red and blue input tabs of the Media node, respectively.

Media nodes are represented in the Media list with brackets ("[ ]") around the input clip names.



(a) Front (indirect) clip (b) Matte (indirect) clip (c) Front and matte indirect media in the Media list

---

TIP You can parent a back clip to the Action node. Although doing so is not necessary, a parented back clip node provides a good visual reference for identifying the Action composite in the process tree.

Adding Direct Media to an Action Node

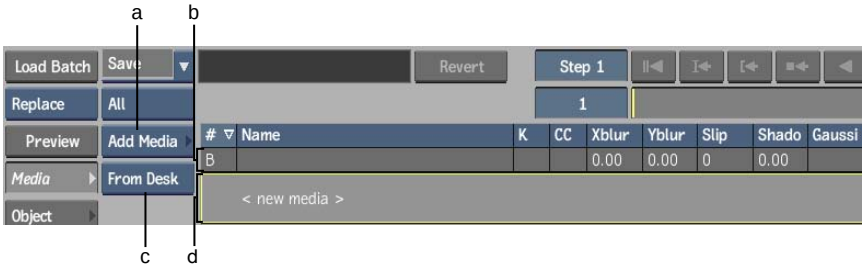
To add direct media to an Action node, you use the Media list.

If you have direct media but decide you want to apply processes to the media before outputting the clip to the Action node, you can convert the media to indirect media.

To add direct media to an Action node:

1 Double-click the Action node and then click Media.

The Media menu appears.



(a) Media List box (b) Back media line (c) Media From box (d) New media line

2 In the Media From box, select whether you want to add clips from the Desktop or from the library.

3 Do any of the following:

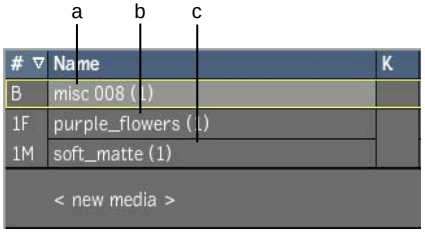
- To add back media, double-click the Back media line, or select the Back media line and then select Add Media from the Media List box.
- To add front and matte media, double-click the New media line, or select the New media line and then select Add Media from the Media List box.

NOTE If you have Front and Matte media lines in the Media list but no media, you can also double-click one or both lines to add media.

4 Select the clips and click Load.

NOTE You can only select two clips (front and matte) when adding media with the New media line.

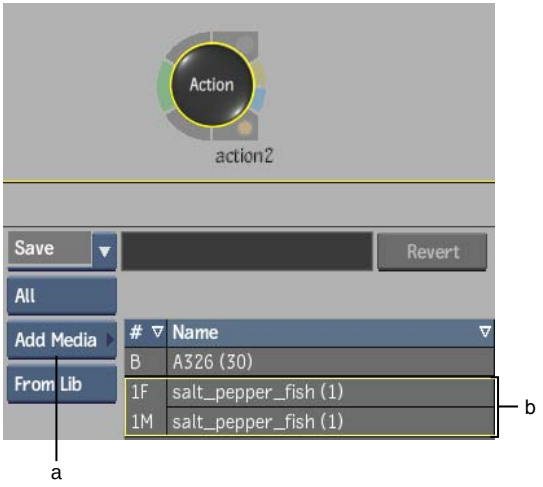
The media is added to the Media list and appears in the Action node.



(a) Back media (b) Front media (c) Matte media

To convert direct media to indirect media:

1 Select the media that you want to convert from the Media list.
In the following example, the front and matte media are selected.

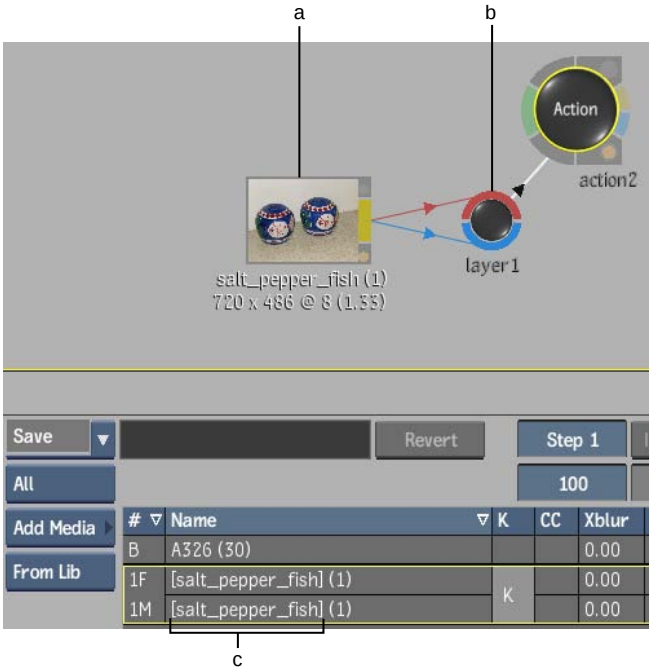


(a) Media List box (b) Selected front and matte media

2 Do any of the following:

- Click Add Input in the Media List box.
- **Ctrl**-click Add Media in the Media List box.

A Media node is added to the Batch node and the selected media is automatically connected to the respective input tabs. In the Media list, brackets appear around the media name, indicating the media is indirect.



(a) Indirect media (b) Media node (c) Parenthesis indicate indirect media

NOTE To replace the media, double-click the clip node in the schematic.

Missing Media

If a clip set to No Media is attached to an Action node and is missing media, the node's output is based on the tab to which the clip is connected, and the status of the node's other inputs.

The following conditions will output transparent Action media if the clip is missing media:

- A front clip set to No media
- A back clip set to No Media
- A matte set to No Media, with a front clip that is also connected to the Action node

If all media in the scene are transparent and no other information is input into the Action node, the node output is set to No Media.

Application-centric Improvements

9

Topics in this chapter:

Keyframe Indicators

New for this release: Improve your speed and workflow using these new keyframe indicators.

A number of colours are used in numeric fields to indicate that keyframes are present in the channels associated with the fields.

Indicator	Example
A blue bar under a numeric field indicates that one or more keyframes are present on this channel.	
A yellow bar under a numeric field indicates that a keyframe is present on this channel at this point in time.	
A dotted yellow bar under a numeric field indicates that this channel is linked to another channel using an expression. In this case, the numeric value is also greyed out, as the value can not be modified. The name of the linking channel is also displayed as part of the tooltip.	
A yellow character in a numeric field indicates that a numeric value is changed, but a keyframe is not set. Once a keyframe is set, the character colour returns to grey, and the keyframe indicator under the value is set.	

When working with keyframes and displaying selected channels, there are also hotkeys available and a contextual menu with options based on the state of the selected field. See Selecting Channels Automatically on page 136.

Selecting Channels Automatically

New for this release: a new contextual menu helps you work with fields and the channel editor.

After you alter a numeric field, the corresponding channel can be selected automatically in the Channel Editor with hotkeys, a contextual menu, or the Auto Select button.

The contextual menu—and corresponding hotkeys also provide keyframe options depending on the state of the field. For example, if a keyframe exists at the current frame, the Delete Keyframe option exists in the contextual menu for the field.

You can also open the channel editor with the desired channel framed and highlighted, by pressing **Shift** and double-clicking on a numeric field. Press

Ctl+Shift and double-click a numeric field to open the channel editor and add the selection to the already selected channel.

To display the contextual menu:

1. Press the Menu button on the keyboard, or the upper side switch on the pen.
2. Click inside a field.

   The contextual menu is displayed. The possible operations available depend on the state of the field.

Operation	State	Hotkey
Set Keyframe (Current Value)	No keyframe on current frame at the current value	**K**+ click field
Set Keyframe (Default Value)	No keyframe on current frame at the default value (and the current value is not already the default value)	**Ctrl**+ click field
Reset (Default Value)	The current value is not already the default value	
Delete Keyframe	A keyframe is present at the current frame	**Del**+ click field
Keep Current Keyframe Only	A keyframe is present at the current frame (all keyframes on the channel are deleted except the current keyframe)	**Shift**+**Del**+ click field
Copy Keyframe	A keyframe is present at the current frame	
Paste Keyframe	A keyframe has been copied and is available to paste	
Select Channel	Always available (the channel becomes selected in the channel hierarchy)	**Shift**+ click field
Add Channel to Selection	Another channel is already selected in the channel hierarchy	**Ctrl**+**Shift**+ click field
Unselect Channel	Channel is already selected in the channel hierarchy	**Ctrl**+**Alt**+ click field

Operation	State	Hotkey
Reset Channel (Current Value)	Channel has keyframes or an expression applied (the channel is deleted, but the current value is kept in the field)	**Alt**+ click field
Reset Channel (Default Value)	Channel has keyframes or an expression applied, and the current value is not the default value	**Ctrl**+**Alt**+ click field
Copy Channel	Always available	
Paste Channel	A channel has been copied and is available to paste	
Link Channel	A channel has been copied and is available to link	

The results of contextual menu and hotkey operations in the channel editor override any User Filter and Auto Frame settings.

Rendered Clip Name

New for this release: In the General section of the Preferences menu, a new preference allows you to add a prefix of suffix to rendered clip names.

Set how rendered clip names are displayed in Inferno.

Rendered Clip Name box Select whether to add an acronym of the module as a prefix or suffix to a rendered clip name, if a setup name does not already exist for the clip. If a setup name does not exist, the rendered name of the clip is the background clip name (or front clip name, if there is no background clip), with the module suffix or prefix. You can also choose Do Not Add.

Controlling Image Display using Exposure and Image Data Type

New for this release: In the View menu and the Player, you can change toggle the image preset to display the clip in RGB mode or Matte Mode. You can also gesturally change the image preset and apply or bypass the image data type directly in the image window.

When you are working in the Player or a module that supports multiple viewports, you can change the display of an image based on the type of image data you are working with. By default, an image is displayed in RGB mode with a transformation for a video image. You can apply transformations to the image to display an optimal view of logarithmic and linear images.

The Matte mode is a preset that allows you to preview the matte with exposure and contrast settings that are independent of those in RGB mode. In Matte mode, a linear transformation is applied to the image by default and 3D LUTs are not processed.

To access the exposure and image data type settings:

➤ Do one of the following:

- In a module with multiple viewports, display the View menu.
- To access settings for the broadcast monitor, open the Preferences menu.

(a) Image Data Type box (b) Preset box

Exposure field Displays the exposure that is used to transform image display in the current image window.

Contrast field Displays the contrast that is used to transform image display in the current image window.

Image Data Type box Select the type of image data you are displaying in the current image window. Your selection determines the type of transformation that 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	Apply a transformation to a 16-bit floating-point image, with a high dynamic range.

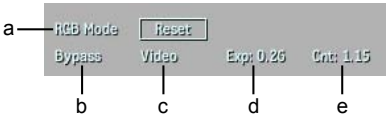
Apply All button In modules with multiple viewports, enable to apply the transformation for the current viewport to all viewports using the same Preset mode.

Bypass button Enable to deactivate display settings in the current image window.

Preset box Select an option to preview the image in either RGB or Matte mode.

Controlling Image Display Gesturally

You can adjust exposure settings and image data type directly in the image window of the Player or a module viewport. The lower left corner of the image in the Player and each viewport display these settings.



(a) Preset Mode (b) Bypass/Active Mode (c) Image Data Type d) Exposure (e) Contrast

To adjust exposure and image display settings directly in an image window:

1. If you are working in a module, select the viewport.
2. Do one of the following:
   - Press **Shift+E** and drag left or right in the viewport to decrease or increase the exposure.

- Press **Shift+C** and drag left or right in the viewport to decrease or increase the contrast.
- Click the Preset mode to alternate between RGB mode and Matte mode.
- Click the Bypass/Active mode to alternate between Bypass mode (colour management is deactivated in the current display window), and Active Mode (settings are enabled).
- Click the image data type to cycle through video, logarithmic, and linear settings.

NOTE Click Reset to restore default exposure and contrast settings. **Ctrl**-click either the exposure or contrast setting to reset it exclusively.

Selecting a Project and User on Start-up

New for this release: When connected to a remote framestore, you can now select a local or remote user.

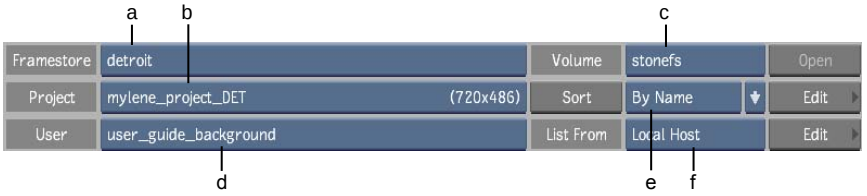
When you start Inferno, the Project Management menu appears. Use the Project Management menu to select a project and user for the current session, to create projects and users, or to manage existing projects and users.

You can work with projects on the current framestore or on a remote framestore. If multiple volumes are available, you can select which one to use. For information on creating volumes, see the *Autodesk Stone and Wire Filesystem and Networking Guide*.

To select a project and user on start-up:

1 Start the Inferno application.

The Project Management menu appears, displaying the framestore, project, and user from the previous session.



(a) Framestore box (b) Project box (c) Volume box (d) User box (e) Sort Order box (f) Host box

---

NOTE If this is the first time you are starting Inferno, there are no existing projects or users.

---

2 Do one of the following:

- To open a project on the current framestore, select a project from the Project box.
- To open a project on a remote framestore, select the framestore from the Framestore box. If the framestore has more than one volume, select a volume from the Volume box. Click Open, and then select a project from the Project box.

NOTE If you have a long list of projects, you can use the Sort Order box and arrow to sort the projects by frame resolution, name, or creation date, in descending or ascending order.

---

3 Select a user from the User box. If you opened a project on a remote framestore, you can use the Host box to select a user on the remote framestore or on your local framestore.

4 Click Start.

The project's Desktop appears. If you try to access a Smoke project that is already open on a remote framestore, an error message appears and you are asked to confirm whether you want to go into that project's clip library.

Creating User Profiles

New for this release: You now have options for copying existing user profiles when creating new user profiles.

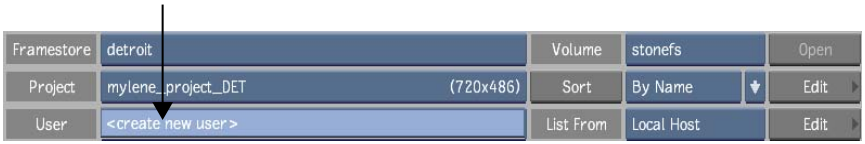
Create a user profile to manage your preferences. When you create a user, you have the option of copying preferences from an existing user. If the user whose preferences you want to copy was created on the same version of the application, you can copy all preferences. If the user was created on an older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

Users do not persist when upgrading from one version of Inferno to another. You need to create new users for the new version. Also, users are not shared between Autodesk Visual Effects and Finishing products.

You can create a user on start-up from the Project Management menu, or during a session from the Preferences menu. By default, a user's preferences are created in the directory */usr/discreet/user/effects/<user name>*.

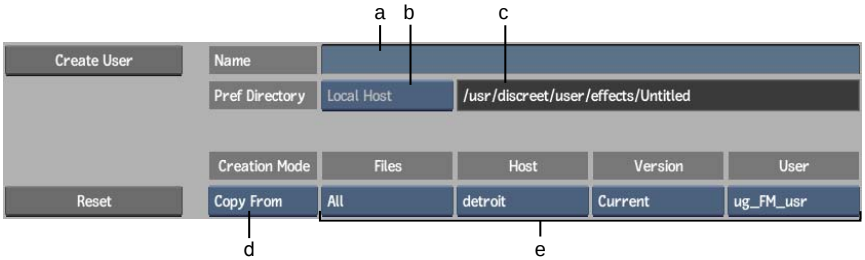
To create a user profile:

1. Do one of the following:
   - If you are creating a user on start-up, select <create new user> from the User box in the Project Management menu.



   - If you are creating a user in the middle of a session, select <create new user> from the User box in the Preferences menu.

The Create User menu appears.



(a) Name field (b) Preferences Directory Host box (c) Preferences Directory field (d) Creation Mode box (e) User Profile Copy From options

TIP To reset all user settings to their default values, click Reset at any time.

2 Enter a name for the user in the Name field.

3 The user's default home directory appears in the Preferences Directory field. If you logged in to a remote system, select whether to save the preferences in the default home directory of the remote or local user by selecting an option from the Preferences Directory Host box.

4 Do one of the following:

- To create a user without copying existing preferences, select New Prefs in the Creation Mode box and then click Create User.
- To copy the preferences of an existing user, select Copy From in the Creation Mode box, and then select the options for the user profile that you want to copy. Click Create User.

  NOTE If you are copying a user profile from a different version of the application, you can only copy hotkey preferences.

  The user is created, and you are returned to the Project Management or Preferences menu.

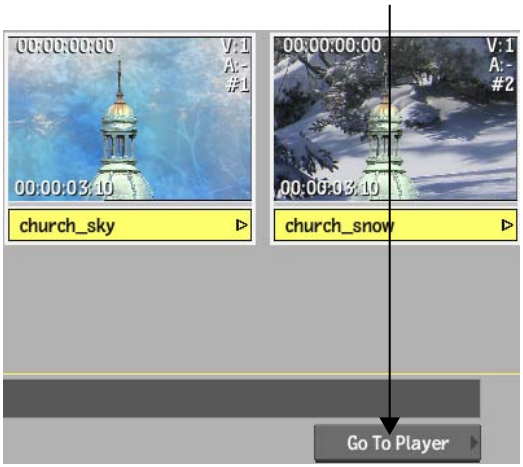
5 To load the user into the current work session, click Load from the Preferences menu. From the Project Management menu, click Start.

Playing Clips from the Library

New for this release: You can play a selection of source clips and processed clips directly from the library.

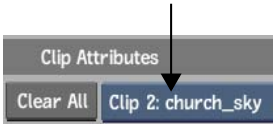
To play clips from the library:

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.

Available Libraries

New for this release: Additions to the clip library give you direct access to Flare libraries and hidden libraries. You can also control whether you want network-accessed libraries to appear in the Clip Library box. It is also now easier to distinguish between library types in the Clip Library box.

You can open libraries that are read-write as well as those that are read-only. You can load clips from either type of library. However, you can save clips only to a read-write library.

To do any type of clip library management, you must have read-write access to the library.

The list of available libraries appear in the Clip Library box. This list may become very long, especially if you accessed libraries belonging to other projects (through the network panel). You can select whether you want network-accessed libraries to appear in the Clip Library box.

You can also make hidden libraries available (for example, **_cache** and **._Backup)**, which are by default not visible.

If you have a Flare system that is connected remotely to your Inferno system, you can enable read-write access to Flare libraries with the R/W button. See Managing Flare Libraries from Inferno on page 32.

To control the libraries displayed in the Clip Library box:

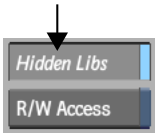
➤ From the clip library, select an option from the Show Library box.

Select:	To display:
All Libraries	All libraries belonging to all projects, including those accessed through the network panel.
Current Project Libraries	All libraries belonging to the current project. Note that if the current library is accessed through the network panel, this option is greyed out.

NOTE If you are in Dual View, each view has its own Show Library box. You can select any option for either view.

To display hidden libraries in the Clip Library box:

➤ Enable Hidden Libs.

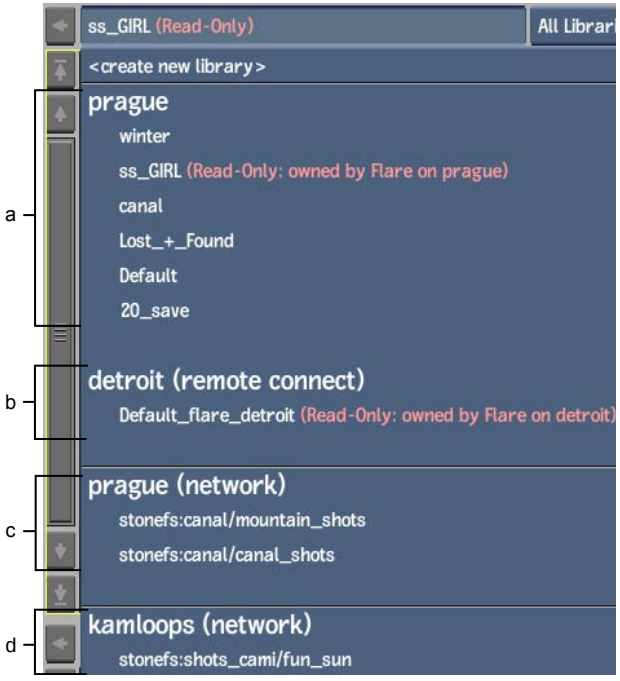


Order of Available Libraries

To make it easier to identify where libraries are located and whether they belong to the current project, libraries appear in the following order in the Clip Library box:

- First, libraries that you own and that belong to the current project appear.
- Secondly, libraries that you do not own and that belong to the current project appear.
- Thirdly, libraries belonging to other projects on the system to which you are connected appear.
- Lastly, libraries belonging to other projects on other systems appear.

All read-only libraries provide more information about their status in the Clip Library box and in the Library status bar.



(a) Current project libraries owned by Inferno (b) Current project libraries not owned by Inferno (c) Network library of another project on the system to which Inferno is connected (d) Network library of another project on another system

Interoperability Workflow Improvements

10

Topics in this chapter:

Importing Final Cut Pro XML

New for this release: A new Skip Recapture button appears in the Import XML menu. This is useful when importing timelines that point to media of a different format (such as when trying to relink to original sources).

Each XML file corresponds to an FCP sequence, including video, audio, and select transitions and effects, that can be opened in the timeline. Some unsupported effects are marked with comments indicating what you have to rebuild in Inferno based on the original offline edit.

FCP XML supports 720/24p, 720/30p, and 720/60p output from Varicam to create XML files. The timecode of the source is always 59.94, but the timelines can be 24p, 30p, or 60p.

Inferno can conform XML (23.976/29.97/59.94) from Varicam material (23.976/29.97/30/59.94/60). Inferno can also remove flagged (non-active) frames when capturing Varicam media. The source material must be at the same framerate as the sequence (for example 23.97 in a 23.97 sequence).

Before importing XML that contains any MXF P2 content, ensure that the P2 recording devices that are used are set to record clip metadata in Type 2. This allows important metadata, such as the User Clip Name, to be assigned properly in the XML file.

When importing FCP XML files that were created in SD resolution and that need to be conformed in HD, you need to reformat the clips.

After importing an XML file, you recapture the footage from the original tapes using the Recapture tool. You can also reload file-based media. Once all the media is captured, imported, or soft-imported, you can relink it to the sequence.

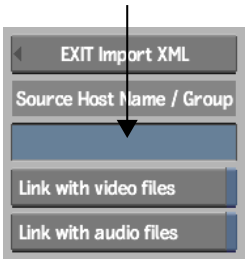
To import or soft-import an FCP XML file:

1 In the clip library, from the Interchange Format box, select XML.

2 Click Import.

3 In the file browser, navigate to an XML file exported from FCP.

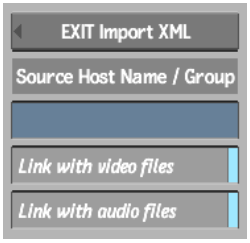
4 If you are importing FCP XML that contains media files, and your files are stored on other computers, you can use Wiretap to access them. Enter the host or group name, as configured in the *sw_wiretap_path_translation_db.xml* file.



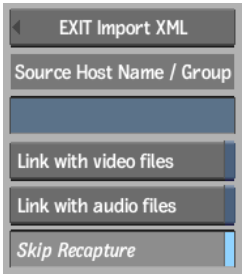
The Wiretap Server must be configured properly to recognise the different hosts that you want to retrieve media from. See the *Autodesk Stone and Wire Filesystem and Networking Guide* or contact your system administrator.

If you do not enter a value in the Source Host Name / Group field, any paths contained in the FCP XML file will be interpreted as pointing to your Inferno workstation. You can change the path in the Recapture dialog box when relinking.

5 If you are importing FCP XML that contains media files of the format and resolution that you want to use, enable Link with video files and/or Link with audio files.



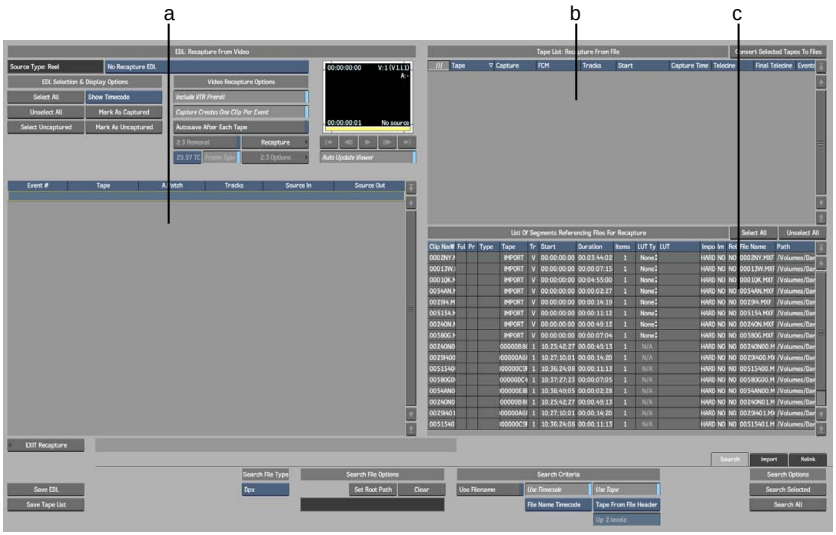
6 If you are importing FCP XML that contains media files of a different file type or resolution than what you want to use (for example, when trying to relink to source media after working with proxies), then enable the Skip Recapture button.



Upon clicking Load, you are taken to the Library menu, where you can apply a Reformat action (through the Tools menu) to your timeline, and then use Recapture to load your source media.

7 Click Load.

If you did not enable Skip Recapture, the Recapture menu appears.



(a) Clip List: List of Clips on Tape (b) Tape List: Recapture From File (c) List of Segments Referencing Files For Recapture

8 If you are importing FCP XML that references media clips on tapes, a list of clips appears in the Clip List. A list of source tapes also appears in the Tape List. This is the same tape list as logged in FCP.

9 If you are importing FCP XML that contains media files, a list of files appears in the List of Segments. This is the list that you need to relink. See Relinking to File-based Media on page 152.

Relinking to File-based Media

New for this release: The Recapture screen now contains updated options for media file search, import, and relink options.

To facilitate file-based conform from FCP XML, the new search feature is able to intelligently and automatically find and read image sequences (DPX) or streaming media (MXF, QT), based on preset search rules and criteria.

Relink problems are reduced with the new Copy from Selected Clip button. This button copies the formatting information of the selected clip into the Resolution parameters; in effect, providing the parameters (resolution, frame rate, bit depth, etc.) by which to “resize” a target FCP XML clip.

After having edited sequences in Final Cut Pro using file-based media, such as QuickTime movies, you can relink the exported XML to these files in Inferno.

If the media originated on tape, you can opt to relink to the captured QuickTime files instead of recapturing the media from tape, if you choose to use the captured resolution from Final Cut Pro.

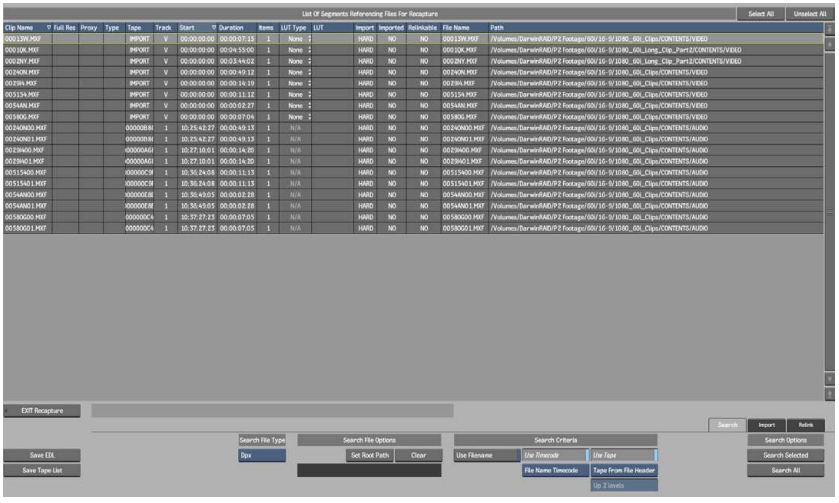
You can relink imported FCP XML files to file-based video or audio media.

Verify that the files you are relinking to are supported in Inferno.

If you are importing FCP XML with Varicam support, it will be identified as such in the Source Type box.

To relink FCP XML to file-based media:

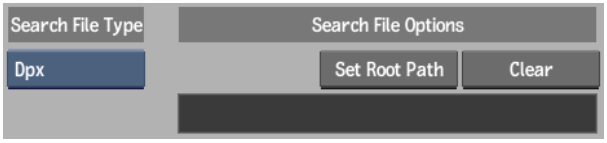
1 Swipe to the right to display the List of Segments Referencing Files For Recapture table across the full screen.



2 Use the Search Criteria to find the source media files.

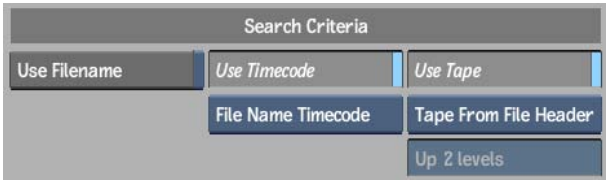
When importing sequences (DPX) or streaming media (MXF, QT) that are referenced by XML files, you can use the advanced Search feature to find and read these image sequences or streaming media, based on preset search rules and criteria. These media files are mostly arranged in hierarchical structures that can be identified and traversed through all the subdirectories from a given root destination. Providing additional criteria, such as file type, tape name, and timecode can help to narrow and pinpoint the search.

3 From the Search File Type box, select the file type to search for.



4 Click Set Root Path to select the root directory where the search will start.

5 Select the criteria that you want to match on.



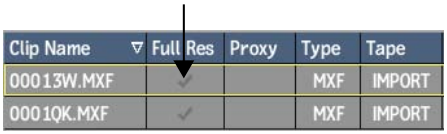
Select	To
Use Filename	Search for the exact file name referenced in the XML.
Use Timecode	Read the metadata from found items to make sure that the starting timecode matches the one in the edit list. Select whether to read the timecode from the file header (MXF and QuickTime) or from the file name (DPX).
Use Tape	Read the metadata from found items to make sure that the tape name matches the one in the edit list. Select whether to read the tape name from the file header (MXF and QuickTime) or from the directory (DPX).

If DPX is selected as a format, Use Timecode and Use Tape are on, but Use Filename is turned off. If MXF or QuickTime is selected, Use Filename and Use Timecode are turned on, but Use Tape is off.

6 Once all settings are made for the selected file type, click Search Selected or Search All.

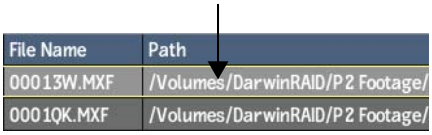
A progress bar appears.

The results update the List of Segments Referencing Files For Recapture table. A checkmark appears in the Full Res column for each clip when the full-resolution version of a file is found. The Type column lists the file type/extension. The Tape and Path columns are also updated.



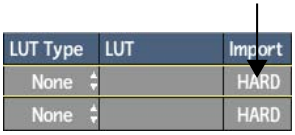
7 If files are not found, you can redo the search by deselecting match criteria. You can also run the search for a different file type. Data for all the previously found files is kept.

8 If the file search still does not find your media, you will have to enter the correct path names manually for each clip. To change the path, select the unfound media segments in the list, and then click the Path field.



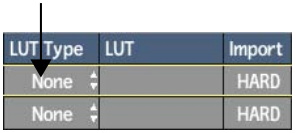
This opens the library browser where you can choose a different path.

9 Optional: To soft-import the media files, click Select All or drag to select the segments in the list and then, in the Import column, drag left or right to toggle between SOFT and HARD.

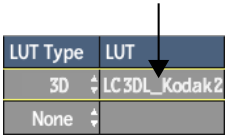


NOTE Not all files can be soft-imported (for example, audio files at 44.1 kHz).

10 Optional: If you want to apply a LUT or gamma correction to your media files, click Select All or drag to select the segments in the list and then, from the LUT Type column, select an option.



If you selected 1D LUT or 3D LUT for LUT Type, click in the LUT column to choose a specific LUT name.



The LUT file name appears in the LUT column.

11 Click the Import tab.

The Import File options appear.

12 Set any options, as needed.

WARNING These options are provided here in case you are having trouble relinking certain media files. If the files are already found, changing any of these options may prevent them from relinking.

13 Optional: Change the name of the reel in the Library Reel Name field.

14 Click Import All Files.

All the files should now be imported or soft-imported.

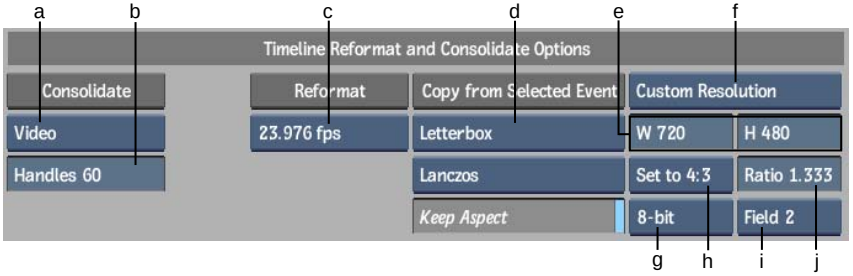
Import	Imported	Relinkable	File Name
SOFT	YES	YES	00013W.MXF
SOFT	YES	YES	0001QK.MXF

The Imported column indicates whether a file was imported or not. The Relinkable column displays whether the file is relinkable. A file can be imported and non-relinkable if a discrepancy exists between the resolution of the XML and the found media. Also, the media may have already been

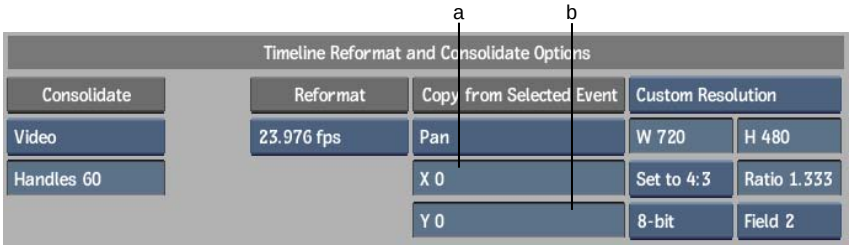
imported previously, in which case this column would already be checked accordingly.

15 Click the Relink tab.

The Timeline Reformat and Consolidate options appear.



(a) Consolidate option box (b) Handles field (c) Frame Code Mode box (d) Fit Method box (e) Width and Height fields (f) Resolution Presets box (g) Bit Depth box (h) Aspect Ratio Presets box (i) Scan Mode box (j) Aspect Ratio field



(a) Pan Start X field (b) Pan Start Y field

16 If you need to reformat the timeline to match the resolution of a given clip, specify the destination resolution by doing one of the following:

- Click the Copy from Selected Clip button to copy the formatting information of a selected clip into the Resolution parameters.
- Select a preset from the Resolution Presets box.
- Specify the dimensions using the Width and Height fields.

17 From the Frame Code Mode box, set the frame rate and drop frame mode as needed.

If you have a clip that contains some linked media and some unlinked metadata, when you change the frame code mode such that the duration of the clip is affected, the unlinked metadata and linked media are treated

differently. The linked media is timewarped to accommodate the new duration. For unlinked metadata, if more material is needed to accommodate the change in duration, it is input when the clip is recaptured. Effects will look identical, although the timing of the clip will be adjusted.

18 Set the aspect ratio, bit depth, and scan mode as needed.

19 If your clip contains video tracks or segments that still contain media (for example, module-processed shots), select a resize fit method from the Fit Method box.

20 Click Reformat and confirm the action. If there are multiple clips to confirm, you can click Confirm All to confirm them all or click Confirm for each clip.

The clip metadata for the timeline clips is updated to the specified values. Any existing media is also converted and resized using the specified fit method. You can now recapture the media associated with these clips in the appropriate format.

21 If consolidation was not performed in FCP, do it now.

1 From the Consolidate box, select Audio, Video, or All Tracks.

This determines which tracks are affected by the consolidate operation.

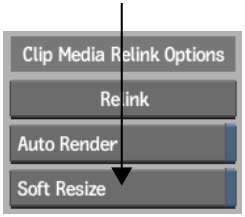
2 In the Handles field, set the maximum number of head and tail frames that you want to retain after consolidating the clip.

3 Click Consolidate and confirm the operation.

22 Ensure that all media files are relinkable. Click the Relinkable header to sort the list and group any NOs at the top.

Some files that are not relinkable may only need to be resized to be compatible. For example, the Soft Resize feature allows you to import Quicktime files, included with the FCP XML, at 720x480 instead of the usual NTSC 720x486.

23 For files that are not relinkable, enable Soft Resize, and then click Import Selected Files.



Soft resize is applied to all clips that need it.

If the files are still not relinkable then they cannot be soft-imported. Try to import the files normally, or check with your system administrator.

24 Click Relink.

A new reel with the XML filename is created in the clip library for each imported XML file. The assembled clip in the new reel has the same name as the original FCP sequence, and is placed in the same reel as the media.

25 Click Exit Recapture to end the session.

Importing AAF Files

New for this release: A new Skip Recapture button appears in the Import AAF menu. This is useful when importing timelines that point to media of a different format (such as when trying to relink to original sources).

Import AAF sequences from Avid in the same way that you import an EDL. Each AAF file corresponds to a sequence that can be opened in the timeline, including video and audio layers, and select transitions and effects. Some unsupported effects are marked with comments indicating what you have to rebuild in Inferno based on the original offline edit.

Avid can support 720/24p, 720/30p, and 720/60p output from Varicam to create AAF files. The timecode of the source is always 59.94 but the timelines can be 24p, 30p, or 60p.

Inferno has the ability to conform AAF (23.976/29.97/59.94) from Varicam material (23.976/29.97/30/59.94/60). Inferno can also remove non-active frames when capturing Varicam media. The source material must be at the same framerate as the sequence (for example, 23.97 in a 23.97 sequence).

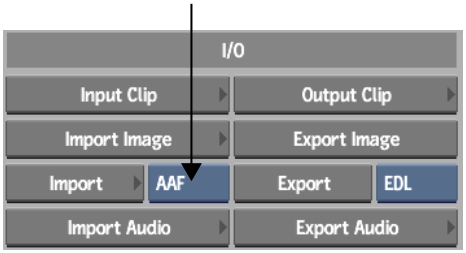
After importing an AAF file, you recapture the footage from the original tapes using the Recapture tool. You can also reload file-based media. Once all the media is captured or reloaded, it is relinked to the sequence.

When importing AAF files that were created in SD resolution and that need to be conformed in HD, you will need to reformat the clips.

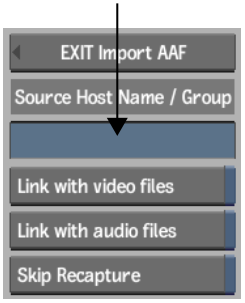
A new reel with the AAF filename is created in the clip library for each imported AAF file. The assembled clip in the new reel has the same name as the original AAF sequence.

To import an AAF file:

1. In the Clip Library menu, from the Interchange Format box, select AAF.



2. Click Import.
3. In the file browser, navigate to an AAF file exported from an Avid application.
4. If you are importing AAF that contains media files, and your files are stored on other computers, you can use Wiretap to access them. Enter the host or group name, as configured in the *sw_wiretap_path_translation_db.xml* file.

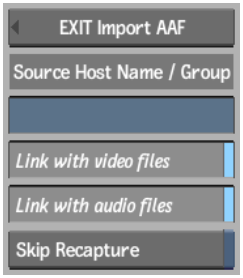


The Wiretap Server must be configured properly to recognise the different hosts that you want to retrieve media from. See the *Autodesk Stone and Wire Filesystem and Networking Guide* or contact your system administrator.

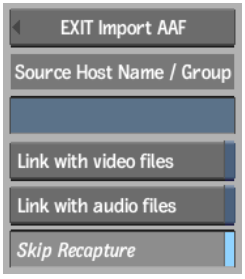
If you do not enter a value in the Source Host Name / Group field, any paths contained in the AAF file will be interpreted as pointing to your

Inferno workstation. You can change the path in the Recapture dialog box when relinking.

5 If you are importing AAF that contains media files of the format and resolution that you want to use, enable Link with video files and/or Link with audio files.



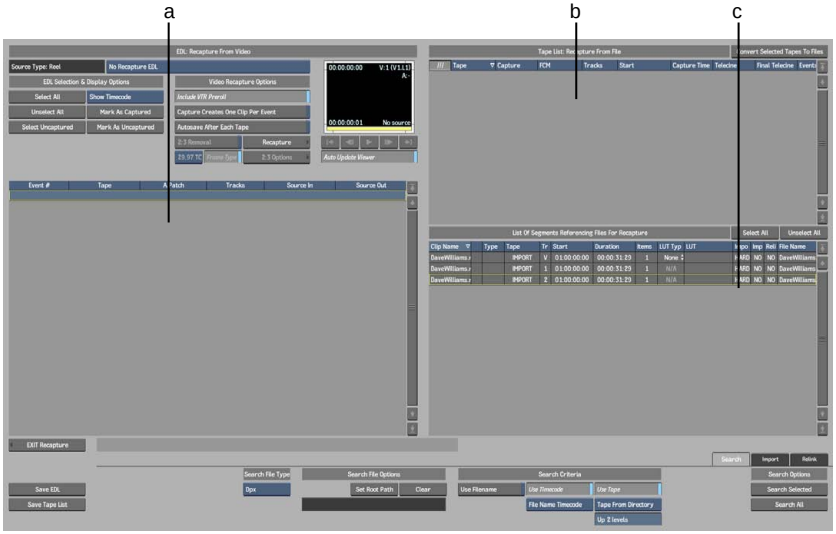
6 If you are importing AAF that contains media files of a different file type or resolution than what you want to use (for example, when trying to relink to source media after working with proxies), then enable the Skip Recapture button.



Upon clicking Load, you will be taken to the Library menu, where you can then apply a Reformat action (through the Tools menu) to your timeline, and then use Recapture to load your source media.

7 Click Load.

If you did not enable Skip Recapture, the Recapture menu appears.



(a) Clip List: List of Clips on Tape (b) Tape List: Recapture From File (c) List of Segments Referencing Files For Recapture

8 If you are importing AAF that references media clips on tapes, a list of clips appears in the Clip List. A list of source tapes also appears in the Tape List. This is the same tape list as logged in your Avid application.

9 If you are importing AAF that contains media files, a list of files appears in the List of Segments. This is the list that you need to relink. See Relinking to File-based Media on page 162.

Relinking to File-based Media

New for this release: The Recapture screen now contains updated options for media file search, import, and relink options.

To facilitate file-based conform from AAF, the new search feature is able to intelligently and automatically find and read image sequences (DPX) or streaming media (MXF, QT), based on preset search rules and criteria.

Relink problems are reduced with the new Copy from Selected Clip button. This button copies the formatting information of the selected clip into the Resolution parameters; in effect, providing the parameters (resolution, frame rate, bit depth, etc.) by which to “resize” a target AAF clip.

After having edited sequences in your Avid application using file-based media, such as MXF files or QuickTime movies, you can relink the exported AAF to these files in Inferno.

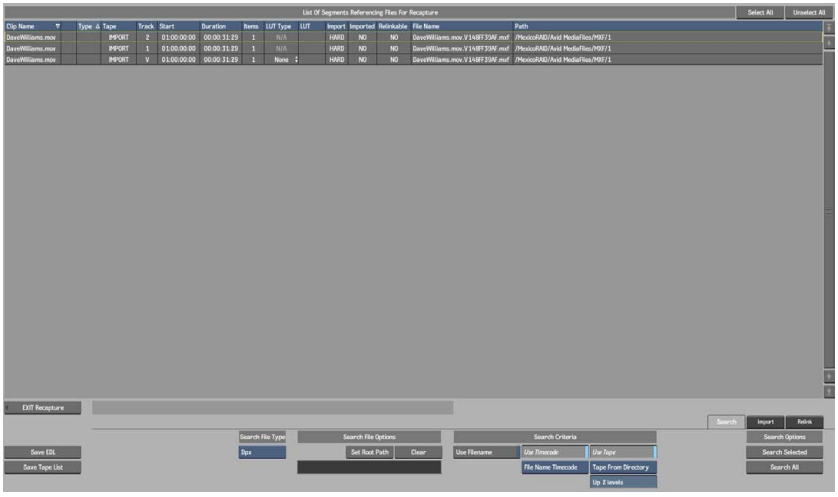
Verify that the media files you are relinking to are supported in Inferno.

You can relink imported AAF files to file-based video or audio media.

If you are importing an AAF file with Varicam support, it will be identified as such in the Source Type box.

To relink AAF to file-based media:

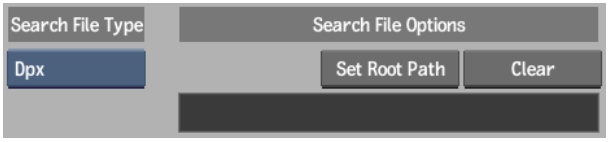
1. Swipe to the right to display the List of Segments Referencing Files For Recapture table across the full screen.



2. Use the Search Criteria to find the source media files.

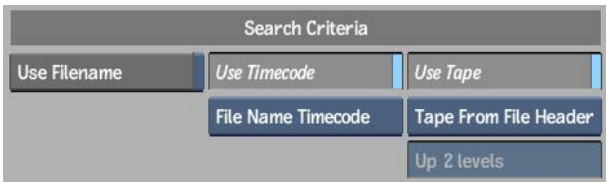
   When importing sequences (DPX) or streaming media (MXF, QT) that are referenced by AAF files, you can use the advanced Search feature to find and read these image sequences or streaming media, based on preset search rules and criteria. These media files are mostly arranged in hierarchical structures that can be identified and traversed through all the subdirectories from a given root destination. Providing additional criteria, such as file type, tape name, and timecode can help to narrow and pinpoint the search.

3. From the Search File Type box, select the file type to search for.



4 Click Set Root Path to select the root directory where the search will start.

5 Select the criteria that you want to match on.



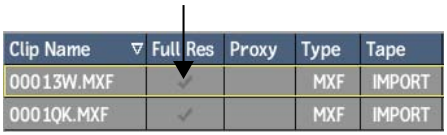
Select	To
Use Filename	Search for the exact file name referenced in the AAF.
Use Timecode	Read the metadata from found items to make sure that the starting timecode matches the one in the edit list. Select whether to read the timecode from the file header (MXF and QuickTime) or from the file name (DPX).
Use Tape	Read the metadata from found items to make sure that the tape name matches the one in the edit list. Select whether to read the tape name from the file header (MXF and QuickTime) or from the directory (DPX).

If DPX is selected as a format, Use Timecode and Use Tape are on, but Use Filename is turned off. If MXF or QuickTime is selected, Use Filename and Use Timecode are turned on, but Use Tape is off.

6 Once all settings are made for the selected file type, click Search Selected or Search All.

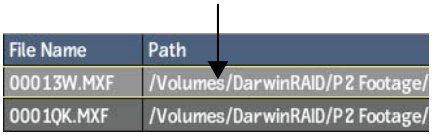
A progress bar appears.

The results update the List of Segments Referencing Files For Recapture table. A checkmark appears in the Full Res column for each clip when the full-resolution version of a file is found. The Type column lists the file type/extension. The Tape and Path columns are also updated.



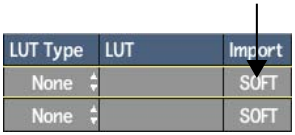
7 If files are not found, you can redo the search by deselecting match criteria. You can also run the search for a different file type. Data for all the previously found files is kept.

8 If the file search still does not find your media, you will have to enter the correct path names manually for each clip. To change the path, select the unfound media segments in the list, and then click the Path field.



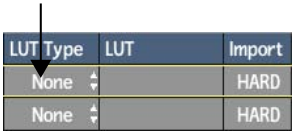
This opens the library browser where you can choose a different path.

9 If you want to soft-import the media files, click Select All or drag to select the segments in the list, and then, in the Import column, drag left or right to toggle between SOFT and HARD.

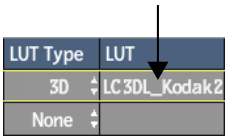


NOTE Not all files can be soft-imported (for example, audio files at 44.1 kHz).

10 Optional: If you want to apply a LUT or gamma correction to your media files, click Select All or drag to select the segments in the list, and then, from the LUT Type column, select an option.



If you selected 1D LUT or 3D LUT for LUT Type, click in the LUT column to choose a specific LUT name.



The LUT file name appears in the LUT column.

11 Click the Import tab.

The Import File options appear.

12 Set any options, as needed.

WARNING These options are provided here in case you are having trouble relinking certain media files. If the files are already found, changing any of these options may prevent them from relinking.

13 Optional: Change the name of the reel in the Library Reel Name field.

14 Click Import All Files.

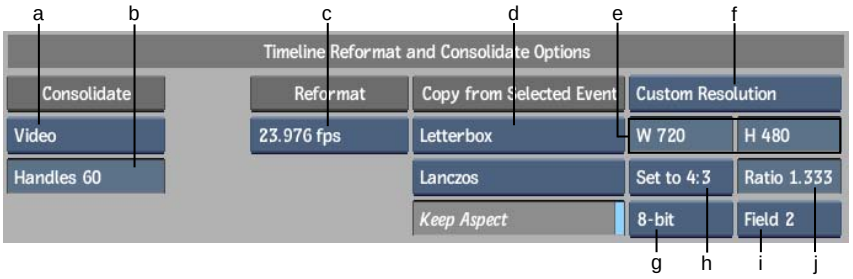
All the files should now be imported or soft-imported.

Import	Imported	Relinkable	File Name
SOFT	YES	YES	00013W.MXF
SOFT	YES	YES	0001QK.MXF

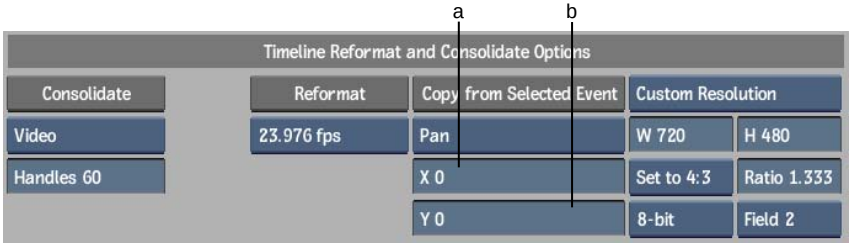
The Imported column indicates whether a file was imported or not. The Relinkable column displays whether the file is relinkable. A file can be imported and non-relinkable if a discrepancy exists between the resolution of the AAF and the found media. Also, the media may have already been imported previously, in which case this column would already be checked accordingly.

15 Click the Relink tab.

The Timeline Reformat and Consolidate options appear.



(a) Consolidate option box (b) Handles field (c) Frame Code Mode box (d) Fit Method box (e) Width and Height fields (f) Resolution Presets box (g) Bit Depth box (h) Aspect Ratio Presets box (i) Scan Mode box (j) Aspect Ratio field



(a) Pan Start X field (b) Pan Start Y field

16 Optional: To reformat the timeline to match the resolution of a given clip, specify the destination resolution by doing one of the following:

- Click the Copy from Selected Clip button to copy the formatting information of a selected clip into the Resolution parameters.
- Select a preset from the Resolution Presets box.
- Specify the dimensions using the Width and Height fields.

17 From the Frame Code Mode box, set the frame rate and drop frame mode as needed.

If you have a clip that contains some linked media and some unlinked metadata, when you change the frame code mode such that the duration of the clip is affected, the unlinked metadata and linked media are treated differently. The linked media is timewarped to accommodate the new duration. For unlinked metadata, if more material is needed to accommodate the change in duration, it is input when the clip is recaptured. Effects will look identical, although the timing of the clip will be adjusted.

18 Optional: Set the aspect ratio, bit depth, and scan mode.

19 Optional: If your clip contains video tracks or segments that still contain media (for example, module-processed shots), select a resize fit method from the Fit Method box.

20 Click Reformat and confirm the action. If there are multiple clips to confirm, you can click Confirm All to confirm them all or click Confirm for each clip.

The clip metadata for the timeline clips is updated to the specified values. Any existing media is also converted and resized using the specified fit method. You can now recapture the media associated with these clips in the appropriate format.

21 If consolidation was not performed in your Avid application, do it now.

1 From the Consolidate box, select Audio, Video, or All Tracks.

This determines which tracks will be affected by the consolidate operation.

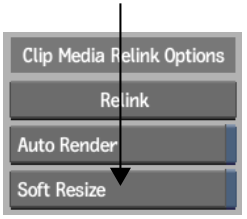
2 In the Handles field, set the maximum number of head and tail frames that you want to retain after consolidating the clip.

3 Click Consolidate and confirm the operation.

22 Ensure that all media files are relinkable. Click the Relinkable header to sort the list and group any NOs at the top.

Some files that are not relinkable may only need to be resized to be compatible. For example, the Soft Resize feature allows you to import Quicktime files, included with the AAF, at 720x480 instead of the usual NTSC 720x486.

23 For files that are not relinkable, enable Soft Resize, and then click Import Selected Files.



Soft resize is applied to all clips that need it.

If the files are still not relinkable then they cannot be soft-imported. Try to import the files normally, or check with your system administrator.

24 Click Relink.

A new reel with the AAF filename is created in the clip library for each imported AAF file. The assembled clip in the new reel has the same name as the original Avid sequence, and is placed in the same reel as the media.

25 Click Exit Recapture to end the session.

Supported and Unsupported Transitions and Effects

New for this release: There is now more support for Avid AAF transitions and effects.

The information in this section indicates how AAF data, transitions, and effects map to Inferno data, transitions, and effects. There are three levels of support that can be indicated for each:

- **Translated** Parameters from the AAF file are read and translated as Inferno Soft Effect Parameters.
- **Converted** The effect type is converted without parameters from the AAF file to a Inferno Soft Effect or Transition. Some effects are recreated with a similar effect. Some are only recreated as an empty effect.
- **Not supported** The effect is not supported and replaced by a comment or by a default transition or effect.

In addition, the following restrictions must also be taken into account:

- Only flattened Photoshop (.psd) files are supported (layered files will be imported as flattened in Inferno).
- Only RGB material is supported, not RGBA.
- Graphics files of resolutions bigger or smaller than the timeline are imported as Center/Crop mode only.
- AAF files that contain MP3 media files cannot be relinked on Inferno for Linux, since the MP3 file format is not supported on Linux.
- Nested effects are not supported.

- All video and audio tracks for each sequence must be imported.

General

The following tables describe how metadata and media are supported in Inferno.

Metadata

Avid	Inferno
All Video / Audio tracks	All Video / Audio tracks
Tape name	Tape name data
Source / Record Timecode	Source / Record Timecode
Drop / Non-Drop Timecode	Drop-Frame / Non-Drop-Frame
Mark In / Out	Not supported
Keycode	Supported
Video locator	Cue mark with locator text (no text data is translated)
23.976 / 29.97 timecode	23.976 / 29.97 timecode data provided (24p sequences are 23.976 for REC and 29.97 for SRC)

Media Import

Avid	Inferno
Video media (JPEG, TIFF, etc.)	Paths and filenames translated
Audio media (WAV, AIFF)	Paths and filenames translated
Embedded/Linked Video media data (OMF, MXF)	Not supported
Embedded/Linked Audio media data (WAV, AIFF)	Not supported

Media Export

Avid	Inferno
Video media (OMF, MXF)	Not supported
Audio media (WAV, AIFF)	Not supported

Sequence

Avid	Inferno
Video only	Video-only sequence
Audio only	Audio-only sequence
Video and audio	Video and Audio sequence

Video and Audio Transitions

The following tables describe how transitions are supported in Inferno.

Transitions marked with an * are also supported with the "Inverse" option set.

Blend

Avid	Inferno
Dip to colour	Translated to Dissolve with colour data value
Dissolve	Converted to Dissolve (linear animation)
Fade from colour	Supported; background is black only, reset manually
Fade to colour	Supported; background is black only, reset manually
Picture-in-picture	Not supported; replaced by Dissolve + cue mark

Film

Avid	Inferno
Film dissolve	Converted to Dissolve (hermite animation)
Film fade	Converted to Dissolve (linear animation)

Box wipe

Avid	Inferno
Bottom box*	Converted to SMPTE 025; softness not supported
Bottom left to top right*	Converted to SMPTE 006; softness not supported
Bottom right to top left*	Converted to SMPTE 005; softness not supported
Left box*	Converted to SMPTE 026; softness not supported
Right box*	Converted to SMPTE 024; softness not supported
Top box*	Converted to SMPTE 023; softness not supported
Top left to bottom right*	Converted to SMPTE 003; softness not supported
Top right to bottom left*	Converted to SMPTE 004; softness not supported

Edge Wipe

Avid	Inferno
Horizontal*	Converted to SMPTE 001; softness not supported
Horz open*	Converted to SMPTE 021; softness not supported
Bottom left diagonal*	Converted to SMPTE 042; animation is inverted, softness not supported
Bottom right diagonal*	Converted to SMPTE 041; animation is inverted, softness not supported
Upper left diagonal*	Converted to SMPTE 041; softness not supported
Upper right diagonal*	Converted to SMPTE 042; softness not supported
Vert open*	Converted to SMPTE 022; softness not supported
Vertical*	Converted to SMPTE 002; softness not supported

Shape Wipe

Avid	Inferno
4 corners*	Converted to SMPTE 007; softness not supported

Avid	Inferno
Horizontal bands	Not supported; replaced by Dissolve + cue mark
Horizontal blinds	Not supported; replaced by Dissolve + cue mark
Vertical blinds	Not supported; replaced by Dissolve + cue mark
Center box*	Converted to SMPTE 101; softness not supported
Circle*	Converted to SMPTE 119; softness not supported
Ellipse*	Converted to SMPTE 120; softness not supported
Clock*	Converted to SMPTE 201; softness not supported
Diamond*	Converted to SMPTE 102; softness not supported

Sawtooth Wipe

Avid	Inferno
Horizontal sawtooth*	Converted to SMPTE 071; softness not supported
Horz open sawtooth*	Converted to SMPTE 073; softness not supported
Vert open sawtooth*	Converted to SMPTE 074; softness not supported
Vertical sawtooth*	Converted to SMPTE 072; softness not supported

Matrix Wipe

Avid	Inferno
Grid*	Not supported; replaced by SMPTE 008
One-way row	Not supported; replaced by SMPTE 001 + comment
Speckle	Not supported; replaced by SMPTE 001 + comment
Spiral	Not supported; replaced by SMPTE 001 + comment
Zig-zag	Not supported; replaced by Cue mark

Xpress 3D Effect

Avid	Inferno
3D ball	Not supported; replaced by Cue mark
3D page fold	Not supported; replaced by Cue mark
3D slats	Not supported; replaced by Cue mark
3D PIP	Translated to Soft Axis (Position / Scaling (ISO, Softness / Crop)

Miscellaneous

Avid	Inferno
Conceal	Converted to Soft Axis (Conceal effect); softness not supported
L-Conceal	Converted to Soft Axis (L-Conceal effect); softness not supported
Squeeze	Converted to Soft Axis (Squeeze effect); softness not supported
Peel	Not supported; replaced by Dissolve + cue mark
Push	Not supported; replaced by Dissolve + cue mark
Spin	Not supported; replaced by Dissolve + cue mark
Video gap	Video gap
Video filler	Video gap
Video match frame edit	Match frame

Video and Audio Effects

The following tables describe how effects are supported in Inferno.

Blend

Avid	Inferno
Picture-in-picture	Supported
Superimpose	Translated to Soft Blend (transparency value is translated)

NOTE Scaling in AAF can be X and Y. Inferno only supports one value (X or Y) for both.

Film

Avid	Inferno
1.66 mask	Supported; bkg is black, no mask, horizontal position off
1.85 mask	Supported; bkg is black, no mask, horizontal position off
16:9 mask	Supported; bkg is black, no mask, horizontal position off
Anamorphic mask	Supported; bkg is black, no mask, horizontal position off
Mask	Supported; bkg is black, no mask, horizontal position off
Blowup	Supported

AVX Plugin

Avid	Inferno
Illusion FX	Not supported; replaced by Cue mark
AVX Plugins	Not supported; replaced by Cue mark

Image

Avid	Inferno
Avid Pan and Zoom	Not supported; replaced by Cue mark
Blur effect	Not supported; replaced by Cue mark
colour Correction	Converted to Soft CC (empty) + cue mark
colour Effect	Converted to Soft CC (empty) + cue mark

Avid	Inferno
Flip	Converted to Soft Axis (Flip effect)
Flip-flop	Converted to Soft Axis (Flip-flop effect)
Flop	Converted to Soft Axis (Flop effect)
Mask	Supported; bkg is black, no mask, horizontal position off
Resize	Supported; background is black, no left and right cropping
Scratch removal	Not supported; replaced by Cue mark
Submaster	Converted to Container

Reformat

Avid	Inferno
14:9 Letterbox	Not supported; replaced by Soft Axis + Cue mark
16:9 Letterbox	Not supported; replaced by Soft Axis + Cue mark
4:3 Sidebar	Not supported; replaced by Soft Axis + Cue mark
Pan and Scan	Not supported; replaced by Soft Axis + Cue mark

Titles

Avid	Inferno
Title	Not supported; only text string is available
Marquee Text	Not supported; only text string is available

Key

Avid	Inferno
Animate	Not supported; replaced by Cue mark
Chroma key	Converted to Soft Axis (empty)
Luma key	Converted to Soft Axis (empty)
Matte key	Converted to Soft Axis (empty)

Avid	Inferno
RGB keyer	Not supported; replaced by Cue mark

Miscellaneous

Avid	Inferno
Timewarp	Converted to Soft TW; recreate the curve type
Motion Effect	Translated to Soft TW (Constant speed, no strobe effect)
3D PIP	Supported
Peel	Not supported; replaced by Dissolve + cue mark
Push	Not supported; replaced by Dissolve + cue mark
Spin	Not supported; replaced by Dissolve + cue mark
Squeeze	Not supported; replaced by Dissolve + cue mark
Video gap	Video gap
Video filler	Video gap
Video match frame edit	Match frame

General Audio

Avid	Inferno
Audio level	Audio gain
Audio dissolve	Audio dissolve
Audio fade in	Audio dissolve
Audio fade out	Audio dissolve
Audio gap	Audio gap
Audio filler	Audio gap
Audio match frame edit	Match frame splice

Audio Suite Plugin

Avid	Inferno
Chorus	Not supported; replaced by Cue mark
D-verb	Not supported; replaced by Cue mark
Compressor	Not supported; replaced by Cue mark
Limiter	Not supported; replaced by Cue mark
Expander-gate	Not supported; replaced by Cue mark
Gate	Not supported; replaced by Cue mark
DeEsser	Not supported; replaced by Cue mark
1-band EQII	Not supported; replaced by Cue mark
4-band EQII	Not supported; replaced by Cue mark
Flanger	Not supported; replaced by Cue mark
Invert	Not supported; replaced by Cue mark
Duplicate	Not supported; replaced by Cue mark
Delay	Not supported; replaced by Cue mark
Multi-tap delay	Not supported; replaced by Cue mark
Normalize	Not supported; replaced by Cue mark
Gain	Not supported; replaced by Cue mark
Ping-pong delay	Not supported; replaced by Cue mark
Reverse	Not supported; replaced by Cue mark
DC offset removal	Not supported; replaced by Cue mark
Signal generator	Not supported; replaced by Cue mark
Time compression exp	Not supported; replaced by Cue mark
Pitch shift	Not supported; replaced by Cue mark

Editorial and Timeline Improvements

11

Topics in this chapter:

Navigating Edit Sequences

New for this release: You can easily navigate through a clip's edit sequence directly on the Desktop. You can navigate between cuts and transitions on all layers and tracks. You can also navigate between tracks and layers and change the clip's focus.

When you navigate between tracks, the track you navigate to becomes the Primary video track. You do not have to manually change the Primary track of the clip's timeline in the Player to see the results of the new track.

When navigating to a new track, you go to the topmost layer of the track.

When you navigate between layers and tracks, only the video tracks and video layers cycle unless you are in the Player timeline. In the Player timeline, navigating between layers and tracks also cycles the audio tracks.

When you have navigated to the topmost or bottommost layer or track, the tracks and layers keep cycling in the same order.

To retain the same video clip length regardless of focus as you navigate, black frames are added to make each layer the same length as the video clip duration. Video on the next layer down is always seen through a gap.

Clips appear in realtime. If soft effects cannot be displayed, the words "Unrendered Frame" appear on the applicable frames.

To navigate clips:

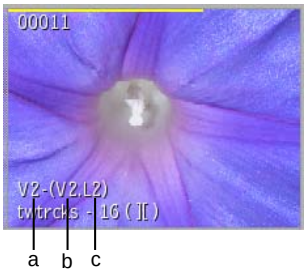
1. Uncollapse the clip on a Desktop reel.
2. With the cursor over the Desktop clip, use any of the following hotkey and Player control combinations.

(a) Previous Clip Player control (b) Next Clip Player control

To go to:	Press:
The next cut or transition on the current layer	**Alt**+Next Clip Player control
The previous cut or transition on the current layer	**Alt**+Previous Clip Player control
The next cut or transition on any layer or track for the duration of the current track	**Shift**+**Alt**+Next Clip Player control
The previous cut or transition on any layer or track for the duration of the current track	**Shift**+**Alt**+Previous Clip Player control
The next layer or track down in the vertical edit	**down arrow**
The next layer or track up in the vertical edit	**up arrow**
The next video track down	**Ctrl**+**Alt**+**down arrow**

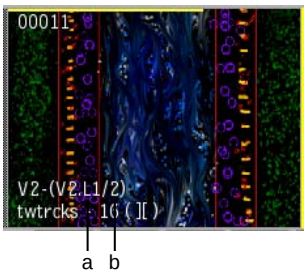
To go to:	Press:
The next video track up	**Ctrl+Alt+up arrow**

As you navigate between layers and tracks, information on the Desktop clip 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

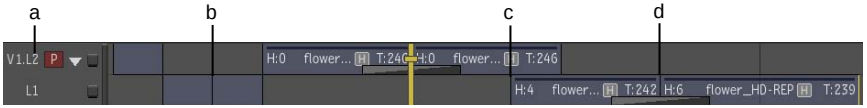
Identifying Cuts and Transitions

As you navigate an edit sequence, new visual markers can help you identify the location where one element ends and another begins.

Cuts and transitions on an uncollapsed clip appear as follows on a Desktop reel:

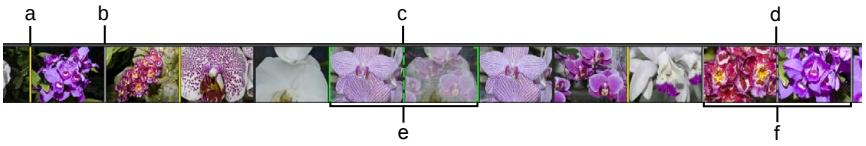
- Cuts between clips on the focus layer in an edit sequence appear as yellow lines.
- Transition focus points on the focus layer appear as dotted green lines.
- Cuts between clips on non-focus layers appear as grey lines.
- Transition focus points on non-focus layers appear as dotted grey lines.

The following examples show the cuts and transitions of a two-layer clip. The first illustration is of the clip displayed in the Player's timeline.



(a) Focus layer (L2) (b) Cut on L1 seen through gap of focus layer (c) Cut on L1 not seen through focus layer (d) Transition on L1 seen through gap of focus layer

The next illustration shows the same clip displayed on the Desktop. The focus is still on the topmost layer, L2. Notice how the cuts and transitions on the timeline in the previous example are colour coded depending on which layer they are on. The only cut not displayed is the one not visible from the focus layer: (c) in previous example.



(a) Cut on focus layer (b) Cut on non-focus layer (L1) (c) Transition focus point on focus layer (d) Transition focus point on non-focus layer (L1) (e) Start and end of transition on focus layer (f) Start and end of transition on non-focus layer (L1)

In addition to the video and layer information on the clip updating as you navigate, the frame count also changes colour. The colour of the frame count on layers lower down from the result layer change to a dimmed version of its original colour. For example, a blue frame count for a mixed resolution layer appears a dimmer blue if the focus is on a layer lower down from the result layer.

EQ

New for this release: There are now a total of six EQ bands, or nodes, available for more precise manipulation of the audio frequencies: one Low node, four Mid nodes, and one High node.

EQ is a soft effect that allows you to perform precise manipulation of the audio frequency content using the EQ Editor, which is based on a graphical display of EQ settings. For example, you can improve noisy audio tracks or enhance vocal tracks.

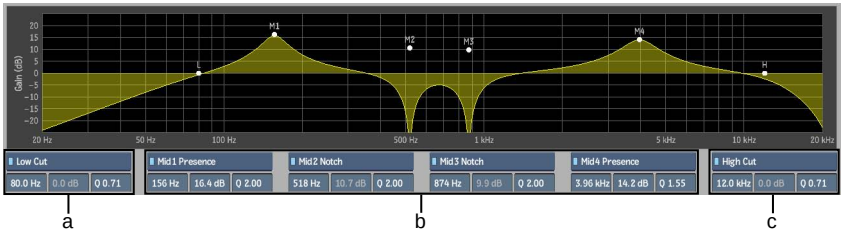
You can use any of six available filter or nodes: one Low node, four Mid nodes, and one High node. The Low node can be set to use either a Low Shelf filter or a Low Cut filter. The four Mid nodes can each be set to either a Mid Notch filter or a Mid Presence filter. The High node can use a High Shelf filter or a High Cut Filter.

These filters can have a dramatic effect on the audio so they should be used sparingly.

To apply EQ effects on a segment:

1. From the Record T/L, select the audio segment that you want to adjust.
2. From the Audio Soft-Effects menu, enable EQ.
3. Click E.

   The EQ Editor appears.



(a) Low Shelf/Cut filter node (b) Mid Notch/Presence filter nodes (c) High Cut/Shelf filter node

4. Enable filters and make adjustments as necessary.

Input/Output Improvements

12

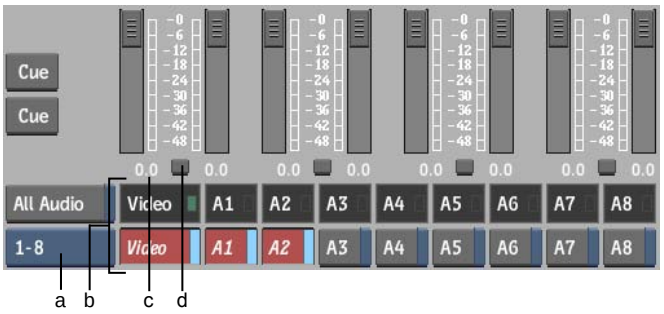
Topics in this chapter:

Adjusting Audio Gain on Output Clip

New for this release: Embedded audio through the AJA video card now supports 16 audio tracks.

Audio gain adjustment is a part of the clip output process only. The clips you are outputting are unaffected by audio gain adjustments made using the Output Clip menu.

On output, you can adjust the audio gain, for example, to restore the levels you had monitored on capture.



(a) Audio Tracks Toggle button (b) Channel Selection buttons and indicators (c) Audio Level fields (d) Fader Lock buttons

All Audio button When enabled, outputs to the audio monitor every audio channels. When disabled, outputs only the enabled audio channels to the audio monitor. The All Audio button has no impact on the audio tracks recorded by the VTR.

Audio Tracks Toggle button Toggles the Channel Selection buttons and indicators between audio tracks 1-8 and 9-16.

Channel Selection buttons and indicators Controls and displays which audio channels are recorded by the VTR. The black boxes with the green LEDs indicate video tracks and audio channels that are part of the clip that you want to output. The red buttons indicate the tracks and channels the VTR records on output.

Audio Level fields Displays the audio gain, in decibels. Adjust using the faders. In the small Output Clip menu, increase or decrease the gain by dragging left or right on the fields. By default, audio gain is 0 db.

Fader Lock buttons When enabled, locks the faders for the corresponding pair of audio channels together.

To adjust the audio gain on output clip:

1. Enable the Fader Lock buttons (so that they are light grey) if you want to apply the identical value to pairs of audio channels.
2. Slide the faders to adjust the audio gain before you start processing. Use the All Audio button to monitor all the audio tracks that are output, regardless of what audio tracks the VTR records.
3. In the Output Clip menu, enable Output All Audio.
4. Select or deselect channels for output by clicking the Channel Selection button for each channel as needed.

NOTE In a multiple clip selection, channel selection is independent for each clip but the gain levels set with the faders are the same for all clips.

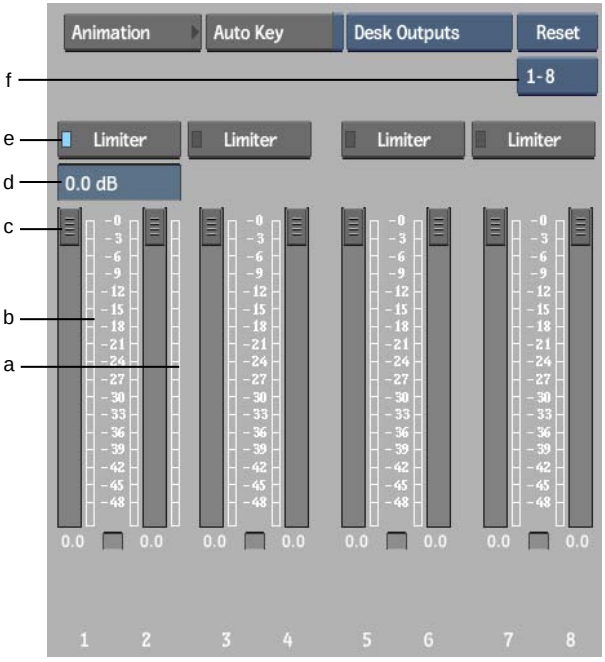
5 Process the clip.

Selected channels are output.

Using Output Strips

New for this release: Embedded audio through the AJA video card now supports 16 audio tracks in the output strips of the AudioDesk menu.

You can use the output strips to control the gain or limit the peaks of the audio output signals.



(a) Limiter meter (b) Output meter (c) Gain Level fader (d) Limiter Level box (e) Limiter button (f) Output Strip Display selection box

Adjusting Output Strip Gain

Use the output strip faders to control the audio output levels. You can adjust the faders while playback is stopped, or during playback to get a dynamic update of audio levels.

To adjust the audio output strip gain:

1. In the AudioDesk, ensure that each audio input strip that you want to work with is assigned to an output strip.
2. Toggle the meters to Desk Outputs.
3. Play the clip.

   The audio output levels are displayed on the meters.
4. Click the fader for the output strip that you want to adjust and drag it to the new level.

   You can also adjust the levels with the playback stopped.

Using the Limiter

The Limiter provides a form of signal compression. It allows audio signals below a set value to pass unaffected, and clips off the peaks of stronger audio signals that exceed the set value. The audio remains untouched unless the limiter is working, in which case only gain is affected. The built-in auto-release mechanism allows for fast recovery, minimizing distortion and pumping. The Limiter is a stereo effect that applies to a pair of output strips.

To set the limiter:

1. In the AudioDesk, ensure that each audio input strip that you want to work with is assigned to an output strip.
2. Toggle the meters to Desk Outputs.
3. Play the clip.
4. Note the audio output levels displayed on the meters.
5. Enable the Limiter button.

   The Limiter meter appears.
6. Adjust the Limiter level to remove any overloads, or to limit the peaks to a desired output level.