Autodesk®
Flint® 2010
A Discreet® Systems product

New Features Guide

Autodesk

© 2009 Autodesk, Inc. All rights reserved. Except as otherwise permitted by Autodesk, Inc., this publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose.

Certain materials included in this publication are reprinted with the permission of the copyright holder.

Autodesk® Inferno®, Autodesk® Flame®, Autodesk® Flint®, Autodesk® Fire®, Autodesk® Smoke®, Autodesk® Backdraft® Conform software

Portions relating to MXF-SDK was developed by Media, Objects and Gadgets – Soluçoes de Software e Hardware, S.A. (http://www.mog-solutions.com) in co-operation with Institut für Rundfunktechnik GmbH (http://www.irt.de).

Portions powered by Automatic Duck. © 2006 Automatic Duck, Inc. All rights reserved.

Portions relating to "dslib" C/C++ Copyright 1988-1989 Eugene Dronek and Rich Morin.

Autodesk® Flare™ software

Portions relating to MXF-SDK was developed by Media, Objects and Gadgets – Soluçoes de Software e Hardware, S.A. (http://www.mog-solutions.com) in co-operation with Institut für Rundfunktechnik GmbH (http://www.irt.de).

Portions powered by Automatic Duck. © 2006 Automatic Duck, Inc. All rights reserved.

Portions relating to "dslib" C/C++ Copyright 1988-1989 Eugene Dronek and Rich Morin.

Portions relating to MPEG Layer- 3; supply of this product does not convey a license under the relevant intellectual property of Thomson multimedia and/or Fraunhofer Gesellschaft nor imply any right to use this product in any finished end user or ready-to-use final product. An independent license for such use is required. For details, please visit http://www.mp3licensing.com.

Autodesk® SystemCentral™ software
Adobe® Flash® Player. Copyright © 1996-2006 Adobe Systems Incorporated. All Rights Reserved.

Autodesk® Inferno®, Autodesk® Flame®, Autodesk® Flint®, Autodesk® Smoke®, Autodesk® Backdraft® Conform

Portions relating to MPEG Layer- 3; supply of this product does not convey a license under the relevant intellectual property of Thomson multimedia and/or Fraunhofer Gesellschaft nor imply any right to use this product in any finished end user or ready-to-use final product. An independent license for such use is required. For details, please visit http://www.mp3licensing.com.

Trademarks

The following are registered trademarks or trademarks of Autodesk, Inc., in the USA and other countries: 3DEC (design/logo), 3December, 3December.com, 3ds Max, ADI, Alias, Alias (swirl design/logo), AliasStudio, Alias|Wavefront (design/logo), ATC, AUGI, AutoCAD, AutoCAD Learning Assistance, AutoCAD LT, AutoCAD Simulator, AutoCAD SQL Extension, AutoCAD SQL Interface, Autodesk, Autodesk Envision, Autodesk Insight, Autodesk Intent, Autodesk Inventor, Autodesk Map, Autodesk MapGuide, Autodesk Streamline, AutoLISP, AutoSnap, AutoSketch, AutoTrack, Backdraft, Built with ObjectARX (logo), Burn, Buzzsaw, CAiCE, Can You Imagine, Character Studio, Cinestream, Civil 3D, Cleaner, Cleaner Central, ClearScale, Colour Warper, Combustion, Communication Specification, Constructware, Content Explorer, Create>what's>Next> (design/logo), Dancing Baby (image), DesignCenter, Design Doctor, Designer's Toolkit, DesignKids, DesignProf, DesignServer, DesignStudio, Design|Studio (design/logo), Design Web Format, Discreet, DWF, DWG, DWG (logo), DWG Extreme, DWG TrueConvert, DWG TrueView, DXF, Ecotect, Exposure, Extending the Design Team, Face Robot, FBX, Filmbox, Fire, Flame, Flint, FMDesktop, Freewheel, Frost, GDX Driver, Gmax, Green Building Studio, Heads-up Design, Heidi, HumanIK, IDEA Server, i-drop, ImageModeler, iMOUT, Incinerator, Inferno, Inventor, Inventor LT, Kaydara, Kaydara (design/logo), Kynapse, Kynogon, LandXplorer, LocationLogic, Lustre, Matchmover, Maya, Mechanical Desktop, Moonbox, MotionBuilder, Movimento, Mudbox, NavisWorks, ObjectARX, ObjectDBX, Open Reality, Opticore, Opticore Opus, PolarSnap, PortfolioWall, Powered with Autodesk Technology, Productstream, ProjectPoint, ProMaterials, RasterDWG, Reactor, RealDWG, Real-time Roto, REALVIZ, Recognize, Render Queue, Retimer,Reveal, Revit, Showcase, ShowMotion, SketchBook, Smoke, Softimage, Softimage|XSI (design/logo), SteeringWheels, Stitcher, Stone, StudioTools, Topobase, Toxik, TrustedDWG, ViewCube, Visual, Visual Construction, Visual Drainage, Visual Landscape, Visual Survey, Visual Toolbox, Visual LISP, Voice Reality, Volo, Vtour, Wire, Wiretap, WiretapCentral, XSI, and XSI (design/logo).

The following are registered trademarks or trademarks of Autodesk Canada Co. in the USA and/or Canada and other countries: Backburner, Multi-Master Editing, River, and Sparks.

The following are registered trademarks or trademarks of Moldflow Corp. in the USA and/or other countries: Moldflow MPA, MPA (design/logo), Moldflow Plastics Advisers, MPI, MPI (design/logo), Moldflow Plastics Insight, MPX, MPX (design/logo), Moldflow Plastics Xpert.

Adobe and Flash are either trademarks or registered trademarks in the United States and/or countries. Automatic Duck and the duck logo are trademarks of Automatic Duck, Inc. FFmpeg is a trademark of Fabrice Bellard, originator of the FFmpeg project. Python is a registered trademark of Python Software Foundation. All other brand names, product names or trademarks belong to their respective holders.

Disclaimer

THIS PUBLICATION AND THE INFORMATION CONTAINED HEREIN IS MADE AVAILABLE BY AUTODESK, INC. "AS IS." AUTODESK, INC. DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE REGARDING THESE MATERIALS.

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

Images Courtesy of: Behavior Communications Inc.

Title: Autodesk Flint 2010 New Features Guide

Document Version: 1

Date: March 9, 2009

Contents

Chapter 1 Introduction 1

About the Documentation 1
Using the New Features Guide 2
Viewing Tooltips 2
Viewing the Help 2
Viewing PDF Documentation 3
Autodesk Media and Entertainment Training 3
Notation Conventions 3
Contacting Customer Support 4

Chapter 2 What's New 5

About This Release 5
Floating Point Workflow 5
Creative Tools 7
Technical Tools 8
Batch Workflow Improvements 9
Application-centric Improvements 11
Interoperability Workflow Improvements 14
Editorial and Timeline Improvements 17
Input/Output Improvements 19

Chapter 3 Modular Keyer 21

About the Modular Keyer 21

Accessing the Modular Keyer 22
Using Nodes 52
Animation in the Modular Keyer 68

Chapter 4 Floating Point Workflow 69

Keyer-Channel Node 69
Keyer-HLS Node 70
Keyer-Luma Node 70
Keyer-RGB Node 72
Keyer-RGBCMYL Node 72
Keyer-YUV Node 73
Colour Picker 74

Chapter 5 Creative Tools 83

Adding a Displacement Map 83
Using UV Mapping 87
Displacing DVE Layer Objects 88

Chapter 6 Action: 3D Text 91

About 3D Text 91
Adding a 3D Text Node 91
Changing 3D Text Properties 92
Changing Character Axis Properties 95
Animating 3D Text 97

Chapter 7 Batch Workflow Improvements 99

Batch Node Bins 99
Creating Custom Nodes 105
Sparks Load Node 108
Saving Sources and Setups 108
Library Node 113
Grouping Nodes 114
Offsetting Clips 118
MUX Node 119
Action Node 120

Chapter 8 Application-centric Improvements 129

Keyframe Indicators 129
Selecting Channels Automatically 130
Rendered Clip Name 132
Controlling Image Display using Exposure and Image Data Type . . . 133
Selecting a Project and User on Start-up 135
Creating User Profiles 136

Playing Clips from the Library 138
Available Libraries 139

Chapter 9 Interoperability Workflow Improvements 143

Importing Final Cut Pro XML 143
Relinking to File-based Media 146
Importing AAF Files 153
Relinking to File-based Media 156
Supported and Unsupported Transitions and Effects 163

Chapter 10 Editorial and Timeline Improvements 173

Navigating Edit Sequences 173
EQ 177

Chapter 11 Input/Output Improvements 179

Adjusting Audio Gain on Output Clip 179
Using Output Strips 181

Introduction 1

Topics in this chapter:

- About the Documentation on page 1
- Using the New Features Guide on page 2
- Viewing Tooltips on page 2
- Viewing the Help on page 2
- Viewing PDF Documentation on page 3
- Autodesk Media and Entertainment Training on page 3
- Notation Conventions on page 3
- Contacting Customer Support on page 4

About the Documentation

Autodesk® Flint® 2010 includes 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/flint-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 Flint. 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 on page 5
- Floating Point Workflow on page 5
- Creative Tools on page 7
- Technical Tools on page 8
- Batch Workflow Improvements on page 9
- Application-centric Improvements on page 11
- Interoperability Workflow Improvements on page 14
- Editorial and Timeline Improvements on page 17
- Input/Output Improvements on page 19

About This Release

This release of Flint 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.

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.

Modular Keyer in Flint

Flint now supports the Modular Keyer node.

See Modular Keyer on page 21.

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 69, Keyer-HLS Node on page 70, Keyer-Luma Node on page 70, Keyer-RGB Node on page 72, Keyer-RGBCMYL Node on page 72, and Keyer-YUV Node on page 73.

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

Colour Management in 3D Histogram

The 3D histogram in the Colour Warper 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 74.

Creative Tools

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

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 83.

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 87.

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 88.

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.
- cascading of animation on individual letters.

See Action: 3D Text on page 91.

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 99 and Creating Custom Nodes on page 105.

For information on the Modular Keyer node bins, see Modular Keyer Node Bins on page 22, Customizing the Modular Keyer Bin on page 23, and Creating Custom Nodes on page 27.

- 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 108.
- 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, Flint setups are saved such that they cannot be overridden by Flare setups.

See Saving Sources and Setups on page 108.

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 113.

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 114.

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 118.

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 119.

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 120.

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 129.

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 130.

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 132.

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 133.

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 Flint 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 135.

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 136.

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.
- 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 138.
- 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 139.

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 143 and Relinking to File-based Media on page 146.

For AAF files, see Importing AAF Files on page 153 and Relinking to File-based Media on page 156.

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 163.

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 Flint).

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 173.

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 177.

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 179 and Using Output Strips on page 181.

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.

Modular Keyer

3

Topics in this chapter:

- About the Modular Keyer on page 21
- Accessing the Modular Keyer on page 22
- Using Nodes on page 52
- Animation in the Modular Keyer on page 68

About the Modular Keyer

New for this release: The Modular Keyer is available on the Desktop, in Action, and in Batch. Use the Modular Keyer to pull a key and to selectively colour correct a clip. The processing pipeline provides an environment in which you can select and arrange the keying tools. You can select from an array of tools to create your key:

- The Keyer lets you key images by extracting a single colour or a range of colours, or by setting the luminance.
- The Master Keyer allows you to refine keys by gesturally modifying the matte, removing colour spill, blending edges, applying patches, and removing grain.
- The 2D histogram to adjust the luminance of the matte.
- Garbage masks, used for revealing or hiding explicit parts of the matte. You can also use the Tracer to key difficult images.

- Colour suppression and edge correction tools.
- Colour correction, including the Colour Warper, a tool for performing localized colour correction.
- Degrain and regrain tools.
- Difference matte.
- Containers for doing multilayered mattes and colour corrections.
- Logic operations to blend the components of matte layers.

Accessing the Modular Keyer

As with the traditional Keyer, you load three clips into the Modular Keyer to create a key: a front clip, a back clip, and a key-in clip. You use the key-in clip, which is usually the same as the front clip, to create the matte for the front clip.

You can access the Modular Keyer from the Desktop, Action, and Batch.

To access the Modular Keyer from the Desktop:

1. In the Main menu, click Effects.
2. In the Effects menu, click Modular Keyer.
3. Select a front, back, and key-in clip. Make sure the front, back, and matte clips are the same resolution. If not, resize them.
4. Select a destination.

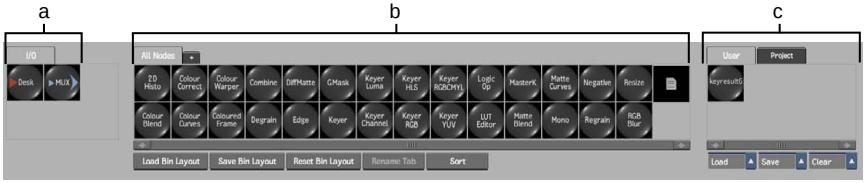
Modular Keyer Node Bins

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 23.
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 105.



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

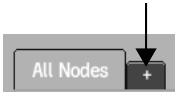
Customizing the Modular Keyer Bin

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.

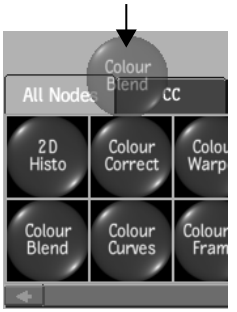


2 Name the tab in the keyboard that appears.

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.

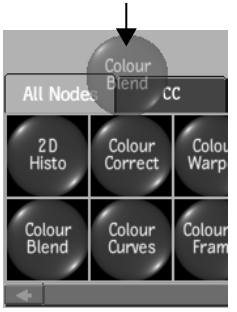
3 Click the destination tab when the standard yellow cursor reappears.

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:

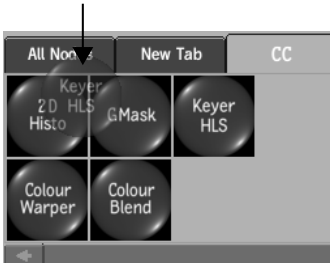
➤ Press **Ctrl+Alt** and drag a node to the schematic.

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

To reorder a node within a bin:

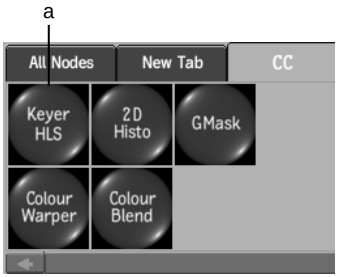
1 Press **Ctrl+Alt** and drag the node to a new location. You can move nodes from one row to another as well as reorganize nodes within a row.

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:

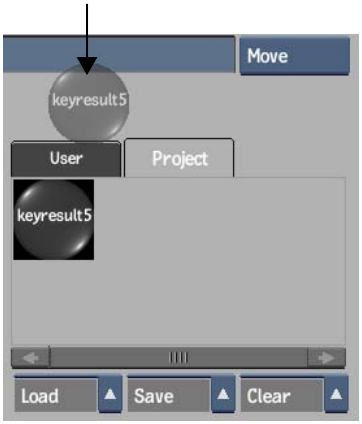
1. In the Selection Mode box, select the part of the process tree that you want to use as a custom node.
2. **Alt**-click a node that is part of your selection, and then drag the selection on top of the User or Project tab. You can also drag the selection directly into the bin if it is the active bin.

   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.

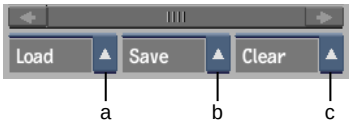
(a) MUX nodes converted from front, back, and matte clips (b) Matte Curves node converted from Result node

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	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.

Select:	To:
the Load dropdown list	
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:

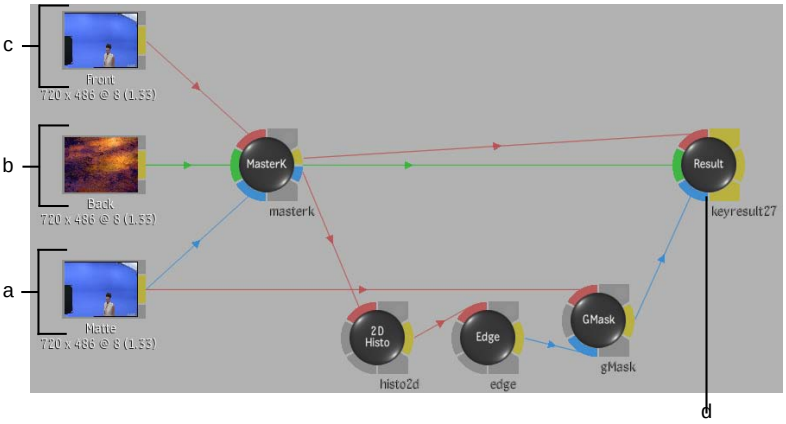
1. Press Ctrl+Alt and drag the node to the bottom of the screen.
2. Release the cursor when it changes to a delete cursor.

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

Setting Up the Processing Pipeline

To build your key, you can use the tools provided on the default processing pipeline as well as add tools. You can also remove tools and set up the pipeline to best suit your needs. Each tool is represented by a process node that you can click to access the node's menu.

The pipeline is made up of three processing branches—one each for the front, back, and matte clips, as shown in the following image.



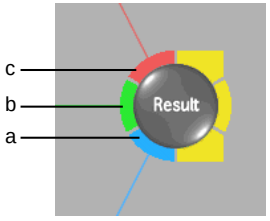
(a) Matte branch (b) Back branch (c) Front branch (d) Result node

Image courtesy of Behavior Communications Inc.

Red is used for the outline of the front clip proxy and branch, green is used for the back clip, and blue is used for the key-in clip. These colours correspond to the colours of the arrow cursor when selecting clips from the Desktop.

The processing pipeline is similar to the process tree of the Batch module, except that its main purpose is to create a key. Unlike Batch, most of the nodes that you use in the processing pipeline have one input and one output. This allows for a more streamlined schematic.

The processes on the pipeline are performed sequentially in the direction of the arrows shown on each branch. The result of each branch is fed into the Front, Back, and Matte tabs of the Result node to create the composite.



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

To process the composite—the output of the Result node—to the destination reel, you click the Process button in the Modular Keyer menu.

The pipeline that you see when you first access the Modular Keyer contains certain nodes. These nodes correspond roughly to the functions of the traditional Keyer:

- On the Matte branch:
  - The MasterK node opens the Master Keyer, where you can pull the key. It is also connected to the Front branch.
  - The 2D Histogram node opens the histogram, where you can adjust the luminance of the matte.
  - The Edge node opens the Edge menu, where you can adjust the edges of the key using shrink, erode, blur, for example.
  - The GMask node opens the Garbage Mask menu, where you can create garbage masks.
- The Result node opens the Matte Curves menu, where you can adjust the front and back mattes.

If the node for the operation you want to perform on a clip is not already on the pipeline, you add it to the appropriate branch. For example, if you want to blur the front clip, add an RGB Blur node to the Front branch.

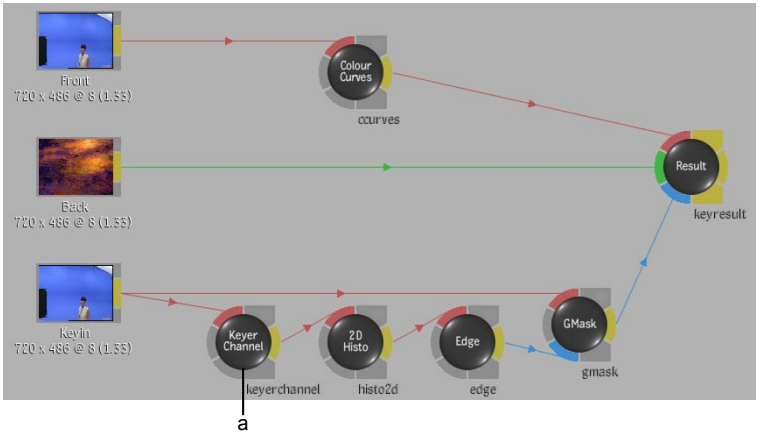
Selecting a Default Processing Pipeline

You can reset the schematic to display a new processing pipeline. You can select to reset the processing pipeline you used when you first opened the Modular Keyer, or you can select one using a different node to pull the key. At the same time, you can change the Keyer selection to replace the default Master Keyer node on the Matte branch with one of the following nodes:

- Keyer-Channel node
- Keyer-HLS node
- Keyer-Luma node
- Keyer-RGB node
- Keyer-RGBCMYL node
- Keyer-YUV node

The default processing pipeline that includes the Master Keyer node does not require a Colour Curves node when it is selected in the start mode, because

the node is set to bypass the Colour Curves node by default. If a different Keyer node is selected, it is connected to the Matte branch only and the Colour Curves node is added to the Front branch. You can use the Colour Curves node to remove colour spill from the front clip, or perform a hue shift, for example.

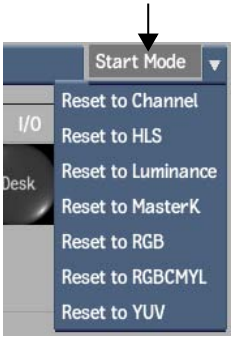


(a) Keyer-Channel node used to pull the key in default processing pipeline

Image courtesy of Behavior Communications Inc.

To change the default processing pipeline:

1 Click the Start Mode dropdown list and select the type of keyer you want to include in the default pipeline.



2 Click Confirm to restore settings and display a new pipeline.

Displaying Menus

Menus are built into the schematic, so you need only click a node to access its menu. The only exceptions are the Gmask (garbage mask) and Colour Corrector nodes. After clicking the GMask or Colour Corrector node, you click another button to open the module. These modules are not integrated into the Modular Keyer in the same way as the other modules are—you cannot access the pipeline or the Modular Keyer menu options during an edit session.

To access a menu for a node:

1 Select Move from the Edit Mode box.

2 Click the node.

The node has a yellow border, and the menu appears in the lower part of the screen.

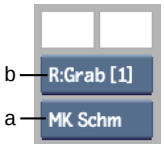
Modular Keyer Menu

The Modular Keyer menu provides the commands you need to:

- Set up the processing pipeline the way you want.
- Change the image displayed in the image window.
- Play and process the clip.
- Save the key setup.

Viewing Clips in the Image Window

Use **Esc** to switch from the schematic (pipeline) to the image window, and vice versa. You can use the view options, or the equivalent hotkeys, to change the image that appears in the image window.



(a) View box (b) Reference View box

Select a view from the View box to display one of the following views in the image window.

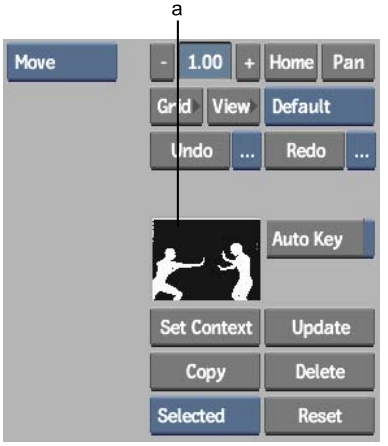
NOTE Hotkeys are shown in parentheses.

Select:	To view:
Front (**F1**)	The front clip of the selected node.
Back (**F2**)	The back clip of the selected node.
Matte (**F3**)	The matte of the selected node.
CurResult (**F4**)	The result of the selected node.
Grab	The reference image created when clicking Grab in the View menu of the Modular Keyer.
MK Schematic (**Esc**)	The processing pipeline. The **Esc** hotkey toggles between the schematic (the pipeline) and the last selected view.
Result	The resulting output of the entire processing pipeline.
Action Result	The Action context point (available only when accessing the Modular Keyer from Action).
Channels	The Channel Editor.
Context 1 (**1**)	The context of context point 1. See Setting a Context Point on page 47.
Context 2 (**2**)	The context of context point 2. See Setting a Context Point on page 47.

Select a view from the Reference View box to display one of the following alternate views in the image window. There are no hotkeys for the reference views.

Select:	To view as a reference:
R:Result	The result of the changes made in the selected node.
R:Context1	Context Point 1. See Setting a Context Point on page 47.
R:Context2	Context Point 2. See Setting a Context Point on page 47.
R:Back	The current back clip of the selected node.
R:Grab [1]	The reference image created when you click Grab in the View menu of the Modular Keyer.
R:Front	The current front clip of the selected node.
R:Matte	The current matte clip of the selected node.

The Current Result proxy shows the Current Result view (see CurResult in the first table). It is updated as you select different nodes on the pipeline or make changes in a menu. You can also click it to update it.

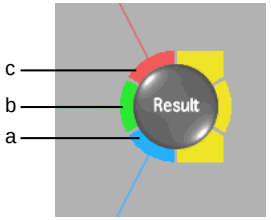


(a) Current Result proxy

About Nodes

You can use single input and multiple input nodes on the processing pipeline.

An input node has two to four coloured tabs used to connect inputs and outputs. The colour tabs on the node's left are called *source tabs*. The source tab colours correspond to the cursor colours when selecting clips from the Desktop. The yellow tab on the node's right is the *result*, or *output tab*.



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

Tab	Colour	Purpose
Front	Red	Use to connect a source for the front clip of the node.
Back	Green	Use to connect a source for the back clip of the node.
Matte	Blue	Use to connect a source for the matte clip of the node.
	Grey	Unused tab.
Result	Yellow	Use to connect the result of the node to one or more other nodes.

The tabs that are available depend on the node. If the node accepts a front, back, and matte clip, all source tabs are available. If the node only accepts a front and back clip, the red and green source tabs are available and the blue source tab is grey. The result tab is always available, except on the Result node, which is the last node in the pipeline.

When source tabs are not connected to anything (clips or other nodes), the tabs are slightly greyed.

Working with Nodes

This section describes how to perform basic operations to set up your pipeline, such as adding nodes to the pipeline and moving them.

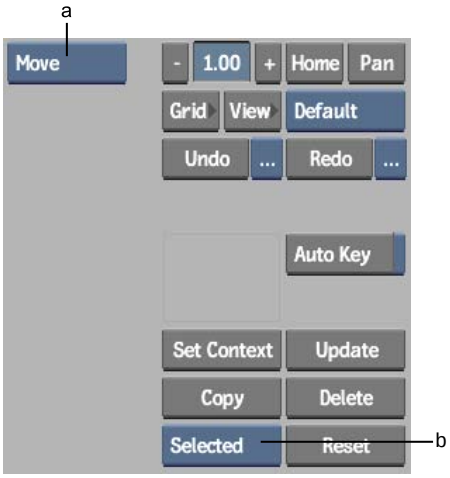
For information on individual nodes and where on the pipeline they should be used, see Using Nodes on page 52.

Selecting Nodes on the Pipeline

You can select individual nodes, branches, all nodes in the pipeline, or all nodes in the schematic. You can also use the **Ctrl** key to select a combination of nodes.

To select nodes in the schematic:

1 Select Move from the Edit Mode box.



(a) Edit Mode box (b) Selection box

2 Select the node(s) as follows.

To select:	Do this:
One node	Select Selected from the Selection box and click the node.
A branch	Select Branch from the Selection box and click anywhere on the branch.
All nodes on pipeline	Select Graph from the Selection box and click anywhere on the pipeline.
All nodes in schematic	Select All from the Selection box and click any node.

To select:	Do this:
A combination of nodes	Press **Ctrl** and drag a rectangle around the nodes. The current option in the Selection box does not affect this type of selection.

Selected nodes have a white border.

NOTE A yellow border indicates both that a node is selected and that its menu is the active menu.

Adding Nodes to the Pipeline

You can add nodes to the pipeline to use while building your key. For example, you can place several garbage mask nodes on the pipeline.

Most nodes can be placed anywhere in the pipeline. There are some nodes that are recommended for use on specific branches because of the way branches operate.

To add a node to branches:

1. Scroll the node bar to display the node you want by clicking on any node and dragging to the left or right.
2. Click the node and drag the cursor to the schematic.

   The cursor changes to an arrow-headed cross.
3. Release the cursor.

   The node appears in the schematic.
4. Drag the node closer to the location on the branch where you want to add it.
5. Select Parent from the Edit Mode box.
6. Drag the cursor across the branch where you want to add the node.

   The portion of the branch between the previous and next items disappears.
7. Drag the cursor from the front of the previous clip or node (or its Result tab) to the back of the new node.

   If the node has source tabs, attach the branch to the front, back, or matte tab, depending on the type of node. See Using Nodes on page 52.

NOTE You can also use the output (result image) of any node on the pipeline as an input source for the new node.

The branch is connected to the new node.

8 Drag the cursor from the result tab of the new node to the back of the next node (or to the appropriate source tab—see note, above).

Moving Nodes

You can move a node from one part of the pipeline to another. For example, you can move a garbage mask to the beginning of a branch on the pipeline.

To move a node:

1 Select Move from the Edit Mode box.

2 Detach the node from the pipeline. Select Parent from the Edit Mode box and drag the cursor across the branches that connect it to the pipeline. Reconnect the branch without the node, then return to Move mode.

3 Drag the node over the pipeline where you want to add it and release the cursor. See Adding Nodes to the Pipeline on page 39.

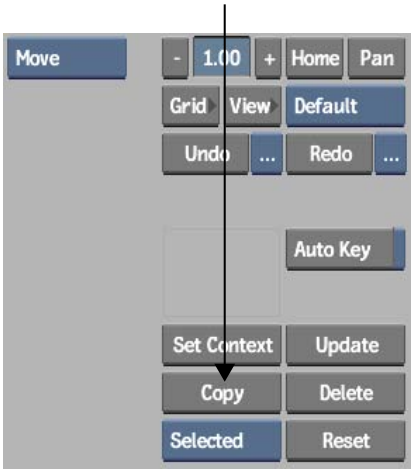
Copying Nodes

You can copy one or more nodes and place the copies elsewhere on the schematic. All the settings of a copied node are included in the copy.

To copy one or more nodes:

1 Select the nodes. See Selecting Nodes on the Pipeline on page 38.

2 Click Copy.



Copies of the nodes appear on the schematic. The copies are selected (shown by the yellow border).

3 Drag the copies over the pipeline where you want to add them and release the cursor. See Adding Nodes to the Pipeline on page 39.

Deleting Nodes

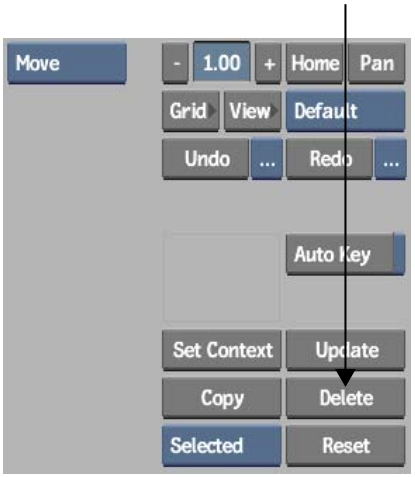
When you delete a node from the pipeline, all the associated information, including any unsaved work, is also deleted. There are several methods you can use to delete nodes:

- Select the nodes and then delete them.
- Switch to Delete mode and then click the nodes you want to delete.
- Drag the nodes off the Desktop.

To select and delete one or more nodes:

1 Select the nodes. See Selecting Nodes on the Pipeline on page 38.

2 Click Delete.



3 Click Confirm to delete the nodes, or click elsewhere to cancel.

To use Delete mode:

1 Select Delete from the Edit Mode box.

2 Click the node you want to delete.

3 Click Confirm to delete the node, or click elsewhere to cancel.

TIP To override Confirm, press **Alt** as you click the node.

4 Repeat steps 2-3 for any other nodes you want to delete.

5 Return to Move mode.

To delete by dragging:

1 From the Edit Mode box, select Move.

2 From the Selection box, select Current, Branch, Graph, or All.

3 In the schematic, click a node and drag either to the top or the bottom of the Desktop.

The cursor changes to a green recycling icon.

4 Click Confirm to delete the nodes, or click elsewhere to cancel.

Updating Nodes

All the nodes in the pipeline are not updated or processed when a change is made to the pipeline. This is to save time if processing is not needed. For example, when you make a change in one node, that node and all previous nodes in the branch are updated immediately, but not the subsequent nodes.

You can opt to have nodes updated on an “as needed” basis, or update them manually:

- With automatic update, when you click a node, that node and all nodes before it on the pipeline are updated since the previous nodes are required to produce the correct result for the node. For example, if you click the Result node, all nodes in the pipeline are updated.
- With manual update, you selectively update nodes.

To set the update mode:

1. Display the Setup menu.
2. Use the Auto Update button to set the update mode.

For:	Do this:
Automatic update	Enable the Auto Update button.
Manual update	Disable the Auto Update button.

NOTE Auto Update only affects the schematic (it does not affect processing while you are working within nodes).

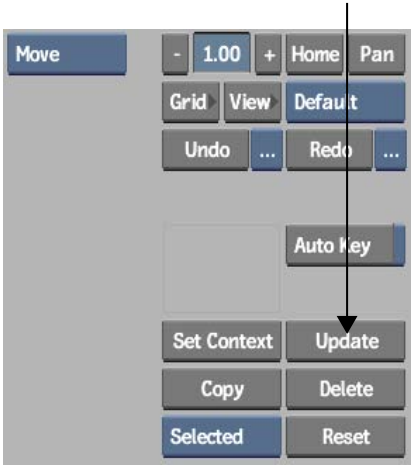
To update a node automatically:

1. Select Move from the Edit Mode box.
2. Click the node.

NOTE If the node does not get updated, make sure that all multiple input nodes in branches leading to the node have the required source inputs.

To update a node manually:

1. Select the node. See Selecting Nodes on the Pipeline on page 38.
2. Click Update.



The selected node and all previous nodes are updated.

NOTE If the node does not get updated, make sure that all multiple input nodes in branches leading to the node have the required source inputs.

Renaming Nodes and Clips

Nodes are given a default name composed of the node name and a number appended to it, for example, gmask1, gmask2. When you select a node or clip, its name is displayed in the Name field.

(a) Name field

Names are also displayed in the schematic, below the clip or node (with the exception of the nodes on blend pipes). Rename nodes and clips to help keep track of them as the pipeline grows.

NOTE You cannot rename clips that are brought into the schematic using the Add Clip button.

To rename a node or clip:

1 Select the node or clip.

2 Click the Name field.

The on-screen keyboard appears.

3 Click Esc to delete the previous name and type a new name.

4 Click Enter to save the new name or click EXIT Keyboard to cancel.

Adding Clips to the Pipeline

You can add clips to the pipeline to use with nodes that require extra clips. For example, when you use the Difference Matte node, you need two clips.

You can also add new clips to use in place of the Front, Back, or Key-In clips you selected when you opened the Modular Keyer. You may want to replace an existing clip temporarily to see the result, or process the final composite with the new clip.

NOTE When you access the Modular Keyer from Action, Action uses the original Front, Back, and Key-In clips. If you want to replace these clips, you must return to Action and replace them using the Layers menu.

To add a clip to the schematic:

1 In the node bar, click the Desk node and drag it to the Modular Keyer schematic.

You are brought back to the Desktop.

2 Select the clip you want to bring into the schematic by clicking its upper-left corner.

You are returned to the Modular Keyer and the clip appears in the schematic.

To attach the new clip to a node:

1 Select Move from the Edit Mode box.

2 Move the clip to the left of the node you want to attach it to.

3 Select Parent from the Edit Mode box.

4 Drag the cursor from the right edge of the new clip to the appropriate source tab of the node. If it is a single input node, drag the cursor to the left side of the node.

The clip is connected to the node.

NOTE For more information on using particular types of nodes, see Using Nodes on page 52.

To replace an existing clip:

1. Select Move from the Edit Mode box.
2. Move the clip near the existing clip.
3. Select Parent from the Edit Mode box.
4. Detach the old clip from the pipeline by dragging the cursor across the part of the pipeline between the old clip and the rest of the pipeline.
5. Connect the new clip to the pipeline by dragging the cursor from its right edge to the appropriate source tab of the first node on the branch.

To replace a clip that was added to the pipeline:

1. Double-click the clip.

   You are brought back to the Desktop.
2. Select another clip.

   It replaces the previous clip in the pipeline.

TIP Do not leave unused clips in the schematic: each time you process or play the clip, the Modular Keyer reads all clips off the framestore, even if they are not a part of the pipeline.

Clip Controls

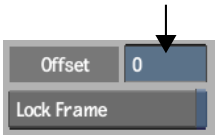
You can make a clip begin at a certain frame by using the slip function. This applies only to clips loaded into the Modular Keyer via the Desk node. When you link a clip loaded via the Desk node into the Modular Keyer pipeline, it retains its clip controls.

To slip a clip in the Modular Keyer:

1. From the Modular Keyer schematic, click the Desk node.

   The Slip field and Lock button appear in the Modular Keyer menu. If the node bar appears, swipe the bar at the left or right of the Modular Keyer menu.

2 Change the value in the Offset field to slip the starting frame of the clip.



For example, a slip value of -15 holds the clip at the first frame and repeats it 15 times before the clip begins. A slip value of 10 begins the clip at frame 11, since a slip value of zero corresponds to frame 1. Slip information is stored with Modular Keyer setups.

3 To lock the currently displayed frame, enable Lock.

Setting a Context Point

As you work with different nodes to create your key, it is often useful to see the effect of your changes on the image at a later part of the pipeline. You do this using context points. You set a context point on a particular node, then view that context point from the node you are working in. For example, you could place a Degrain node at the beginning of the Matte branch and set a context point on the MasterK node. You could then degrain the clip while viewing the context point, in this case, the Current Result of the Master Keyer.

NOTE To view the context point in this example, you must have pulled a key with the MasterK Keyer.

You can set two context points in the pipeline.

To set context points:

1 Press and hold = and click the node.

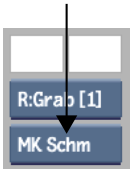
A green dotted border appears around the node and (C1) appears below the label.

2 To set the second context point, repeat step 1 on another node. The second context point has the same green border and (C2) appears below the label.

When you next set a context point, the context point that was first set disappears. The (C2) context point is changed to (C1) and the new one becomes (C2). The oldest set context point is always Context Point 1, and the newest one is always Context Point 2. The oldest context point is always the one to get removed when you add a new one.

To view a context point:

1. Set a context point, as described in the previous procedure.
2. Select the node you want to work with.
3. Select Context1 or Context2 from the View box, according to the context point you want to see. Alternatively, press **1** for Context Point 1 or **2** for Context Point 2.



As you make changes, you see the effect the changes are having on the selected context point.

Comparing Colours on Different Images

Use the colour pots in the Modular Keyer menu to compare colours from two different images, or from the same image as it appears in different parts of the pipeline. To do this, you pick the colours from each image and apply them to the two colour pots.

You can also apply a colour that you copied to a colour pot to a colour pot in one of the node menus. For example, apply the colour to the Suppress Colour pot in the Colour Curves menu. For more information on using the colour picker, see Colour Picker on page 74.

To apply a colour to a colour pot:

1. Display the image with the colour you want to sample.
2. Click the colour pot.
   The colour picker appears.
3. Click Pick and then click in the image on the colour.
4. Click in the colour pot again.

To copy a colour to a colour pot in another menu:

1. Open the node menu containing the colour pot you want to copy a colour to.
2. Click in the colour pot in the node menu.

   The colour picker appears.
3. Click Pick, then click in the Modular Keyer colour pot containing the sampled colour.
4. Click in the node menu colour pot again.

Panning the Schematic

When setting up the pipeline, you may need more space than the visible screen provides. You can pan the schematic to access more space for the pipeline.

To pan the schematic:

1. Click Pan, or press and hold the **spacebar**.
2. Drag anywhere in the schematic.

   Notice that the schematic extends “underneath” the menu area.

Cropping the Key Area

You can create a crop box to limit the area where the key setup is applied.

Using a crop box as you create the key speeds up interactivity, which is especially useful when working at high resolutions. You can also process the composite with a crop box. By setting the crop box to surround just the foreground subject, you can save processing time. Everything outside the crop box will be processed as black on the matte.

The crop box is applied to the Front and Matte branches, but not to the Back branch. With the exception of the GMask node, you can see the crop box as you work in different nodes. When you open the Garbage Mask module, you no longer see a defined crop box. However, once you return to the Modular Keyer, you see the crop box, and can use it as before, regardless of any defined garbage masks.

To use the crop box:

1. Press **Esc** to display the image window.
2. Click Setup.

   The Setup menu appears.

3. Click Crop to enable the crop box.
4. Click Draw.
5. Draw the crop box on the image by selecting a point on the image and dragging the cursor diagonally. Drag the crop box handles to adjust the size and aspect ratio of the box.

   NOTE As an alternative to using the Draw button, you can press **Shift+Alt** and draw the crop box. The advantage of using this hotkey is that you can redraw the crop box when any other menu is displayed. The Crop button must be enabled to use either of these methods.

6. You can now go into any node menu and create the key. You will only see the results of your changes within the crop box.
7. If you want to remove the crop box when you have finished creating the key, go back to the Setup menu and disable the Crop button.

   If you want to process the clip with the crop box, simply leave the Crop button enabled when you process.

Backing Up the Key Setup

Enable the Backup button in the Setup menu to automatically back up your key setup. When this button is enabled, a backup copy of current key settings is saved in the *_session.Bak* file in the modularKeyer directory.

You can set the amount of time between backups using the Backup Frequency field next to the Backup button. The Backup Frequency is in minutes.

Saving the Key Setup

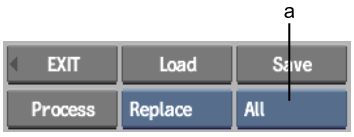
You can save a key setup that includes the settings in all nodes used to create the key. You can save part of a setup and reload it, or load and save nodes separately. To do this, enable All or Selected (proxies are not saved when using the selection options). You can also load setups using the Replace and Append options.

For Modular Keyer setups that contain RGB Blur and Edge nodes, width and height values are scaled according to the clip resolution of the current session.

To save the key setup:

1 Click Setup in the Modular Keyer menu.

The Setup menu appears.



(a) Save option box

2 From the Save option box, select All or Selected.

3 Click Save.

TIP You can go directly to the Load Setup browser by holding **Alt** when you drag a node to the Desktop.

Saving Individual Nodes

You can save and reload certain individual nodes, as well as view menus for them. This applies to Colour Warper, Colour Correct, LUT Editor, Regrain, Degrain, and, as in the following example, GMask nodes.

To save a node:

1 Click NodeSetup in the Modular Keyer menu.

The Node Setup menu appears.

2 Click Save Node.

Using Nodes

This section describes the nodes you can use to enhance your key, with the following exceptions.

To learn about:	Refer to:
The Keyer	Keyer.
Floating-point keying nodes	Keyer-Channel Node on page 69, Keyer-HLS Node on page 70, Keyer-Luma Node on page 70, Keyer-RGB Node on page 72, Keyer-RGBCMYL Node on page 72, and Keyer-YUV Node on page 73.
Garbage masks	Garbage Masks.

Before reading this section, you should be familiar with the basic node operations described in Working with Nodes on page 37.

2D Histogram Node

Use the 2D histogram node to display the luminance of the matte.

To display the matte with the 2D histogram:

1 Select Move from the Edit Mode box.

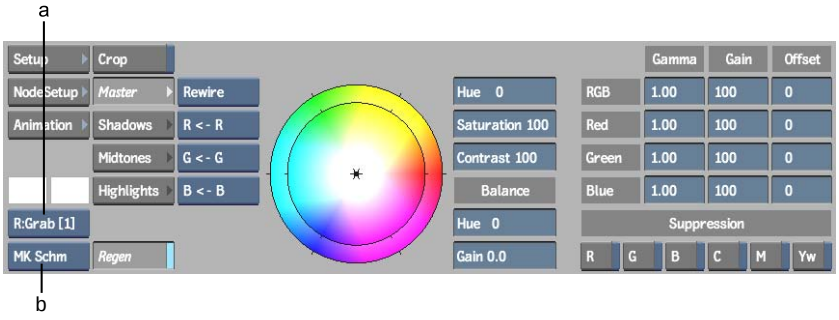
2 Select the 2D histogram node.
The histogram appears.

3 Press **Esc** to view the image.
The histogram is identical to the histogram in the traditional Keyer.

Colour Correct Node

Use this node to apply colour corrections to the front, back, or key-in clip using the Colour Corrector.

When using the Colour Corrector from the Modular Keyer, select a view from the Reference box and View box, while subsequently using the reference clip to compare your result.



(a) Reference box (b) View box

The View box includes views for Context Point 1 and Context Point 2 while colour correcting the clip. See Setting a Context Point on page 47.

To use the Colour Correct node:

1. Add a Colour Correct node to the pipeline:
   - If you are colour correcting an entire image, attach a single input to the Front tab.
   - If you are colour correcting a portion of the image, attach the front image to the Front tab and the matte to the Matte tab.
   - If you are colour correcting a composite, attach the front, matte, and back images to the Front, Matte, and Back tabs. This allows you to colour correct the front image only (while viewing the whole composite).
2. Click the Colour Correct node.

   The Colour Corrector menu appears in the menu area.

Colour Curves Node

Use the Colour Curves node to adjust the colour in the front, back, or key-in clip. For example, increase or decrease a colour in an image by plotting the colour and adjusting the curve. When you move the first or last keyframe, the other keyframes move by the same amount so that curves are continued from beginning to end.

The Colour Curves menu is identical to the Colour menu in the traditional Keyer.

Colour Warper Node

The Colour Warper node accepts a front, back, and key-in clip as input. Use the Colour Warper node to access the Colour Warper module for colour correcting clips and refining mattes.

Difference Matte Node

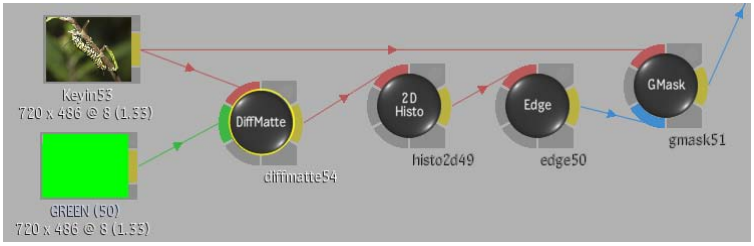
Use the Difference Matte node to create a matte based on the differences between two clips. You can generate a matte from two clips with the same

background but different foreground elements. For example, the first clip could be the blue or green screen shot you want to key, and the second clip could be a “clean plate”—a shot of the blue or green screen with no foreground subject.

This node functions in the same way as the Difference command in the Processing menu.

To use the Difference Matte node:

1. If needed, add the clip you want to use as the back clip to the schematic. See Adding Clips to the Pipeline on page 45.
2. Add a Difference Matte node to the schematic. See Adding Nodes to the Pipeline on page 39.
3. Attach a source to the Front tab and a source to the Back tab of the node.
4. Attach the output of the node to the pipeline at the appropriate position. For example, you could set up the node as follows.



5. Click the node to view the Difference Matte menu.

Degrain and Regrain Nodes

You can use the Degrain node to remove grain from any clip in the pipeline. For example, degrain the KeyIn clip by using a Degrain node at the beginning of the Matte branch to make it easier to extract the matte. Similarly, add the Regrain node to the pipeline to add grain to an image. Keep in mind the following when working with Degrain and Regrain nodes:

- You can add the Degrain node to any part of the processing pipeline.
- You can go directly to the Load Setup browser by holding **Alt** when you drag a node to the Desktop.

The Degrain and Regrain menus in the Modular Keyer contain the same set of controls as the Degrain and Regrain menus accessed from the Processing menu on the Desktop (only the menu layout is slightly different).

Edge Node

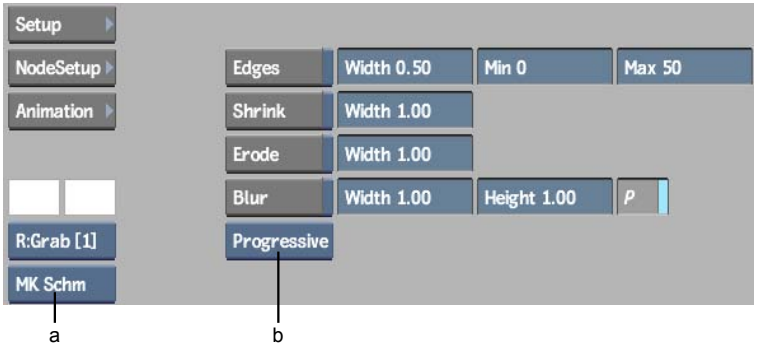
The Edge menu provides controls to:

- Create a greyscale image composed of the edges in an image. You can apply edge-detection to both colour and monochrome clips.
- Modify the edges of the key. You can apply the Erode, Shrink, and Blur filters to the edges of the matte.

Using Field Mode

Video clips with fast-moving objects can contain artifacts characterized by edge misalignments. These artifacts occur when the odd and even fields are combined into frames. Applying Erode, Blur, or Shrink effects to images containing misalignments does not produce good results. To solve this problem, use Field mode. In Field mode, frames are separated into fields and the effects are applied to the individual fields. The fields are then recombined into frames. This is all done "behind the scenes"—that is, fields are not displayed in the image window.

There are four matte controls in the Edge menu you can use to enhance the edges of the key—Edges, Shrink, Erode, and Blur.



(a) View box (b) Scan Format box

These controls are processed sequentially: Edges is processed first, then Shrink, Erode, and lastly, Blur.

Use:	To:
Edges	Create an image composed of the edges in an image. The resulting greyscale image may be used as a matte or to produce special effects. You can apply the Edges tool to a colour or monochrome clip. This tool has many applications; among the most useful are: ■ To delimit a portion of the matte to perform colour corrections on the front clip. ■ To create special effects by using it directly on the front or back clip.
Shrink	Remove pixels from the edge of the matte. It should not be used when the object in the front clip has soft edges such as hair. A negative value expands the matte.
Erode	Blend the light and dark edges of the matte.
Blur	Apply a softening Gaussian blur filter to the edge of the matte.

To transform an image into edges:

1. Select CurResult view from the View box.
2. Enable Edges.
3. Set a value for the edge-detection sensitivity in the Width field to the right of the Edge button. This control affects how the edge-detection algorithm determines whether or not each pixel forms part of the edge.
4. Set the Minimum Input Level in the Min field.

   The Minimum Input Level sets the start of the range of luminance values in the image. Pixels with luminance values below the Minimum Input Level are mapped to black (0).
5. Set the Maximum Input Level in the Max field.

   The Maximum Input Level sets the end of the range of luminance values in the image. Pixels with luminance values greater than the Maximum Input Level are mapped to white (255 in 8-bit mode and 4095 in 12-bit mode).
6. Enable Shrink.

7 Set the width of the edge using the Shrink Width field. Enter a negative Shrink value to expand the border, and therefore widen the edge. Enter a positive Shrink value to decrease the width of the edge.

The edge before adjusting the width using the Shrink Width field

The edge after adjusting the width using the Shrink Width field

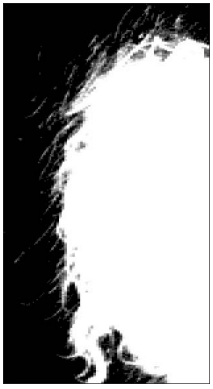
To shrink the edge of the matte:

1 Select CurResult view from the View box.

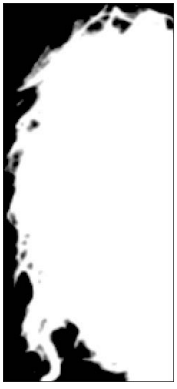
2 Enable Shrink.

3 Set a value in the Width field next to the Shrink button.

This value specifies the width of the border, in number of pixels, that is removed from the edge of the matte.



The matte before applying the Shrink filter



The matte after applying a shrink width value of 1.00

To erode the edge of the matte:

1. Select CurResult view from the View box.
2. Enable Erode.
3. Set a value in the Erode Width field.

   This value specifies the width of the matte border, in number of pixels, that will be softened.

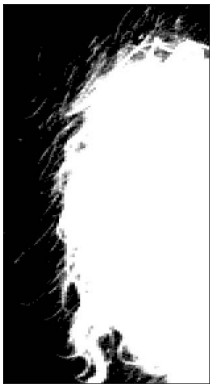
The matte before applying the erode filter

The matte after setting the erode width value to 1.00

To blur the edge of the matte:

1. Select CurResult view from the View box.
2. Enable Blur.
3. If you want the image to be equally blurred horizontally and vertically, click P to enable a proportional blur.

4 Set the width and height of the blur. When P is enabled, enter a value in either field. The higher the number, the greater the blur.



The matte before applying a blur

The matte after setting the blur width and heigh values to 1.00

Logic Op Node

Use the Logic Op node to generate a clip whose luma (brightness) values are calculated according to the luma values of two source clips. For example, you can add, subtract, or multiply the luma values of corresponding pixels on two clips to create the result clip.

To use the Logic Op node:

1 If needed, add a second clip to the schematic. See Adding Clips to the Pipeline on page 45.

2 Add a Logic Op node to the schematic. See Adding Nodes to the Pipeline on page 39.

3 Attach a source to the Front tab and a source to the Back tab of the node.

4 Attach the output of the node to the pipeline at the appropriate position.

5 Click the node to view the Logic Op menu.

LUT Editor Node

A LUT converts logarithmic images to linear images, or linear images to logarithmic images, while maintaining colour accuracy.

When you import or export clips, default LUT parameters are used and the direction of the conversion is detected automatically. Use the LUT Editor to modify these default settings interactively or to import an existing LUT. You can also export a LUT from the LUT Editor.

Master Keyer Node

Use the Master K node to access the Master Keyer in order to create a complete key by first extracting colours to generate a matte, and then refining the result. The MasterK node can be added anywhere in the Modular Keyer pipeline.

NOTE The Master Keyer is also available through the Range menu in the Keyer.

The Modular Keyer allows you to bypass the MasterK node by designating either its front, back, matte, or key-in clip input as the result.

To bypass a MasterK node:

1. In the Modular Keyer schematic, select a MasterK node.
2. In the Bypass box, select the clip you want to pass as the input to the next node.

The MasterK node's operations are bypassed. The clip selected in the Bypass box passes to the next node(s).

Matte Curves Node

The Matte Curves node displays the same menu that is displayed when you select the Result node—the Matte Curves menu. Use this menu to adjust the luminance of the front and back mattes:

- Use the menu from the Result node to adjust the front and back matte curves for the final composite.
- Use the menu from the Matte Curves node to adjust the matte curves at other parts of the pipeline.

See Result Node on page 63 for instructions on using the Matte Curves menu.

Monochrome Node

The Monochrome node generates a monochrome copy of the front clip. Use the Channel box to select the monochrome channel for the clip.

MUX Node

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.

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.

Negative Node

The Negative node generates the negative of a source image. The colour values of each pixel in the source image are inverted to produce the negative image. In the Modular Keyer, this is mainly used to invert the matte.

You can add a Negative node to any part of the processing pipeline.

Resize Node

The Resize node changes a clip's resolution, frame depth, and aspect ratio. You can then select and animate the portion of the source clip that appears as the destination clip to create the result image.

You can input a back clip with a different resolution from the front and matte clip to the Modular Keyer node in Batch. However, when you enter the Modular Keyer editor, the Result node does not process a full composite for mixed resolution input. Add a Resize node to change the destination resolution so all input has the same resolution.

Result Node

The Result node is the last node in the pipeline—you cannot connect other nodes to its Output tab. Use the Result node to:

- Adjust the luminance curves of the front and back mattes.
- Process the composite.

When you click the Result node, the Matte Curves menu appears. Use this menu to adjust the luminance curves and select processing options.

Adjusting the Front and Matte Luminance Curves

When you create a matte for the front clip, the Modular Keyer automatically creates a matte for the back clip to specify which part of the back clip is used for the composite. By default, the back matte is the inverse of the front matte.

You can adjust the luminance of the front matte and back matte separately in the Matte Curves menu. For example, increase the luminance of the back matte so that more of the back clip shows through at the edges of the key. This creates a better blend at the edges.

To learn how to set the matte curves to create an additive key, see Creating an Additive Key for CG Images on page 65.

The following calculation is applied to each pixel of the image to create the composite. The calculation is applied in three passes, one each for the R, G, and B values of the front and back images, and the pixel is given the resulting R, G, and B values.

Result = F * FrontLUT + B * BackLUT

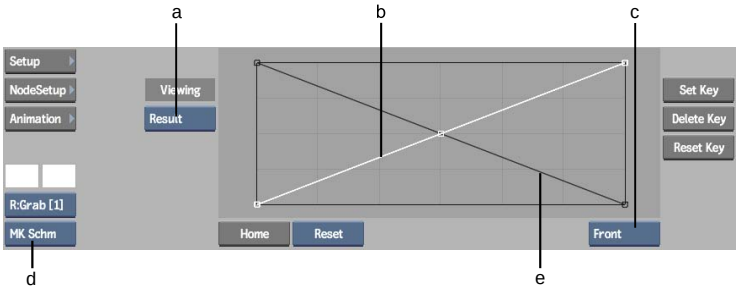
where:

- F = the R, G, and B values of the front image
- B = the R, G, and B values of the back image
- FrontLUT is the front matte pixel value, re-mapped according to any luminance curve change made in the Matte Curves menu. The value is expressed as a decimal, where, for example:
  - in 8-bit mode, 0 = 0, 127.5 = 0.5, and 255 = 1
  - in 12-bit mode, 0 = 0, 2047.5 = 0.5, and 4095 = 1
- BackLUT is the back matte pixel value, re-mapped according to any luminance curve change made in the Matte Curves menu. The value is expressed as a decimal, as is the FrontLUT.

To adjust the luminance curves:

1. In Move mode, click the Result node in the pipeline. Alternatively, if you added a Matte Curves node elsewhere on the pipeline, click the Matte Curves node.

   The Matte Curves menu appears.



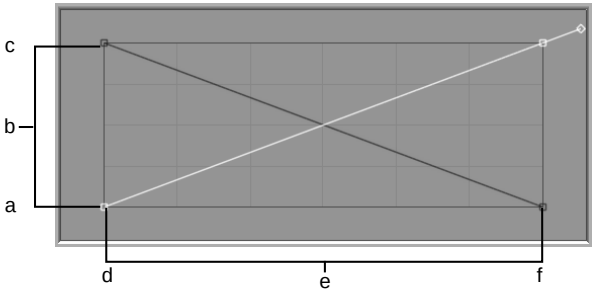
(a) Result box (b) Front matte curve (c) Matte box (d) View box (e) Back matte curve

2. Select Result view from the View box. This allows you to use the Result box to view a particular image as you adjust the curve.

3 From the Result box, select the image you want to view as you adjust the curves.

Select:	To view:
Result	The composite clip.
Matte	The front matte. You cannot see changes made to the back matte when this view is selected.
Bmatte	The back matte. You cannot see changes made to the front matte when this view is selected.
Comp	The composite with a coloured background. The default colour is white. To select a different colour, click the colour swatch below the Edit Mode box. The colour picker appears.

4 To adjust the luminance curve for the front matte, select Front from the Matte box. To adjust the back matte curve, select Back. Alternatively, click a curve to select it.



(a) 255 (White) (b) Output (remapping of luminance values) (c) 0 (Black) (d) 0 (Black) (e) Input (current luminance values) (f) 255 (White)

In Move edit mode, click a point to display its tangent handle and drag the handle to adjust the curve. Use other modes in the Edit Mode box (Add, Delete, or Break, for example) to further adjust the curve, adding or deleting points, or breaking tangent handles as needed.

Creating an Additive Key for CG Images

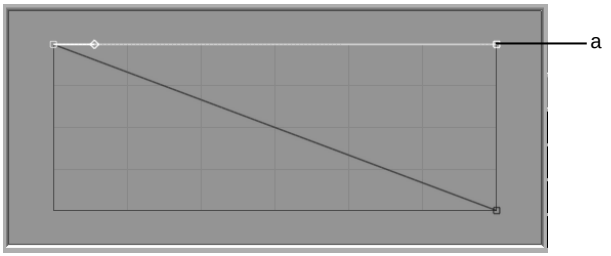
Computer graphics clips with a pre-multiplied alpha channel in which the area to be keyed out is black, should only use the back matte. This is because the front curve calculation has already been applied to the front image.

To use only the back matte, re-map the entire front matte curve to white (255). This causes the curves calculation to ignore the front matte and use the entire front image. The curves calculation maps all the black areas of the front clip as transparent.

To create an additive key, adjust the front matte curve using the Matte Curves menu in the Result node.

To create an additive key:

1. Pull the key and perfect it.
2. Select the Result node.
   The Matte Curves menu appears.
3. Select the front matte curve from the Matte box.
4. Drag the vertex at the beginning of the curve from the lower-left corner of the graph to the upper-left corner of the graph.



(a) Front matte curve

All pixels in the front matte are mapped to white (255 in 8-bit mode, 4095 in 12-bit mode). The black pixels on the front clip are calculated as transparent.

NOTE This is the equivalent of selecting Punch Back Only in the Setup menu of the traditional Keyer.

Processing Options

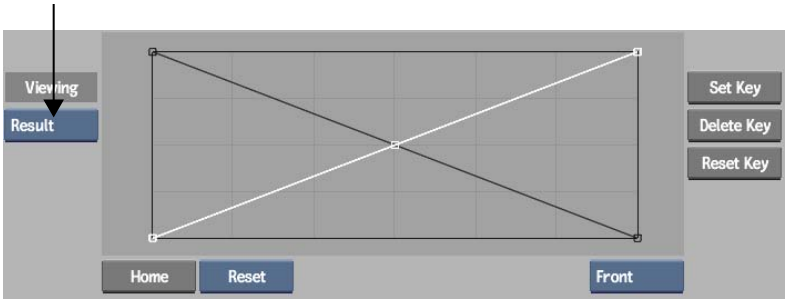
You can process the composite clip, the front matte, the back matte, or the composite using a coloured background instead of the back clip.

When using the Modular Keyer node in Batch, the Result node only outputs a full composite when all of the input is the same resolution. Otherwise, the

Result node will discard information for the back clip and process a composite with a coloured background instead.

To process the composite:

1. Display the pipeline and select the Result node to display the Matte Curves menu.
2. Select the type of clip you want to process in the Result box.



Select:	To process:
Result	The composite clip.
Matte	The front matte.
Bmatte	The back matte.
Comp	The composite with a colour background. The default colour is white. To select a different colour, click the colour swatch to the left of the Result box. The colour picker appears.

NOTE When accessing the Modular Keyer from Batch, the Result option can only be selected when the front, back, and matte input for the Result node have the same resolution.

3. Click Process in the Modular Keyer menu.

NOTE When accessing the Modular Keyer from Action, the Process button is not available; you must return to Action and process it there.

RGB Blur Node

Use the RGB Blur node to apply a Gaussian blur to colour images. To blur a matte, use the Blur filter in the Edge node. Both the RGB and Edge node blurs are Gaussian.

You can animate a blur using the Channel Editor.

To blur a colour image:

1. Place the RGB Blur node at the appropriate point in the pipeline. See Adding Nodes to the Pipeline on page 39.
2. In Move mode, click the RGB Blur node to display the RGB Blur menu.
3. If you want the image to be equally blurred horizontally and vertically, click Proportional to enable a proportional blur.

4. Set the width and height of the blur. When Proportional is enabled, enter a value in either field. The higher the number, the greater the blur.

NOTE Increasing the blur increases the processing time.

Animation in the Modular Keyer

In the Modular Keyer, you typically create animations for garbage masks, key elements, and blue screen colour:

- Follow a moving element that you want to remove from the matte using a garbage mask.
- To animate the tolerance, softness, and patch ranges in a clip, animate a key.
- To animate saturation use the animation controls in the Colour Corrector.

Floating Point Workflow

4

Topics in this chapter:

- Keyer-Channel Node on page 69
- Keyer-HLS Node on page 70
- Keyer-Luma Node on page 70
- Keyer-RGB Node on page 72
- Keyer-RGBCMYL Node on page 72
- Keyer-YUV Node on page 73
- Colour Picker on page 74

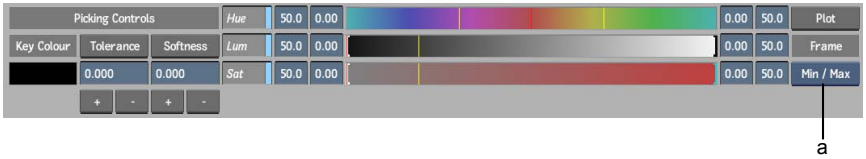
Keyer-Channel Node

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

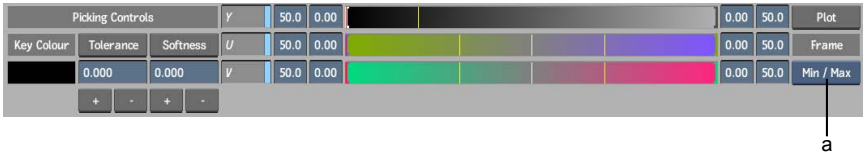
The Picking Controls are the same as those found in the Keyer module's RGBCMYL menu. Set softness and tolerance ranges using the red, green, blue, cyan, magenta, and yellow channels.

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.

Colour Picker

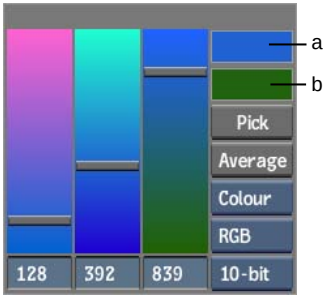
New for this release: The colour picker now supports 16-bit floating point colours.

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.



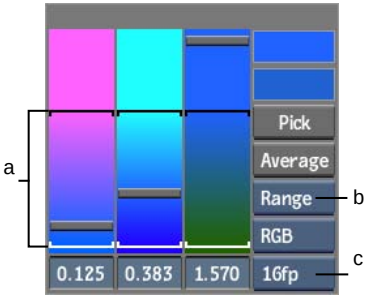
(a) Current Colour pot (b) Reference Colour pot

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.

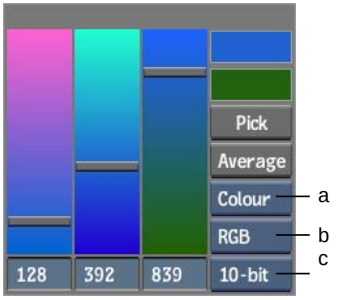
The Mode box, Colour Model box, and Bit Depth box settings, and channel value units (percentages or bit-values) are saved on a per-user basis, at the end of each session.

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.

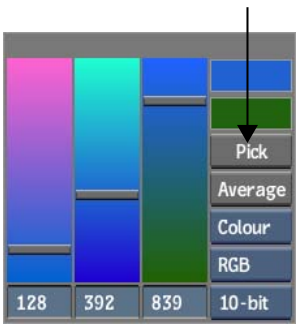
NOTE Applying a LUT or changing exposure/contrast affects the display, but colour picking is done using the original values of the media. Disabling LUTs and resetting exposure/contrast values will show the actual pixel values.

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.

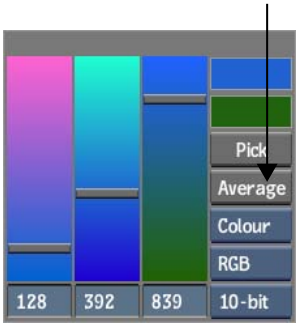
TIP From any colour pot, **Shift**-click to enter Pick mode. Clicking while dragging the Pick icon over an image will display its RGB values. Click again to transfer the colour to the colour pot.

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

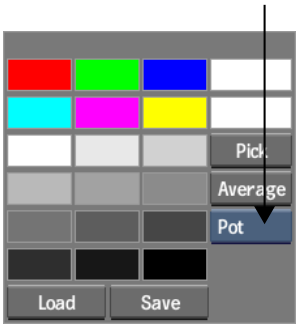
Select from 1 of 18 preset colours in the colour pots. You can also store custom colours in the colour pots.

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

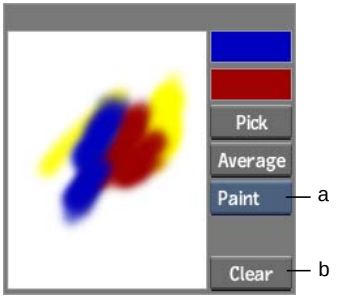
You can use the colour picker to mix colours on a palette.

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:

- Adding a Displacement Map on page 83
- Using UV Mapping on page 87
- Displacing DVE Layer Objects on page 88

Adding a Displacement Map

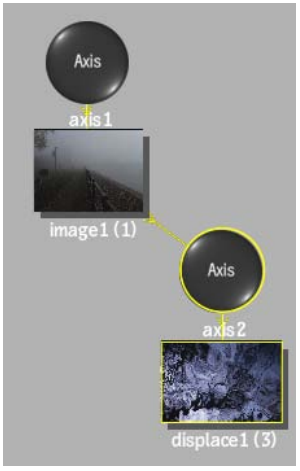
New for this release: In Action, Displacement mapping is removed from the Surface menu, and becomes an object in the Node bin, improving the accessibility of displacement maps for surfaces and 3D geometries. The previous method of using displacement maps in the Surface menu has been removed. Any setups 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 88.

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.

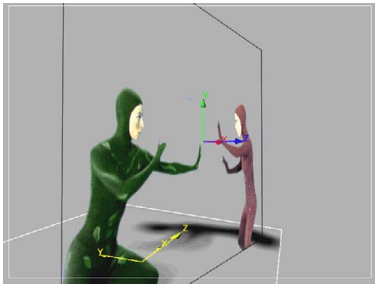
Displacement axes Specifies the amount of displacement in pixel units along the X, Y, and Z axes. Use positive values for displacement on the positive axis, and negative values for displacement on the negative axis.

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

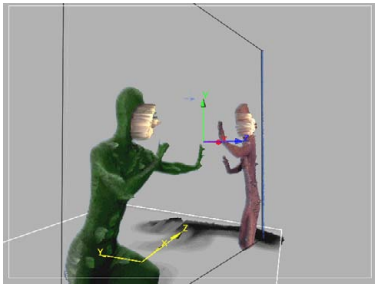
Fill to Surface button Enable to resize the displacement map to the resolution of its parent surface.

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 87.

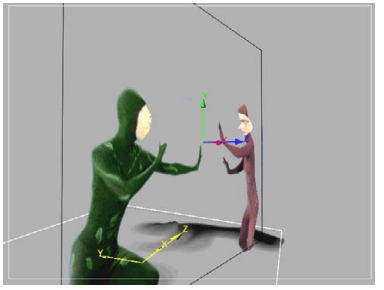
The following figure illustrates a possible use of displacement mapping and shows the difference between displacing with and without softness.



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.

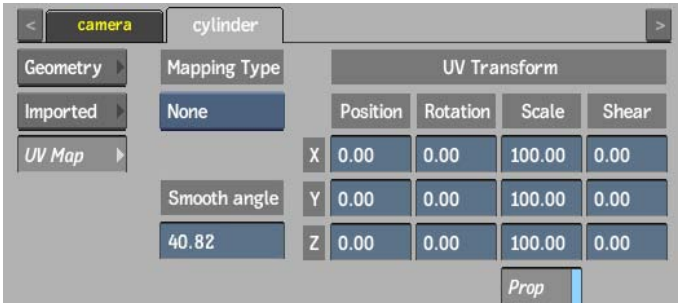
4 In the Media menu, click Apply.

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 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 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 or Texture node.

When a Displace node is attached to a geometry, a UV mapping type other than None is needed for the displace 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.

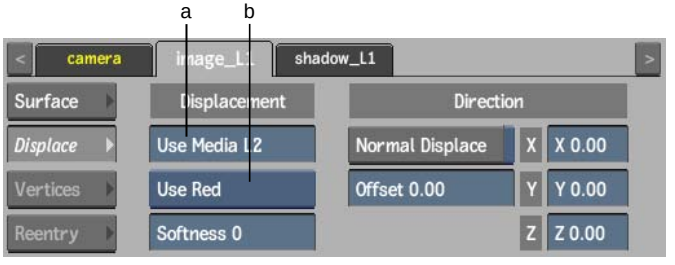
Prop Scale button Scales the X, Y, and Z UV axes proportionally.

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.

Action: 3D Text

6

Topics in this chapter:

- About 3D Text on page 91
- Adding a 3D Text Node on page 91
- Changing 3D Text Properties on page 92
- Changing Character Axis Properties on page 95
- Animating 3D Text on page 97

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 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

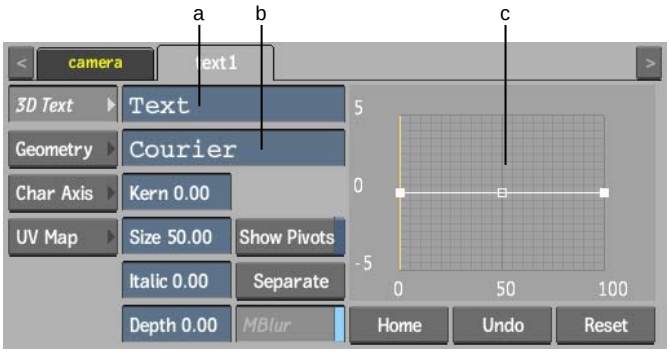
When you add a 3D Text node to your scene, the default text string "Text" appears. You can easily change this text string.

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).

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 94.

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 94.

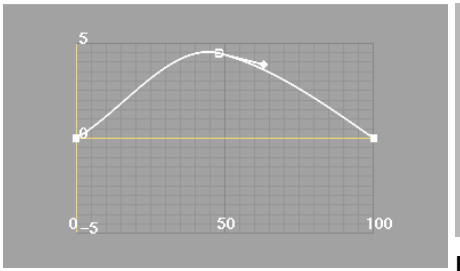
Home button Resets the Bevel curve viewer to show the whole curve.

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

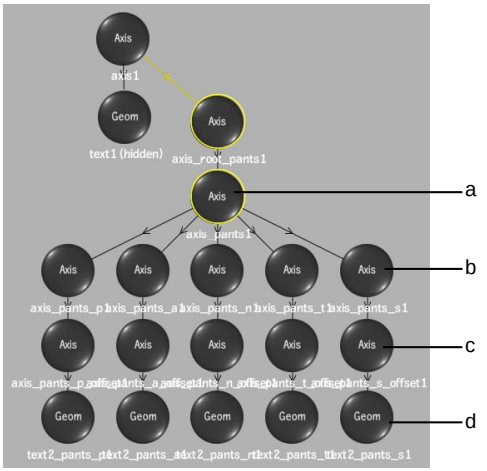
Separating Text

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 92.
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.



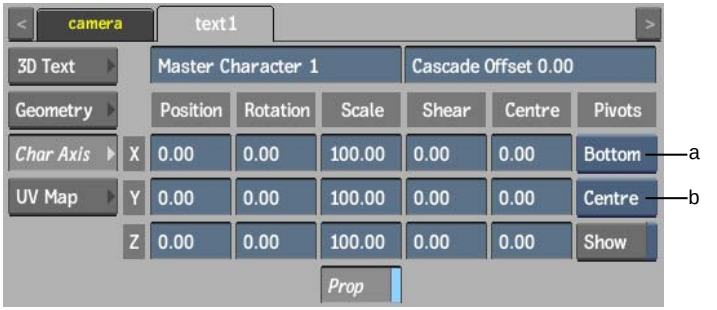
(a) Root axis for word (b) Root axis for letter (c) Offset per letter (d) Letter geometry

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.



(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.

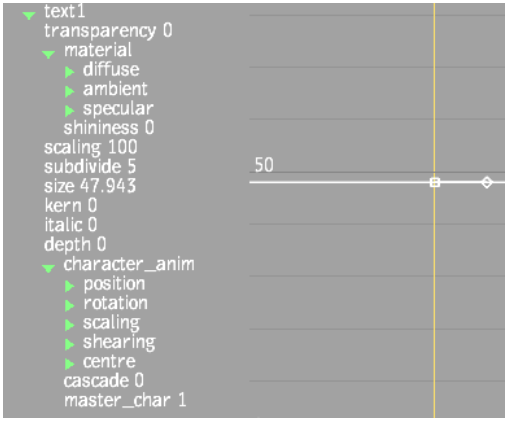
Horizontal Pivot box Select the horizontal position of the pivot point for the selected text characters.

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.



Batch Workflow Improvements

7

Topics in this chapter:

- Batch Node Bins on page 99
- Creating Custom Nodes on page 105
- Sparks Load Node on page 108
- Saving Sources and Setups on page 108
- Library Node on page 113
- Grouping Nodes on page 114
- Offsetting Clips on page 118
- MUX Node on page 119
- Action Node on page 120

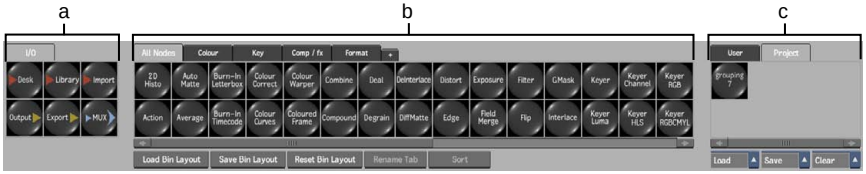
Batch Node Bins

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 105.

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:

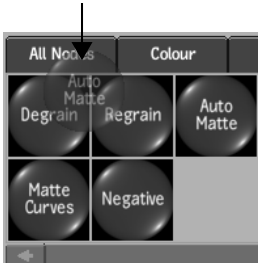
➤ Press **Ctrl+Alt** and drag a node to the schematic.

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

To reorder a node within a bin:

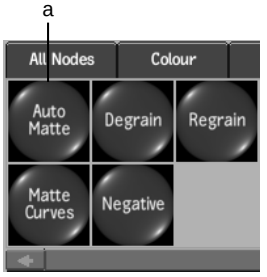
1 Press **Ctrl+Alt** and drag the node to a new location. You can move nodes from one row to another as well as reorganize nodes within a row.

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:

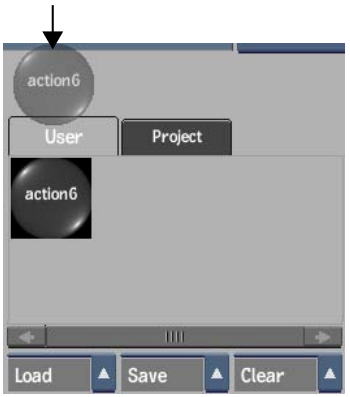
1. In the Selection Mode box, select the part of the process tree that you want to use as a custom node.
2. **Alt**-click a node that is part of your selection, and then drag the selection on top of the User or Project tab. You can also drag the selection directly into the bin if it is the active bin.

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

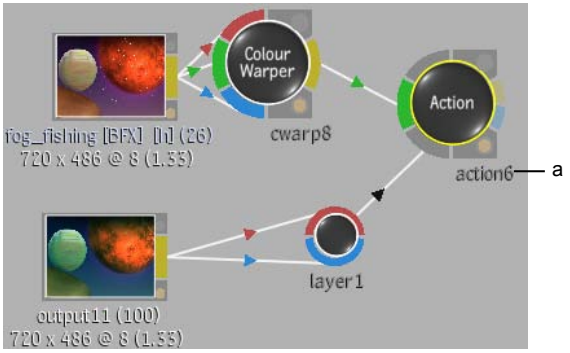
NOTE No two custom nodes can share the same name. Attempting to drop a node into a custom bin with a similarly named node is not possible.

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.

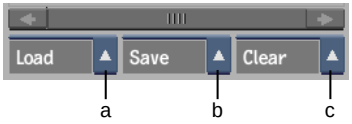


(a) Branch used to create custom node

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.

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 Flint not being able to find source clips associated with a setup.

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

- A Flint 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.

Flint setups are saved in a Flint subdirectory as follows:

~/batch/flint/<my_setup_name>

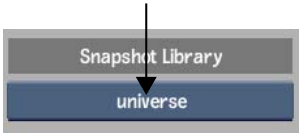
Flare setups are saved to a Flare subdirectory as follows:

~/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 Flint 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 Flint 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 Flint 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 Flint 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.

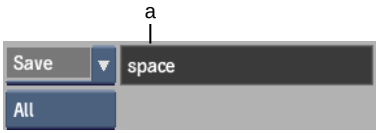
Save Setup As & Snapshot Saves the setup with a new name, which you enter in the keyboard that appears, and takes a snapshot of the sources

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.

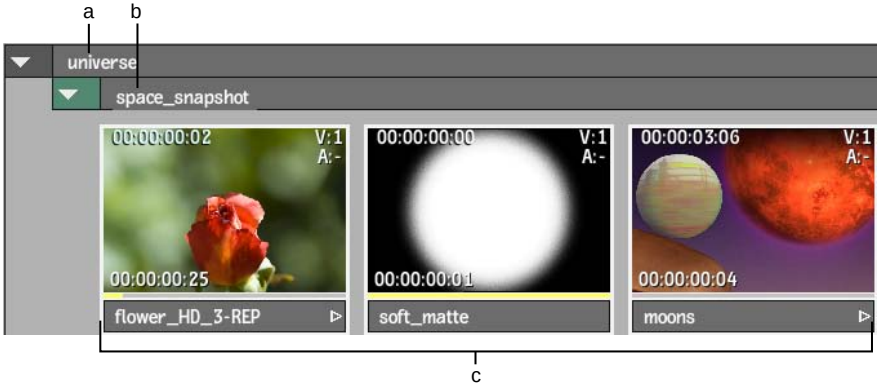
With this option, you can change or create a new Snapshot library directly from the Save & Snapshot browser without having to go back to the Setup menu.

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.

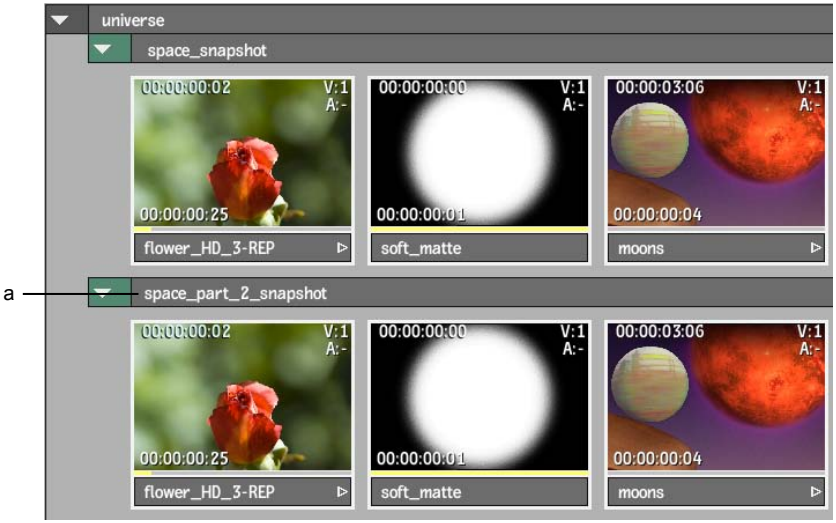


(a) Snapshot library specified in Batch Setup menu (b) Reel entry named according to Batch setup name (c) Source clips saved with Save Setup & Snapshot operation

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.

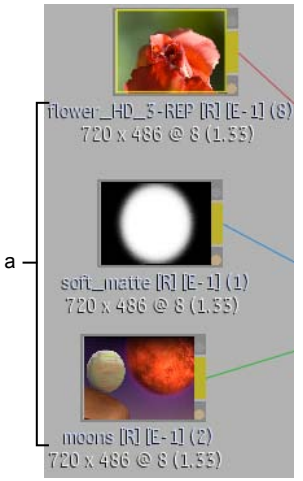
NOTE To overwrite the previous snapshot (and not create a new reel), select Save Setup & Snapshot.



(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.

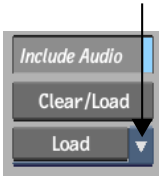
   The clip library appears.

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.

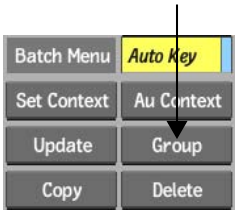
You can group nodes to simplify cluttered schematics. A group is represented by a single dynamic node that displays the group's contents, inputs, and outputs. You can use a group node as a parent to, or child of, other clips and nodes.

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:

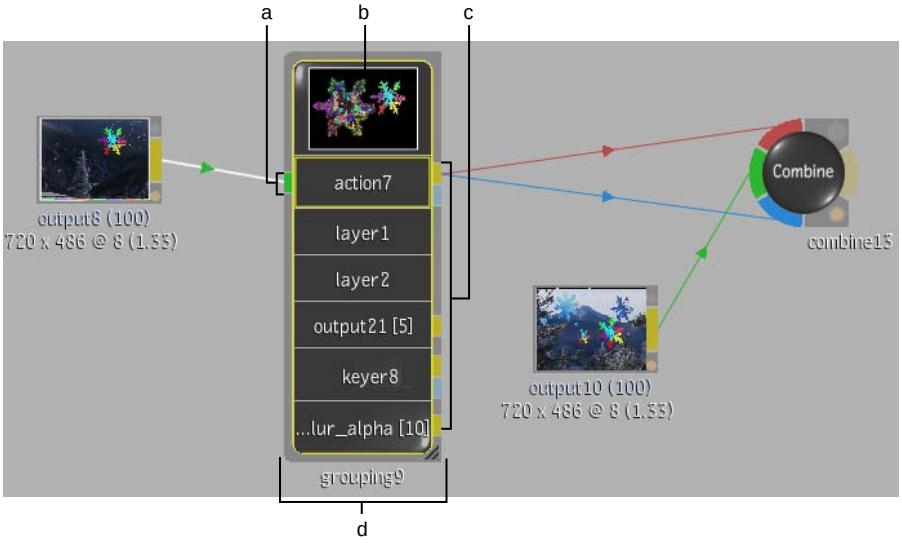
1 **Ctrl**-drag to select the nodes you want to group.
Selected nodes are outlined in white.

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

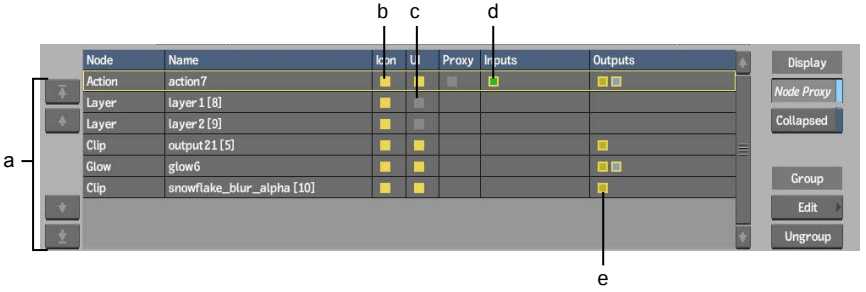
Customize the group node using the Group List menu. The Group List menu lists all the nodes and connections contained in a group. You can rename and hide the contents of the group, as well as define which input and output connection sockets are visible and available for connection in the schematic.

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.

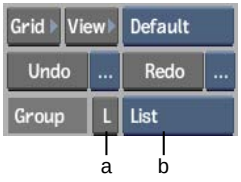
Editing a Group

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.

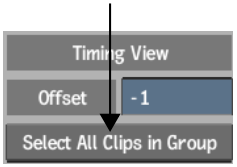
Adjust the timing of clips by offsetting them in the Timing View. Timing View displays the timing of all clips in the current Batch schematic.

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.

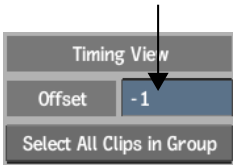
You can display or hide negative frames as you offset a clip. You can also offset clips brought into Batch to their record timecode.

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.



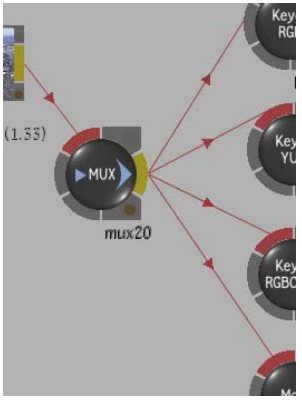
All selections are offset by the same amount.

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

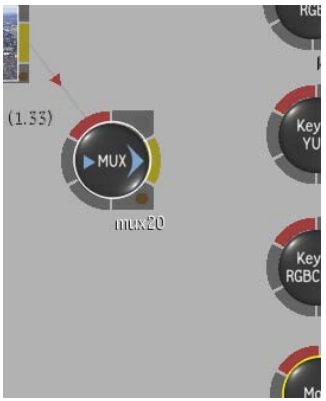
MUX Node

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.

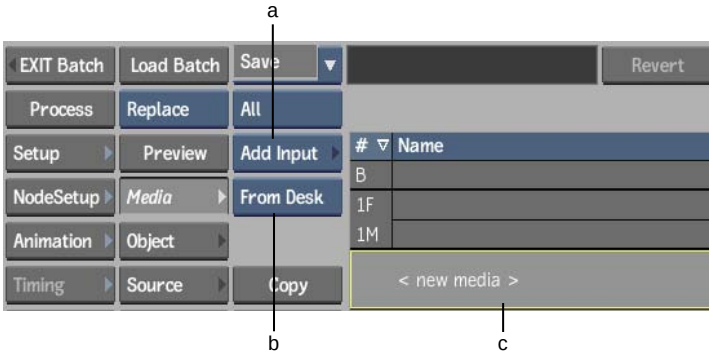
Setting up indirect media is a good way to use pre-processes (a precomposite or the result from multiple nodes) that you want to input into media.

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.



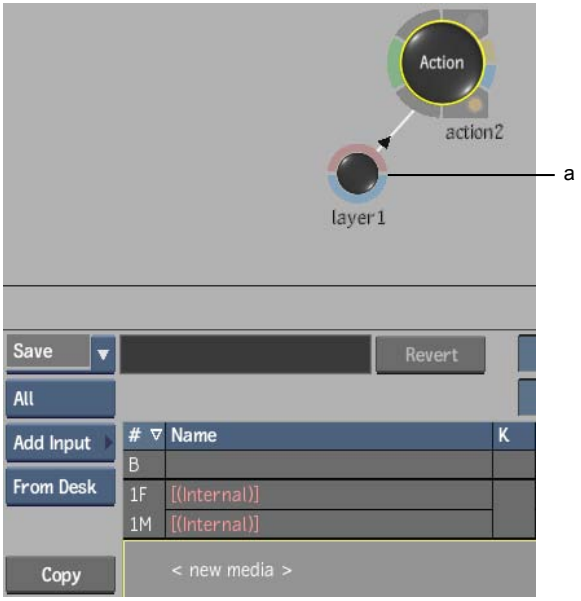
(a) Media List box (b) Media From box (c) New media line

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.

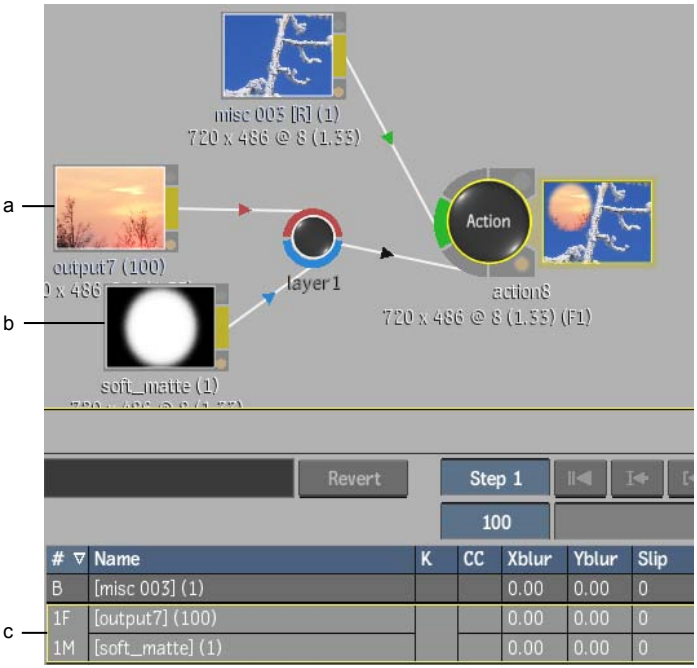
The Media node is added to the Action node.



(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

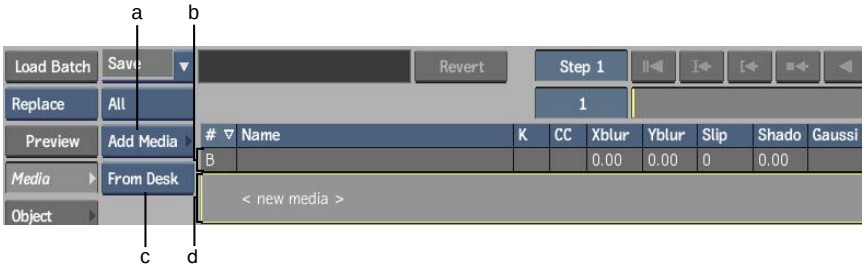
To add direct media to an Action node, you use the Media list.

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:

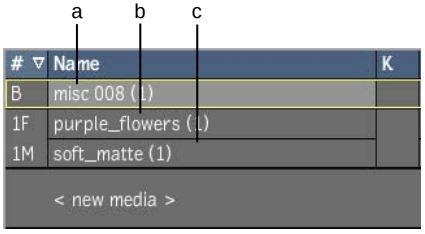
- To add back media, double-click the Back media line, or select the Back media line and then select Add Media from the Media List box.
- To add front and matte media, double-click the New media line, or select the New media line and then select Add Media from the Media List box.

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.

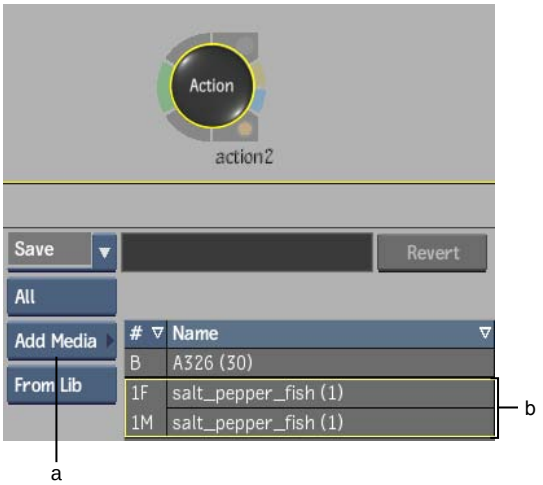
The media is added to the Media list and appears in the Action node.



(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.

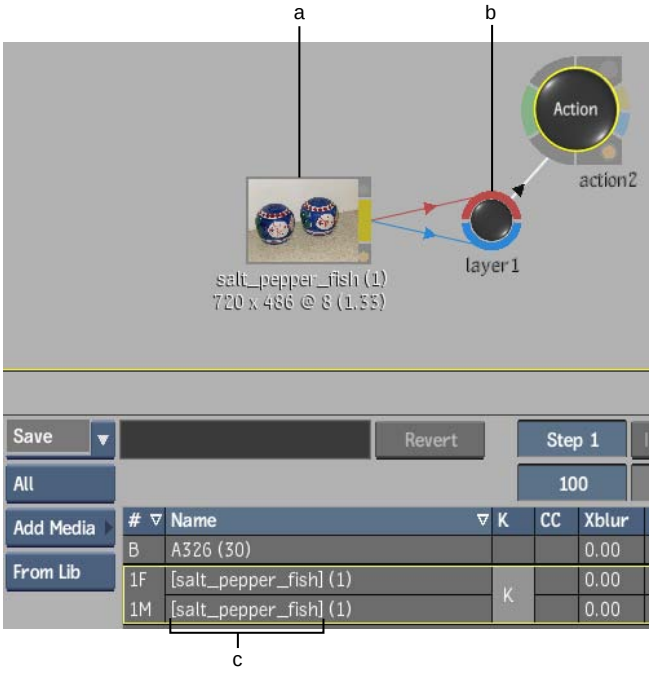


(a) Media List box (b) Selected front and matte media

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

8

Topics in this chapter:

- Keyframe Indicators on page 129
- Selecting Channels Automatically on page 130
- Rendered Clip Name on page 132
- Controlling Image Display using Exposure and Image Data Type on page 133
- Selecting a Project and User on Start-up on page 135
- Creating User Profiles on page 136
- Playing Clips from the Library on page 138
- Available Libraries on page 139

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 130.

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 Flint.

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.

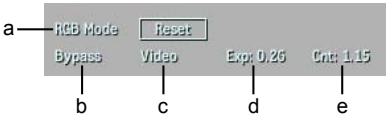
Apply All button In modules with multiple viewports, enable to apply the transformation for the current viewport to all viewports using the same Preset mode.

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.

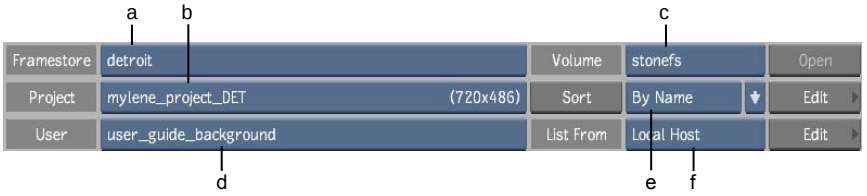
When you start Flint, the Project Management menu appears. Use the Project Management menu to select a project and user for the current session, to create projects and users, or to manage existing projects and users.

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 Flint 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 Flint, 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.

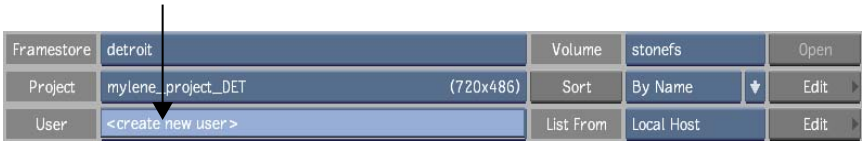
Create a user profile to manage your preferences. When you create a user, you have the option of copying preferences from an existing user. If the user whose preferences you want to copy was created on the same version of the application, you can copy all preferences. If the user was created on an older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

Users do not persist when upgrading from one version of Flint to another. You need to create new users for the new version. Also, users are not shared between Autodesk Visual Effects and Finishing products.

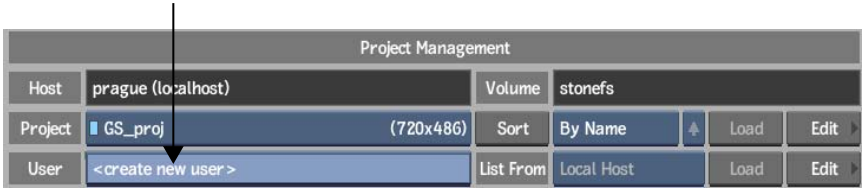
You can create a user on start-up from the Project Management menu, or during a session from the Preferences menu. By default, a user's preferences are created in the directory */usr/discreet/user/effects/<user name>*.

To create a user profile:

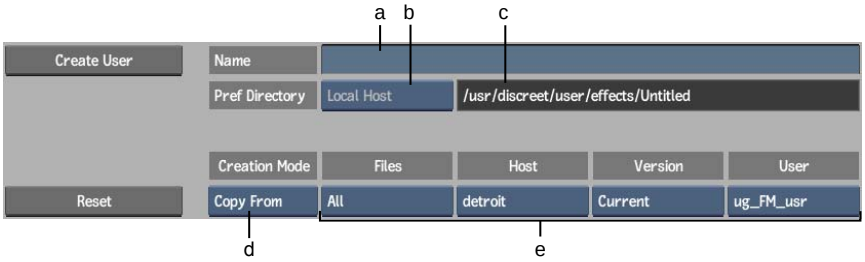
1. Do one of the following:
   - If you are creating a user on start-up, select <create new user> from the User box in the Project Management menu.



   - 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.

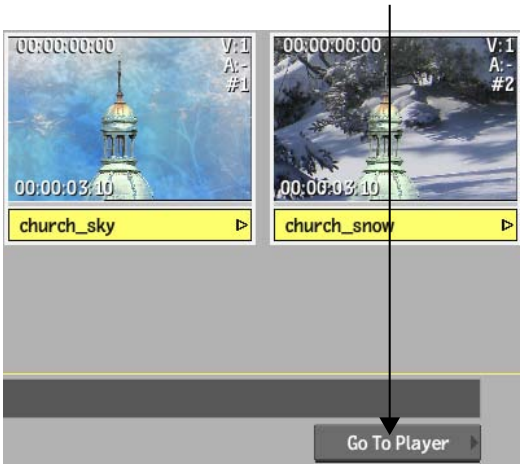
5 To load the user into the current work session, click Load from the Preferences menu. From the Project Management menu, click Start.

Playing Clips from the Library

New for this release: You can play a selection of source clips and processed clips directly from the library.

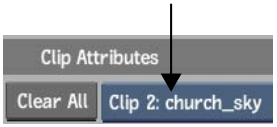
To play clips from the library:

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.

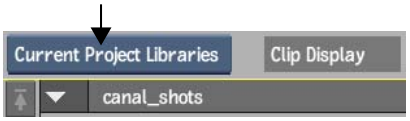
The list of available libraries appear in the Clip Library box. This list may become very long, especially if you accessed libraries belonging to other projects (through the network panel). You can select whether you want network-accessed libraries to appear in the Clip Library box.

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 Flint system, you can enable read-write access to Flare libraries with the R/W button.

To control the libraries displayed in the Clip Library box:

➤ From the clip library, select an option from the Show Library box.

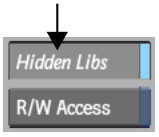


Select:	To display:
All Libraries	All libraries belonging to all projects, including those accessed through the network panel.
Current Project Libraries	All libraries belonging to the current project. Note that if the current library is accessed through the network panel, this option is greyed out.

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.

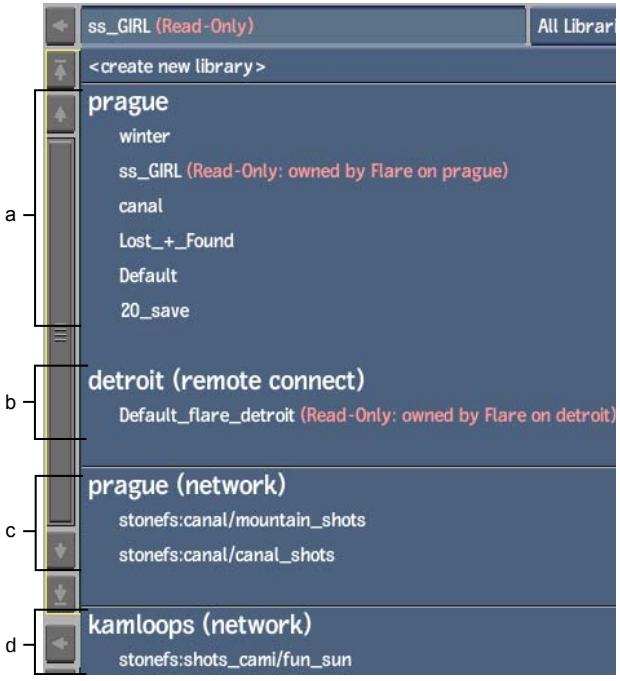


Order of Available Libraries

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

Interoperability Workflow Improvements

9

Topics in this chapter:

- Importing Final Cut Pro XML on page 143
- Relinking to File-based Media on page 146
- Importing AAF Files on page 153
- Relinking to File-based Media on page 156
- Supported and Unsupported Transitions and Effects on page 163

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 Flint 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.

Flint can conform XML (23.976/29.97/59.94) from Varicam material (23.976/29.97/30/59.94/60). Flint 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.

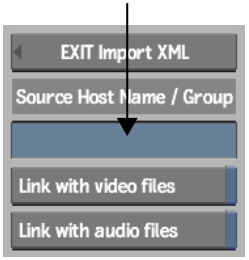
To import or soft-import an FCP XML file:

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.

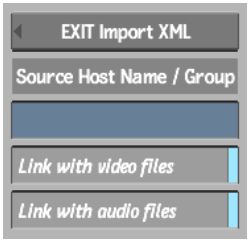
4 If you are importing FCP XML that contains media files, and your files are stored on other computers, you can use Wiretap to access them. Enter the host or group name, as configured in the *sw_wiretap_path_translation_db.xml* file.



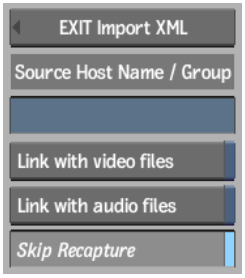
The Wiretap Server must be configured properly to recognise the different hosts that you want to retrieve media from. See the *Autodesk Stone and Wire Filesystem and Networking Guide* or contact your system administrator.

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 Flint 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.



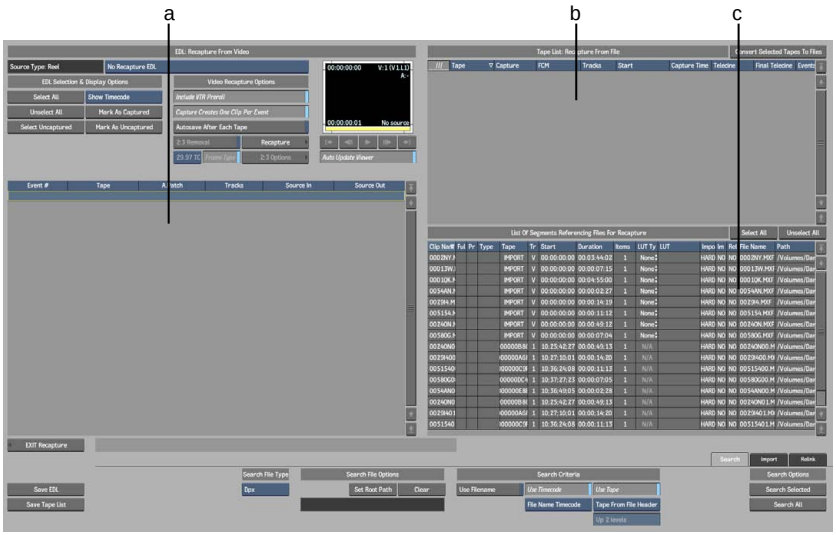
6 If you are importing FCP XML that contains media files of a different file type or resolution than what you want to use (for example, when trying to relink to source media after working with proxies), then enable the Skip Recapture button.



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 146.

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 Flint.

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.

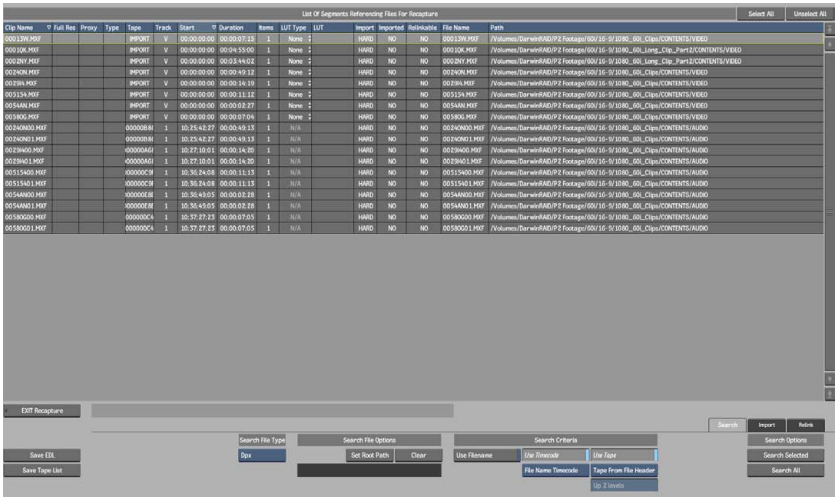
You can relink imported FCP XML files to file-based video or audio media.

Verify that the files you are relinking to are supported in Flint.

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:

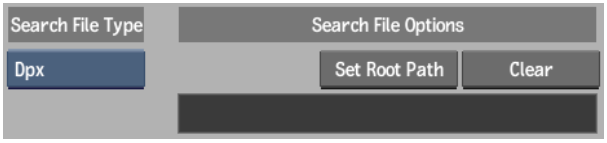
1 Swipe to the right to display the List of Segments Referencing Files For Recapture table across the full screen.



2 Use the Search Criteria to find the source media files.

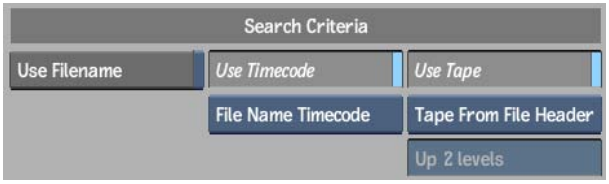
When importing sequences (DPX) or streaming media (MXF, QT) that are referenced by XML files, you can use the advanced Search feature to find and read these image sequences or streaming media, based on preset search rules and criteria. These media files are mostly arranged in hierarchical structures 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.



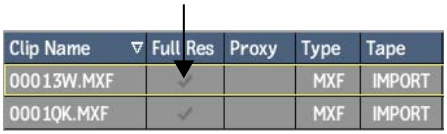
Select	To
Use Filename	Search for the exact file name referenced in the XML.
Use Timecode	Read the metadata from found items to make sure that the starting timecode matches the one in the edit list. Select whether to read the timecode from the file header (MXF and QuickTime) or from the file name (DPX).
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.

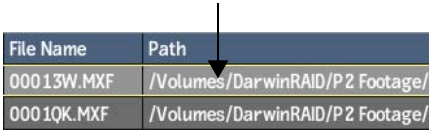
A progress bar appears.

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.



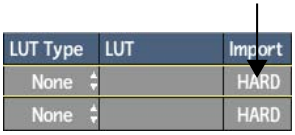
7 If files are not found, you can redo the search by deselecting match criteria. You can also run the search for a different file type. Data for all the previously found files is kept.

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.



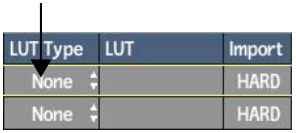
This opens the library browser where you can choose a different path.

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.

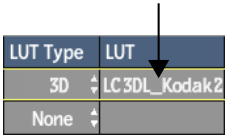


NOTE Not all files can be soft-imported (for example, audio files at 44.1 kHz).

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.

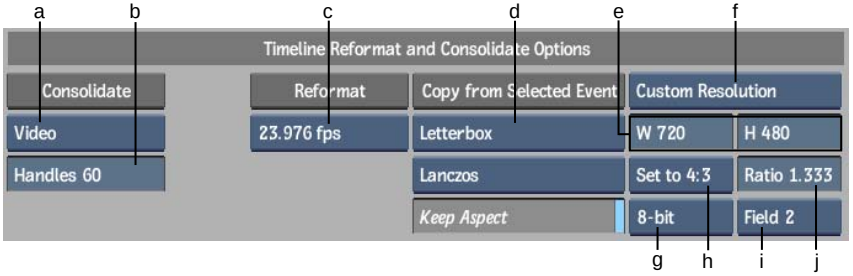
Import	Imported	Relinkable	File Name
SOFT	YES	YES	00013W.MXF
SOFT	YES	YES	0001QK.MXF

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

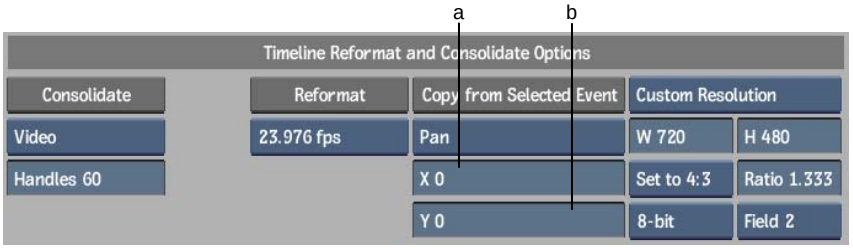
imported previously, in which case this column would already be checked accordingly.

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.

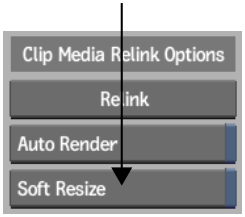
2 In the Handles field, set the maximum number of head and tail frames that you want to retain after consolidating the clip.

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 Flint 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.

Flint has the ability to conform AAF (23.976/29.97/59.94) from Varicam material (23.976/29.97/30/59.94/60). Flint 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).

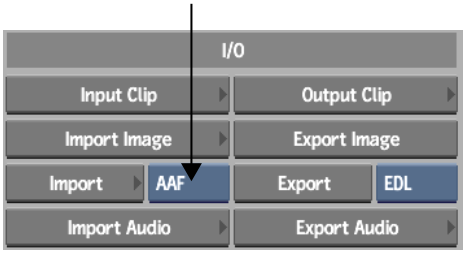
After importing an AAF file, you recapture the footage from the original tapes using the Recapture tool. You can also reload file-based media. Once all the media is captured or reloaded, it is relinked to the sequence.

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.

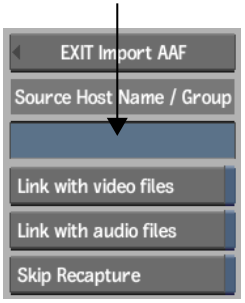
A new reel with the AAF filename is created in the clip library for each imported AAF file. The assembled clip in the new reel has the same name as the original AAF sequence.

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.

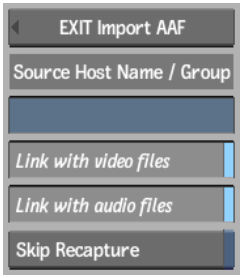


The Wiretap Server must be configured properly to recognise the different hosts that you want to retrieve media from. See the *Autodesk Stone and Wire Filesystem and Networking Guide* or contact your system administrator.

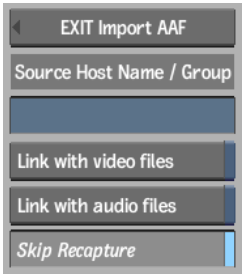
If you do not enter a value in the Source Host Name / Group field, any paths contained in the AAF file will be interpreted as pointing to your

Flint 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.



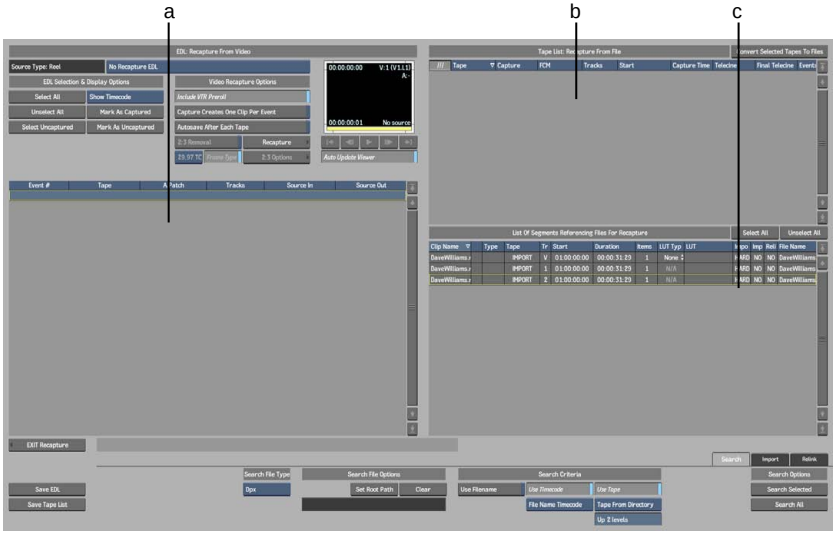
6 If you are importing AAF that contains media files of a different file type or resolution than what you want to use (for example, when trying to relink to source media after working with proxies), then enable the Skip Recapture button.



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 156.

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 Flint.

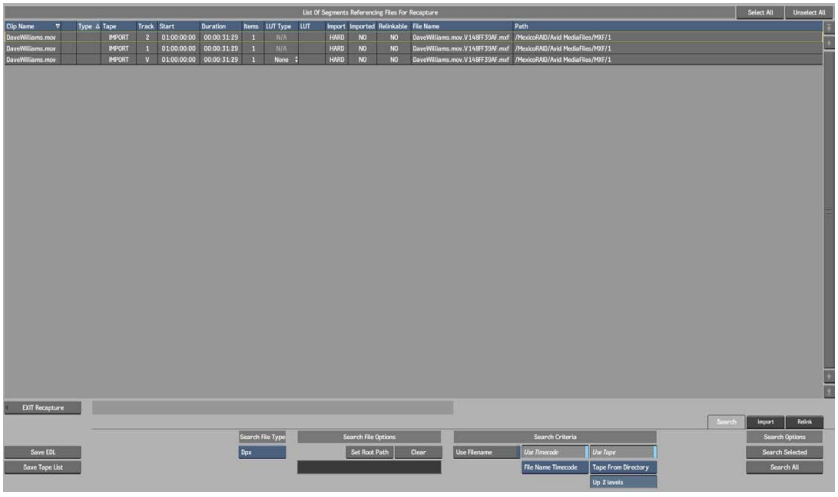
Verify that the media files you are relinking to are supported in Flint.

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:

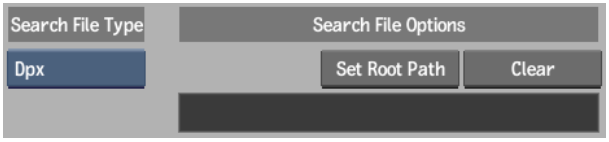
1. Swipe to the right to display the List of Segments Referencing Files For Recapture table across the full screen.



2. Use the Search Criteria to find the source media files.

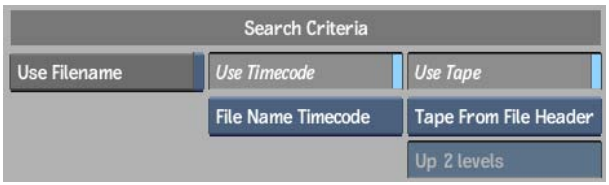
   When importing sequences (DPX) or streaming media (MXF, QT) that are referenced by AAF files, you can use the advanced Search feature to find and read these image sequences or streaming media, based on preset search rules and criteria. These media files are mostly arranged in hierarchical structures 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.



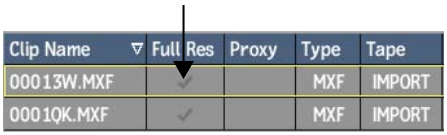
Select	To
Use Filename	Search for the exact file name referenced in the AAF.
Use Timecode	Read the metadata from found items to make sure that the starting timecode matches the one in the edit list. Select whether to read the timecode from the file header (MXF and QuickTime) or from the file name (DPX).
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.

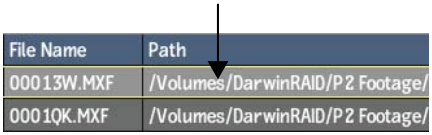
A progress bar appears.

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.



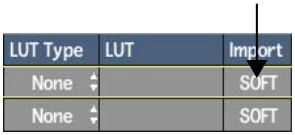
7 If files are not found, you can redo the search by deselecting match criteria. You can also run the search for a different file type. Data for all the previously found files is kept.

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.



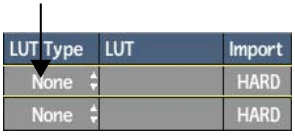
This opens the library browser where you can choose a different path.

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.

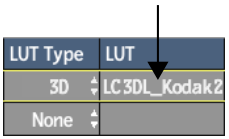


NOTE Not all files can be soft-imported (for example, audio files at 44.1 kHz).

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.

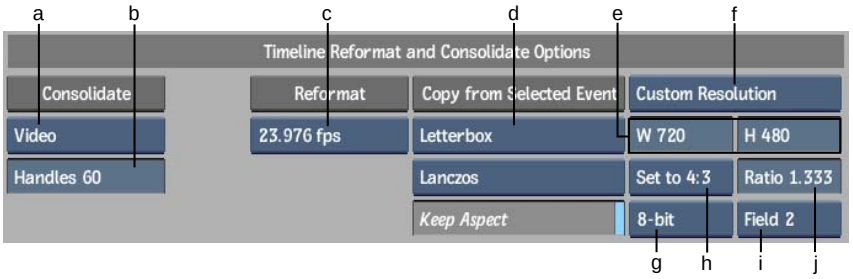
All the files should now be imported or soft-imported.

Import	Imported	Relinkable	File Name
SOFT	YES	YES	00013W.MXF
SOFT	YES	YES	0001QK.MXF

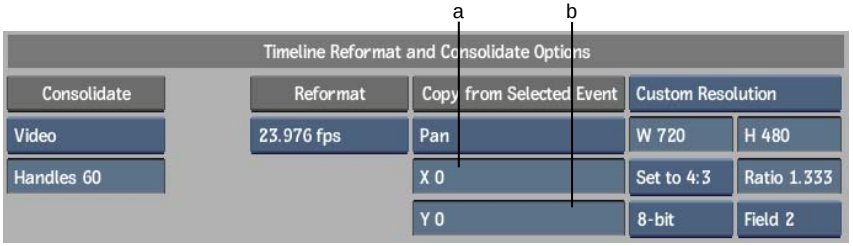
The Imported column indicates whether a file was imported or not. The Relinkable column displays whether the file is relinkable. A file can be imported and non-relinkable if a discrepancy exists between the resolution of the AAF and the found media. Also, the media may have already been imported previously, 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.

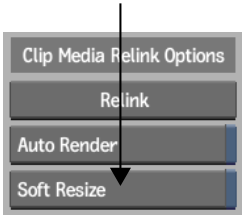
2 In the Handles field, set the maximum number of head and tail frames that you want to retain after consolidating the clip.

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 Flint 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 Flint Soft Effect Parameters.
- **Converted** The effect type is converted without parameters from the AAF file to a Flint 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 Flint).
- 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 Flint 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 Flint.

Metadata

Avid	Flint
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	Flint
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	Flint
Video media (OMF, MXF)	Not supported
Audio media (WAV, AIFF)	Not supported

Sequence

Avid	Flint
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 Flint.

Transitions marked with an * are also supported with the “Inverse” option set.

Blend

Avid	Flint
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	Flint
Film dissolve	Converted to Dissolve (hermite animation)
Film fade	Converted to Dissolve (linear animation)

Box wipe

Avid	Flint
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	Flint
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	Flint
4 corners*	Converted to SMPTE 007; softness not supported

Avid	Flint
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	Flint
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	Flint
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	Flint
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	Flint
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 Flint.

Blend

Avid	Flint
Picture-in-picture	Supported
Superimpose	Translated to Soft Blend (transparency value is translated)

NOTE Scaling in AAF can be X and Y. Flint only supports one value (X or Y) for both.

Film

Avid	Flint
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	Flint
Illusion FX	Not supported; replaced by Cue mark
AVX Plugins	Not supported; replaced by Cue mark

Image

Avid	Flint
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	Flint
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	Flint
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	Flint
Title	Not supported; only text string is available
Marquee Text	Not supported; only text string is available

Key

Avid	Flint
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	Flint
RGB keyer	Not supported; replaced by Cue mark

Miscellaneous

Avid	Flint
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	Flint
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	Flint
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

10

Topics in this chapter:

- Navigating Edit Sequences on page 173
- EQ on page 177

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:

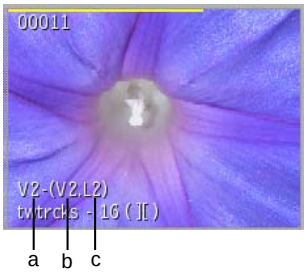
1. Uncollapse the clip on a Desktop reel.
2. With the cursor over the Desktop clip, use any of the following hotkey and Player control combinations.

(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**

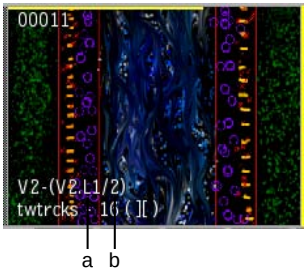
To go to:	Press:
The next video track up	**Ctrl+Alt+up arrow**

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

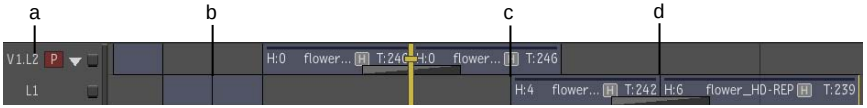
Identifying Cuts and Transitions

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:

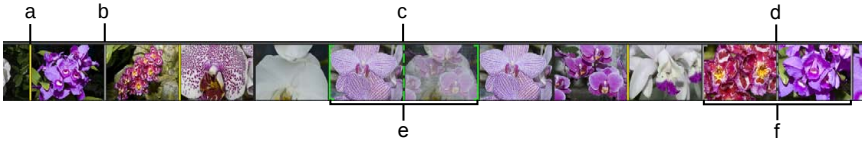
- Cuts between clips on the focus layer in an edit sequence appear as yellow lines.
- Transition focus points on the focus layer appear as dotted green lines.
- Cuts between clips on non-focus layers appear as grey lines.
- Transition focus points on non-focus layers appear as dotted grey lines.

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.

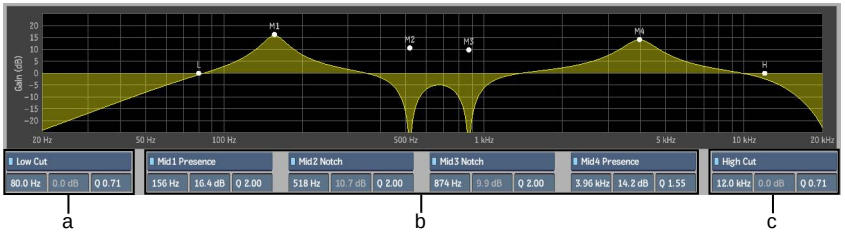
You can use any of six available filter or nodes: one Low node, four Mid nodes, and one High node. The Low node can be set to use either a Low Shelf filter or a Low Cut filter. The four Mid nodes can each be set to either a Mid Notch filter or a Mid Presence filter. The High node can use a High Shelf 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

11

Topics in this chapter:

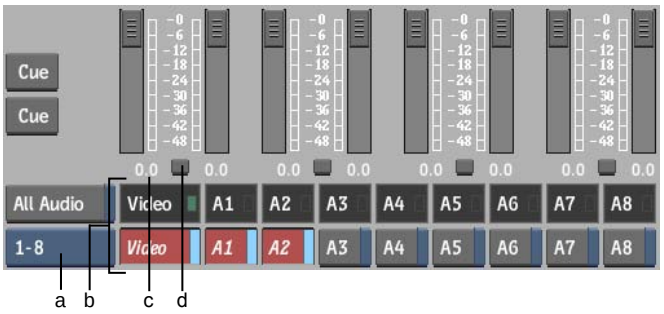
- Adjusting Audio Gain on Output Clip on page 179
- Using Output Strips on page 181

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.

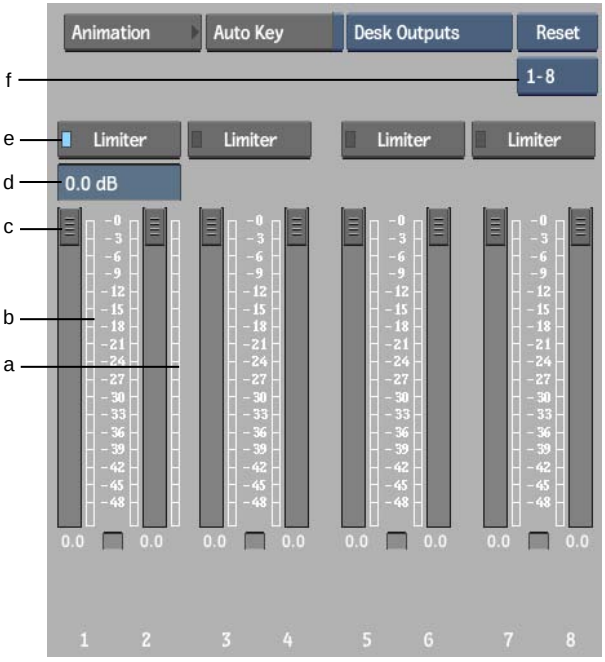
5 Process the clip.

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.