Autodesk®

Inferno® 2009 Extension 1

A Discreet® Systems product

New Features Guide

Autodesk®

the name of the NumPy Developers nor the names of any contributors may be used to endorse or promote products derived from this software without specific prior written permission. THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Portions relating to PostgreSQL v. 7.4.8-1 Copyright ©1996-2003, PostgreSQL Global Development Group.

Portions relating to PostgreSQL v. 7.4.8-1 Copyright © 1994, The Regents of the University of California. Permission to use, copy, modify, and distribute this software and its documentation for any purpose, without fee, and without a written agreement is hereby granted, provided that the above copyright notice and this paragraph and the following two paragraphs appear in all copies. IN NO EVENT SHALL THE UNIVERSITY OF CALIFORNIA BE LIABLE TO ANY PARTY FOR DIRECT, INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES, INCLUDING LOST PROFITS, ARISING OUT OF THE USE OF THIS SOFTWARE AND ITS DOCUMENTATION, EVEN IF THE UNIVERSITY OF CALIFORNIA HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGE. THE UNIVERSITY OF CALIFORNIA SPECIFICALLY DISCLAIMS ANY WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE SOFTWARE PROVIDED HEREUNDER IS ON AN "AS IS" BASIS, AND THE UNIVERSITY OF CALIFORNIA HAS NO OBLIGATIONS TO PROVIDE MAINTENANCE, SUPPORT, UPDATES, ENHANCEMENTS, OR MODIFICATIONS.

Portions relating to PostgreSQL Server v. 7.4.8-1 Copyright ©1996-2003, PostgreSQL Global Development Group.

Portions relating to PostgreSQL Server v. 7.4.8-1 Copyright © 1994, The Regents of the University of California. Permission to use, copy, modify, and distribute this software and its documentation for any purpose, without fee, and without a written agreement is hereby granted, provided that the above copyright notice and this paragraph and the following two paragraphs appear in all copies. IN NO EVENT SHALL THE UNIVERSITY OF CALIFORNIA BE LIABLE TO ANY PARTY FOR DIRECT, INDIRECT, SPECIAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES, INCLUDING LOST PROFITS, ARISING OUT OF THE USE OF THIS SOFTWARE AND ITS DOCUMENTATION, EVEN IF THE UNIVERSITY OF CALIFORNIA HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGE. THE UNIVERSITY OF CALIFORNIA SPECIFICALLY DISCLAIMS ANY WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE. THE SOFTWARE PROVIDED HEREUNDER IS ON AN "AS IS" BASIS, AND THE UNIVERSITY OF CALIFORNIA HAS NO OBLIGATIONS TO PROVIDE MAINTENANCE, SUPPORT, UPDATES, ENHANCEMENTS, OR MODIFICATIONS.

Portions relating to Mesa v. 6.5.1 Copyright © 1999-2007 Brian Paul. All Rights Reserved. Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions: The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software. THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL BRIAN PAUL BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.

Portions relating to pySerial Copyright © 2001-2004 Chris Liechti cliechti@gmx.net; All Rights Reserved.

Portions relating to OpenExr v. 1.2.1 Copyright © 2004, Industrial Light & Magic, a division of Lucasfilm Entertainment Company Ltd. Portions contributed and copyright held by others as indicated. All rights reserved. Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met: Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. Neither the name of Industrial Light & Magic nor the names of any other contributors to this software may be used to endorse or promote products derived from this software without specific prior written permission. THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.

Portions relating to libtiff are Copyright © 1988-1997 Sam Leffler. Copyright © 1991-1997 Silicon Graphics, Inc. Permission to use, copy, modify, distribute, and sell this software and its documentation for any purpose is hereby granted without fee, provided that (i) the above copyright notices and this permission notice appear in all copies of the software and related documentation, and (ii) the names of Sam Leffler and Silicon Graphics may not be used in any advertising or publicity relating to the software without the specific, prior written permission of Sam Leffler and Silicon Graphics. THE SOFTWARE IS PROVIDED "AS-IS" AND WITHOUT WARRANTY OF ANY KIND, EXPRESS, IMPLIED OR OTHERWISE, INCLUDING WITHOUT LIMITATION, ANY WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. IN NO EVENT SHALL SAM LEFFLER OR SILICON GRAPHICS BE LIABLE FOR ANY SPECIAL, INCIDENTAL, INDIRECT OR CONSEQUENTIAL DAMAGES OF ANY KIND,

Trademarks

Disclaimer

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

Images Courtesy of: Behavior Communications Inc., The House, Topix / MadDog

Title: Autodesk Inferno 2009 Extension 1 New Features Guide

Document Version: 1

Date: September 29, 2008

Contents

Introduction

Topics in this chapter:

About Inferno Documentation

Inferno® 2009.1 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/inferno-documentation*.

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

Using the New Features Guide

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

Viewing Tooltips

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

To view tooltips:

1. Move the cursor over the object.

   After a few seconds, the tooltip displays.

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 Inferno is a Help system that you can view in a Web browser. The Help is installed automatically and is accessible anywhere within Inferno.

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

To view the Help from Inferno:

1. Start Inferno.
2. On the bottom-right of the screen, click Help.

   You can also click Preferences to display the Preferences menu and click Help.

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

   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 Inferno *documentation* directory. You can view any of the PDF files in that directory from Inferno.

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

To view the PDF files from Inferno:

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.

Notation Conventions

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

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

Convention	Example
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 Guide

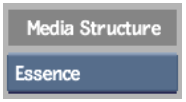
This New Features Guide describes the new and updated features for this release of Inferno. For a quick look at the new features, see the following sections. Some

major features also have more information in this guide — just follow the links in this chapter.

New Features Overview

This table provides a quick overview of the main new features for this release. Follow the links for more information about the features.

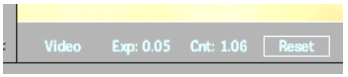
Inputting and Outputting Media New features include enhancements for media import and export to provide better support for MXF, QuickTime, DPX, and Cineon files. See What's New in Inputting and Outputting Media on page 7.



Real-time Deliverables Use this new time-saving feature to output to many output formats without rendering. See What's New in Real-Time Deliverables on page 10.

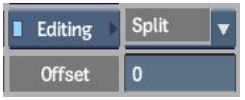
Clip Workflow Improvements Work more efficiently with updates to keycode, Resize, and for viewing clips in modules or your broadcast monitor. See What's New in Clip Workflow Improvements on page 10.

Colour Management Enhancements include improvements with working with LUTs and viewing images. See What's New in Colour Management on page 13.



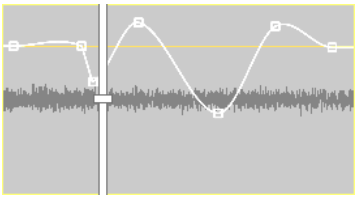
Batch Nodes New features include floating point (OpenEXR) support for more Batch nodes, as well as improvements to Auto Stabilize, and the introduction of the Motif node. See What's New in Batch Nodes on page 14.

Batch Setups New features include improvements to displaying head and tail frames, committing Batch FX, and copying

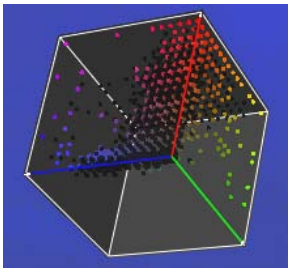


BFX clips. See What's New in Batch Setups on page 17.

Audio Workflow improvements have been made to editing gain curves, and setting interpolation on audio animations. See What's New in Audio on page 18.

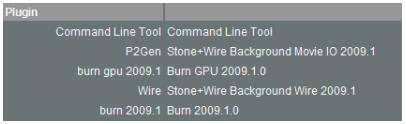


Effects New features include floating point (OpenEXR) support for the Colour Corrector and Colour Warper, as well as improvements to Distort. See What's New in Effects on page 19.



Action Enhancements include soloing objects in the schematic, auto 3D tracker updates, and the introduction of particle presets. See What's New in Action on page 20.

Infrastructure Support has been added for Burn background rendering of floating point jobs on GPU-equipped Render Nodes. See What's New in Infrastructure on page 22.



What's New in Inputting and Outputting Media

The following changes apply to inputting and outputting media.

New QuickTime Features

QuickTime support has been enhanced with the following features.

- You can now apply a gamma correction when you export and recapture QuickTime files, in addition to when you import them. See Gamma Correction for QuickTime on page 27.
- You can now import and export QuickTime files using the Motion JPEG-A codec. Note that on export of Motion JPEG-A, only progressive frames are exported.
- Importing QuickTime Apple Animation files with Alpha channel is now supported. Ensure that the QuickTime is exported with a depth of Millions of Colors+.
- When importing QuickTime files, the tape name and timecode are now read directly from the file header.
- When exporting QuickTime files using the MPEG4 codec, you now have a choice of three new sample Video Codec Presets to choose from: MPEG4_LOW_QUALITY, MPEG4_MID_QUALITY, and MPEG4_HIGH_QUALITY.

New LZW Compression when Exporting Tiff

When exporting Tiff files, enabling Compress will now activate LZW compression. LZW is a lossless compression algorithm that is faster than the previous RLE algorithm and creates a smaller output file.

Field Dominance upon Import

Field dominance can now be read from the file upon import. This can be enabled, in the Processing group, by setting Scan Mode to File Header. See Field Dominance upon Import on page 28.

Importing Proxies for DPX and Cineon

For DPX and Cineon images, a film scanner generates full-resolution scans that are often complemented by lower resolution proxy scans. You now have the choice of importing these existing proxies, or generating new proxies,

when recapturing or importing DPX or Cineon images. See About the Proxy Import Settings on page 29.

New Supported MXF File Codecs

When importing P2 MXF media, the AVC-Intra 50 and AVC-Intra 100 codecs at 10 bits are now supported.

When importing XDCAM MXF media, the MPEG-2 IMX 30, MPEG-2 IMX 40, MPEG-2 IMX 50 (for XDCAM), and MPEG-2 long-GOP (for XDCAM HD) codecs are now supported.

Importing MXF Media

When importing MXF media, the browser can now read the P2 and XDCAM content structure and display the assembled clips, which include the XML, video, and audio files combined. See Importing and Soft-Importing Video Files on page 31.

Clip Input/Output

When outputting multiple clips, you are prompted to select the timing to use from a list of the available clip timings. See Outputting Multiple Clips on page 34.

Also, the following changes apply to the Output Clip menu:

- A list of clips allows you to select the clips and layers to output, resolve timecode conflicts, and activate and deactivate logo overlay. See Output Clip Menu Controls on page 36.
- You can overlay logos on clips during output. You can also preview the logo in the Output Clip menu Player. This feature is available only on workstations with the NVIDIA FX 5600 SDI graphics card. See Outputting Clips with a Logo Overlay on page 40.
- When using Real-Time Deliverables, you can edit the Deliverables settings directly from the Output Clip menu. See Clip Output Using Real-Time Deliverables on page 55.
- The Player sports a jog area and controls, available when the Deliverables tab or the Logo tab is selected. In this case, the Player displays the clip to output, not the content of the tape.

Publishing Media

The Publishing menu has been improved. Buttons are now grouped so that commands are easier to locate. You can now control the name of output files with shot padding, and the name of published clips with an editable extension. See Publishing on page 47.

What's New in Real-Time Deliverables

Real-Time Deliverables allows you to output from one master source format to many output formats directly to tape, without rendering.

The Real-Time Deliverables feature is only available on workstations using the NVIDIA FX 5600 SDI graphics card. Users working with clips with Deliverables on platforms without the required graphics card cannot view the Deliverables, but can still work with the original clip.

The following are some characteristics of Real-Time Deliverables:

- The Deliverables menu, available from the Player, is used to set timing, resize, and LUT settings for real time output.
- A Deliverable is a virtual clip that can be output like any other clip.
- A Deliverable can be processed to create a new clip.
- Deliverable settings can be edited from the Output Clip menu.
- A Deliverable can be previewed in the Player and in the Deliverables tab of the Output Clip menu.
- Deliverables can be copied, pasted, deleted, loaded, and saved as Deliverable templates.
- You can search for Deliverables in the library and on the Desktop.

See Clip Output Using Real-Time Deliverables on page 55.

What's New in Clip Workflow Improvements

The following clip workflow improvements apply to Inferno.

New Features in Resize

A number of improvements have been made to Resize to improve the workflow and consistency across the application.

- The crop box is now handled as a single object (rather than four lines) with parameters for horizontal and vertical positioning and scaling.
- Resize filter options are improved and expanded to include a number of new methods.
- Resize is now available from Player on the Deliverables tab.
- The arrangement of the Resize parameters has been improved to allow for more consistency and ease of use, including the addition of some new options. See Resize Settings on page 77.

New Keycode Support

When working with clips that contain embedded keycode, a number of new keycode emulsions are supported, including Eastman 5219/7219 Vision3 500T and a number of Fuji ETERNA film stocks.

The 8-perf VistaVision format in keycode is also supported.

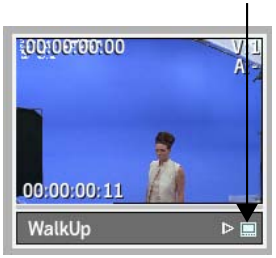
Broadcast Monitor Output

In the Preferences menu, you can select the Show Selected Item option in the Broadcast Monitor box to display the current clip selection in the broadcast monitor. You can also set the broadcast monitor to display the Result view of a selected item. Additional preferences are displayed for image data type and broadcast LUTs.

See Broadcast Settings Preferences on page 85.

In modules with multiple viewports, the Monitor symbol now appears in each viewport. The blue Monitor symbol indicates that the graphics signal is sent to the broadcast monitor. You can send any viewport to the broadcast monitor by clicking its Monitor symbol.

In the clip library, the monitor indicator appears on clips when you select Show Selected Item in the Broadcast Monitor box. Click the indicator to send the clip to the broadcast monitor.



Clip Library Expand Options

You can hide and display selected clips, and expand or collapse selected clips with real-time settings, history, or soft edits using the Expand Options group of dropdown lists. You can also toggle the display of clips with an associated deliverable, history, or source clip by clicking an indicator on a clip.

See Expanding and Collapsing the Contents of a Clip on page 89.

Updated Default Preferences and Setup Options

A number of default preferences and setup options have been updated to better meet user needs. The following table summarizes these new defaults.

Area/Module	Option	Default
Action	Regen button	Enabled
Action	Play Lock button	Disabled
Action	Camera Motion Path button	Disabled
Batch FX	Set to BFX In/Out button and Set to Media Range button	Previous selection within the session
Garbage Mask	Regen button	Enabled
Glow	Regen button	Enabled
Lens Distort	Regen button	Enabled

Area/Module	Option	Default
RGB Blur	Regen button	Enabled
Resize	Regen button	Enabled

New Project Copy Feature

There is a new script that you can use to copy projects from a previous release to Inferno 2009.1. This simplifies the steps necessary to upgrade your projects from a previous release.

See the Compatibility chapter of your user guide for more information.

What's New in Colour Management

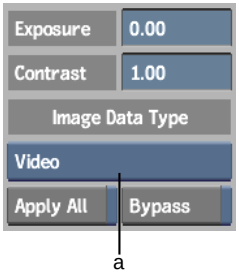
The following enhancements have been made to managing LUTs:

- All LUT types are now configured in the LUT Preferences menu. The entries in the 1D LUT list and 3D LUT list correspond to hot keys that can be used to apply a specific gamma or 1D LUT to the display monitor, or 3D LUT to the image window. See LUT Preferences on page 91.
- 1D LUTs can now be applied to the graphics monitor from the View menu, or the Clip & Setup tab in the Player. See Applying 1D LUTs to the Monitor on page 94.
- When the LUT Editor is accessed from the Import Image menu, you must export the 1D LUT file upon exit. See Exporting LUTs on page 95.
- You can apply a 3D LUT to a clip in the Import Image and Export Image menus. See Applying a LUT on Import or Export on page 96.

Image Display Viewer

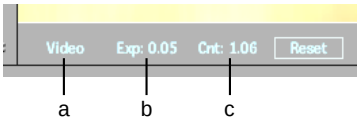
You can now change the display of an image in a viewport or the Player, based on the type of image data you are working with. By default, all images are displayed with a transformation for video images. You can apply transformations to the image to display an optimal view of logarithmic and linear images.

You can set image display setting in a module's View menu or Clip & Setup tab.



(a) Image Data Type box in the View menu

The lower left corner of each viewport displays image data type, exposure, and black level settings. When you are not in the View menu, settings can be adjusted interactively in a module viewport or the Player image window.



(a) Image Data Type (b) Exposure (c) Contrast

See Controlling Image Display using Exposure and Image Data Type on page 98.

What's New in Batch Nodes

The following enhancements apply to working with Batch nodes.

Floating Point Support

The following nodes have been added to the list of nodes that accept floating point (OpenEXR) data as input. This file format has advantages over other formats, such as a high dynamic range and high-quality colour resolution.

- Auto Stabilize
- Average
- Burn-in Letterbox

- Burn-in Timecode
- Combine
- Compound
- Exposure
- Filter
- Flip
- GMask
- Monochrome
- Motif
- Optics
- Text

For a list of all modules that support floating point images, see the Floating Point Support Table on page 23.

Auto Stabilize Node

The following workflow improvements have been made to the Auto Stabilize node:

- When performing an analysis to rectify stabilization issues, the viewport now defaults to the Front view. You can also monitor progress by the keyframes that are created during analysis. When analysis is complete, track points are displayed on the clip to indicate how the stabilization was tracked.
- The stabilization can be customized further using the Trackers section of the Auto Stabilize menu.
- You can further refine stabilization within a region of interest using new settings in the Analysis section of the Auto Stabilize menu.

See Auto Stabilize Node on page 101.

Exposure Node

You can now add the Exposure node to the Batch schematic. Use it to apply plausible exposure and contrast settings suitable to the image data type of input clip. The values can be adjusted separately for each colour channel or controlled simultaneously. See Exposure Node on page 106.

GMask Node

The GMask node supports 16-bit floating-point (OpenEXR) data as input. The Stabilizer and pickers are not available when this type of input is attached to the node in Batch or the Modular Keyer.

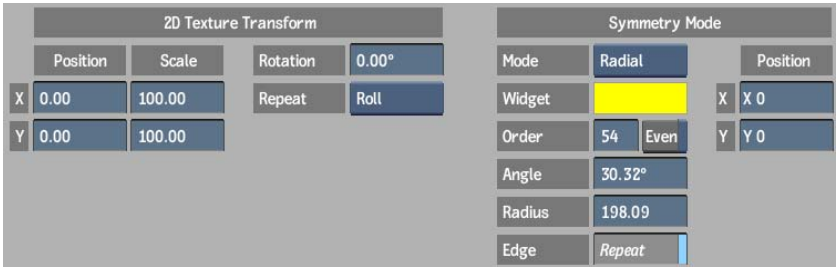
For more information, see the Batch: Node Reference chapter in your user guide.

Motif Node

Use the new Motif node to create a tiled symmetrical texture. Source clips can be transformed before symmetrical effects are applied. Two types of symmetry modes can be used on the transformed image.

The radial symmetry mode uses a sector of a polygon as the originating tile for the creation of a kaleidoscopic texture. The region of interest effect uses a user-defined rectangular selection as the originating tile to create a basic mirrored texture.

The Motif Node menu displays 2D Source Transform controls that allow you to simultaneously change the position, scaling, rotation, and type of image padding. The Symmetry Mode controls display a colour pot for the original tile selection.



See Motif Node on page 107.

What's New in Batch Setups

The following workflow enhancements have been made to Batch setups.

Clip and Timecode Display

Several improvements have been made to clip and timecode display in Batch.

- You can now change the timecode of a clip brought into Batch so that it corresponds to Batch time.
- You can display or hide negative frames that result from offsetting the starting frame of a clip in main Batch.
- When you load a clip into Batch, you can now display head and tail frames in the Batch timebar. Previously you could only display head frames.

See BFX Timecode and Clip Frame Length on page 113.

Committing Results

You can now commit BFX or BFXa results while preserving soft effects. See Committing Batch FX on page 117.

Copying Clips

You can now send copies of BFX clips to the Desktop. See Copying Clips to the Desktop on page 118.

Split Option Renamed

With the introduction of the new dropdown list, the Split options for creating one clip per layer have been renamed. To activate an option, select it from the dropdown list.

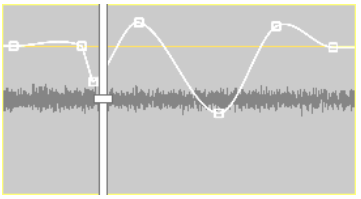
New Name:	Replaces:
Split Tracks and Layers	Split Layers
Split and Convert to Action Layers	Split+Action
Split and Convert to Batch Tree	Split&Extract

For information about each option, see Creating One Clip Per Timeline Layer on page 120.

What's New in Audio

The following changes apply to audio.

- You can now animate the gain for individual segments (and view their animation curves) in the timeline, using a simplified version of the Channel Editor.

  You can also edit the gain curves of two or more audio segments simultaneously, provided that you select all segments before you begin adjusting the curve. The segments must start at the same frame.

  See Gain on page 123.

- All keyframe animation applied to audio segments uses linear interpolation by default. You can change the default in the Preferences menu.

What's New in Effects

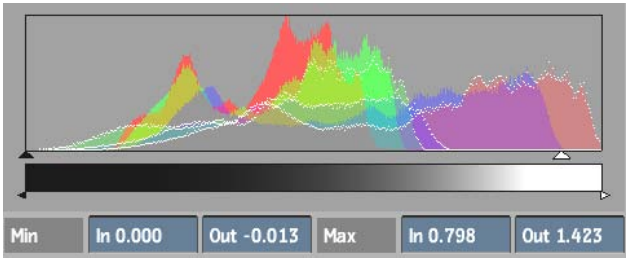
This section describes enhancements made to various effects modules.

Floating Point Support in the Colour Corrector and Colour Warper

16-bit floating point OpenEXR clips are now supported in the Colour Corrector and Colour Warper. This file format has advantages over other formats, such as a high dynamic range and high-quality colour resolution.

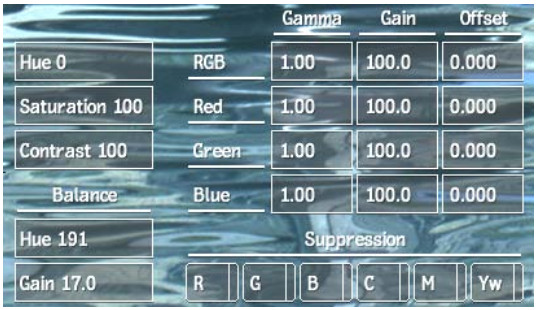
Some of the menus and controls in the Colour Corrector have been updated to accommodate the new capabilities of floating-point images. See Adjusting the Colour Balance on page 125, Adjusting the Colour Range on page 128, and Remapping Colour Values on page 134.

Similarly, there are some differences in the Colour Warper menus to accommodate floating-point images. See Generating Mattes on page 136.



Overlay User Interface in the Colour Corrector and Colour Warper

When working in the Colour Corrector and Colour Warper modules, you can now use the Overlay user interface to make menu controls transparent on top of your image. This provides you a greater viewing area when working with large or zoomed-in images. See Overlay User Interface on page 141.



Distort Improvements

The following enhancements have been added to Distort to help you work quicker by reusing existing elements:

- You can now import saved garbage mask setups into Distort to use as splines. See Using Garbage Masks as Distort Splines on page 144.
- When loading saved setups in Distort, you can now choose to replace any existing objects (such as splines or axes) in your Distort scene with the loaded setup, or you can append the setup to your existing Distort scene.
- When you add multiple axes above a pair of source and destination splines, each axis can be used to manipulate a separate instance of the pair of splines. See Instancing Distort Splines on page 146.

Filter Module and Node

The Filter module and Batch node feature a new menu layout tailored to a widescreen interface.

Modular Keyer Improvements

A Resize node can now be added to any part of the processing pipeline, except the blend node pipes. See Resize Node on page 146.

What's New in Action

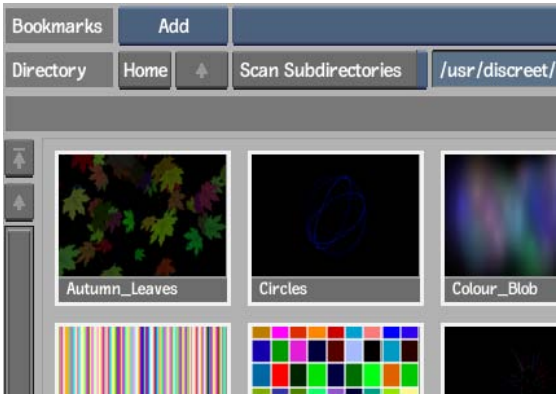
The following enhancements have been made in Action.

Setting Action Resolution to Background

In the Resolution Presets box of the Action Setup menu, you can now select Background Resolution to set the resolution to that of the background clip. The settings in the Resolution section change automatically to reflect the background clip resolution. If there is no background clip, the settings revert to the project resolution.

Particle Presets

To help take some of the complexity out of creating particles, a new library of close to 100 particle presets is available from within Action. You can use the particle presets as is, or as a starting point to explore particles and create custom effects. For details on how to apply the presets in Action, see Using Particle Presets on page 164. For a list of all the presets with usage comments, see the Particle Presets Catalogue on page 167.



Automatic Crop Softness

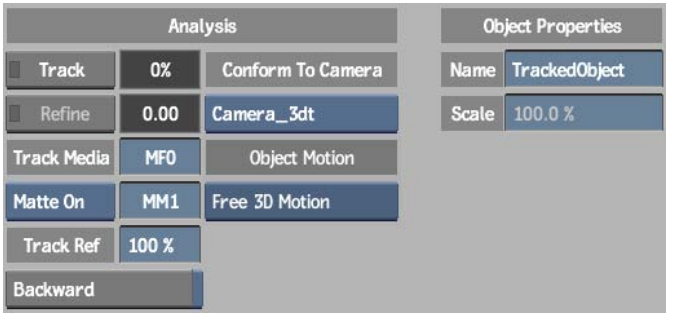
Cropping a clip in the Media list automatically enables crop softness, saving you time by not having to enable the softness separately.

Soloing Objects in the Schematic

You can now solo an object or branch in the schematic and temporarily hide all other objects in the schematic. This saves time by not having to hide multiple objects one by one. See Soloing Objects on page 149.

3D Camera Tracking Improvements

A new mode of automatic 3D tracking has been introduced: Object Tracking, allowing you to track multiple objects in a scene relative to the same camera. When auto camera tracking, you can also now track the scene or an object and generate a fixed camera. See Auto 3D Tracking on page 151.



Animated Garbage Masks as 3D Geometry

In the previous release, the ability to import saved garbage mask setups into Action as 3D models was introduced. A further improvement to this feature has been added: you can now import animated garbage masks as 3D geometries, and all shapes are imported and available to use.

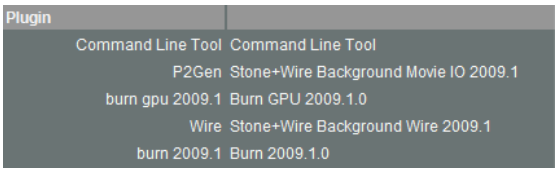
What's New in Infrastructure

The following enhancements apply to infrastructure.

GPU-Accelerated Burn Nodes

On Render Nodes equipped with GPU-accelerated graphics cards, Burn now supports background rendering of complex jobs, such as floating point jobs (for example, unclamped colours in Action, directional RGB blur, or radial RGB blur).

You can use Backburner Monitor or Backburner Web Monitor to see the hardware capabilities of each node in your rendering network.



For information about the supported GPU-accelerated graphics cards, as well as installation and configuration instructions, see the Autodesk *Burn 2009 Extension 1 Installation and User Guide*.

Floating Point Support Table

The following table summarizes the features and nodes which support 16-bit floating point OpenEXR images for all Autodesk Visual Effects and Finishing products.

Action	GMask node
Add Pulldown	Import
Archive	Interlace
Auto Stabilize	Interleave
Average	Lens Distort
Burn-In	Library
Burn-In Letterbox	LUT Editor
Colour Correct	Monochrome
Colour Warper	Motif
Coloured Frame	Motion Analysis
Combine	MUX
Compound	Motion Blur
Deal	Negative
Deinterlace	Optics
Difference Matte	Output

Dominance	Posterise
Export	Remove Pulldown
Exposure	Resize
Field Merge	RGB Blur
Filter	Sparks
Flip	Text
Glow	Vector Viewer

New Hot Keys For This Release

The following table summarizes new or updated hot keys for this release.

Area/Module:	Press:	To:
Any numeric field	**spacebar-drag**	Change the value of the field slower than by dragging alone
Any numeric field	**Alt+spacebar-drag**	Change the value of the field faster than by dragging alone
Action	**Q**	Select hidden bicubic vertex
Batch FX	**Ctrl+Alt+1** This hot key combination replaces **Alt+Shift+1**	Display C:Main level context view (the result of the current Batch FX in context of the final output)
Batch FX	**Ctrl+Alt+2** This hot key combination replaces **Alt+Shift+2**	Display C:Level-up context view (the result of the current Batch FX in context of the vertical editing one level up in the pipeline)
Colour Corrector	**Z-drag**	Adjust Shadows colour balance directly in the image window
Colour Corrector	**C-drag**	Adjust Highlights colour balance directly in the image window

Area/Module:	Press:	To:
Colour Corrector	**X-drag**	Adjust Midtones colour balance directly in the image window
Colour Corrector	**V-drag**	Adjust Master colour balance directly in the image window
Colour Corrector/Colour Warper	**Ctrl+'**	Toggle between the Overlay user interface and the regular user interface
Colour Corrector/Colour Warper	**Ctrl+Alt+Num+**(plus sign on numerical keypad)	Increase the opacity of the Overlay user interface
Colour Corrector/Colour Warper	**Ctrl+Alt+Num-** (minus sign on numerical keypad)	Decrease the opacity of the Overlay user interface
Colour Corrector/Colour Warper	**Ctrl+Alt+Num*** (star sign on numerical keypad)	Swap background and foreground colours of buttons in the Overlay user interface
Colour Warper	**V-drag**	Use trackball appearing in Trackball option box
LUTs	**Alt+Shift+(1-0)**	Apply a gamma correction or 1D LUT based on its position in the 1D LUT list
LUTs	**Alt+Shift+'** (on the tilde key)	Toggle ID LUT display mode
LUTs	**Ctrl+Shift+(1-0)**	Apply a the 3D LUT based on its position in the 3D LUT list
LUTs	**Ctrl+Shift+'** (on the tilde key)	Toggle 3D LUT display mode
Module Viewport/Player	**Shift+C+drag**	Modify the contrast of the image display
Module Viewport/Player	**Shift+E+drag**	Modify the exposure of the image display

New Tooltips For This Release

We have added tooltips for all new features in this release. In addition, we have added tooltips to the following:

- Batch nodes
- Colour Corrector
- Colour Warper
- Compositor and Optics
- Clip Library Tools menu
- Distort
- Filters
- LUTs
- Resize and Resize soft effects
- Stabilizer
- Text and Text soft effects

Inputting and Outputting Media

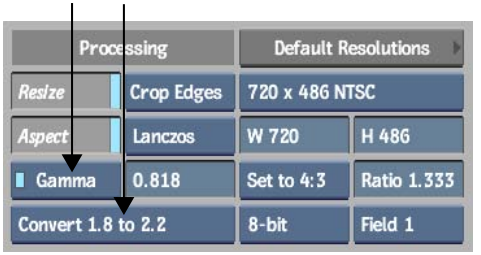
3

Topics in this chapter:

Gamma Correction for QuickTime

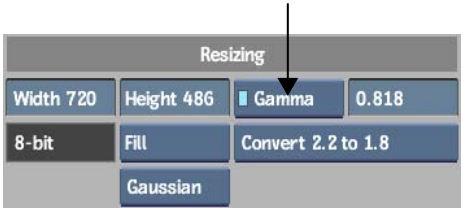
You can now apply gamma correction when you import, export, or recapture QuickTime files.

If you are importing QuickTime files originating from a Mac® computer, apply gamma correction to bring the brightness of the image in line with the rest of your project. From the Colour Correction Type box, select Gamma, then from the Gamma Presets box, select Convert 1.8 to 2.2.



The Gamma Correction field displays the gamma correction value to be applied. It can be modified to a custom value.

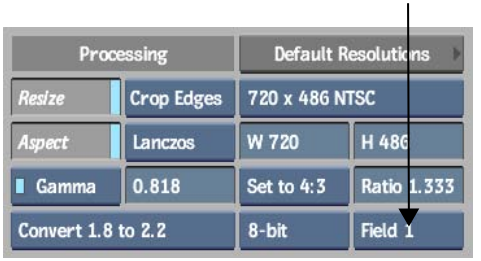
To apply gamma correction on export, load it from Export Image, using the Colour Correction Type box.



For exporting files to a Mac, set Gamma Presets to Convert 2.2 to 1.8.

Field Dominance upon Import

Field Dominance can now be read from the file upon import. This can be enabled, in the Processing group, by setting Scan Mode to File Header.



Use the Scan Mode box to select an option to set the order in which the fields of interlaced material is scanned.

Select:	To:
Progressive	Scan a frame-based clip with no interlacing.

Select:	To:
Field 1	Scan Field 1 first, followed by Field 2.
Field 2	Scan Field 2 first, followed by Field 1.
File Header	Read the field dominance from the file.

About the Proxy Import Settings

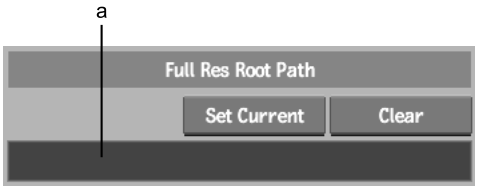
For DPX and Cineon images, a film scanner generates full-resolution scans that are often complemented by lower resolution proxy scans. You now have the choice of importing these existing proxies, or generating new proxies, when recapturing or importing DPX or Cineon images.

Use the Proxy Import settings to configure where the proxy scans are found for the selected Cineon or DPX full-resolution scans. The Proxy Import settings appear when, in the Clip Media Group, you enable the Proxy Import button and click Settings. The options are divided into three functional groups:

- Full Resolution Root Path Group
- Proxy Root Path Group
- Proxy Search Criteria Group

Full Resolution Root Path Group

Use these options to determine the directory location of the full-resolution scans.



(a) Full Resolution Root Path field

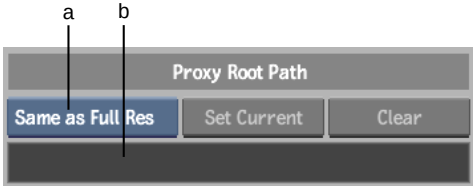
Set Current button Sets the Full Resolution Root Path field to the path displayed in the browser.

Clear button Clears the Full Resolution Root Path field.

Full Resolution Root Path field Displays the path location of the selected full resolution images. This is set using the browser and the Set Current button.

Proxy Root Path Group

Use these options to determine the directory location of the proxy scans.



(a) Proxy Path Selection box (b) Proxy Root Path field

Proxy Path Selection box Choose how the proxy path is determined. When set to Same as Full Res, the path is taken from the Full Resolution Root Path field. When set to Other Mount Point, the Proxy Root Path can be determined using the browser.

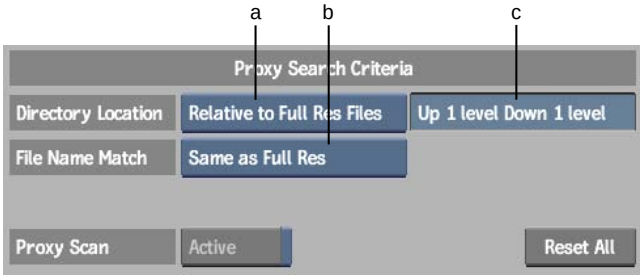
Set Current button Sets the Proxy Root Path field to the path displayed in the browser. Active when Proxy Path Selection is set to Other Mount Point.

Clear button Clears the Proxy Root Path field. Active when Proxy Path Selection is set to Other Mount Point.

Proxy Root Path field Displays the path location of the selected proxy images. This is set using the browser and the Set Current button.

Proxy Search Criteria Group

Use these options to specify the search criteria for finding proxies, including the relative directory location and file name.



(a) Directory Location box (b) File Name Match box (c) Relative Directory Level field

Directory Location box Select whether the proxies can be found in a directory location that mirrors that of the full-resolution scans or in another directory location that is relative using the Relative Directory Level field.

Relative Directory Level field Drag left or right or click to enter the level difference between the directory location of the proxy scans, relative to the directory location of the full-resolution scans. Active when Directory Location is set to Relative to Full Res Files.

File Name Match box Select whether the proxy file names use the same name as the full-resolution files or if they use names that are appended with a prefix/suffix, in which case the File Name Search field appears.

File Name Search field Enter a prefix/suffix that differentiates the proxy file name from the full-resolution scan file name. Use any characters that are legal in Unix file names. Use the asterisk (*) as a placeholder for the current full-resolution scan file name.

Proxy Scan Active button Enable to start the proxy scan using the proxy search criteria entered.

Reset All button Restores all proxy import settings to default.

Importing and Soft-Importing Video Files

Use the Import Image menu to import or soft-import one or more QuickTime movies or MXF media files to the current clip library. Both video and audio tracks are supported.

Audio files that are not 48 KHz are automatically resampled to 48 KHz on import. For video files that contain an alpha channel, you can maintain the alpha channel in the imported clip for use in compositing.

Before importing any P2 MXF 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 timecode and the User Clip Name, to be assigned properly in the MXF file.

Note that you cannot import or soft-import P2 MXF content that contain 2:3 or 2:3:3:2 pulldown, or variable framerates (such as from Varicam cameras).

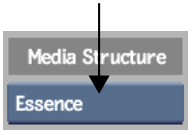
You can scan and import P2 MXF clips whose directory names are either all uppercase or all lowercase.

NOTE To import files created using unsupported video file formats or codecs, you can use Cleaner® XL to encode the files into a supported format. For example, use Cleaner XL to import video as a sequence of compatible still images. See the guide, *Using Cleaner XL with Autodesk Visual Effects and Finishing Applications*.

To import or soft-import a QuickTime movie or MXF media file:

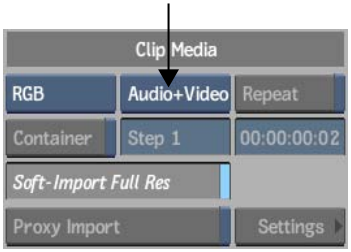
1. From the Library menu, click Import Image and select a destination reel.
2. From the File Format box, select the format of the clip that you want to import.

3. If you chose MXF, then select an option from the MXF Options box. Select P2 or XDCAM to import the timeline and all of the associated video and audio files (which are embedded in the timeline). Select Essence to import only specific audio or video files.



You can also import the entire MXF content and include only the audio files or only the video files. This allows you to preserve the clip timeline

while having the option of leaving out unwanted media. From the Clip Media group, select an option from the Media Stream box.



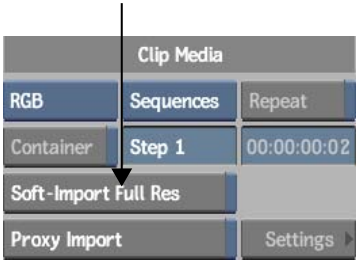
4 Browse to the directory containing the files that you want to import. If you are importing MXF content, then browse to the root directory containing the contents of the P2 card or Professional Disc.

5 Enable Scan Subdirectories to display all the subdirectories and their contents.

For MXF content, the available clip names appear in the browser.

6 Select the files.

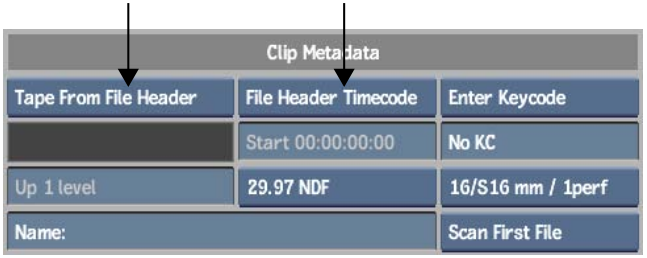
7 Enable Soft-Import Full Res if you want to import a reference to the media, but you do not want to store the media on the local storage.



NOTE Certain options are disabled if you choose to soft-import the media.

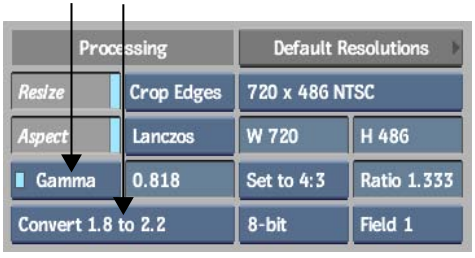
8 Set the tape name for the imported clips and adjust any other Clip Metadata properties.

For QuickTime files, you can read the tape name and timecode from the file header, or enter the tape name and timecode manually.



NOTE A clip that is imported using an unsupported frame rate is converted to the frame rate specified for the current project. This causes the clip to appear out of sync with its audio. A comment specifying the original frame rate is added to the clip notes, providing the ratio needed if you choose to re-time the video.

9 If you are importing QuickTime files originating from a Mac® computer, apply gamma correction to bring the brightness of the image in line with the rest of your project. From the Colour Correction Type box, select Gamma, then from the Gamma Presets box, select Convert 1.8 to 2.2.



10 In the Movie attributes area, ensure that the codecs used in the video file are displayed.

11 Click Load.

The media files are imported to the Desktop and to the specified library reel.

Outputting Multiple Clips

You can output more than one clip to a VTR in a single pass. When you load multiple clips into the Output Clip menu, you must define in and out points (and any other output clip options) for each clip before starting the clip output process.

When multiple clips in the same session are selected for output, the application performs a validation check to determine if certain parameters match the project settings and whether or not the video device is capable of outputting these clips. The following outcomes are possible:

- If the clips have mixed timing settings, you are prompted to select which timing you want to use. Clips having timings different from the one you select are discarded.
- Clips with field dominance mismatch are discarded or kept, depending on your selection.
- Clips exceeding the maximum bit depth are discarded from the selection. A selection containing 8- and 10-bit clips can be output.
- If all the clips are discarded from the selection after the validation check, a message appears allowing you to confirm.

If some of the clips have overlapping timecodes, their timecodes are highlighted in red in the clip list. To fix overlapping timecodes, edit the timecodes in the clip list, or use the timecode fields in the Output tab.

To output multiple clips with the large Output Clip menu:

1. Load multiple clips into the Output Clip menu.
2. If the clips have conflicting timings, you are prompted to select the timing you want to use. Clips that do not match this timing are discarded.
3. To ouput a clip, its status field must have the *selected* indicator. Click the Status field to alternate between the *selected* indicator and an empty field.
4. Set output options for each clip. See Outputting a Single Clip in the Clip Input/Output Using a VTR chapter.
5. Optional: Click the column headers to sort the list. This changes the order of the output sequence.
6. To output the clip to the VTR, select Insert from the Output box.

   The clips with Status selected are output to tape. During output, the Status column is updated to reflect the status of each clip:

   - Pending: the clip is waiting to be output to tape.
   - Output: the clip is being output to tape.
   - Done: the clip has been ouput to tape.

7 After the transfer is complete, verify that it was successful by playing the transferred clips: cue to the in timecodes and click the Play button in the VTR Transport controls. Clips that have been output are highlighted in the clip list of the large Output Clip menu.

8 When finished, click EXIT Output Clip.

When outputting multiple clips with the small Output Clip menu, note the following differences:

- Use the Previous and Next buttons to navigate the clip list.
- Once output options have been set for each clip, make sure you are viewing the first clip you want to output.
- When you select Insert in the Output box, the clip you are viewing, and all subsequent clips, are output to tape. Clips that precede the clip you are viewing are not output to tape.

Output Clip Menu Controls

You can view the Output Clip menu in large or small form depending on which tab is selected.

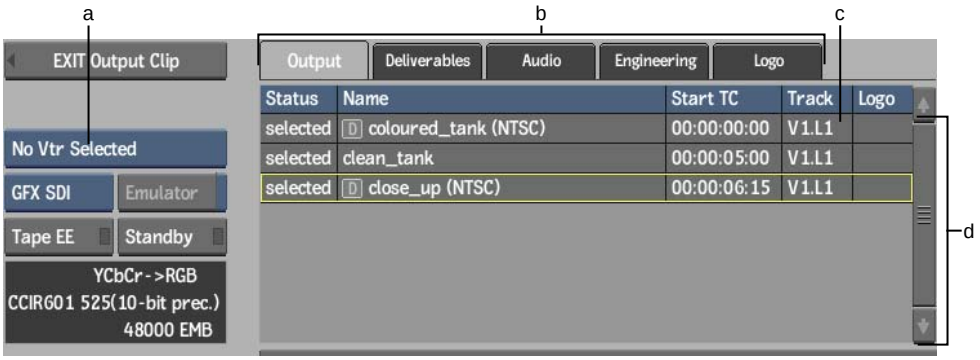
When the Output, Audio, or Engineering tab is selected, both the large and small forms are available. To toggle between the large and small forms, **Ctrl**-swipe the bottom of the screen.

When the Deliverables or Logo tab is selected, only the large form is available, but additional Pan and Zoom controls are available.

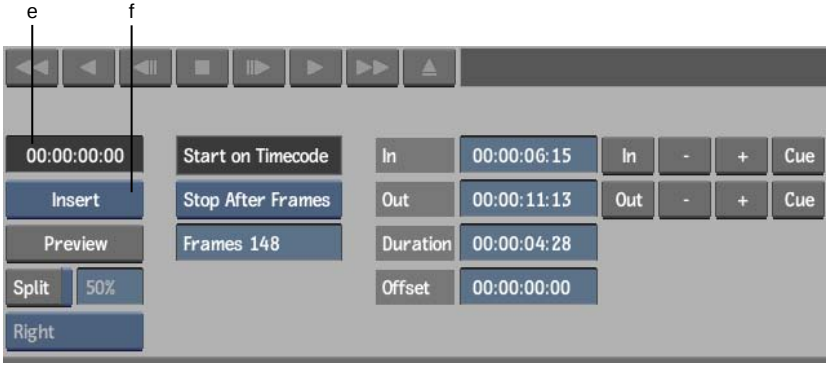
If you are using the large Output menu with an HD clip, the menu automatically switches to the smaller form during clip output, and then switches back when output is complete. This gives you an unobstructed view of the clip during output.

Small Output Clip menu (left portion):

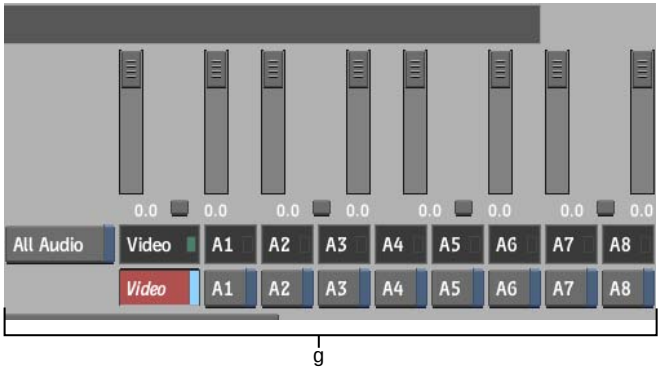
Large Output Clip menu (broken into three parts):



(a) Device Name box (b) Navigation tabs (c) Video Layer field (d) Output Clip list



(e) Current Timecode field (f) Output box



(g) Audio controls

In this documentation, the large menu is described. Differences present in the small menu are noted where applicable.

Device Name box Displays the options for each uncommented VTR keyword line in the software initialisation configuration file. Select the option corresponding to the VTR to which you want to output clips. See Selecting a VTR Device For Output in the Clip Input/Output Using a VTR chapter.

Navigation tabs Switch between different Output Clip menus.

Select:	To:
Output	Configure the output settings described in this section. This is the default menu.
Deliverables	Set up Real-Time Deliverables on output, such as Letterboxes or LUTs. See Outputting Clips With a Letterbox Overlay in the Clip Input/Output Using a VTR chapter, and Clip Output Using Real-Time Deliverables on page 55.
Audio	Set Audio preferences. Changes are reflected in the Audio section of the Preferences menu, and vice versa.
Engineering	View the output clip Engineering menu. See Setting Video Input and Output Engineering Menu Controls in the Clip Input/Output Using a VTR chapter.
Logo	Set up a logo on output. See Outputting Clips with a Logo Overlay on page 40.

NOTE The Deliverables and Logo tabs are only available if the workstation uses an NVIDIA Quadro FX 5600 SDI graphics card. See Hardware Requirements for Real-Time Deliverables on page 57.

Output Clip list Displays information about the clip selected for output. If there are multiple clips, you can sort them by clicking the column headings. This changes the output sequence order. You can also edit the Timecode field in this list. See Outputting Multiple Clips on page 34. If the list includes Deliverables and you do not have the hardware required by Real-Time Deliverables, the Deliverables are greyed out. See Outputting Deliverables on page 64.

Video Layer field Indicates the track to output in a multi-track clip. Drag the field to browse through the video tracks and video layers. This field is red when the selected layer is not the top layer of the selected video track; this does not prevent output.

Graphics Card box If you have the NVIDIA Quadro FX 5600 SDI graphics card for using Real-Time Deliverables, switch between it and the standard AJA_OEM2K card using this box. See Hardware Requirements for Real-Time Deliverables on page 57.

Tape EE button When lit, indicates that the VTR is in E-to-E mode (electronic to electronic). This means that the VTR output is showing its input signal. When E-to-E is off, the VTR shows the contents of the tape it contains. Click this button to toggle E-to-E on and off.

Standby button When lit, indicates that the VTR is in standby mode. Click this button to toggle between on and off.

VTR Status display Indicates the current status of the VTR. See Verifying the VTR Status in the Clip Input/Output Using a VTR chapter of your user guide.

Current Timecode field Indicates the current timecode of the tape in the VTR.

Output box Switch between insert or assemble mode. Click to perform the selected action.

Preview button Triggers a simulation of the output process. The VTR behaves as if it is inserting material, however no material is recorded to tape.

Split View button Enable to simultaneously monitor the clip selected for output and the contents of the tape. See Monitoring Video During Clip Input and Output Operations in the Clip Input/Output Using a VTR chapter of your user guide.

Start On Timecode field A locked field indicating that clip output begins at the timecode entered in the In Timecode field.

Stop Mode box Determines the stop mode for clip output.

Select:	To output the current clip until:
Stop On Timecode	A timecode on the tape is reached (indicated in the Out Timecode field).
Stop After Frames	A specified number of frames is output. When you select this option, a field appears in which you enter the number of frames to output.

In Timecode field Indicates the timecode on the tape at which point the clip output process begins.

Out Timecode field Indicates the timecode on the tape at which point the clip output process ends.

Duration field Indicates the duration, in timecode, between the clip output in and out points.

Offset field Indicates the offset, in timecode, by which the selected clip is output. For example, an offset of 00:00:00:05 indicates that the first five frames of the clip to be output are skipped. The first frame to be output is frame 5 of the clip (counting frames from 0).

In/Out Point controls Use to enter, adjust, and cue the in and out points. See Setting Input and Output In and Out Points in the Clip Input/Output Using a VTR chapter.

All Audio button Enable to output all audio channels for monitoring, even if only some audio channels are enabled for recording to tape.

Audio Channel buttons Enable and control audio output signals. See Adjusting Audio Gain on Clip Output in the Clip Input/Output Using a VTR chapter in your user guide.

Outputting Clips with a Logo Overlay

You can add a logo overlay on output, in real time. This feature is available only if your workstation is equipped with an NVIDIA Quadro FX 5600 SDI graphics card. The configuration for using this card is described in the following documents:

- *Autodesk Visual Effects, Finishing, and Colour Grading HP xw8600 Workstation Hardware Setup Guide*
- *Autodesk Visual Effects and Finishing 2009 HP xw9400 Workstation Hardware Setup Guide*

To add a logo, make sure the FX 5600 SDI card is selected in the Output Clip engineering menu. If AJA_OEM2K is selected, you cannot use the Logo Overlay feature.

To select the FX 5600 SDI card in the Output Clip menu:

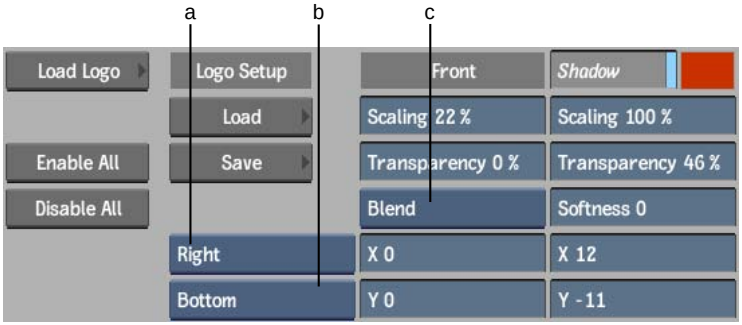
1. Open the Output Clip menu.
2. From the Graphics Card box, select GFX SDI.

If you do not see GFX SDI as an available option in the Graphics Card box, it is either not installed, or it is not set up or configured properly. See one of the hardware guides previously cited.

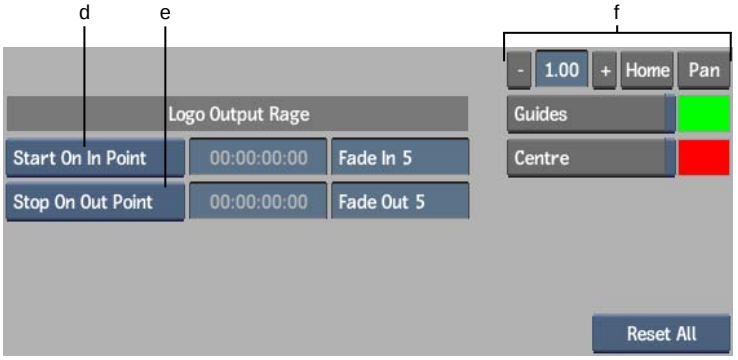
About the Logo Controls

The logo controls are found in the Output Clip menu, under the Logo tab.

NOTE When you select the Logo tab, you see the clip selected in the Output Clip list instead of the VTR feedback.



(a) Horizontal alignment button (b) Vertical alignment button (c) Blend Mode box



(d) In point (e) Out point (f) Zoom controls

Logo Enables or disables burning of the selected logo on output.

Load Logo Opens the Clip Select menu where you can select logo clips. See Inserting Logos for Real-Time Output on page 45.

Enable All Use to display the logo on all clips to output.

Disable All Use to remove the logo from all clips to output.

Scaling (Front) Changes the size of the Logo clip. The range of values is 0 to 100, with 100 being the actual size of the Logo clip.

Transparency (Front) Changes the transparency of the Logo (front clip). 0% is Opaque. 100% is Transparent.

Position controls (X and Y fields) Moves the logo along the X and Y axes.

Blend Mode box Selects how the front and matte clips or the front and back clips are combined:

Select:	To:
Add Trans	Compensate for the soft or anti-aliased edge on an object in a front and matte clip media, with transparency.
Blend	Punch the matte through the front. This blends the edge of the front clip and adds additional softness to the media.

Shadow Uses the Matte clip to add a Shadow. The default position is set to X:-5 and Y:-5. A colour pot on the right allows for the shadow colour to be changed.

Scaling (Shadow) Changes the size of the shadow. The range of values is 0 to 100, with 100 being the actual size of the shadow.

Transparency (Shadow) Changes the transparency of the Shadow. 0% is Opaque. 100% is Transparent.

Softness Softness can be added to the Shadow. The range of values is 0 to 200.

Horizontal alignment Positions the logo along the X-axis using one of the following options:

Select:	To:
Horizontal Centre	Center the logo on the X-axis.
Left	Snap the left edge of the clip to the Safe Title guide on the left.
Right	Snap the right edge of the clip to the Safe Title guide on the right.

Vertical alignment Positions the logo along the Y-axis using one of the following options:

Select:	To:
Vertical Centre	Center the logo on the Y-axis.
Top	Snap the top edge of the logo to the Safe Title guide at the top.
Bottom	Snap the bottom edge of the clip to the Safe Title guide at the bottom.

Guides Enables the display of the Safe Action and Safe Title guides within the Clip Preview window. Use the colour pot to change the colour of the guides.

Centre Enables the display of the Centre guide within the Clip Preview window. Use the colour pot to change the colour of the Centre guide.

In point Select an option to specify when the logo appears on the clip:

Select:	To:
Start On In Point	Have the logo appear at the in point of the clip.
Start After Frames	Specify after how many frames the logo appears. Enter the number of frames in the adjacent field.

Out point Select an option to specify when the logo disappears from the clip:

Select:	To:
Stop on Out Point	Have the logo disappear at the out point of the clip.
Duration	Specify how long the logo remains on screen. Enter the duration in the adjacent field.
Stop Before Frames	Hide the logo before the end of the clip, by the number of frames specified in the adjacent field. For example, selecting this option and entering a value of 5 makes the the logo disappear by frame 15 in a 20-frame clip.

Fade In Defines the length of the logo fade in. A value of 0 indicates there is no fade in.

Fade Out Defines the length of the logo fade out. A value of 0 indicates there is no fade out.

Zoom controls The small output clip menu is not available from the Logo tab. Use the Zoom controls to view oversized clips.

Reset All Use to reset the Logo controls to default:

Select:	To:
Reset All	Reset every field, and delete the logo.
Reset	Reset every field, but keep the logo.

Inserting Logos for Real-Time Output

Keep in mind the following when selecting clips to use as logos:

- If the clip has more than one frame, only the first frame is used.
- A clip that is Burn pending, unlinked or unlinked HiRes, cannot be used.
- The higher the resolution for the clip, the more impact it has on the capacity of the application to output in real time.

Logo settings are persistent, and are reloaded the next time you open the Output Clip menu.

NOTE The same logo, with its settings, is applied to all clips for which you have checked the Logo column in the Output Clip list.

To insert a logo in real time:

1. Access the Output Clip menu.
2. In the Output Clip menu, use the Logo column to indicate the clips on which to overlay the logo.
3. Click the Logo tab to access the logo controls.
4. Select one or two clips to be used as your logo:
   - If you select one clip, it is used as the front clip and a luminance key is applied to it. If the clip has multiple frames, only the first frame is used.
   - If you select two clips, the first is used as the front and the second as the matte.
5. Use the logo controls to position and configure the appearance of your logo, and to define in and out points. See About the Logo Controls on page 41.
6. Output the clip.

Publishing

4

Topics in this chapter:

About Publishing

Publishing allows you to share media with other applications and users.

There are a number of updates in the Publishing menu that improve the workflow of this feature.

Accessing the Publish Menu

Access the Publish menu from the clip library.

To access the Publish menu:

1 In the clip library, click Tools.

2 From the Tools box, select Publish.

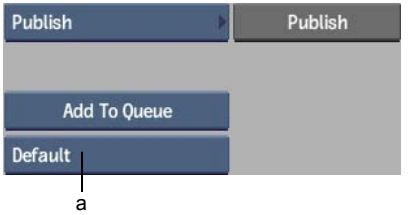
The Publish options appear.

About the Publish Menu Options

Use the options in the Publish menu to specify the attributes for the published file.

File Operations Group

Use the File Operation options to publish your media.



(a) Destination Library option box

Publish button Writes media to shared storage.

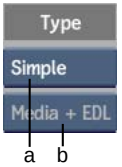
Add To Queue box Select a method to publish in the background.

Select:	To:
Add To Queue	Add the job to the queue.
Add To Queue & Execute	Add the job to the queue and execute it automatically. If there are any jobs in progress, this job (and any others already added with the Add to Queue & Execute option) is triggered upon the completion of its predecessor.

Destination Library option box Select the library where the clips are published.

Type Group

Use the Type Group options to control the output of the Publish tool.



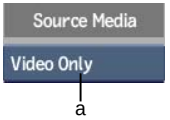
(a) Publish Type option box (b) EDL button

Publish Type option box Select the type of publishing.

EDL button Enable to publish only an EDL file, or EDL and media. This allows you to regenerate the EDL file of a clip that has been published and later modified without republishing all the media. The EDL button is available only when Publish Type is set to Flatten.

Source Media Group

Use the Source Media Group options to control whether you want to publish video or audio.

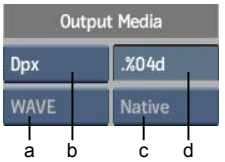


(a) Media Type option box

Media Type option box Select the media type to publish. Video Only is enabled when you publish a flattened clip.

Output Media Group

Use the Output Media options to control the format of the published media.



(a) Audio File Format option box (b) Image File Format option box (c) Audio Sample Bit Depth option box (d) File Name Padding field

Image File Format option box Select the type of video file to publish.

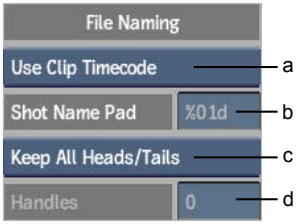
File Name Padding field Edit the padding format string that adds leading zeros before the number in the published file name. Use file name padding to ensure the images are listed and stored in the correct order. The default padding format scheme is %04d, where the "0" indicates that leading zeros are added to the filename for each frame when there are fewer than four digits in the sequence number. The "4d" indicates that four digits are maintained. Choose a value according to the duration of your clips before publishing.

Audio File Format option box Select a format for the published audio file. The Audio File Format box is enabled when Media Type is set to Video/Audio or Audio Only.

Audio Sample Bit Depth option box Select an audio sample bit depth. The Audio Sample Bit Depth box is enabled when Media Type is set to Video/Audio or Audio Only.

File Naming Group

Use the File Naming options to control the naming of published media.



(a) File Name option box (b) Shot Name Padding field (c) Handles option box (d) Handles field

File Name option box Select how the name of the published clip is created.

Select:	To:
Use None	Use only a number to name the published files.
Use Both	Use timecode, clip name, and shot name to name the published files.
Use Clip Timecode	Use only the timecode's corresponding frame count to name the published files.
Use Clip Name	Use the name of the clip and the shot to name the published files.

Shot Name Padding field Edit the padding format string that adds leading zeros before the shot number in the published filename. The default padding format scheme is %01d, where the "0" indicates that leading zeros are added to the filename for each frame, and the "1d" indicates that one digit is maintained. You can add the shot name to the published file when you have a Complex or Flatten Publish Type and you select Clip Name or Both from the File Name option box.

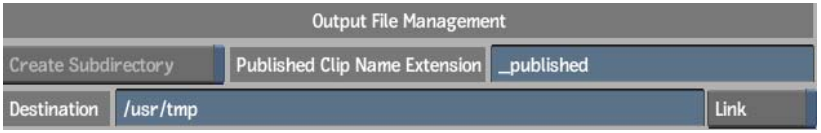
Handles option box Select an option for handling head and tail frames.

Select:	To:
Keep All Heads/Tails	Include all heads and tails of every element in the published clip.
Consolidate	Limit heads and tails to a specific number of frames. When selected, the Handles field becomes active.
No Heads/Tails	Include no heads or tails in the published clip.

Handles field Enter a value for the number of head and tail frames added to the published clip. The Handles field is active when you select Consolidate from the Handles option box.

Output File Management Group

Use the Output File Management options to control the destination of published media.



Create Subdirectory button Enable to create subdirectories in the following format: */basedirectory/clipname/tapename/resolution*. The Create Subdirectory button is available when Publish Type is set to Complex or Flatten.

Published Clip Name Extension field The value in this field is appended to the name of the published clip in the library. The default is *_published*.

Destination field Enter the destination path of the file, which is usually the location of the shared storage. By default, the destination path is */usr/tmp*.

Link button Enable to link the published clips to the original file, located on the same filesystem. A link at the filesystem level is created to the media

already on the destination filesystem, which saves space because the media is not duplicated.

When you publish media from one filesystem to another, the media is copied even if the Link option is enabled. If you are publishing some media from the same filesystem and some from a different filesystem with the Link option enabled, any media that is on the same filesystem is linked to the original media. Media that is on a different filesystem is duplicated.

Naming Published Files

The Publish Type and File Naming options you select determine how exported files are named.

The following tables show how a published clip is named based on the options you select. The example is based on a simple, single-layer, 2-shot timeline.

Simple Publish with File Naming Option:	**Generated File Name:**
Use None	<nnnn>.dpx
Use Both	<clipname>.<timecode>.dpx
Use Clip Timecode	<timecode>.dpx
Use Clip Name	<clipname>.<nnnn>.dpx

Complex Publish with File Naming Option:	**Generated File Name:**
Use Both	<clipname>_shot<0001,0002>.<shot_timecode>s<1,2>.dpx
Use Clip Timecode	<timecode>s<1,2>.dpx
Use Clip Name	<clipname>_s<0001,0002>.nnnn.dpx

Flatten Publish with File Naming Option:	**Generated File Name:**
Use Both	<clipname>_shot<0001,0002>.<shot_timecode>s<1,2>.dpx
Use Clip Timecode	<timecode>s<1,2>.dpx

Flatten Publish with File Naming Option:	Generated File Name:
Use Clip Name	<clipname>_s<0001,0002>.<nnnn>.dpx

Publishing a Clip Using the Publish Tool

Publish a clip so that you can collaborate with others on the same media.

When you publish a clip, the media is written to the destination on the shared storage. Another clip is generated that soft-imports the newly created media on the shared storage.

You see changes in the published clip when other users in other applications change the published media. For example, if a user uses a paint program to modify the content of one of the files in the shared storage and replaces the file with the modifications, you see these changes in your clip. If you are using proxies, regenerate the proxies to see the changes made in other applications.

If you change the clip in Inferno, users outside of Inferno are not be able to see these changes. Republish the clip for users to see changes you make.

NOTE If published media contains source files that are deleted or moved, the frames of the missing files are replaced with a checkerboard.

To publish a clip:

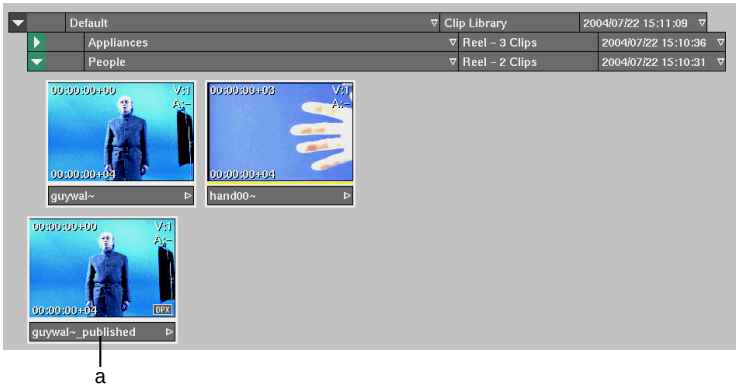
1. Access the Publish menu from the clip library. See Accessing the Publish Menu on page 47.
2. In the clip library, select the clips that you want to publish.
3. From the Publish Type box, select the publish type.
4. Set any other Publish options using the corresponding controls. See About the Publish Menu Options on page 48.
5. In the Destination field, enter the location for the published files.
6. From the Destination Library box, select a library for the published clip.

NOTE If your clips are not rendered, do not press **Alt**+**Tab** after you click Publish. Your application uses the output of your graphics card to render the final image file. If there are any other application windows on top of your Inferno window, undesired artefacts may appear in your rendered image files.

7 Click Publish.

A new clip is published in the clip library, which references the published image sequence. This clip is also identified by a soft-imported icon in the lower-right corner, indicating that its media resides outside the framestore.

Each frame of the clip is published to the destination directory. See Naming Published Files on page 52.



(a) Published clip

Images courtesy of Behavior Communications Inc.

NOTE A published clip does not have history.

Clip Output Using Real-Time Deliverables

5

Topics in this chapter:

About Real-Time Deliverables

The Real-Time Deliverables mastering solution allows you to perform certain types of operations on material during clip output without having to first render the clips. For example, you can output from a master source format to many output formats directly to tape without rendering. Using the concept of the Deliverable—a virtual clip containing real-time operations and linked to its original source clip—you can create and manage multiple real-time output formats from one clip.

Supported operations include:

- Resize
- Pan and Scan
- Application of 3D and 1D LUTs
- Framerate change with deceleration or acceleration with 2:3 insertion
- Audio timewarp
- Letterbox overlay

The Real-Time Deliverables feature supports source material up to 2K (10-bit) depending on the hardware setup. Good performance may be achieved beyond 2K (10-bit), but is not guaranteed. Soft effects are not supported for real-time output, unless processed before output. Embedded audio is also not supported.

Real-Time Deliverables sections explain important concepts and workflows for how to manage, preview, and output real-time operations, and references other sections of the user guide for more details, where appropriate.

Make sure to follow the hardware requirements described in Hardware Requirements for Real-Time Deliverables on page 57.

What Is a Deliverable?

The settings created in the Deliverables menu are saved as a Deliverable, which is central to the Real-Time Deliverables workflow.

A Deliverable is a virtual clip that is associated with the source clip from that point on, in the library or on the Desktop. A clip can have multiple Deliverables, and each Deliverable that you create is itself treated like a clip—a modified version of the original clip. As with clips, you can enter the Output Clip menu and output a Deliverable to tape. See Managing Deliverables on page 66.

Hardware Requirements for Real-Time Deliverables

To follow the Real-Time Deliverables procedures, you must be using an NVIDIA Quadro FX 5600 SDI graphics card. The configuration for using this card is described in the following documents:

- *Autodesk Visual Effects, Finishing, and Colour Grading HP xw8600 Workstation Hardware Setup Guide*
- *Autodesk Visual Effects and Finishing 2009 HP xw9400 Workstation Hardware Setup Guide*

When installing this release of the application for use with Real-Time Deliverables, it is important to use the settings in the new version of the *init.cfg* file. These settings specify the new hardware to be used as your preview device. For example:

```
VideoPreviewDevice 1280x720@599 nvidia, 1, 1280, 720, 5994p, DTV
sync
```

If there are settings in your previous *init.cfg* file that you want to keep, port them to the new version during installation.

To use Real-Time Deliverables, make sure the FX 5600 SDI card is selected in the Output Clip menu.

To select the FX 5600 SDI in the Output Clip menu:

1. Open the Output Clip menu as described in the Outputting Clips To a VTR chapter.
2. From the Graphics Card box, select GFX SDI.

When GFX SDI is selected, the FX 5600 SDI is used for video output.

When AJA_OEM2K is selected, the AJA OEM-2K is used for video output. The AJA OEM-2K does not support Real-Time Deliverables, and the corresponding options are not displayed when it is selected.

The AJA OEM-2K is always used for clip input, and the FX 5600 SDI (if it is installed) is always used for broadcast monitor display.

GFX SDI is not an available option in the Graphics Card box if:

- The FX 5600 SDI is not installed.
- The FX 5600 SDI is not set up or configured properly.

To install or configure an FX 5600 SDI, see one of the hardware guides previously cited in this topic.

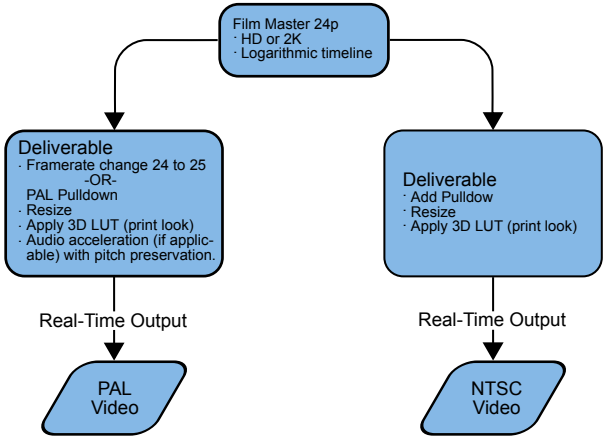
Real-Time Deliverables Use Cases

This section provides two examples of how Real-Time Deliverables can be applied.

Film Master 24p, HD, or 2K Logarithmic Timeline, to Video

An important case addressed by Real-Time Deliverables is when an organization wants to take a 24p film master with an HD or 2K logarithmic timeline, and create real-time output to PAL and NTSC video format.

Use Case 1
Film master 24p, HD or 2K logarithmic timeline, to Video



To output to PAL video, you would create a Deliverable that does the following:

- Increases the framerate from 24 to 25, or uses PAL Pulldown
- Resizes the clip
- Applies a print-look 3D LUT
- Accelerates the audio while preserving the pitch, if necessary

To output to NTSC video, you would create a Deliverable that does the following:

- Adds 2:3 pulldown
- Resizes the clip
- Applies a print-look 3D LUT

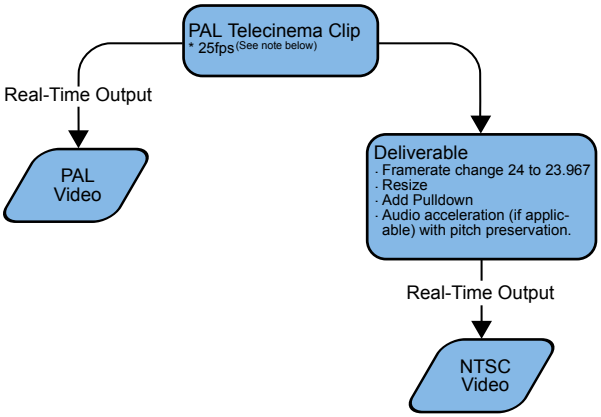
From the Clip Output menu, the clip is output to tape with no need to render the preceding settings.

PAL Telecinema to Video

In this example, NTSC and PAL video are output, but the source material is a PAL Telecinema clip. Because a PAL Telecinema clip is not field-based, the

resulting video can be treated as progressive. Its video fields are spatially complementary.

Use Case 2 - PAL Telecinema to Video



Note: A PAL Telecinema clip is not field-based, so the resulting video can be treated as progressive. Video fields are spacially complimentary (correlated).

To output to PAL video, you would not need to create a Deliverable. You can output the clip directly to tape from the Output Clip menu.

To output to NTSC video, you would create a Deliverable that does the following:

- Decreases the framerate from 25 to 23.967
- Resizes the clip
- Adds 2:3 pulldown
- Accelerates the audio while preserving the pitch, if necessary

From the Clip Output menu, the clip is output to tape with no need to render the preceding settings.

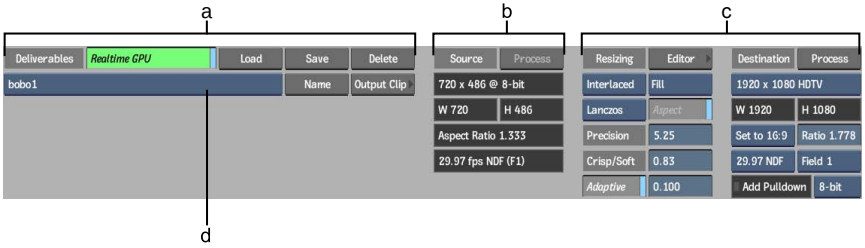
Accessing the Deliverables Menu

Use the Deliverables menu to manage Real-Time Deliverables, and to configure real-time operations such as resize, pulldown, and so on. Many of these settings can also be altered in the Output Clip menu.

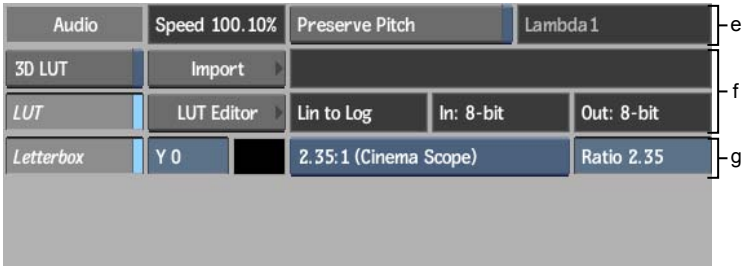
To access the Deliverables menu:

1. Select the clip you want to work on and open the Player.
2. Click the Deliverables tab.

The Deliverables menu appears (broken into two parts here).



(a) Deliverable controls (b) Source clip information (c) Resize and framerate controls (d) Deliverable box



(e) Audio controls (f) LUT controls (g) Letterbox controls

Most of the controls in the Deliverables menu are common to various parts of the software, and their functions are explained in their respective sections. The following does not describe how to use the controls, but rather how to manage the results using Deliverables and output them to tape in real time.

The settings you create in the Deliverables menu can be previewed in the Player and are applied in real time during clip output.

To configure real-time operations, see Managing Deliverables on page 66.

NOTE Settings in the Deliverables menu override clip settings from other modules. These settings are overridden, not discarded. For example, if a Deliverable has LUTs set, the LUTs from the source clip are not used.

Source clip information Displays information about the source clip before real-time operations are applied.

Resize controls Displays values set in the Resize editor, accessible with the Editor button. See the Resize and 2:3 Pulldown chapters in your user guide. You can edit the values directly in the Deliverables menu. The available destination resolutions and timings depend on the format of the source material.

Audio controls Displays the audio speed. To preserve audio pitch, enable Preserve Pitch. See the Audio chapter in your user guide.

LUT controls (3D and 1D) Enables or disables the use of LUTs. You can specify a 3D LUT to use, and access the LUT editor. See the Colour Management with LUTs chapter in your user guide.

Letterbox controls Enables or disables a letterbox overlay. See Outputting Clips to a VTR chapter in your user guide.

Previewing and Processing Material for Real-Time Output

Preview Deliverables to verify your settings. If necessary, process source clips or Deliverables before outputting in real time.

Previewing Deliverables

The real-time operations you create in the Deliverables menu can be previewed in the Player.

To preview the result of a Deliverable in the Player:

1 From the Playback Resolution box, select the Deliverable option.

With this option selected, your real-time settings are displayed during playback or when you jog or scrub through the clip.

You can also preview real-time operations in the Clip Output menu. See Outputting Deliverables on page 64.

Processing Source Clips and Deliverables

The purpose of Real-Time Deliverables is to be able to output material without having to process first. However, there are exceptions where, before outputting, you must process:

- Source material that has an unrendered soft effect applied to it.
- A Deliverable whose settings place unusually high demands on hardware and cannot be output in real time.

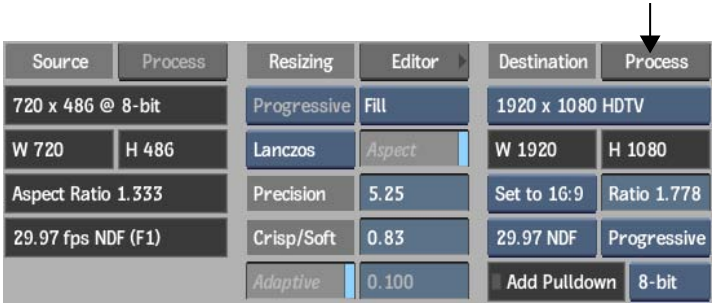
If you open the Deliverables menu with a clip that has an unprocessed soft effect applied to it, you can configure Deliverable settings for it, but you cannot output the Deliverable until the effect is processed. To do so, click Process in the source clip information section of the Deliverables menu.

Clicking Process here is the same as clicking Timeline Process. See Processing Soft Effects and Batch FX in the Processing and Finishing chapter.

Once you set up your Deliverable, it is possible that your hardware cannot output the Deliverable to tape in real time. The status of the GPU Benchmark button helps you decide whether to process the Deliverable:

Status:	Processing status:
Green	You can output the Deliverable in real time, without firs rendering.
Yel-low	You should process the Deliverable before going to Output Clip, as real-time output might not be possible.
Red	Real-time output is not possible: you must process the Deliverable before going to Output Clip.

If the GPU Benchmark button indicates that real-time output is impossible, click Process in the resize and framerate group of the Deliverables menu.



A new clip is created with "_<Deliverable Name>" appended to the clip name. Any settings from the Deliverables menu are burned into the clip. Because it is a new clip, it does not have any associated Deliverables associated. Enter the Output Clip menu with the clip and output it to tape.

Outputting Deliverables

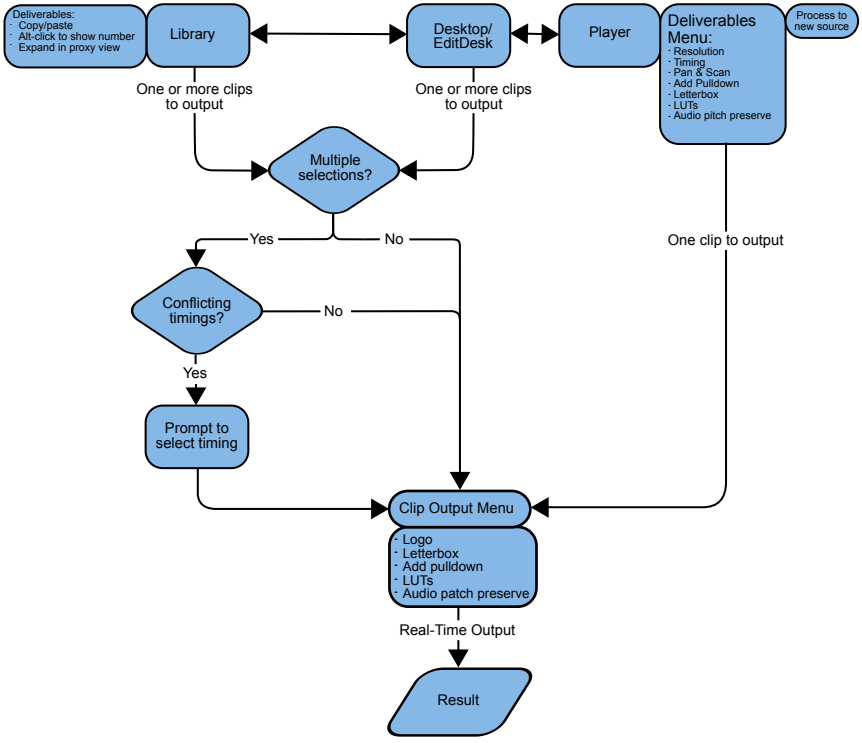
The workflow for outputting Deliverables is like the standard clip output workflow described in the Outputting Clips to a VTR section of the Clip Input/Output Using a VTR chapter.

Each Deliverable that you create is treated like a clip. You can enter the Output Clip menu from the library with a Deliverable and output it to tape as you would any clip.

From the Deliverables menu, click the Output Clip button to enter the Output Clip menu with the current Deliverable.

If you enter the Output Clip menu from the library with multiple items (clips, Deliverables, or both), the resulting behaviour is the same as when outputting multiple clips. If there are conflicting timings, you are prompted to select the timing you want to use, and items that do not match this timing are discarded. See Outputting Clips to a VTR in the Clip Input/Output Using a VTR chapter.

If you select a clip in the library to bring it into the Output Clip menu, all its Deliverables are also selected. If there are conflicting timings, you are prompted to select the timing to use. This can result in the exclusion of the original clip.



Once in the Output Clip menu, you can alter Letterbox and LUT settings in the Deliverables tab, as well as preserve audio pitch. You can also add a logo in the Logo tab, although it does not require a Deliverable.

These changes are saved to the Deliverable. The next time you work with that Deliverable in the library or Player, the changes you made in the Output Clip menu are loaded.

NOTE When you select the Deliverables tab, you do not see the VTR feedback; instead you see the clip selected in the Output Clip list.

To create real-time settings for a clip and output to tape in real time:

1. From the Player, create a Deliverable. See Accessing the Deliverables Menu on page 61.
2. Do one of the following:
   - From the Deliverables menu, access the Output Clip menu
   - From the library or Desktop, select the Deliverable or multiple clips and Deliverables, and access the Output Clip menu. If you selected multiple items and they have conflicting timings, you are prompted to select which timing you want to use.
3. In the Output Clip menu, make adjustments to real-time settings if necessary.
4. To preview real-time operations before outputting, use the Player controls in the Deliverables or Logo tabs. (In the Output tab, the Player controls display VTR material.)
5. Output the material to tape as described in the section Outputting a Single Clip and the section Outputting Multiple Clips in the Clip Input/Output Using a VTR chapter.

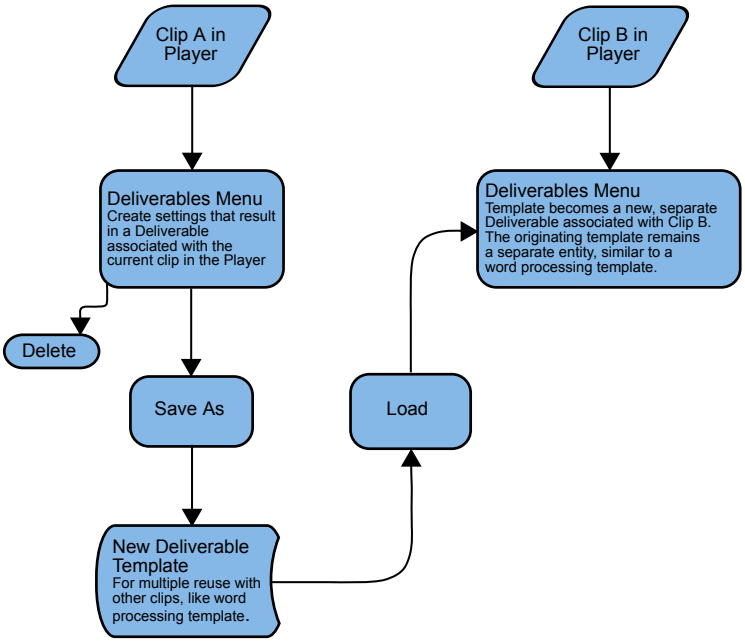
Managing Deliverables

A Deliverable results when you create real-time settings for a clip, in the Deliverables menu. Deliverables are virtual clips associated with the original clip. You can:

- Rename or delete a Deliverable.
- Create multiple Deliverables for one clip, so that you can output many different formats of that clip.
- Save a Deliverable as a reusable Deliverable template.
- Load a Deliverable template onto another clip to apply the settings of that Deliverable to the clip.
- View information about Deliverables in the Desktop and library.
- Copy and paste deliverables between clips in the library and on the Desktop.
- Transfer clips and their Deliverables to other systems.

NOTE Any change to the framerate (timing) or resolution of the clip deletes all Deliverables attached to that clip.

The following illustrates how Deliverables can be used.



In the illustration, the Deliverable for Clip A is saved as a Deliverable template. Think of this template as an independent, reusable instance of the real-time settings that were created for Clip A (like how a word-processing template stores styles and formats).

When the template is applied to Clip B, it becomes a new Deliverable for clip B. This new Deliverable has the same real-time settings as Clip A, until you start changing it. For this operation to work, Clip A and B must have the same resolution, framerate, and aspect ratio.

Creating and Modifying Deliverables

From the Deliverables menu, you can create, rename, and delete Deliverables.

To create a Deliverable:

1. Access the Deliverables menu, as described in Accessing the Deliverables Menu on page 61.
2. From the Deliverable box, select <New Deliverable>.
3. Type a name for the Deliverable and press **Enter**.

Once the Deliverable is created, set up real-time operations in the Deliverables menu. The operations you set up are automatically stored in the Deliverable you created.

You can create multiple Deliverables for a clip. If you create another Deliverable:

- The settings of the previous Deliverable are cleared from the Deliverables menu.
- The new Deliverable name is appended to the list in the Deliverables box.

To rename a Deliverable:

1. With the Deliverable loaded in the Deliverables menu, click Name.
2. Type a new name for the Deliverable and press **Enter**.

To delete a Deliverable:

1. With the Deliverable loaded in the Deliverables menu, click Delete.
2. Click Confirm.

The Deliverable is deleted, and the previously-loaded Deliverable (if there was one) is reloaded.

Exporting and Loading Deliverables as Templates

Use the Save and Load buttons in the Deliverables menu to export a Deliverable as a template and load it on another clip.

You can only load a Deliverable on clips that have identical resolutions and timings with the original clip.

When you load a Deliverable template:

- Its settings are loaded to the Deliverables menu.
- A new Deliverable is appended to the list in the Deliverables box.

Viewing Deliverables in the Desktop and Library

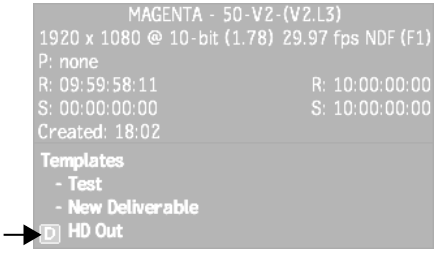
Clips with Deliverables are indicated by a “D” overlay on their proxies.

There are two types of “D” overlays, each indicating the selection in the Playback Resolution box:

- **Grey “D”** The Player displays the Deliverable. See Previewing Deliverables on page 62.
- **Black “D”** The Player displays Proxies or Full Res.

To open the Deliverables menu from the clip proxy on the Desktop, double-click the “D” overlay.

To see quickly a list of Deliverables for a clip, **Alt**-click the clip proxy on the Desktop.



The list of Deliverables is included in the clip information. A “D” appears beside the active Deliverable (the Deliverable most recently selected in the Deliverables menu).

In the library, to expand Deliverable proxies for a clip, do one of the following:

- Double-click the “D” overlay.
- Click Deliverables at the upper right of the screen, and select Show Deliverables.

In the library, Deliverable proxies are black and include the Deliverable name. Playing the Deliverable gives you the actual deliverable result.

Copying Deliverables from the Desktop

Copy and paste Deliverables from one clip to another. If a clip has multiple Deliverables, they are all copied. The source and target clips must have the same framerate, resolution, and aspect ratio.

To copy Deliverables from one clip to another from the Desktop:

1. On the Desktop, click Copy and select Deliverables.
2. Click the clip whose Deliverables you want to copy.
3. Optional: If the clip has more then one Deliverable attached, select the one to copy or enable All Deliverables to copy all attached deliverables.
4. Click the clip where you want to paste the Deliverables.

Copying and Loading Deliverables from the Library

You can copy, load, save, and delete Deliverables from clips in the library. This is useful for rapidly assigning and removing Deliverables just before loading them to the Desktop, or before accessing Output Clip.

To manage deliverables in the clip library:

1. Click Tools to display the Tools menu, and then select Deliverables.
2. In the clip library, select either a clip or a Deliverable.

3 Select an operation to perform.

Select:	To (with a clip selected):	To (with a Deliverable selected):
Load	Load a Deliverable to attach it to the selected clip.	N/A
Save	Save the Deliverable attached to the clip. If there is more than one Deliverable attached, select the one to save in the Deliverables selection box.	Save the selected Deliverable.
Delete	Delete the Deliverable attached to the clip. If there is more than one Deliverable attached, select the one to delete in the Deliverables selection box.	Delete the selected Deliverable.
Copy	Copy the Deliverable attached to the clip. If there is more than one Deliverable attached, select the one to copy in the Deliverables selection box.	Copy the selected Deliverable.
Paste	Paste the copied Deliverable to the selected clip.	N/A

Searching for Clips with Deliverables

You can search for clips with Deliverables on the Desktop or in the clip library. The Search tool provides an easy way to find and manage Deliverables.

To search for clips with deliverables on the Desktop:

1 Click Search.

The Search controls appear.

(a) Search Mode box (b) Match Criteria box (c) Deliverables Search options (d) Range box

2 In the Search Mode box, select Clip.

3 In the Match Criteria box, select Deliverables.

4 In the Deliverables Search option box, select one of the following.

Select:	To:
Contains	Search for clips that contain deliverables.
Does Not Contain	Search for clips that do not contain deliverables.

5 In the Range box, select Desktop or Library.

6 Click Search.

A message appears in the message bar indicating if any matches were found.

7 If there are matches, click a destination reel.

The matched clips are moved (if found on the Desktop), or loaded (if found in the clip library) to the destination reel.

To search for clips with deliverables in the clip library:

1 Cick Search.

The Search menu appears.

(a) Search option box (b) Argument option box

2 Select Deliverables from the Search option box.

3 Use the Argument option box and field to supply the necessary arguments. Select one of the following options.

Select:	To:
Contains	Search for clips that contain deliverables.
Does Not Contain	Search for clips that do not contain deliverables.

4 Click Search.

A message appears in the message bar indicating if any matches were found and the matches are displayed in the library.

Transferring Deliverables to Other Systems

When a clip is transferred to another system using Wire, its Deliverables are moved as well.

If the other system does not have the hardware needed for Real-Time Deliverables, users of that system will see that the clip has Deliverables, but will not be able to edit or preview them. The Deliverables will be greyed out in the Output Clip menu.

Users on such systems will be able to work with the original clip and transfer it back to the original system, with its Deliverables intact. Any change to the framerate (timing) or resolution of the clip deletes all Deliverables attached to that clip.

You cannot export Deliverables to other file formats. You cannot use Deliverables with EDLs.

Real-Time Deliverables Timing Specifications

When you set up a Deliverable to modify the timing of a clip during real-time output, the timing is changed according to the following tables.

NOTE In the following tables, N/A stands for *not applicable* and NC stands for *no change in timing*.

Destination timings 23.967psf through 25p:

Source	Destination							
	23.967psf	**23.967p**	**24psf**	**24p**	**PAL**	**50i**	**25psf**	**25p**
23.967psf	N/A	NC	Speed up	Speed up	Speed up to 25 or speed up to 24 + PAL Pulldown	Speed up	Speed up	Speed up
23.967p	NC	N/A	Speed up	Speed up	Speed up to 25 or speed up to 24 + PAL Pulldown	Speed up	Speed up	Speed up
24psf	Slow down	Slow down	N/A	NC	Speed up to 25 or apply PAL Pulldown	Speed up	Speed up	Speed up
24p	Slow down	Slow down	NC	N/A	Speed up to 25 or apply PAL Pulldown	Speed up	Speed up	Speed up
PAL	Slow down	Slow down	Slow down	Slow down	N/A	NC	NC	NC
50i	Slow down	Slow down	Slow down	Slow down	NC	N/A	NC	NC
25psf	Slow down	Slow down	Slow down	Slow down	NC	NC	N/A	NC
25p	Slow down	Slow down	Slow down	Slow down	NC	NC	NC	N/A

NOTE When going from 23.967 to PAL, you can choose between increasing the framerate to 25 or increasing to 24 and adding PAL Pulldown. When going from 24 to PAL, you can choose between increasing the framerate to 25 or adding PAL Pulldown.

Destination timings NTSC through 60p:

Source	Destination						
	NTSC	**29.97p**	**30 psf**	**30p**	**60i**	**59.94p**	**60p**
23.967psf	Add Pulldown	Not supported	Not supported	Not supported	Speed up to 24 + Add Pulldown	Not supported	Not supported

Source					Destination		
	NTSC	**29.97p**	**30 psf**	**30p**	**60i**	**59.94p**	**60p**
23.967p	Add Pull-down	Not supported	Not supported	Not supported	Speed up to 24 + Add Pull-down	Not supported	Not supported
24psf	Slow Down to 23.976 + Add Pull-down	Not supported	Not supported	Not supported	Add Pulldown	Not supported	Not supported
24p	Slow Down to 23.976 + Add Pull-down	Not supported	Not supported	Not supported	Add Pulldown	Not supported	Not supported
PAL	Slow Down to 23.976 + Add Pull-down	Not supported	Not supported	Not supported	Slow Down to 24 + Add Pull-down	Not supported	Not supported
50i	Slow Down to 23.976 + Add Pull-down	Not supported	Not supported	Not supported	Slow Down to 24 + Add Pull-down	Not supported	Not supported
25psf	Slow Down to 23.976 + Add Pull-down	Not supported	Not supported	Not supported	Slow Down to 24 + Add Pull-down	Not supported	Not supported
25p	Slow Down to 23.976 + Add Pull-down	Not supported	Not supported	Not supported	Slow Down to 24 + Add Pull-down	Not supported	Not supported
NTSC	N/A	NC	Speed up	Speed up	Speed up	Not supported	Not supported
29.97p	NC	N/A	Speed up	Speed up	Speed up	Not supported	Not supported

Source	Destination						
	NTSC	***29.97p***	**30 psf**	**30p**	**60i**	**59.94p**	**60p**
30 psf	Slow down	Slow down	N/A	NC	NC	Not supported	Not supported
30p	Slow down	Slow down	NC	N/A	NC	Not supported	Not supported
60i	Slow down	Slow down	NC	Field merge	N/A	Not supported	Not supported
59.94p	Not supported	Not supported	Not supported	Not supported	Not supported	N/A	Speed up
60p	Not supported	Not supported	Not supported	Not supported	Not supported	Slow down	N/A

A destination of 50p is not supported.

Clip Workflow Improvements

6

Topics in this chapter:

Resize Settings

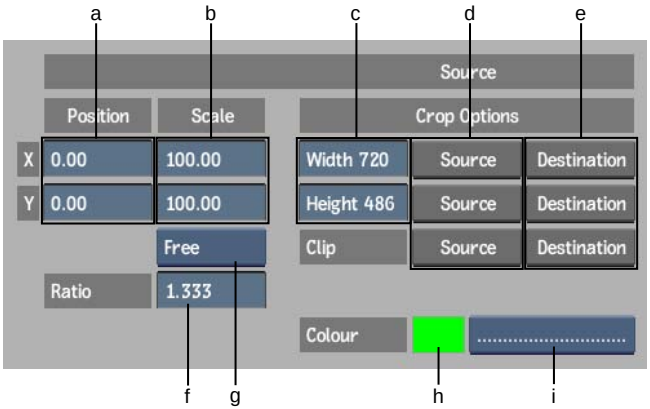
New for this release, the arrangement of the Resize parameters has been improved to allow for more consistency and ease of use, including the addition of some new options. The new options include improved and expanded resize filter options, and a crop box that is now handled as a single object (rather than four lines) with parameters for horizontal and vertical positioning and scaling.

The following sections show the resize source and destination settings available in the Resize module or in functions that support Resize.

The Source, Resizing, and Destination settings are available in all resize procedures; however, the layout of these controls may change and some options are not available in some cases.

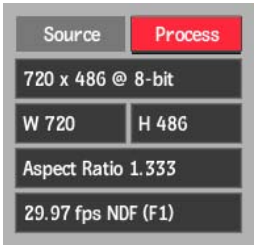
Source Settings

Use the Source settings to set or animate the position and size of the crop box.



(a) X and Y Position fields (b) X and Y Scale fields (c) Crop Box Width/Height fields (d) Source Frame Crop Box buttons (e) Destination Frame Crop Box buttons (f) Source Ratio field (g) Crop Box Mode box (h) Crop Box colour pot (i) Crop Box Line Type box

The Source settings displayed in the Deliverables tab are not editable, since the crop box is not available. In this case, use the Resize button to access the full set of resize settings.



X Position field Displays the horizontal position from the centre of the crop box relative to the centre of the source frame. Drag left or right, or click to enter a new X Position value.

Y Position field Displays the vertical position from the centre of the crop box relative to the centre of the source frame. Drag left or right, or click to enter a new Y Position value.

X Scale field Displays the horizontal scale of the crop box relative to the Crop Box Width field value, as a percentage. Drag left or right, or click to enter a new X Scale value.

Y Scale field Displays the vertical scale of the crop box relative to the Crop Box Height field value, as a percentage. Drag left or right, or click to enter a new Y Scale value.

Crop Box Width field Displays the currently set width of the crop box, in pixels. Drag left or right, or click to enter a new width value.

Crop Box Height field Displays the currently set height of the crop box, in pixels. Drag left or right, or click to enter a new height value.

Source buttons Click to fit the crop box width, height, or both to the source frame.

Destination buttons Click to fit the crop box width, height, or both to the destination frame.

Crop Mode box Select a mode to determine the scaling behaviour of the crop box while repositioning or rescaling.

Select:	To:
Free	Adjust the crop box freely, with no constraints on position or scale.
Prop	Use the current Crop Box Width and Crop Box Height settings. When adjusting the crop box, these proportions are maintained.
Source	Use the source clip aspect ratio for the crop box. When adjusting the crop box, this aspect ratio is maintained.
Destination	Use the destination clip aspect ratio for the crop box. When adjusting the crop box, this aspect ratio is maintained.

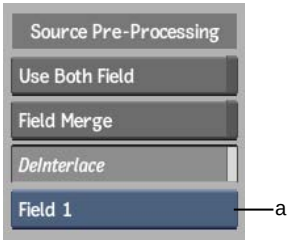
Source Ratio box Displays the aspect ratio of the crop box in the source frame. Editable.

Crop Box Border colour pot Displays the current colour of the crop box border. Editable.

Crop Box Line Style box Select the line style for the crop box.

Source Pre-Processing

Use the Source Pre-Processing settings to determine whether resize is performed on full progressive frames, or on either (or both) of the interlaced fields.



(a) Source Fields Toggle button

Use Both Field button Enable to use both fields of an interlaced clip to process the resizing. Disabled when Field Merge or Deinterlace are enabled.

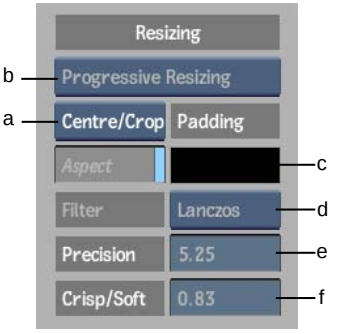
Field Merge button Enable to merge the two fields of an interlaced clip to process the resizing. Disabled when Use Both Field or Deinterlace are enabled.

Deinterlace button Enable to select one of the two fields in an interlaced clip to process the resizing. Disabled when Use Both Field or Field Merge are enabled.

Source Fields Toggle button Select Field 1 or Field 2 when processing the resizing on only one field of an interlaced clip. Active when Deinterlace is enabled.

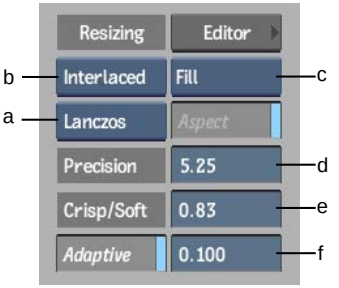
Resizing Settings

The following settings are displayed for resizing in Batch, the Resize module.



(a) Fit Method box (b) Field Format box (c) Padding colour pot (d) Filter box (e) Precision field (f) Crisp/Soft field

The following settings are displayed for Resizing in Real-Time Deliverables.



(a) Filter box (b) Field Format box (c) Fit Method box (d) Precision field (e) Crisp/Soft field (f) Adaptive field

Fit Method box Select a fit method to be applied to the selected clip.

Select:	To fit:
Centre/Crop	The source image, centred, over the destination resolution frame. If the source is larger than the destination, it is cropped. If the source is smaller than the destination, it is surrounded by a black border.
Crop Edges	One edge of the source into the destination resolution frame without stretching or squashing the frame. Excess parts of the source frame after resizing are cropped. If the source—after the one edge is resized—is wider than the destination, its overhanging left and right edges are cropped. If the source is taller than the destination, the upper and lower edges are cropped.
Fill	The source, width, and height, into the destination resolution frame. This process, if the source and destination resolutions do not have the same aspect ratio, can distort the image.
Letterbox	The source to the destination resolution frame without squashing or stretching it, and without cropping the source. If the source is wider than the destination, black bars fill the top and bottom of the destination frame. If the source is narrower than the destination, black bars fill the right and left sides of the frame. In all cases, the entire source frame is fit into the destination frame.

Filter box Provides eight resizing filters. This box does not appear if you selected the Centre/Crop Fit method.

Select:	To get:
Impulse	Quick, low-quality results.

Select:	To get:
Triangle	Moderate results with little processing overhead.
Mitchell	Best results when resizing a clip to a higher resolution.
Bicubic	Very good results for resizing soft-looking images. Use to sharpen the image.
Quadratic	Good results for resizing simple images with straight edges. Similar to Gaussian but with more blurring. Use to soften the image.
Gaussian	Excellent results when resizing a clip with no patterns and a lot of straight edges to a lower resolution. Useful for softening some detail.
Shannon	Excellent results when resizing a clip to a lower resolution. Very similar to Lanczos, but results are a little softer.
Lanczos	Best results when resizing a clip containing a variety of patterns and elements to a lower resolution. It is the most complex with the longest processing time.

NOTE Defining a filter method in a Resize menu will override the default setting specified in the Preferences menu.

Keep Aspect button Enable to maintain non-square pixel formats. This button only appears if you selected the Crop Edges or Letterbox methods.

Padding colour pot Select a colour to use for padding out the frame when centre-cropping, cropping edges, or letterboxing. The default padding colour is black.

Adaptive button Enable to use adaptive de-interlacing to minimize the creation of artifacts associated with resizing interlaced material.

Adaptive field Displays the Adaptive de-interlacing setting.

Destination Settings

These settings define the destination format into which the source sample is fit.

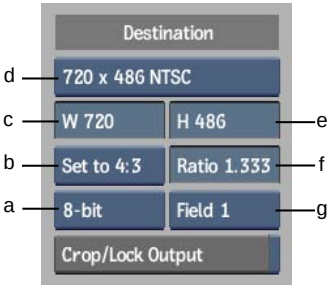
The following settings are displayed for resizing in the Resize module.

(a) Frame Depth box (b) Aspect Ratio Presets box (c) Frame Width field (d) Resolution Presets box (e) Frame Height field (f) Aspect Ratio field (g) Scan Mode box

The following settings are displayed for resizing in the Batch FX Output node and in the Output Batch node.

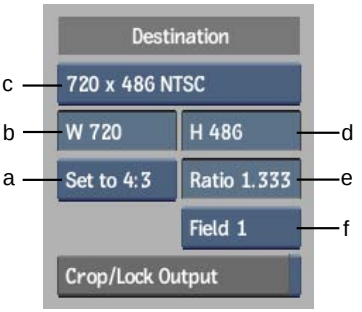
(a) Aspect Ratio Presets box (b) Frame Width field (c) Resolution Presets box (d) Frame Height field (e) Aspect Ratio field (f) Scan Mode box

The following settings are displayed for resizing in the Resize Batch node.



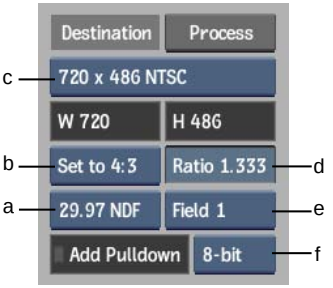
(a) Frame Depth box (b) Aspect Ratio Presets box (c) Frame Width field (d) Resolution Presets box (e) Frame Height field (f) Aspect Ratio field (g) Scan Mode box

The following settings are displayed for resizing from clips in Batch or in the Export Batch node.



(a) Aspect Ratio Presets box (b) Frame Width field (c) Resolution Presets box (d) Frame Height field (e) Aspect Ratio field (f) Scan Mode box

The following settings are displayed for resizing in Real Time Deliverables.



(a) Frame Rate box (b) Aspect Ratio Presets box (c) Resolution Presets box (d) Aspect Ratio field (e) Scan Mode box (f) Frame Depth box

The following settings are displayed for resizing in soft effects.

Fit Crop Resolution button Adjusts the destination resolution to fit the current crop box.

Crop/Lock Output button Enable to have the destination resolution per frame match the crop box. Use this option to animate the resolution of a clip. This option is only available when you access Resize settings from Batch nodes.

Frame Depth box Provides five frame depth options.

Resolution Presets box Provides options for many standard resolutions, as well as a Custom option that you can use to specify non-standard resolutions.

Width/Height fields Display the frame width and height of the selected resolution preset. If you select Custom from the Resolution Presets box, use these fields to enter the frame width and height values you want to use.

Aspect Ratio Presets box Provides standard frame aspect ratio options as well as a w:h option to set the clip to use square pixels. Also provides a Custom option so you can enter a custom frame aspect ratio in the Ratio field.

Scan Mode box Select an option to set the order in which the fields of interlaced material are scanned.

Select:	To resize:
Progressive	A frame-based clip to another frame-based clip.
Field 1	A clip by drawing Field 1 followed by Field 2.
Field 2	A clip by drawing Field 2 followed by Field 1.

For interlaced material, you can specify whether the resize needs to be done from both fields or just from one of the two. In the latter case, the result is a progressive clip made from the same two fields.

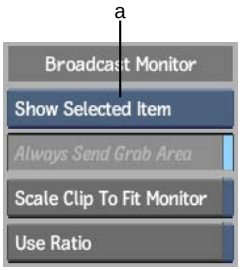
Ratio field Enter a custom frame aspect ratio in this field, if necessary.

Broadcast Settings Preferences

Broadcast monitor settings can be applied globally for Inferno in the Player and modules that support multiple viewports. Settings can be changed on-the-fly while you are working. These preferences are stored in the project directory.

Broadcast Monitor Preferences

The broadcast monitor outputs a complete image or selected viewport. View output in different contexts, to suit multiple types of workflow.



(a) Broadcast Monitor box

Broadcast Monitor box Select what to preview on your broadcast monitor.

Select:	To display:
Show Selected Item	The proxy or full-resolution version of the selected clip.
Screen Grab	The contents of a selected image viewport or the Player.
Off	Nothing on the broadcast monitor.

Always Send to Grab Area button Enable to display the Player or viewport. This button is active when the Screen Grab option is selected.

Scale Clip To Fit Monitor button Enable to resize the clip to fit the broadcast monitor. This button is active when the Show Selected Item option is selected.

Use Ratio button Enable to maintain the original proportions of the clip. Disable to use the monitor ratio. This button is active when the Show Selected Item option is selected.

Displaying a Clip in the Broadcast Monitor

Use the Show Selected Item option in the Broadcast Monitor box to display the clip you selected in the broadcast monitor. Use the Broadcast Monitor buttons to resize and set the proportions of the clip. If the clip is zoomed in on the application monitor, the clip is still displayed in its entirety on the broadcast monitor.

When the Show Selected Item option is selected, you can manage additional preferences. See Broadcast Multiview Preferences on page 87, and Image Data Type Preferences on page 87.

Displaying the User Interface in the Broadcast Monitor

Use the Screen Grab option in the Broadcast Monitor box to display content you select in the Player or module viewport in the broadcast monitor. The content is only displayed in the broadcast monitor when a module viewport or the Player are displayed in the application monitor.

If the output is zoomed in or panned in on the application monitor, the broadcast monitor displays the output in the same state.

When this broadcast option is selected, the Always Send Grab Area button appears. Enable this button to display Player or viewport output, including times when neither is in use.

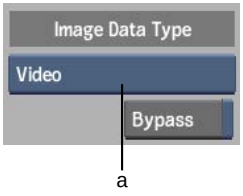
Broadcast Multiview Preferences

This preference is active when Show Selected Item is selected in the Broadcast Monitor box. See Broadcast Monitor Preferences on page 85.

Always Display Result Enable this button when working in modules with multiple views to display an image of the Result viewport in the broadcast monitor, including when a different view is selected in the application monitor. When a view without clips is selected, the Always Display Result button is enabled by default.

Image Data Type Preferences

These preferences are active when Show Selected Item is selected in the Broadcast Monitor box and the Always Display Result button disabled. See Broadcast Monitor Preferences on page 85 and Broadcast Multiview Preferences on page 87.



(a) Image Data Type box

Image Data Type box Select the type of image data you are displaying in the broadcast monitor. Your selection determines the type of transformation that is applied to the clip to modify the contrast.

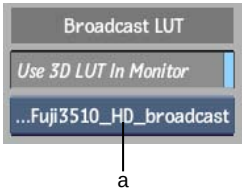
Select:	To:
Logarithmic	Apply a transformation to a logarithmic film scan.
Video	Apply a transformation to a video clip.
Linear	Apply a transformation to a 16-bit floating point image, with a high dynamic range.

Bypass button Enable to deactivate the image data type display settings in the broadcast monitor.

Broadcast LUT Preferences

These preferences appear when Show Selected Item is selected in the Broadcast Monitor box. See Displaying a Clip in the Broadcast Monitor on page 86.

Use the Broadcast LUT preferences to apply and select a 3D LUT you can use in the broadcast monitor. This selection does not affect the LUT settings in the application monitor.



(a) 3D LUT List box

Use 3D LUT In Monitor button Enable to apply a 3D LUT from the 3D LUT list in the broadcast monitor.

3D LUT List box Displays the list of 3D LUTs that you imported in the LUT menu.

Expanding and Collapsing the Contents of a Clip

You can expand or collapse selected clips with real-time settings, history, or soft edits from a clip libary. You can also display or hide selected clips. Select options from the Expand Options group of dropdown lists or click a clip indicator to toggle the display of the associated deliverable, history, or source clip display for the clip.

In Proxies mode, the expanded portion of a clip is slightly smaller in order to distinguish it at a glance from other types of clips. In Titles View mode, these entries are indented.

To expand or collapse a clip:

1. Select the clip you want to expand or collapse.
2. Do one of the following:

Select an option from:	To:
All	Display all hidden clips or hide selected clips.
Deliverables	Display or hide the deliverables associated with the selected clips.
History	Display or hide the clip history associated with the selected clips.
Edits	Display or hide source clips associated with the selected clips.

NOTE Double-click a Deliverable, History or Soft Edits indicator on a clip to toggle its display/hide settings.

Colour Management 7

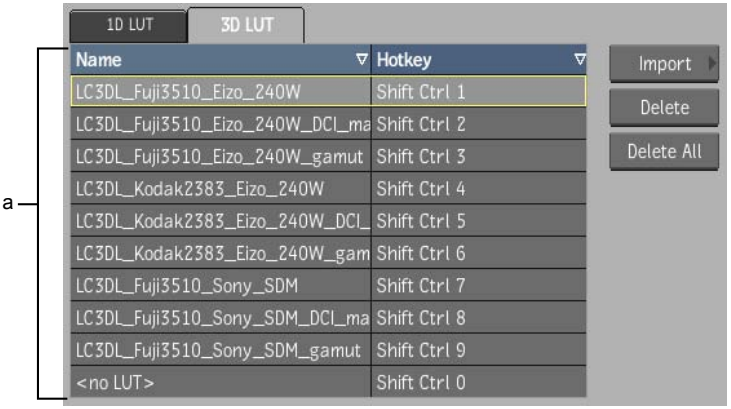
Topics in this chapter:

LUT Preferences

Use the LUT preferences to import and list the 1D and 3D LUTs you can use in Inferno.

You can use a hot key to switch between the first ten LUTs in a 1D LUT list. Press **Alt+Shift+1** for the first LUT, **Alt+Shift+2** for the second LUT, and **Alt+Shift+0** for the tenth LUT in the list.

You can use a hot key to switch between the first ten LUTs in a 3D LUT list. Press **Ctrl+Shift+1** for the first LUT, **Ctrl+Shift+2** for the second LUT, and **Ctrl+Shift+0** for the tenth LUT in the list.



(a) 3D LUT list

3D LUT List box Displays the list of 3D LUTs that you imported for use in Inferno.

Import button Opens the Import LUT browser. Navigate to the 3D LUT file you want to use, and select it to load it to the 3D LUT list.

Delete button Deletes the selected LUT from the 3D LUT list.

Delete All button Deletes all LUTs from the 3D LUT list.

(a) LUT Type box (b) 3D LUT list

1D LUT List tab Displays the list of gamma LUTs that you defined and 1D LUTs that you imported for use in Inferno.

LUT Type box Select whether you want to define a gamma LUT or import a 1D LUT file in the current row and assign the selected hot key.

Import button Opens the Import LUT browser. Navigate to the file you want to use, and select it to load it to the 1D LUT list. This button is active when the LUT Type box is set to LUT file.

Value field Displays the gamma value. This field is active when the LUT Type box is set to Gamma.

Delete button Deletes the selected gamma or 1D LUTs from the list.

Delete All button Deletes all LUTs from the 1D LUT list.

Select button Apply the selected gamma or 1D LUT.

To define a gamma LUT:

1. Click the 1D LUT tab to display the 1D LUT list.
2. Select the row with the hot key you want to assign to the gamma LUT.
3. Enter a gamma value in the Value field.

To apply a LUT from the 1D LUT list:

1. Click the 1D LUT tab.
2. Select the row with LUT you want to apply to the display.
3. Click Select.
   The filename of the applied LUT and its assigned hot key display in blue.

NOTE When a ID or gamma LUT is applied it cannot be deleted, unless you click the Delete All button to clear the entire 1D LUT list.

To import a 1D or 3D LUT:

1. Click a tab to display the list for the type of LUT you want to import.
2. Select the row with the hot key you want to assign to the LUT.
3. If the 1D LUT list is displayed, in the LUT file box, select LUT file.
4. Click the Import button.
5. In the file browser, navigate to the file you want to import.
6. Select the LUT file.

The LUT file appears in the LUT list in the selected row.

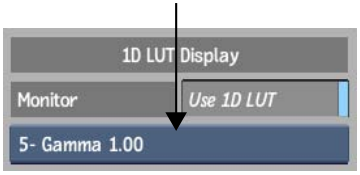
Applying 1D LUTs to the Monitor

Gamma and 1D LUTs configured in the Preferences can be applied to the monitor. See LUT Preferences on page 91. They can also be applied from the View menu or the Player.

3D LUT display mode is disabled when 1D LUT display mode is activated.

To apply a 1D LUT to the monitor:

1. Enable Use 1D LUT (**Alt+Shift+~**) to enable 1D LUT display mode.
   The 1D LUT displayed in the 1D LUT List box is applied to the display.
2. To change 1D LUTs, do one of the following:
   - In the View menu or the Clip & Setup tab in Player, select the 1D LUT from the list in the Monitor LUT List box.



   - Use hot keys to switch between 1D LUTs defined in the 1D LUT list.

Press:	To apply:
Alt+Shift+1	The first 1D LUT defined in the 1D LUT list.
Alt+Shift+[2-9]	The second through ninth 1D LUT defined in the 3D LUT list.
Alt+Shift+0	The tenth 3D LUT defined in the 1D LUT list.

3. To toggle the current 1D LUT on and off, press its associated hot key (**Alt+Shift+[1-0]**).
4. To disable 1D LUT display mode, press **Alt+Shift+~**.

Exporting LUTs

Once you complete your custom LUT, you can use it to convert images. To do so, you must first save or export it. Saving a LUT preserves its menu settings and automatically creates an invert LUT. A LUT's menu settings include values for its basic curves, as well as its advanced editing curves. Exporting combines the basic curves and the advanced editing curves into a single set of curves. However, exporting provides the opportunity to change the bit depth of the LUT. Both saved and exported LUTs are applied to an image sequence or a clip in Batch in the same manner.

When you export a LUT, the settings that correlate with the basic curves and the advanced editing curves are merged to create a single set of RGB conversion curves and the independence of these settings is lost. Whether you load or import the LUT afterwards, it appears as a basic conversion curve only. However, like all basic LUT types, you can then alter it, for example, by using the advanced editing curves.

Exported LUTs serve as a good interchange format for colour consistency between Autodesk and non-Autodesk products.

Exporting is useful when you want to change a LUT's bit depth. If you develop a LUT for importing 10-bit logarithmic film data, for example, you can easily convert it to work for 12-bit logarithmic data. When exporting a LUT, you can scale both the input and output bit depths.

NOTE If you accessed the LUT Editor from the Import Image menu, you must export the LUT when you exit.

To export a LUT:

1. Access the LUT Editor.

   If you accessed the LUT Editor from a node in Batch, select the node and display the LUT editing menu with which you created the LUT.

2. Do one of the following:
   - Click Export.

- If you are using LUT Editor in the Import Image menu, click Exit.

The Export LUT menu appears.

3. Set a location for the LUT.
4. Select the source and destination bit depths for the exported LUT.
5. Enable Generate Inverse LUT to create an inverse LUT along with the normal lut. You can apply an Inverse LUT to restore original LUT settings.
6. Click Export.

   The LUT is exported to the specified location.

Applying a LUT on Import or Export

Applying a LUT to a film clip on import or export converts the clip's images according to the contents of the LUT.

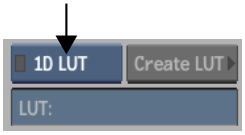
To apply a LUT on image import or export from the Desktop:

1. In the Main menu, click Library.

2 In the Library menu, click Import Image or Export Image.

- If you clicked Import Image, select a destination reel for the imported images.
- If you clicked Export Image, select the clip that you want to export.

3 In the LUT Type box, select an option.



Select:	To:
Gamma	Apply gamma correction to a clip. This option only appears when a QuickTime movie is selected.
1D LUT	Apply a 1D LUT to a clip.
3D LUT	Apply a 3D LUT to a clip.

4 If the LUT field is inactive, click the LED indicator to enable it.

5 Click the LUT field.

The file browser appears. The Files list contains the LUTs currently residing in your project's *~/lut* directory. These include the samples provided with Inferno, as well as any LUTs you may have saved or exported to this directory. By default, Lustre Color Management LUTs are stored in the */usr/discreet/Lustre_Color/lut/Lustre_Color_3DLUTs* directory.

Sample 1D LUTs include the following.

LUT File Name	Conversion Process
10logto12	Converts a 10-bit logarithmic image to a 12-bit linear image.
10logto8	Converts a 10-bit logarithmic image to an 8-bit linear image.
8to10log	Converts an 8-bit linear image to a 10-bit logarithmic image.

LUT File Name	Conversion Process
12to10log	Converts a 12-bit linear image to a 10-bit logarithmic image.
invertRGB	Inverts the RGB values of the image.

6 If you want to switch to a different LUT type without exiting the file browser, use the LUT Type box.

7 Select a LUT in the Files list or navigate to the file you want to use.

The Import Image or Export Image menu reappears. The LUT field displays the name of the new LUT. It will be applied to the clip you import or export when the LUT is enabled.

Controlling Image Display using Exposure and Image Data Type

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 with a transformation for a video image. You can apply transformations to the image to display an optimal view of logarithmic and linear images.

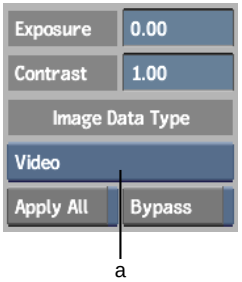
All transformations applied to images are for display purposes only. As you work with your transformed image, you will be able to better determine the decisions required to achieve a desired effect. For example, you can set an image window to display more realistic contrast values for a logarithmic image. If you apply an RGB blur or perform a colour correction, you will see the results on the transformed image, but when the clip is processed, only the RGB blur or colour correction information will be processed.

NOTE If you apply a 3D LUT that expects logarithmic input, you can apply it to any image data type, and still display a plausible result. This includes all print stock LUTs, including Lustre Color 3D LUTs. If you work with a different type of LUT, enable the Bypass button to work with the image with no transformation.

To access the exposure and image data type settings:

1 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 in the View menu

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.

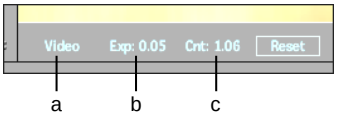
Select:	To:
Logarithmic	Apply a transformation to a logarithmic film scan.
Video	Apply a transformation to a video clip.
Linear	Apply a transformation to a 16-bit floating-point image, with a high dynamic range.

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

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

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) Image Data Type (b) Exposure (c) 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:
   - If you are in a module, click the image data type to cycle through video, logarithmic, and linear settings.
   - 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.

NOTE Click Reset to restore default exposure and contrast settings.

Batch Nodes

8

Topics in this chapter:

Auto Stabilize Node

The Auto Stabilize node has been enhanced with a number of workflow improvements in this release.

Use the Auto Stabilize node to rectify stabilization issues. For example, you can remove camera jitter or lock an object's position over a sequence of frames to make it appear motionless. Attach a front clip to this node to analyse its movement. Matte input can be attached to this node to create an output matte, or to use black pixels on the matte to exclude areas from the stabilization algorithm. Use the parameters in the Auto Stabilize menu to refine the stabilization.

You can use the Negate Stabilization feature to apply the inverse transformation to the input. For example, you can apply a stabilization analysis to a clip and connect the output to a Paint node to touch up a portion of it. You can copy the Auto Stabilize node and revert to the motion of the original input.

You can perform a two-dimensional analysis, which takes X and Y position, rotation, and scaling into account. You can select a three-dimensional analysis, which also analyses and compensates for perspective distortion. Select this option to stabilize a flat object moving in three-dimensional space, or if you are stabilizing a pan or tilt camera movement.

The X-axis, Y-axis, rotational, and scaling stabilization curves are calculated during analysis. If a perspective analysis is performed, perspective curves are also calculated. The data from all or a selection of these curves can be applied to the final stabilization of the result clip and output matte.

This node supports floating-point (OpenEXR) data as input.

(a) Stabilization Method box (b) Stabilization Parameter buttons

Analyse button Click to run the stabilization analysis.

Scene Flexibility field Determines the level of focus on rigid objects that move in the clip. If the analysis has no scene flexibility, the stabilization will focus on a single rigid object as it moves in the clip. If the value is set to 100%, all visible motion paths are analysed.

Use Matte button Use the black areas of the matte input to exclude those areas from analysis on the front.

From and To fields Displays the range of frames included in the analysis.

Stabilization Method box Provides options to perform a two-dimensional or perspective analysis.

Negate Stabilization button Enable to apply the inverted parameter values.

Region of Interest (ROI) button Displays a rectangle that can be resized to indicate the region to be analysed on the front clip.

Auto button Enable to automatically track motion within the region of interest. At each frame, the position of the region of interest will be updated based on the motion within the area. If this button is disabled, and the region of interest was not previously tracked, the region of interest remains static.

X and Y Position fields Displays the centre of the region of interest.

X and Y Scale fields Displays the horizontal and vertical dimensions of the region of interest as a percentage of the total image. When both fields are set to 100, the dimensions of the region of interest and the image are equivalent.

Smoothness field Displays the level of smoothness of the stabilization curves generated by the analysis.

Amount field Displays the percentage of smoothness used in the final stabilization output.

Stabilization Parameter buttons Enable to include a parameter in the stabilization. Enable the Lock button to link the parameter value at the reference frame to all frames in the analysis.

Set Frame button Makes the current frame the reference frame for the stabilization.

Display Size field Displays the pixel width and height of track points.

Show Cloud button Enable to display the track point cloud or press **Alt+D**.

Delete and Update button Deletes selected track points and updates stabilization curves.

2D Transform Offset fields Displays the X and Y position, rotational and scaling offsets applied to the clip. The offset is applied independently of the analysis if the corresponding Stabilization Parameter button is disabled.

Perspective Offset fields Displays the perspective offsets applied to the clip. These fields are enabled after a three-dimensional analysis.

Padding box Provides fill options to pad the empty portions of the frame with the last line of pixels, a repeated (rolled) image, a mirror image, or black pixels.

Resize Fit Mode box Resize the clip with user-defined width and height values.

Width field Displays the width of the image.

Height field Displays the height of the image.

Hardware Filtering Enable the graphics processing unit to filter subpixel information.

Anti-aliasing button Enable to display anti-aliasing.

(AA) Anti-aliasing Sampling box Select the anti-aliasing sampling level.

Anti-aliasing Softness field Displays the level of softness used in anti-aliasing.

To analyse a clip:

1. Enable Use Matte if you want to use the matte to constrain the region of analysis.
2. Select an option from the Stabilization Method box.
3. Enter a percentage in the Scene Flexibility field. The following range of values are recommended.

Select:	To:
0%-20%	Stabilize a pan, tilt, zoom, or the motion of a single rigid object that is visible during the entire analysis.
30%-70%	Stabilize objects that change their shape or depth, and exclude objects that are smaller or moving rapidly.
80%-100%	Stabilize all visible movements.

4. You can enable Region of Interest (ROI) to display a rectangle that constrains the analysis to the region of interest. Enable Auto track motion within the region of interest; at each frame, the position of the region of interest will be updated based on the motion within the area. The rectangle can be resized and animated, and you can use the colour pot to change its colour.
5. In the In and Out fields, enter the range of frames to analyse.
6. Click Analyse.

   During analysis, the viewport defaults to the Front view, and the frame advances as the stabilization is calculated. Progress can be monitored by the keyframes that have been created during analysis and the percentage of completion displayed next to the Analyse button. Track points are displayed on the clip to indicate how the stabilization was tracked.

When the analysis is finished, the stabilization can be customized further using the additional settings in the Auto Stabilize menu. Track points can be deleted to modify the stabilization curve.

To delete track points:

1. Toggle track point selection:
   - Click a track point to select it for stabilization (green) or mark it for deletion (red).
   - **Ctrl**-drag to toggle points within a rectangular area.
   - Press **Ctrl** and either drag or click to toggle additional points and retain other existing selections.
2. Click Delete & Update.

   The red track points are deleted and the stabilization curve is recalculated using the remaining track points.

You can change the centre of rotation and scaling for the image. By default, the centre of rotation and scaling is the centre of the image, represented by a yellow crosshair.

To change the centre of rotation and scaling:

1. Display the Front view (**F1**).

The yellow crosshair appears.

2 Hold down **C** and click the new centre point.

Exposure Node

The Exposure node is new for this release.

Use the Exposure node to apply plausible exposure and contrast settings suitable to the image data type of an input clip. The values can be adjusted separately for each colour channel or controlled simultaneously.

The Exposure node accepts a front clip and a matte clip. The matte input is used to limit the areas of the front clip that are affected by exposure and contrast changes.

This node supports floating-point (OpenEXR) clip input.

NOTE To adjust the exposure, contrast, and image data type of the image for display purposes only, set the image display viewer options in the View menu or edit them gesturally in the current viewport.

Front view of a 16-bit floating point image

Result view of the image after the exposure and contrast have been modified

Calculations for exposure and contrast adjustments are based on the type of image that is selected in the Exposure menu.

(a) Input Data Type box

Input Data Type box Select the type of image data being input to the node. Your selection determines the most suitable type of transformation to apply to the clip. The algorithm used to calculate the result is dependent on this option.

Select:	To:
Logarithmic	Apply a transformation to a logarithmic film scan.
Video	Apply a transformation to a video clip.
Linear	Apply a transformation to a 16-bit floating-point image, with a high dynamic range.

Exposure fields Display the exposure offset for each colour channel.

Contrast fields Display the contrast level for each colour channel.

Pivot fields Display the channel value used as a pivot when generating contrast. The pivot value affects the way the contrast is calculated. The default value is mid-gray on a logarithmic scale.

Proportional buttons Enable to change a parameter for a channel and update the parameter values for the other channels proportionally.

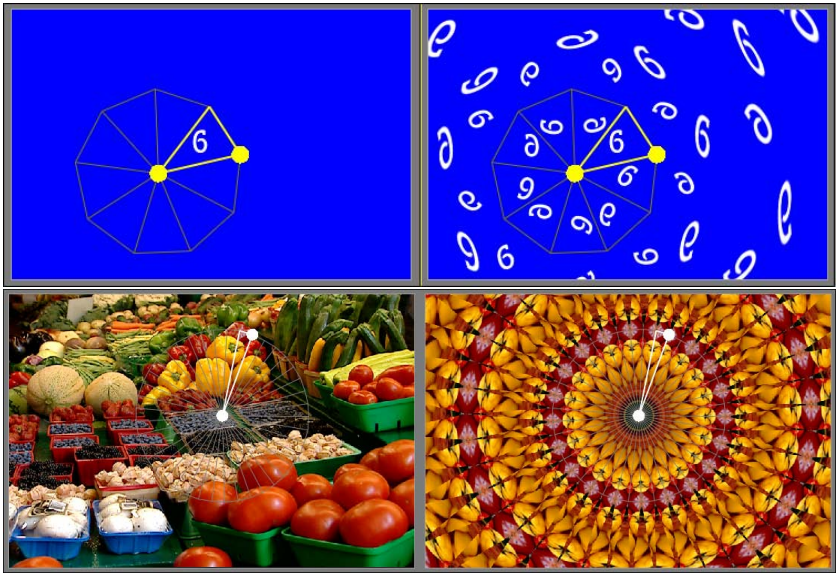
Motif Node

The Motif node is a new feature in this release.

Use the Motif node to create a tiled symmetrical texture. The Motif node accepts a front and a matte input. The node outputs a result clip and output matte. This node accepts floating-point (OpenEXR) data as input.

Source clips can be transformed before symmetrical effects are applied. Transformations are applied to both the front and matte clips simultaneously. You can select the type of symmetry mode to use on the transformation and define its parameters.

The radial symmetry mode displays a user-defined widget, which is a polygon with an equal number of sectors to the order of symmetry. You select the sector to use as the originating tile, the basis for a kaleidoscopic texture. Additional parameters allow you to define the center of symmetry of the image, and the angle and radius of the originating tile.

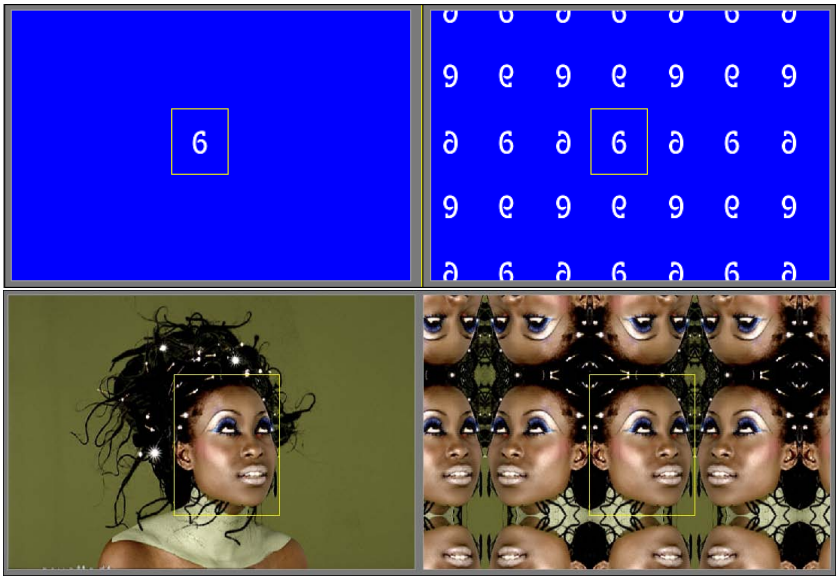


Front and Result views in radial symmetry mode

In Radial mode, you can also reuse the pixel on the edge of a sector to pad the space between outer tiles.

Result views with the repeating disabled and enabled

The region of interest (ROI) effect uses a user-defined rectangular selection as the originating tile to create a basic mirrored texture.



Front and Result views in region of interest symmetry mode

Image courtesy of The House

The Motif menu displays 2D Texture Transform controls that allow you to simultaneously change the position, scaling, rotation, and the type of image padding. The Symmetry Mode controls display a colour pot for the original tile selection.

X and Y Position fields Display the number of pixels by which the input and matte clips are offset.

X and Y Scale fields Display the percentage of horizontal and vertical scaling to apply to the front and matte clips.

Rotation field Displays the angle of rotation of the front and matte clips.

Repeat Mode box Provides fill options to pad the empty portions of the frame with the last line of pixels, a repeated (rolled) image, or black pixels.

Mode box Determines the type of symmetry effect to apply to the transformation.

Widget colour pot Displays the colour used to highlight the originating tile.

Order field Displays the order of symmetry. It also indicates the number of sectors or sides of the polygonal widget. An order of 2 creates a basic mirrored image. An order of symmetry can have an odd number as a value. Tiles are mirrored in a counter-clockwise direction, therefore the sector to the right of the highlighted sector may not be its mirror image. Displayed in Radial mode.

Even button Enable to maintain an even order of symmetry. If the order of symmetry is an odd value, an extra tile will be added, ensuring that adjacent tiles are mirror images of each other. Displayed in Radial mode.

Angle field Displays the angle of the axis of symmetry. Displayed in Radial mode.

Radius field Displays the pixel length of the sector radius. Displayed in Radial mode.

Repeat button Enable to use the pixel colour on the sectors' edge to pad the space between outer tiles. Displayed in Radial mode.

X and Y Centre Position fields Displays the horizontal and vertical position of the centre of symmetry.

X and Y ROI Scale fields Displays the horizontal and vertical dimensions of the region of interest as a percentage of the original dimensions of the clip. Displayed in Region of Interest (ROI) mode.

Modifications to the originating tile and symmetry mode parameters can be made gesturally in any view.

Drag:	To:
Outer edge of the region of interest	Change the width or height of the selection.
Corner of the region of interest	Change the width and height of the selection.
Inside the region of interest or widget	Move the selection.
The widget centre	Change the centre of symmetry.

Drag:	To:
Circle on the outer edge of the widget	Change the radius and the angle of symmetry.

Batch Setups

9

Topics in this chapter:

BFX Timecode and Clip Frame Length

When you load a clip into a BFX level, its timecode automatically keeps the timecode of the timeline one level up (main or BFX). Head frames are loaded as negative frames. You can use these extra frames to offset the starting frame of a clip. All subsequent clips brought into the same BFX level are loaded with head and tail frames intact.

Both the BFX timebar positioner and the clip timeline positioner represent the main timecode for the current frame. If you set record and source clips to edit material from one clip to another, the current time position of the source clip serves as an implicit in point.

You can set the BFX timebar to display or hide head and tail frames. Display head and tail frames if you need the extra frames for editing or if you load subsequent clips that are longer in length. If you use the extra frames, you will

have to trim out the clip when you exit back to the main timeline. If you offset a clip using the extra frames, the main timeline will reflect the new offset.

From main Batch, you can also display or hide the negative frames that result from offsetting the starting frame of a clip as well as offset clips to their record timecode.

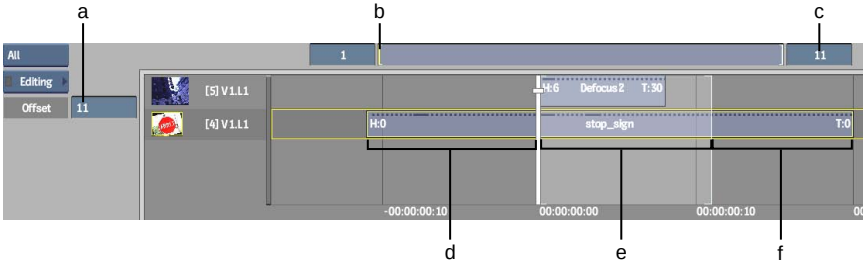
To set the BFX timebar to display the frames defined by a clip's in and out points:

1 From the Timebar Range area of the BFX Setup menu, click Set to BFX In/Out.

The BFX timebar displays the range of frames of the clip as defined by the in and out points set on the main timeline.

NOTE Set to BFX In/Out is the default setting.

In the following example, a clip (stop_sign) with head and tail frames is loaded into a BFX level from the main timeline at timecode 00:00:00:00. The head frames are loaded into the Offset field. The BFX timebar positioner is set at the beginning of the clip—frame 1. The BFX timebar does not show the head and tail frames.



(a) Head frame number (b) BFX timebar positioner (c) Clip length without head and tail frames (d) Head frames (e) Frames between in and out points (f) Tail frames

NOTE In Timing view, shown in the previous illustration, select the layer or clip proxy to load the head frames into the Offset field.

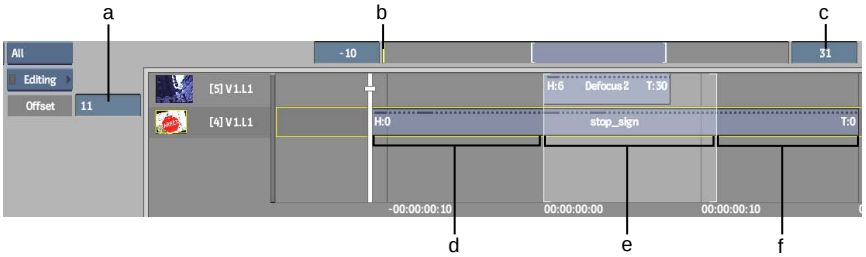
To set the BFX timebar to display a clip's entire range of frames:

1 From the Timebar Range area of the BFX Setup menu, click Set to Media Range.

The BFX timebar displays all the frames of the clip brought in from the main timeline, including head and tail frames.

NOTE This setting remains persistent throughout the Inferno session.

In the following example, a clip (stop_sign) with head and tail frames is loaded into a BFX level from the main timeline at timecode 00:00:00:00. The head frames are loaded into the Offset field. The BFX timebar includes both the head and tail frames. The BFX timebar positioner is set at the first head frame—negative frame 10.



(a) Head frame number (b) BFX timebar positioner (c) Clip length including head and tail frames (d) Head frames (e) Frames between in and out points (f) Tail frames

NOTE In Timing view, shown in the previous illustration, select the layer or clip proxy to load the negative frames into the Offset field.

Main Batch clips have a default frame length of 100. You can set the Batch duration to that of the loaded clip.

The first clip loaded into a BFX level determines the duration of that BFX level. If you load another clip, you can change the BFX duration to that of the new clip. The new duration applies only to that BFX level. Back on the main timeline, you must trim the clip to display its new length.

To set the Batch duration:

1. From main Batch or a BFX level, select a clip in the schematic.
2. Press **T** and click the clip in the schematic.

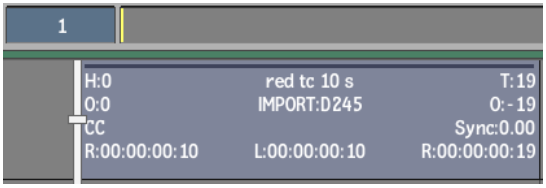
   The Duration field updates to reflect the new length.

NOTE You can also click the Set to Media Range preference after loading the new clip.

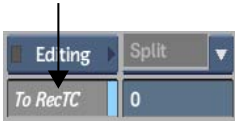
To offset a clip to its record timecode:

1 Select a clip in main Batch and display its timeline.

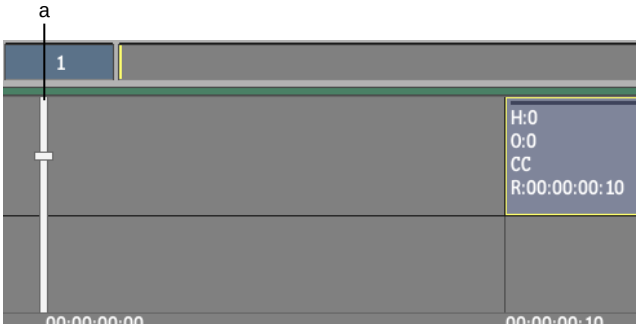
Note that the following clip was brought into Batch with a timecode of 00:00:00:10. The timeline positioner is set to frame 1.



2 Enable To RecTC.



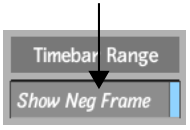
Frame 1 is offset to 00:00:00:00 to correspond to Batch time, as illustrated in the following example.



(a) Positioner at frame 1

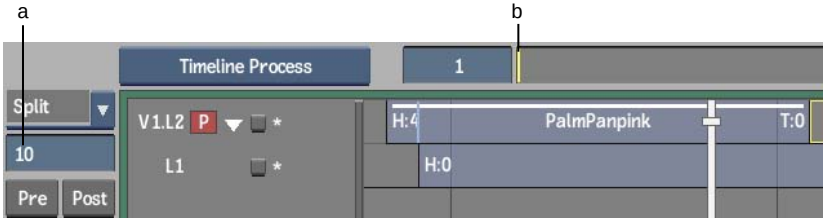
To set the Batch timeline to display negative frames:

1 In the Batch Setup menu, enable Show Neg Frame.



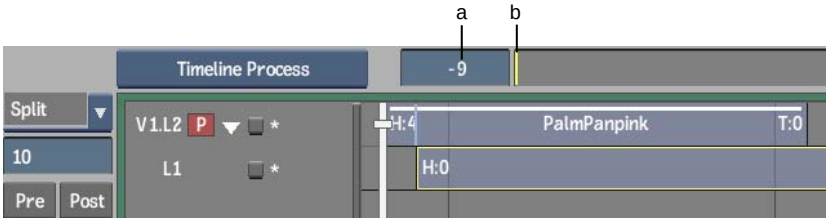
The Batch timeline positioner changes location to take into account negative frames resulting from the segment being offset before frame 1.

The following example shows the Batch timeline with Show Neg Frame disabled. The clip is offset by 10 frames. The negative frames are loaded into the Offset field. The Batch timebar positioner is set at the first head frame—frame 1.



(a) Offset field (b) Batch timebar positioner

The next example shows the Batch timeline with Show Neg Frame enabled. The Batch timebar positioner is set at the first head frame—negative frame 9.



(a) First frame—negative frame 9 (b) Batch timebar positioner

Committing Batch FX

When you output from a Batch FX output node to the timeline, you can commit the RGB results (BFX) or the matte results (BFXa).You can commit Batch FX imported from another application. Any soft effects on the timeline are preserved when you commit the Batch setups.

To commit Batch FX:

1 Select the timeline element with the BFX or BFXa setup that you want to commit.

2 Select Commit BFX from the Effect Operations box.

If you committed RGB results, the BFX element is committed. If you committed matte results, the BFXa element is committed into a container.

Copying Clips to the Desktop

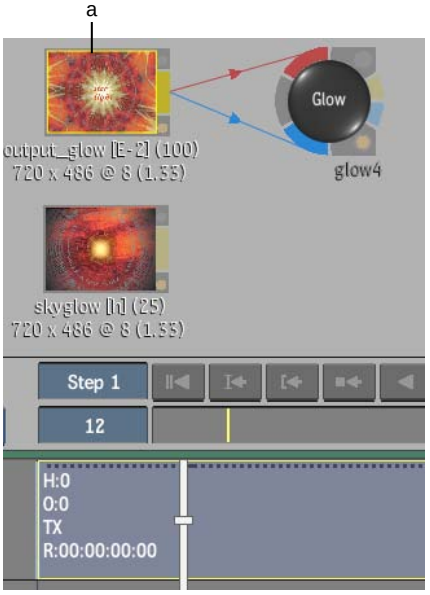
From Batch or a BFX level, you can send a copy of any clip to the Desktop. If the clip contains soft effects, the soft effects are preserved with the clip.

To save a BFX setup, you must save it separately. To save a setup with the clip copied back to the Desktop, make sure you save the setup after copying the clip.

To copy a clip to the Desktop:

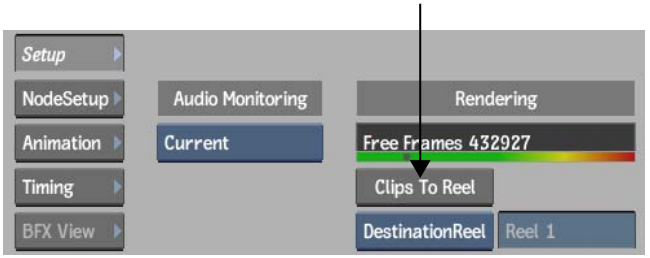
1 Select the clips to copy in the schematic.

In the following example, the selected clip has a Text soft effect.



(a) Selected clip with Text soft effect

2 From the Setup menu, click Clips to Reel.



NOTE To copy the clips to a reel other than the destination reel, select Custom Reel and enter the reel number.

If you loaded the clips from the library, the clip names turn from blue to white indicating the clips are Desktop clips.

3 Exit to the Desktop.

A copy of the selected clip appears on the Desktop. Note that the Text soft effect in the clip is preserved.

To save a BFX setup:

1 Inside a BFX level, click Save, name the setup, and click Save again.

NOTE If you sent a copy of the clips in Batch to the Desktop using Clips to Reel, make sure you save the setup after sending the copy so that the links from the setup are to the copied clips.

The BFX setup is saved. The clips are not saved with the setup; however, the links to the original clips are saved. When you load the BFX setup, the Desktop is searched first for the clips and then the library is searched.

If the clips are not found, the setup is loaded with black proxies for the missing clips, and the clip name appears in red.

Creating One Clip Per Timeline Layer

If you bring multiple layers into a Batch setup and want to apply a separate flow graph to each layer, you can split the layers and create one clip per layer. Soft effects and transitions from the original timeline layers are kept with each clip.

If you want to work with the clips in an Action setup, and they are not affected by vertical compositing from the original timeline, you can have each split layer automatically converted to an indirect layer of an Action node.

If the clips are affected by vertical compositing from the original timeline, you can have the applicable soft effects automatically extracted and converted to the corresponding Batch nodes. This extraction process attempts to preserve the same visual result as the original timeline. The same rules of conversion apply to soft effects whether they are extracted automatically or manually.

You can split both layers and tracks into individual clips.

To create one clip per layer:

1. Load a multi-layer clip into a BFX level and display its timeline.

   In the following example, a three-layer clip is loaded with all soft effects preserved. The Axis soft effects on the top two layers create the vertical compositing.

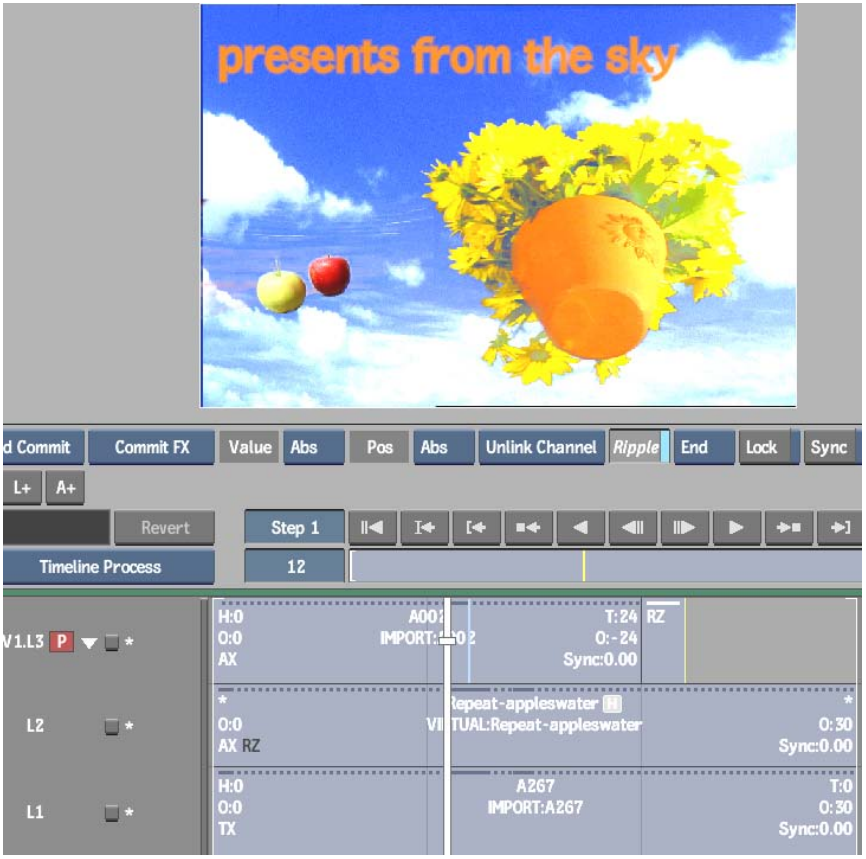


Image courtesy of Topix / MadDog

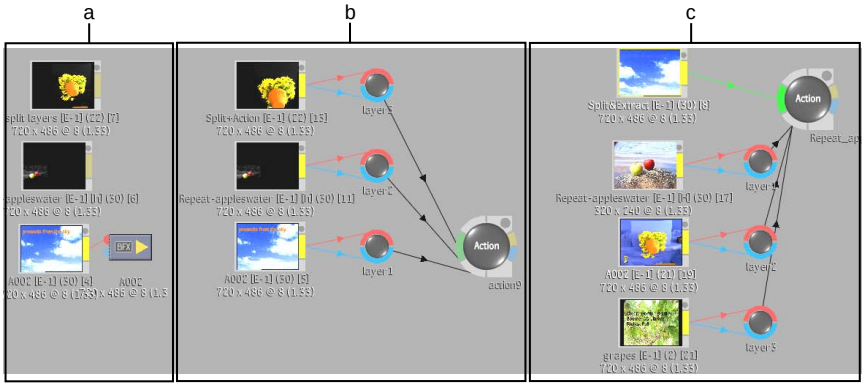
2 Select the clip in the schematic, and then select an option from the Split dropdown list.

Select:	To:
Split Tracks and Layers	Create one clip for each timeline layer and track.

Select:	To:
Split and Convert to Action Layers	Create one clip for each timeline layer and track and have each clip connected to an indirect layer of an Action node. Visual results from soft effects may be lost in this operation.
Split and Convert to Batch Tree	Create one clip for each segment and have soft effects that affect the vertical editing of the original timeline converted to corresponding Batch nodes.

Clips are created for each layer and are connected in the Batch flow graph as illustrated in the following examples.

In example a, separate clips are created for each layer of the original timeline. In example b, an indirect Action layer is created for each clip. In example c, separate clips are created for each segment of the original timeline. The Action indirect layers are converted from the Axis soft effects so as to maintain the same visual result as the original timeline. The Text soft effect is not extracted since it is not affected by vertical compositing.



(a) Result from Split Tracks and Layers option (b) Result from Split and Convert to Action Layers option (c) Result from Split and Convert to Batch Tree option

NOTE The timeline of the original clip loaded into the BFX level is replaced by a separate timeline for each clip. To see the timelines of all clips at once, use Timing view.

3 Add or modify nodes to create your process tree.

Audio

10

Topics in this chapter:

Gain

Use the Gain soft effect to adjust the gain for individual segments in an audio track. Gain is the measure, in decibels, of how much a circuit amplifies a signal. You can set the segment gain to any value from -96.0 dB to +24.0 dB. The segment gain is combined with the input strip gain in the AudioDesk.

New for this release, you can also animate the segment gain in the timeline, using a simplified version of the Channel Editor.

To adjust the segment gain:

1. From the Record T/L, select the audio segment that you want to adjust.
2. From the Audio Soft-Effects menu, enable Gain.
3. Enter the new gain level (from -96.0 dB to +24.0 dB) in the Segment Gain field that appears.

To animate the segment gain in the timeline:

1. From the Record T/L, select the audio segment that you want to adjust.

   You can also change the segment gain curve of two (or more) audio segments simultaneously. Simply select all segments before tweaking the curve. The segments must all start at the same frame to do this.

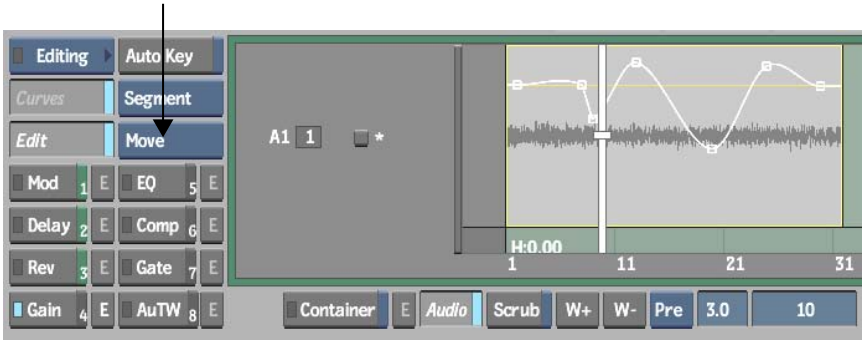
2. From the Audio Soft-Effects menu, enable Gain.

   The Edit button becomes active.

3. Enable Edit.

   The segment turns grey. The segment gain level is represented by a white line, which you can modify just like in the Channel Editor, but with a limited set of functionality.

4. From the Edit Mode box, use the available options to edit keyframes on your segment gain level.



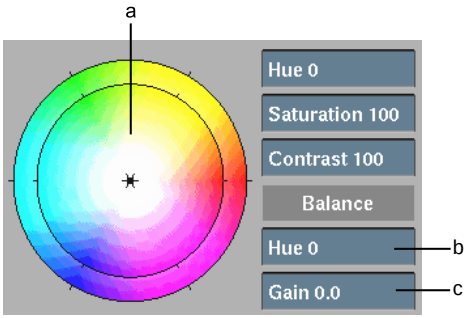
Effects

Topics in this chapter:

Adjusting the Colour Balance

New for this release: When working with 16-bit floating point images in the Colour Corrector, you can add values outside of the colour wheel.

You can adjust the colour balance in an image using the colour wheel or the Balance Tools Hue and Gain fields. The value of Hue determines the colour to add and the value of Gain specifies the amount of colour to use.



(a) Colour wheel (b) Hue field (c) Gain field

Pure red is the 0-degree point for hue on the colour wheel. When you increase the Hue value, you move counterclockwise on the colour wheel. When you decrease the Hue, you move clockwise on the colour wheel.

The centre of the colour wheel represents 0 Gain. As you increase the value of Gain, you move towards the edge of the colour wheel and add more of the selected colour to the image.

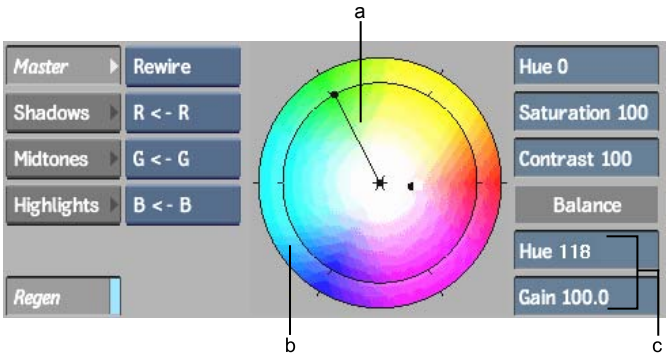
NOTE If Saturation is set to 0, no colour balancing is performed, regardless of the Gain value. In the Channel Editor, Balance channels are labelled hue_balance and gain_balance, respectively.

Creating and Modifying a Colour Balance Point

You can make colour balance points for each range (highlights, midtones, and shadows) or for all ranges simultaneously (master). Click inside the colour wheel to create a balance point for the current tonal range.

The current range's colour balance point is connected to the hub of the colour wheel by a line. You can constrain how the point is modified according to how you select it. If you select the line, the radius of the line Strength is constrained. If you select the point, the direction of the line Tint is constrained. You can make unconstrained modifications by clicking anywhere in the wheel, or by entering values in the Balance fields.

NOTE As you move the colour balance point on the colour wheel, the values in the Balance Tools Hue and Gain fields update automatically. You can also set the values for Tint and Strength directly in these fields.



(a) Current colour balance point (b) Hue shift/saturation comparison (outside ring is Result; inside ring is Source) (c) Balance tools

Colour balancing is performed in RGB colour space. The luma of the image is not changed. For example, if Hue (tint) is set to 0 and Gain (strength) is set to 100%, pure red is added to the image. However, a black pixel remains black since it keeps the same luma value.

To change the colour balance in the image:

1. Enable Regen so that the image will update as you change the colour balance.
2. Select the range you want to modify.
3. On the colour wheel, click and drag toward the colour you want to use.

   As you drag the cursor, a colour balance point follows the cursor. Depending on the range you are modifying, the colour balance points are shaded to make for easier identification (from light grey to black, in this order: Highlights, Shadows, Midtones, Master).

   You can continue dragging outside of the colour wheel. The colour balance point is replaced by a triangle in the outer circle of the colour wheel. The triangles are also shaded depending on the range selected (from white to grey, in this order: Master, Highlights, Shadows, Midtones).

   The image dynamically updates as you drag.

TIP You can interact directly in the image window to change the colour balance. While holding the range hot key (**V** for Master; **Z** for Shadows; **X** for Midtones; or **C** for Highlights), drag directly in the image window. The colour wheel and Balance Tools fields update accordingly.

---

4 If you are not satisfied with the resulting image, click and drag the colour balance point or triangle again. You can continue to move the colour balance point or triangle until the desired effect is achieved.

5 Click Process to apply the changes to the clip.

Adjusting the Colour Range

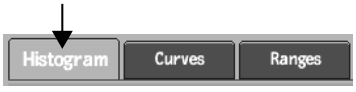
New for this release: With the support of 16-bit floating point images in the Colour Corrector, the Histogram, Curves, and Ranges menus have been redesigned, and some new options have been added.

You can use the Histogram menu to adjust the range of values used for the red, green, blue, or luminance channels of an image. The histogram shows the colour distribution of pixels in the front and result image for the selected channel. The horizontal axis represents the values of pixels, from black at the left to white at the right. The vertical axis represents the number of pixels with these colour values.

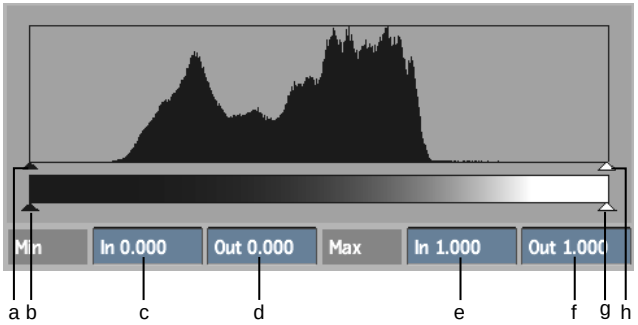
When working with 16-bit floating point images, the values of the histogram are represented on a logarithmic scale. When working with integer images (8, 10, or12-bit), the values of the histogram are represented on a linear scale.

To access the Histogram menu:

1 Click the Histogram tab.



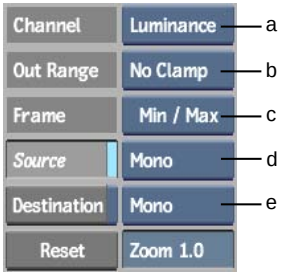
The Histogram menu options are described as follows.



(a) Minimum Input slider (b) Minimum Output slider (c) Minimum Input field box (d) Minimum Output field (e) Maximum Input field (f) Maximum Output field (g) Maximum Output slider (h) Maximum Input slider

Input controls Use the minimum and maximum input sliders or fields to set the input levels of your image.

Output controls Use the minimum and maximum output sliders or fields to set the output levels of your image.



(a) Channel Selection box (b) Out Range Option box (c) Frame Selection box (d) Source View Option box (e) Destination View Option box

Channel Selection box Select which channel you want to work with in the histogram: Luminance, Red, Green, or Blue.

Out Range Option box When using 16-bit floating point images, you can select Clamp to clamp colour and luminance values, or No Clamp to allow pixel floating point values to be less the 0 or more than 1.

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

Select	To Frame:
Min / Max	Based on the minimum and maximum slider values.

Select	To Frame:
Full Range	The complete range of histogram values.
Plot Colour	The plot and reference colours.
Home	Based on a [0:1] horizontal and vertical range.

Source button Enable to show a histogram of the colour values in the front, or source clip. The source colour values are obtained from the current frame. If you input both front and matte clips, the colour values in the front clip that are defined by the white part of the matte are displayed.

Source View Option box Select whether to display all RGB Source histograms or only the histogram you are working on (Mono).

Destination button Enable to show a histogram of the colour values in the result, or destination clip. The destination colour values are obtained from the current frame. If you input both front and matte clips, the colour values in the front clip that are defined by the white part of the matte are displayed.

Destination View Option box Select whether to display all RGB Destination histograms or only the histogram you are working on (Mono). When viewing the Source and Destination histograms at the same time in Mono mode, the Destination histograms are lighter; in RGB mode, the Destination histograms are displayed with a white outline.

Reset button Click to reset the settings defined by the Histogram. The Basic properties of the Colour Corrector are unaffected.

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

Increasing Image Contrast

The slider controls directly below the histogram are the Input Level controls. These controls are used to set the range of colour values used in the image:

- The white triangle on the right sets the maximum value for the range.
- The black triangle on the left sets the minimum value for the range.

You can use the Input Level controls to increase the contrast in the image. For example, if you set the minimum value to 50, pixels with colour values less than 50 are remapped to 0 (black). Pixels with colour values greater than 50 are remapped to the appropriate values. This darkens the image and increases the contrast in the shadow areas.

You can also set the maximum and minimum limits for the colour range by entering the values directly in the Minimum and Maximum Input fields on either side of the histogram.

To increase the contrast in an image:

1. Display the Histogram menu.
2. Click the Master, Shadows, Midtones, or Highlights button to select the parts of the image you want to modify.
3. From the Channel Selection box, select Luminance, Red, Green, or Blue as the channel you want to work with.
4. Position the cursor on the black triangle below the histogram. Drag right to darken the shadow areas in the image.

   The value of the lower limit for the colour range appears in the Minimum Input field.
5. Position the cursor on the white triangle below the histogram. Drag left to brighten in the highlight areas in the image.

   The value of the upper limit for the colour range appears in the Maximum Input field.

   NOTE If you are in Luminance, you can invert the image by reversing the order of the black and white triangles.
6. Click Process to apply the changes to the clip. To cancel the changes, click Reset Basics.

Reducing Image Contrast

The slider controls on the gradient bar below the histogram are the Output Level controls. These controls are used to set the range of colours used in the image:

- The white triangle on the right sets the maximum value for the range.

- The black triangle on the left sets the minimum value for the range.

You can use the Output Level controls to decrease the contrast in the image. For example, suppose that you set the minimum value to 100. A pixel with a colour value of 0 is remapped to 100. Pixels with colour values greater than 0 are remapped to the appropriate values. This has the effect of lightening the image and decreasing the contrast in the shadow areas.

Suppose that you set the maximum value for the range to 200. A pixel with a colour value of 255 is remapped to 200. Pixels with colour values less than 255 are remapped to the corresponding values. This has the effect of darkening the image and decreasing the contrast in the highlight areas.

You can also set the maximum and minimum limits for the colour range by setting the values directly in the Minimum and Maximum Output fields.

To reduce the contrast in an image:

1. Display the Histogram menu.
2. Click the Master, Shadows, Midtones, or Highlights button to select the parts of the image you want to modify.
3. From the Channel Selection box, select Luminance, Red, Green, or Blue as the channel you want to work with.
4. Position the cursor on the black triangle below the Output Level bar. Drag right to lighten the shadow areas in the image.

   The value of the lower limit for the colour range appears in the Minimum Output field.
5. Position the cursor on the white triangle below the gradient bar. Drag left to decrease the brightness in the highlight areas.

   The value of the upper limit appears in the Maximum Output field.

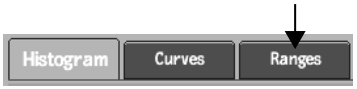
Defining Luma Ranges

Use the histogram in the Ranges menu to modify the shadow, midtone, or highlight ranges.

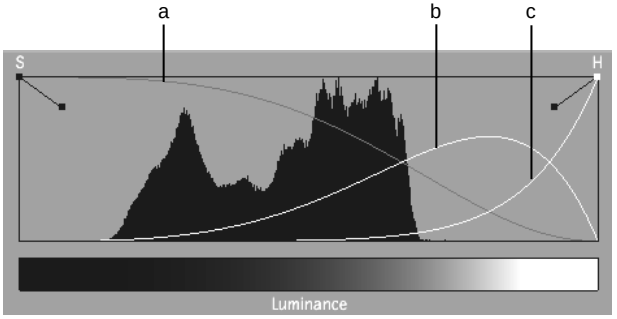
When working with 16-bit floating point images, the values of the histogram are represented on a logarithmic scale. When working with integer images (8, 10, or12-bit), the values of the histogram are represented on a linear scale.

To access the Ranges menu:

1 Click the Ranges tab.



The Ranges menu options are described as follows.



(a) Shadows curve (b) Midtones curve (c) Highlights curve

The X-axis is luma and the Y-axis is weight. The Shadow curve is labelled "S", the Highlight curve is labelled "H", and the Midtones curve is not labelled.

(a) Frame Selection box (b) Source View option box (c) Destination View option box

Frame Selection box Choose how you want to frame the Ranges Histogram.

Select	To Frame:
Min / Max	Based on the minimum and maximum slider values.
Full Range	The complete range of histogram values.
Plot Colour	The plot and reference colours.

Select	To Frame:
Home	Based on a [0:1] horizontal and vertical range.

Source button Enable to show a histogram of the colour values in the front, or source clip. The source colour values are obtained from the current frame. If you input both front and matte clips, the colour values in the front clip that are defined by the white part of the matte are displayed.

Source View Option box Select whether to display all RGB Source histograms or only the histogram you are working on (Mono).

Destination button Enable to show a histogram of the colour values in the result, or destination clip. The destination colour values are obtained from the current frame. If you input both front and matte clips, the colour values in the front clip that are defined by the white part of the matte are displayed.

Destination View Option box Select whether to display all RGB Source histograms or only the histogram you are working on (Mono). When viewing the Source and Destination histograms at the same time in Mono mode, the Destination histograms are lighter; in RGB mode, the Destination histograms are displayed with a white outline.

Reset button Click to reset the settings defined by the Ranges Histogram. The Basic properties of the Colour Corrector are unaffected.

Zoom button Select a vertical zoom value for the histogram display.

To see the effect of adjusting these curves:

1. Adjust the colour balance for each of the Shadows, Midtones, and Highlights channels. Note the effect that this setup has on the image.
2. Display the Ranges menu. Adjust the curves using the tangent handles.

   You should see that the resulting image is different from that in step 1. The difference is the result of the changes you made to the luma curves for the shadows, midtones, and highlights.

Remapping Colour Values

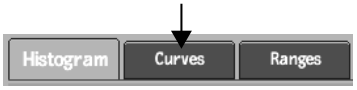
New for this release: With the support of 16-bit floating point images in the Colour Corrector, the Curves menu has been redesigned.

Like the Histogram menu, the Curves menu allows you to remap the colour values for the individual red, green, blue, and luminance channels of the image. However, instead of adjusting the colour values by resetting just the maximum and minimum values for the colour range, the Curves menu allows you to remap any value in the colour range precisely.

To reset any changes to the Curves chart, click Reset. To reset all the changes in the Colour Corrector, click Reset All. The Curves menu also has its own Undo button that is separate from the overall Colour Corrector Undo.

To access the Curves menu:

1 Click the Curves tab.



There is one colour curve for each of the red, green, blue, and luminance channels of an image. The colour curves are generated by plotting the input values for the source image versus the output values for the resulting image.

For integer images, the input values range from 0 to 255, and are plotted along the horizontal axis. The output values range from 0 to 255, and are plotted along the vertical axis. The range for both the input values and the output values is from 0 to 255 in 8-bit mode, and from 0 to 4095 in 12-bit mode.

For 16-bit floating point images, the input values range from 0 to 1, and are plotted along the horizontal axis. The output values range from 0 to 1, and are plotted along the vertical axis. By default, there are 2 points on the curve, mapping 0 to 0 and 1 to 1. Use the Out Range option box to define whether the curves are constant (select Clamp) or linear (select No Clamp) before the first point of the curve and after the last point of the curve.

The default curve for each channel is a diagonal line that extends from the lower-left corner to the upper-right corner of the graph. The default curve represents the initial state in which the colour values for the pixels of the source image are equal to the values for the corresponding pixels of the resulting image. For example, all pixels that have a value of 100 in the source image also have a value of 100 in the resulting image.

The colour values of an image are remapped when you change the shape of a colour curve. Use the Curves Editor to add, delete, or move vertices on the curve. For example, move the end vertices on the curve to change the maximum and minimum values for the colour range. Add a vertex anywhere on the curve to remap a particular colour value.

The Curves Editor behaves in much the same way as an animation curve in the Channel Editor. To edit colour curves, select Add, Delete, or Move from the Edit Mode box.

Generating Mattes

New for this release: With the introduction of 16 bit floating-point support in the Colour Warper, some of the controls in the hue cube and gradient have been updated. See step 7 in the following procedure.

Mattes can be used for several purposes. For instance, use a matte to remove colour spill, hue shift an object in an image for artistic effect, or match colours in a specific range. You can also output a matte for use in other modules such as Action or the Keyer. Use mattes to define the range of colour you want to modify in the result clip.

You can generate up to three mattes using the Selective menu.

TIP To get the best key, make sure you have enough processing speed to see the result in real time while interacting with the Tolerance and Softness boundary handles.

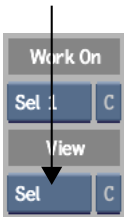
To generate a matte for selective colour correction:

1. Enable Plot and then sample the image.

   A black dot representing the sample appears on the hue cube. A red line also appears in the luma range and the sample is outlined in the 2D or 3D vectorscope. When you create the matte, you can modify the softness and tolerance boundaries to include or exclude the plotted colour.

2. From the Work On box, select a selective (Sel 1, Sel 2, or Sel 3). With each selective, you define a range for a matte by selecting colours in the front clip.

3. From the Selective View box that appears, select Sel.



   The front clip is displayed as a greyscale image.

4 Click one of the Define controls to set the initial softness and tolerance for the matte.

Click:	To define:
Pick Custom	The tolerance range based on a sample from the image. You must drag the cursor over the image to define initial tolerance.
R, G, B, C, M, or Yw	The tolerance range based on the selected colour channel. For example, click Yw to use the yellow channel to set the tolerance. The luma range is set to default tolerance and softness values.
Shadows, Midtones, Highlights	The tolerance range based on the selected luma range. These buttons expand the tolerance and softness boundaries to include all ranges of colours in the image.

The initial softness and tolerance is set for the matte. The range you use to define the matte becomes visible through the greyscale image. The unselected colours remain greyscale.

(a) Original colour visible through the greyscale display.

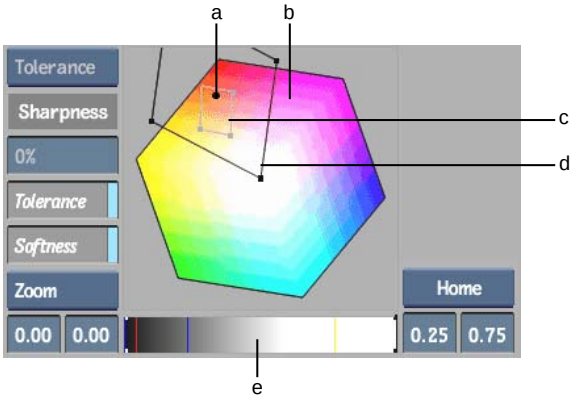
The Active button associated with the selective is enabled when you set the initial softness and tolerance. When an Active button is enabled, the selective's matte will be applied to the result clip. You can disable the Active button at any time if you do not want to apply this matte to the result clip.

TIP To redefine a matte based on a different Define control, click the control you want to use. The matte is reset according to your selection.

---

5 Enable Tolerance and Softness.

The range of colour used to define the matte is shown on the hue cube with tolerance and softness indicators. The light grey outline shows tolerance and the black outline shows softness. When you define a matte with a luma range, it is shown in the luma range—the white line indicates the tolerance and the yellow line indicates softness.



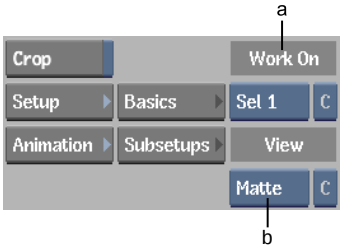
(a) Plotted colour (black dot) (b) Hue cube (c) Tolerance range (d) Softness range (e) Luma range

---

TIP Select Home from the Frame Options box to reset the hue cube to its original size and position.

---

6 To view the matte while you refine it, select Matte from the Selective View box.



(a) Work On box (b) Selective View box

The matte appears in the image window. The black and grey areas of the matte can be colour corrected. The white areas will remain unaffected.

TIP Select Matte view to output a matte for use in another module in Batch. The selective must also be selected in the Work On box.

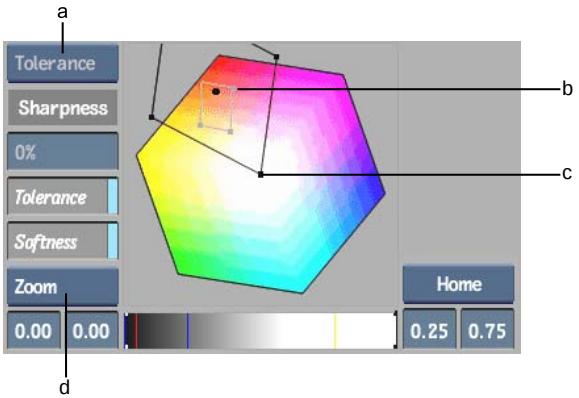
---

7 To refine the matte, do any of the following:

- Adjust softness and tolerance by selecting options from the Adjusting box and then sampling the result clip.

Select:	To:
Tolerance	Add tolerance to the matte.
+Softness	Add softness to the matte.
-Softness	Remove softness from the matte.

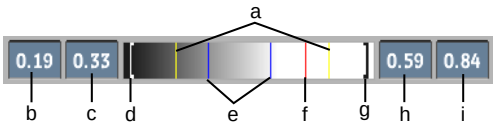
- Adjust softness and tolerance by selecting Move from the Move/Zoom box and then moving the handles of the tolerance or softness outlines on the hue cube. The Tolerance and Softness buttons must also be enabled.



(a) Adjusting box (b) Tolerance handle (c) Softness handle (d) Move/Zoom box

TIP You can zoom the hue cube by selecting Zoom in the Move/Zoom box and then dragging the cube. Alternatively, **Ctrl+spacebar**-drag to zoom. You can pan the hue cube by pressing **spacebar** and dragging.

- Adjust the softness and tolerance in the luma range by dragging the Softness and Tolerance fields.



(a) Softness range (b) Low Softness field (c) Low Tolerance field (d) Low bracket (e) Tolerance range (f) Plot line (g) High bracket (h) High Tolerance field (i) High Softness field

When working with 16-bit floating point images, you can press **spacebar** to pan the gradient bar, and **Ctrl+spacebar** to zoom the gradient. Select Home from the Frame Options box to reset the gradient to the default 0:1 position. If you have softness or tolerance values out of the 0:1 range, select Autoframe from the Frame Options box to view the complete gradient range. Select Plot Colour from the Frame Options box to enlarge the gradient to include the plot and reference colours.

- If the matte appears grainy, drag the Sharpness field to adjust softness and reduce noise.

8 To apply a Gaussian blur, enable G. To apply a box blur, disable G and set the width and height of the blur with the X and Y fields.

9 To invert the matte, enable Inv Selection.

10 Continue fine-tuning the matte until you are satisfied.

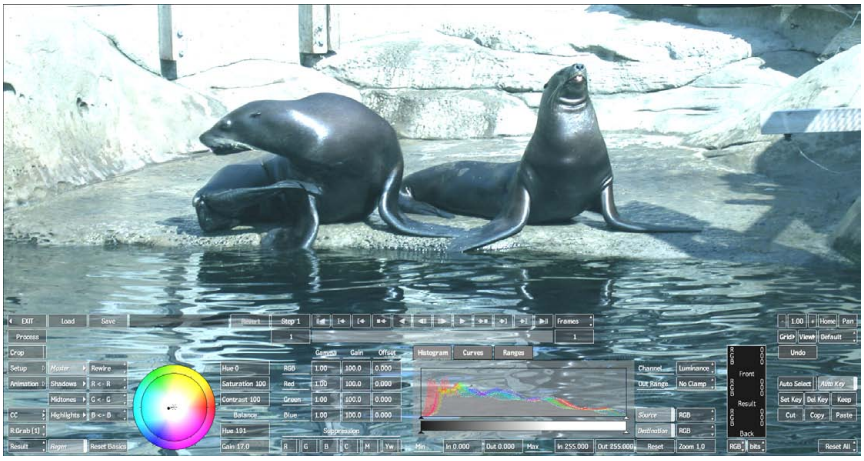
You can select the selective in the Work On box from any menu in the Colour Warper to perform advanced colour corrections on the range defined by the matte. You can also change the view in the Selective View box.

TIP When you switch from the Selective menu to any other menu in the Colour Warper, Result appears in the Selective View box. Select Result to view the result clip.

Overlay User Interface

The Overlay user interface is new for this release. The Overlay user interface is specific to the Colour Corrector and Colour Warper. Its controls are identical to those on the regular user interface except that they are transparent and appear on top of the image. This allows you a greater viewing area when working with large or zoomed-in images.

The Overlay user interface is available when accessing the Colour Corrector or Colour Warper module from the Desktop or from other modules. It is not available when accessing the Colour Corrector or Colour Warper as a Batch node or as a soft effect.



To toggle between the Overlay user interface and the regular user interface:

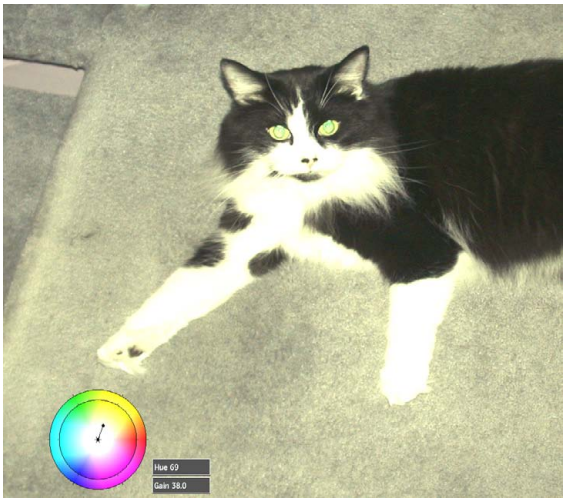
1 Press **Ctrl+`**.

To hide the Overlay user interface:

1 Press **Alt+`**.

In addition to the increased area provided by the transparent controls, you can view even more of your image depending on the action you are performing. For example:

- Panning and zooming in the clip causes the Overlay user interface to disappear completely until the pan or zoom is complete.
- When you play the clip, the Overlay user interface disappears except for the timebar and the current frame number.
- When adjusting a menu parameter, all other user interface elements disappear.



The Overlay user interface does not support multiple views. When you switch to Overlay user interface while using multiple views, your viewport automatically goes to 1-Up until you toggle back to the regular user interface.

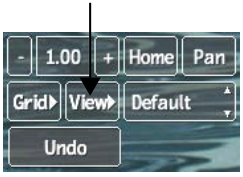
When you access a submenu in the Colour Corrector that does not support Overlay user interface, such as Load or Save, the Overlay user interface is restored when you return to the Colour Corrector.

Changing the Look of the Overlay User Interface Buttons

You can adjust the opacity of the Overlay user interface buttons to suit your preference, and even invert the elements in the interface so that white outlines and text appear in a muted grey.

To set preferences for the Overlay user interface:

1. While in the Overlay user interface from the Colour Corrector module, click View.



The Overlay user interface preferences are displayed.

Use the Opacity field to increase or decrease the opacity of the buttons. An opacity of 1.0 displays the buttons as they look in the regular user interface. You can also use the following hot keys:

Press:	To:
Ctrl+Alt+(num pad)+	Increase the opacity of the Overlay user interface.
Ctrl+Alt+(num pad)-	Decrease the opacity of the Overlay user interface.

TIP You can repeatedly press the hot keys to increase or decrease opacity, or simply hold down the hot keys until the desired opacity is reached.

Enable Invert (or press **Ctrl+Alt+(num pad)*** to switch the Overlay user interface outlines and text from white to grey.

Using Garbage Masks as Distort Splines

New for this release: You can import garbage mask setups into Distort to use as splines.

Garbage mask setups are created through the Keyer or the GMask node in Batch. You can import a saved garbage mask setup into Distort and use it as a spline. Note the following about how Distort interprets settings in the imported garbage mask setup:

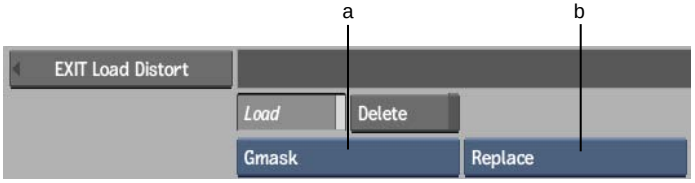
- If the imported garbage mask has animation on its axes, the animation is recreated in axes above the main axis in the Distort schematic.

- If the imported garbage mask has shape animation or explicit animation on the gmask splines, the same animation on the source and destination splines is transformed into shape animations.
- If the imported garbage mask includes tracking data, the offset parameters on the geometry are duplicated on the offset parameters of both the source and destination splines.
- Garbage mask setups that use multiples axes to manipulate instances of splines are translated to Distort instances. See Instancing Distort Splines on page 146.

To import a garbage mask setup into Distort:

1. Do one of the following:
   - In the Distort menu, click Load to open the file browser.
   - From the Distort node in Batch, click NodeSetup to display Distort's Setup menu, then click Load Node to open the file browser.
2. From the Load Type box, select Gmask.

   The browser automatically points to the default Gmask folder.



(a) Load Type box (b) Load Mode box

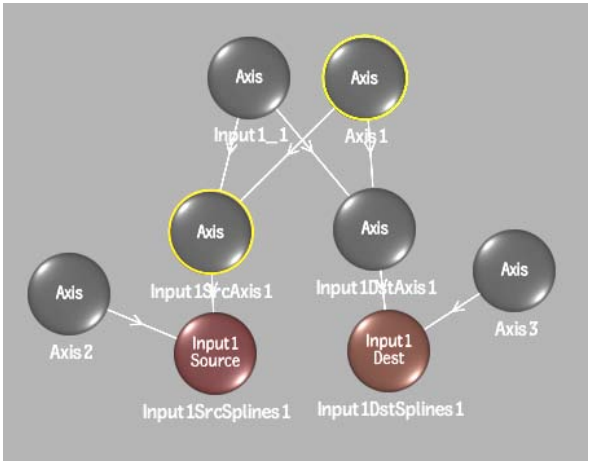
3. From the Load Mode box, select whether you want the Gmask setup to append to or replace any existing Distort splines in your scene.
4. From the file browser, select the name of the setup to load.

TIP A library of preset garbage mask setups is available. To load a preset garbage mask, navigate to the */usr/discreet/<product_name>/gmask/default* directory.

Instancing Distort Splines

New for this release: Due to the support of garbage mask splines in Distort, you are now able to use instances of splines to better manage your Distort schematic.

When you add multiple axes above a pair of source and destination splines, each axis can be used to manipulate a separate instance of the pair of splines. You can then use the Distort settings for each axis to manipulate the instance of each spline. In the following example, the Input1_1 and Axis1 axes can be used to change instances of the source and destination splines. Axis2 and Axis3 cannot be used in this manner, since they are only attached to one input spline.



Resize Node

The Resize node in the Modular Keyer 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.

See Resize Settings on page 77.

New for this release: You can add a Resize node to any part of the pipeline except to the blend node pipes. Typically, you would add it to the branch for the back clip, since the resolution of the front and matte clip are usually the same.

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.

Action

12

Topics in this chapter:

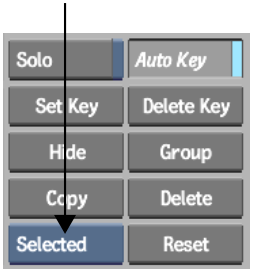
Soloing Objects

In Action, you can now easily isolate an object while hiding all other objects.

Use the Solo button to hide all other objects except the selected object. Using Solo is useful for identifying an object in a scene with many objects, without having to hide all of the other objects .

To solo an object:

1. In the schematic, select the object that you want to solo.
2. From the Selection box, choose Selected.



3 Enable Solo.

All other objects in the scene are hidden, regardless of whether they are set to Hide or not. When Solo is disabled, the Hide settings are restored.

TIP You can leave Solo enabled and select different objects in the schematic to view each object separately.

To solo a branch:

1 In the schematic, select the parent of the branch that you want to solo.

2 From the Selection box, choose Branch.

3 Enable Solo.

All other objects in the scene are hidden, regardless of whether they are set to Hide or not. When Solo is disabled, the Hide settings are restored.

Auto 3D Tracking

The automatic 3D tracker can track the motion in your image based on camera properties or the motion of objects in the scene. When you track based on camera properties, you define settings based on the device used to acquire the images you are tracking. You can also generate a fixed camera that results in a moving cloud of 3D points. Object tracking allows you to track moving objects in the scene and conform the results to any available camera.

Depending on the result of your camera or object tracking, you can fine-tune the track with various filtering options.

Automatic Camera Tracking

You can perform automatic 3D tracking based on the camera properties of the device used to acquire the image sequence you want to track. You can specify these properties, or let the 3D tracker automatically detect the best solutions for the analysis. Even if you want to perform object tracking on specific moving areas of the scene, you should start with an analysis using camera tracking to create a properly tracked camera.

To create an automatic 3D camera track:

1. From the 3D Tracker option box, select Automatic, then select Camera Tracking to display the Camera Tracking menu.
2. In the Track Media field, specify the media number of the front clip you want to track. A value of 0 indicates that the Back clip is tracked.

3 Decide if you need to use a matte in the analysis. For example, in an image sequence of a busy street, you can create a matte of moving elements (such as cars and people) to isolate this area from the analysis.

Select:	To:
Matte On	Use a matte to delimit the tracking results. White areas of the matte are considered for calculating the solution, and black areas are ignored.
Matte Invert	Use an inverted matte to delimit the tracking results. Black areas of the matte are considered for calculating the solution, and white areas are ignored.
Matte Off	Not use a matte.

4 If you are using a matte, specify which matte in the Matte Media field.

NOTE Use a matte made from real frames. Keyer or Gmask outputs do not work as a matte for 3D tracking. Alternatively, in Batch, you can use connected input and matte clips as the media to be tracked. In this case, tracking becomes a foreground process.

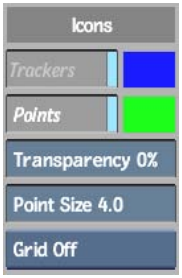
5 Enable Backward to track the image sequence backward after the forward tracking has completed. This option takes longer, but you may get better results.

6 Set Camera and Film Back options. See Setting Camera Properties for Automatic 3D Camera Tracking on page 155.

7 If needed, adjust the scale of the trackers in the Scale field.

Smaller trackers can speed up the calculation, while larger trackers make the analysis more robust with regard to image noise and variations. A general rule is to increase the scale of the trackers when tracking high-resolution footage (2K or larger) that contains more noise or less sharpness.

8 From the Icons section, select tracker display options.



Enable:	To display:
Trackers	2D tracks. Use the colour pot to select the colour of the tracks.
Points	3D points. Use the colour pot to select the colour of the points. The points option is also available in the Display section of the Setup menu (or with the hot key **Alt+D**), so that you can enable the display of 3D points while working in other Action menus. 3D points are viewable in 3D in all Camera views in Action (Side, Front, Top, or Camera) to help you position objects in the reconstructed scene.

You can also adjust the transparency of the trackers and points and the size of the points in your image.

Use the Grid box to display a grid so that you can position objects in the scene more accurately. The same Grid box is available in the Display section of the Setup menu.

9 Click Track. You can see a progress indicator beside the Track button. You can interrupt the analysis and resume it by clicking Track again.

Most of the time, tracking occurs in the background, allowing you to continue working while tracking. In Batch, you can use connected input and matte clips as the media to be tracked. In this case, tracking becomes a foreground process.

After tracking has completed and you press Confirm, the Track button changes to Calibrate, and you can see the 2D tracks (the blue squares in the following example) and 3D points (green crosses) in your image.

Image courtesy of Behavior Communications Inc.

A camera called *Camera_3dt_sync* is also generated when an analysis is performed. This camera synchronizes to the results of your auto 3D tracking, and any further changes you make to the 3D track are reflected in this camera.

NOTE Do not manually modify the *Camera_3dt_sync* parameters before finalizing your auto tracking results by converting points to axes and resetting the tracker. These modifications will be lost, as *Camera_3dt_sync* is linked to the 3D tracker. If you reset the 3D tracker settings, the synchronized camera becomes a regular Action camera that you can modify.

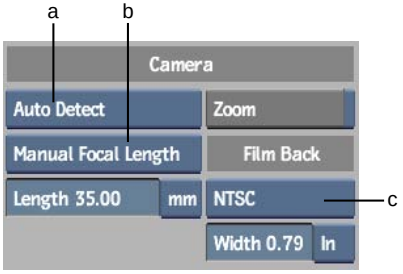
---

10 If you are satisfied with the results of the tracking analysis, proceed to Defining the Auto Track Ground Plane on page 156 and Converting the 3D Auto Tracking Results on page 163. If you want to tweak your track results, see Fine-tuning the 3D Auto Track on page 161.

Setting Camera Properties for Automatic 3D Camera Tracking

Before analysing the 3D motion, define your camera properties. The camera represents the device used to acquire the sequence of images you are tracking. For each camera parameter, you can choose to let the auto 3D tracker calculate the value automatically or you can specify the value yourself. This information can help the automatic 3D camera tracker calculate better results.

NOTE You can set the camera properties before initial tracking, or after initial tracking when calibrating or refining the track.



(a) Camera Type box (b) Focal Length box (c) Film Back box

The Camera and Film Back controls are described as follows.

Camera Type box Select the type of camera used to shoot the scene.

Select:	To:
Free 3D Motion	Track a clip shot with a free-moving camera.
Pan and Tilt	Track a rotating clip shot with a camera on a tripod.
Auto Detect	Automatically detect the camera type and track accordingly (default value).

Focal Length box If you know the properties of the camera that shot the scene, switch to Manual Focal Length, and set the length and film back settings.

NOTE Before initial tracking, if Zoom is enabled, Auto Focal Length is the only choice. You can switch to Manual Focal Length only after initial tracking to clean up the FOV animation before refining or calibrating.

Length field Enter the focal length.

Focal Length Units box Specify inches or millimeters to be used as the length.

Zoom button Enable to calculate the zoom value of the reconstructed camera for each frame (assuming the camera that shot the tracked clip has a variable zoom value). By default, Zoom is disabled (that is, the track analyses with a fixed zoom).

Film Back box Select the film back size preset of the camera that shot the scene.

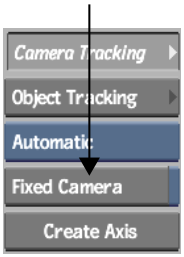
Width field Enter the width of the film back.

Width Units box Specify inches or millimeters to be used as the width.

Generating a Fixed Camera

New for this release:

Typically, 3D tracking results in a moving camera and fixed points, but you can set the camera to be fixed and the points to move. Depending on the footage you are tracking, you can generate a fixed camera and a moving 3D point cloud of generated axes that are attached to a single parent axis. To do this, enable the Fixed Camera button either before of after the initial analysis.

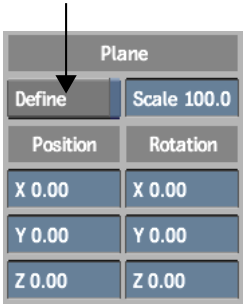


Defining the Auto Track Ground Plane

Although it is not mandatory to define a ground plane in your image, it helps orient the reconstruction of the cameras.

To define the auto track ground plane:

1 In the Plane section of the Camera Tracking menu, enable Define.



NOTE If you selected Pan and Tilt as the camera type (or if the Auto Detect switched to Pan and Tilt) the Plane Define button is not available. The rest of the settings in this group are still available, but only for orientation purposes.

2 Select a minimum of three points in the image that represent the plane of the X/Z axes, such as the ground, a table, or any flat surface.

The selected points appear as red squares with white crosses.

TIP You may want to disable the tracker temporarily and point display options (or raise the transparency level) to help you find and select the plane points.

3 Use the Position and Rotation controls to define the orientation of the ground plane.

4 Use the Scale field to specify the scale of the scene.

5 Disable Define.

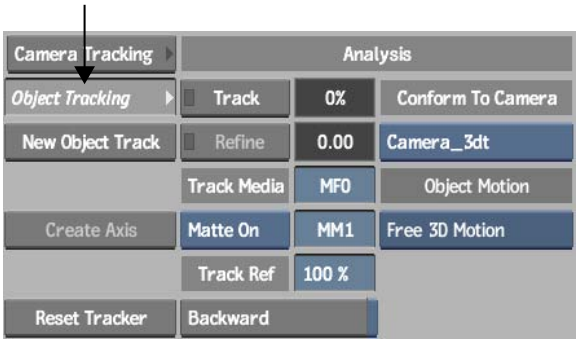
Automatic Object Tracking

New for this release: Use the 3D tracker to track multiple moving objects in your scene.

If the image sequence you are tracking has multiple moving objects, you can perform object tracking to track these objects relative to the same camera. You can select the camera to which the object tracking results conform, or use the camera generated from previous camera tracking analysis. For example, you can perform a camera tracking analysis on the complete scene, then use mattes to perform multiple object tracking passes focusing on various moving objects in the scene. Each result can be converted to separate axes, but all results conform to the camera created by the original camera tracking.

To create an automatic 3D track based on object properties:

1 From the 3D Tracker option box, select Automatic, then click Object Tracking.



2 Click New Object Track.

3 If previous tracking data exists, a message appears indicating that the data will be discarded. After you confirm, the Track Media field and Conform To Camera box are automatically set to match the previous 3D track data. You can change these settings.

4 Select either to use a matte or matte invert, and specify the Matte media. If you are performing multiple object tracking passes, it is a good idea to use a matte to exclude the objects to be tracked next.

White areas of the matte are considered for calculating the solution; black areas are ignored (or vice-versa for Matte Invert).

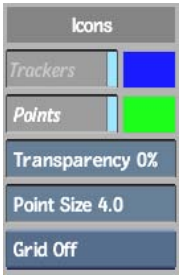
5 Enable Backward to track the image sequence backward after the forward tracking has completed. This option takes longer, but you may get better results.

6 From the Object Motion box, make a selection about the object you are tracking in relation to the camera.

Select:	To:
Free 3D Motion	Track an object moving independantly from the camera.
Orbit Around Cam	Track an object rotating around the camera, or far away from the camera.
Auto Detect	Automatically detect the motion type of the object and track accordingly. For small objects, Auto Detect may not be able

Select:	To:
	to establish the proper motion. In this case, select Free 3D Motion or Orbit Around Cam.

7 From the Icons section, select tracker display options.



Enable:	To display:
Trackers	2D tracks. Use the colour pot to select the colour of the tracks.
Points	3D points. Use the colour pot to select the colour of the points. The points option is also available in the Display section of the Setup menu (or with the hot key **Alt+D**), so that you can enable the display of 3D points while working in other Action menus. 3D points are viewable in 3D in all Camera views in Action (Side, Front, Top, or Camera) to help you position objects in the reconstructed scene.

You can also adjust the transparency of the trackers and points, and the size of the points in your image.

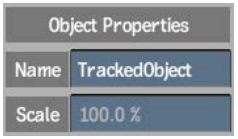
Use the Grid box to display a grid so that you can position objects in the scene more accurately. The same Grid box is available in the Display section of the Setup menu.

8 Click Track.

Most of the time, tracking occurs in the background, allowing you to continue working while tracking. In Batch, you can use connected input and matte clips as the media to be tracked. If this case, tracking becomes a foreground process.

You can see a progress indicator beside the Track button. You can interrupt the analysis and resume it by clicking Track again. After tracking has completed and you press Confirm, the Track button changes to Calibrate, and you can see the 2D tracks and 3D points in your image.

9 Set the name and scale of the track in the Object Properties section.



Name field Displays the name to be given to the parent axis after you convert the analysis into axes. You can rename it to a better name to reflect which object you are tracking.

Scale field Displays the scale of the tracked object. After tracking has occurred, you can set the scale of the tracked object. Since you are tracking a specific object as part of an image, setting the relative scale of the object in relation to the image helps you to position objects in the reconstructed scene when you convert the 3D points into axes.

NOTE After creating axes, if the tracker is not reset, the axes stay in sync with the 3D tracker. Therefore, changing the scale value also has an impact on the created axes.

---

10 If you are satisfied with the results of the tracking analysis, proceed to Converting the 3D Auto Tracking Results on page 163. If you want to tweak your track results, see Fine-tuning the 3D Auto Track on page 161.

Fine-tuning the 3D Auto Track

If the initial auto tracking does not give desired results, you can use some or all of the Filter options to calibrate and refine your track analysis. These procedures are not necessarily required, but depending on your image and the initial tracking, may give better tracking results.

To fine-tune the 3D auto track:

1 Use the Quality slider to adjust the number of good trackers kept. The higher the quality setting, more low quality trackers are selected, such as trackers that drift off their initial reference point. Click Delete to delete the selected trackers.

Trackers of lower quality may hinder the accuracy of the camera tracking.

NOTE After you have made a change that requires the 3D tracking analysis to be refined or calibrated, notice that the LED next to the Refine and Calibrate buttons turns yellow. This signifies that a Refine or Calibrate is required, but you do not have to perform it until you have completed your tracker selections.

---

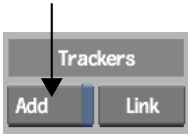
2 Adjust the Short slider to select short duration trackers, that is, trackers that only track a feature for a few frames. Click Delete to delete the selected trackers, leaving the longer duration trackers intact.

3 You can manually select and delete trackers from the image that you feel are not tracking properly. Do one of the following:

- To select an individual tracker, click the tracker, and then click Delete.
- To select multiple trackers in the same area, **Ctrl**-drag a selection box over a series of trackers, and then click Delete.
- To add another tracker to a multiple selection, **Shift**-click the tracker, and the click Delete.

- With Delete mode selected in the Edit Mode box, select trackers in the image.

4 If you want to add a tracker manually, enable Add and click an area of the image to track from this area automatically.



A tracker may or may not be added, depending on the ability of the track analysis algorithm to find an appropriate feature to track in this area.

5 If the analysis creates different trackers that refer to the same feature in the image, you can link these trackers. Press **Shift** and select two or more trackers from the image, and then click Link.

For example, an element leaving the scene at frame 28 and returning at frame 50 may result in two different trackers attached to the same element in the image. In this case, select the trackers and click Link to teach the algorithm that these trackers are related to the same feature in the scene.

To refine or calibrate the 3D track:

1 Depending on the changes you have made, you can choose to refine or calibrate the 3D track. Do one or both of the following:

- Click Refine.

(a) Refine button (b) Pixel Error value

The track analysis uses the current results as a starting point, and refines from this point.

Click Refine again to stop the process once an acceptable pixel error value is reached. The pixel error value is a representation of the distance of the 2D tracks from the computed 3D points.

TIP The refine process is footage-dependant, so your acceptable pixel error value may change depending on what is tracking. Since the refine process continues until you stop it, as a general rule, if the pixel error value does not change for a length of time (for example, 30 seconds), you can stop the refine process. The lower the pixel error value, the more accurate the reconstructed track is.

- Click Calibrate.

NOTE The calibrate operation deletes all previous 3D points and starts over based on the new information. Depending on your footage, and how many trackers you added, deleted, or linked, multiple calibrations may yield different results.

Converting the 3D Auto Tracking Results

When you are satisfied with the results of the 3D auto camera or object tracking analysis, you can convert the selected reconstructed points to actual axes in your scene.

To create axes from the 3D tracking results:

1 Select the points in the image that you want converted to axes. Selected points are displayed as green squares with red crosses.

TIP Press **ctrl** and draw a rectangle over multiple points to select them. If any points are already selected, they become unselected.

2 Click Create Axis.

If you performed Camera Tracking, selected points are converted to axes with a parent axis called *Points_3dt_sync*. If you performed Object Tracking, the parent axis name corresponds to the name in the Object Properties Name field. The axes synchronize to the results of your auto 3D tracking, and any further changes you make to the 3D track are reflected in these axes.

NOTE If you want to apply settings manually to a synchronized axis, change its name so that it does not update automatically when 3D Track settings are changed. If you reset the 3D Track settings, the synchronized axes become regular Action axes.

3 Exit the 3D Track menu. Use the created axes to view the reconstructed camera motion.

You can attach objects such as surfaces, 3D text, and 3D models to the new axes to help position them in 3D space.

You can export the newly created camera, as well as the axes and points created from the 3D tracker analysis, to *.fbx* format for use in other 3D applications.

Using Particle Presets

A number of particle presets are included in Action, such as a fog effect to add to your scene. These presets can also help you learn how the particle system works. The presets are divided into a number of categories and are easily added to your scene from the Action node bar.

Note the following when using particle presets:

- The presets are optimized for the HD format. If you are working in a different resolution, a preset attempts to scale to the resolution, but may render slightly differently.

- Depending on the preset loaded, some Action Setup menu Rendering settings may need to be changed, such as shading, geometry resolution, colour clamping, and z-buffer settings. These changes occur automatically after you confirm.

- Some presets have *.psd* textures associated with them, and are added to the Media list and Desktop when the preset is loaded into Action. You should save these textures from the Desktop into an appropriate library.

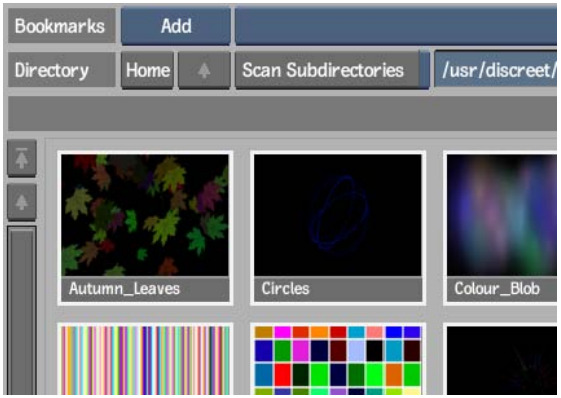
To add a particle preset:

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

   The Particle Preset menu and file browser appear, pointing to the default location of the presets: *usr/discreet/<product home>/particle_presets*.

2. Navigate through the subfolders to find the particle preset you want to load.

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



3 Once you select a preset, you may be asked to confirm certain Setup menu changes. The particle preset is then appended to your Action scene.

NOTE If you choose not to confirm the changes, the preset still loads, but you may not see the intended results.

For a listing and description of all available particle presets, see Particle Presets Catalogue on page 167.

Particle Presets Catalogue

13

Topics in this chapter:

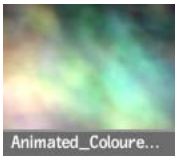
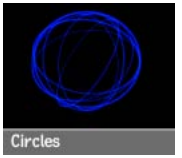
Introduction

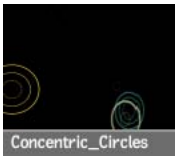
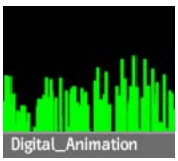
A number of particle presets are included in Action, such as fog or fireworks effects to add to your scene. These presets can also help you learn how the particle system works. The presets are divided into a number of categories and are easily added to your scene from the Action node bar.

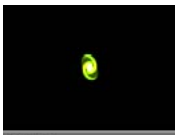
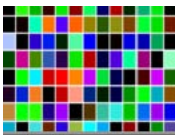
Use the following tables to get a quick overview of the particle presets, along with best use comments, if applicable.

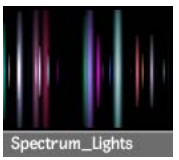
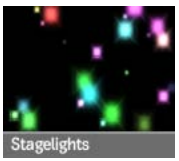
Abstract/Backdrops

Use these presets as backdrops or to create abstract art particle effects in your scene.

Name/Preview	Description/Comments
	**Animated coloured background** Use Number and Lifetime to slow down or speed up the animation. Try with Z Rotation and Spin parameters.
	**Falling maple leaves** Use Number, Size, and Speed settings. Try with the Rotation and Spin parameters.
	**Rotating circles** Use Number, Lifetime, and Size settings. Try with the Spin parameter.
	**Animated coloured gradients** Use Number, Lifetime, and Size settings. Try with Blending modes and Lighting.
	**Coloured cloud texture** Use Number and Size settings. Try with Z Rotation and Lighting.
	**Animated coloured vortex** Play with circ_gradient_ax (especially Y Scaling). Try with Z Spin parameter. For a nice spiral effect, try Timesteps = 2 and Trail values up to 50.

Name/Preview	Description/Comments
 Concentric_Circles	**Animated concentric circles** Use circles_emitter Number and Lifetime fields to set the amount of circles.
 Concentric_Circles_2	**More concentric circles** Use Number and Size. Speed up or slow down animation with Lifetime.
 Crazy_Serpentine	**Random circular patterns** Try with the size manipulator (Magnitude). Modify the Power animation in vortex manipulator.
 Crazy_Stars	**Pulsing coloured stars** Use Number and Size settings.
 Digital_Animation	**Animated rectangles** Use Number, Lifetime, Size, and Speed settings. Change colour using light_colour.
 Dots_Spiral	**Glowing spiral dots** Animate Size setting. Try with Magnitude in size manipulator. Change colour using the Geometry parameters of dots_generator.

Name/Preview	Description/Comments
	**Spiral dots** Use the Size setting. Try with Magnitude in the size manipulator and with Power in the vortex manipulator (try Power = 5). Move the camera and see the effect of gravity.
	**Moving horizontal glows** Use Number, Lifetime, and Size settings to characterize the glow. Try with Magnitude in the size manipulator to scale the glows.
	**Hypnotic spiral** Try with the camera settings.
	**Circular illuminated glow** Change colour by colour correcting the white media. Use Number and Size to characterize the ring. Use the pos_scale axis to set the position and rotation of the ring in the scene.
	**A motif of coloured lines** Speed sets the amount of stripes. Try with Colour variance in motif_generator menu.
	**Pop Art-like effect** Try with a close-up shot of a face as REPLACE_ME media. Zoom in to have fewer coloured squares in the frame, or zoom out to have more.

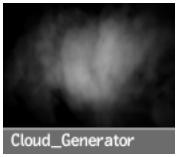
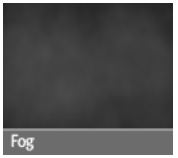
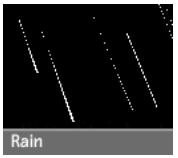
Name/Preview	Description/Comments
	**Moving psychedelic texture** Use Magnitude in the size manipulator to set pattern size. Try with the Trailsize field in the Generator menu.
	**Random animated textures** Use Number, Lifetime, Size, and Speed settings to characterize the texture. Try with the Spin parameters to set the animation.
	**Black sphere emitting coloured streaks** Use Number and Speed settings. Try with the Tailsize field.
	**Moving cloudy sky** Use Number and Colour_V settings to characterize the sky. Use Position parameters in the sky_motion manipulator to set how the clouds are moving.
	**Animated vertical glows** Use Number, Lifetime, and Size to characterize the lights. Try with Magnitude in the size manipulator to control the scaling amplitude of the lights.
	**Stagelight** Use Number, Lifetime, and Size. Try with Magnitude in the size manipulator to control the scaling amplitude of the stagelights.

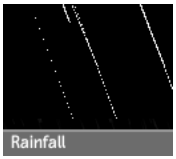
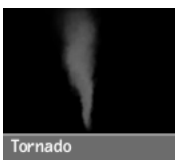
Name/Preview	Description/Comments
 Star_Spiral	**Spiral made of stars** Use Numer, Lifetime, and Size. Try with Magnitude in the size manipulator to control the scaling amplitude of the stars.
 Text_Atom	**Animated 3D Text birth**
 Text_Distort	**Distorted 3D Text** Use Number, Lifetime, and Size. Try with Rotation_V, Spin, and Spin_V parameters. Try also to change the light's colour and the glow_generator's Material parameters.
 Text_Dots	**3D Text made of dots** Use Number, Lifetime, and Size. Try with Rotation_V, Spin, and Spin_V parameters. Try also to change the light's colour and the glow_generator's Material parameters.
 Text_Glow	**Glowing 3D Text** Use the Size setting. Try with Magnitude in the size manipulator.
 Texture	**Texture creation** Use the Size setting. Try with the Rotation_V parameter. Play with the texture_ax parameters.

Name/Preview	Description/Comments
 Glow_Scanline	**Vertical and horizontal moving glows** Use Number, Lifetime, and Size to characterize horizontal & vertical glows. Use Magnitude in the V_size and H_size manipulators to set the scaling's amplitude. Use Magnitude in the V_horizontal_osc manipulator to set the horizontal motion's amplitude. Change colour with the light.

Atmospheric

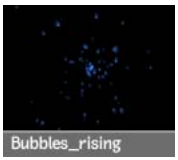
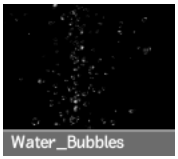
Use these presets to create weather-like effects, such as clouds, rain, or snow.

Name/Preview	Description/Comments
 Cloud_Generator	**A one-frame cloud** Use Number and Size settings. Try with the texture's Axis settings and the geometry's Magnitude and Scaling settings.
 Fog	**Fog effect** Use Number, Lifetime, Size, and Speed to characterize the fog.
 Heavy_Fog	**Heavy fog effect** Use Number, Lifetime, Size, and Speed to characterize the fog.
 Rain	**Rain effect** Use Number and Speed to trigger the amount of rain. Colour correct the white media's Front, and set rain_generator Transparency to improve blending.

Name/Preview	Description/Comments
 Rainfall	**Rainfall effect** Use Number and Speed to trigger the amount of rain. Colour correct the white media's Front, and set rain_generator Transparency to improve blending. Use gravity's Rotation_Y to make it windier.
 Snow	**Snow effect** Use Number, Size, and Speed for characterizing the snow.
 Snow_Graphical	**Graphical snow** Use Number, Size, and Speed for characterizing the snow. Play with lighting.
 Snow_Windy	**Snow storm** Use Number, Size, and Speed for characterizing the snow. Try with wind's Magnitude and axis animation to mimic the wind.
 Surfing_Clouds	**Flying into the clouds** Use gravity's Rotation_X expression to stay above clouds, or dive into them.
 Tornado	**Tornado effect** Use Magnitude in the size manipulator to scale the tornado. Use Magnitude and Power in the twist manipulator to alter the shape of the tornado.

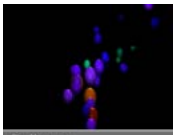
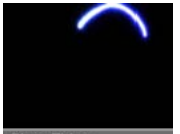
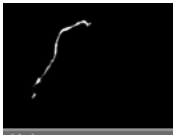
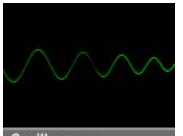
Liquids

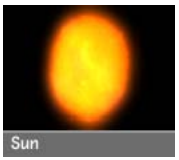
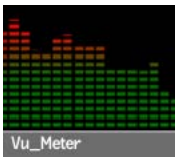
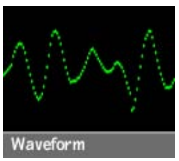
Use these presets to create fluid effects, such as bubbles.

Name/Preview	Description/Comments
	**Big bubbles** Use Number, Size, and Speed settings to characterize the bubbles. Process with Motion Blur enabled for better rendering.
	**Bubbles coming toward camera** Use Number, Size, and Speed settings to characterize the bubbles. Use Mass_V to randomize the bubble motion.
	**Sun rays under water** Use water_generator for the water and rays_generator for the rays coming from the sun above the surface.
	**Water surface** Try the Spin_V parameters to animate the sea. Customize by adding more lights to the scene.
	**Bubbles in water** Use Number, Size, and Speed to characterize the bubbles. Try the Spin & Spin_V parameters to control the flow.

Miscellaneous

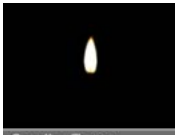
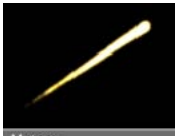
Use the miscellaneous presets to add dirt or scratches to your scene, as well as many other effects.

Name/Preview	Description/Comments
 Balloons	**Releasing balloons** Use Number, Size, and Speed settings to characterize the balloons. Try the Spin & Spin_V parameters to control the flow.
 Blue_Thing	**Moving glow following a path** Use Number, Lifetime, and Size settings to characterize the glow. Try with Rotation and Spin for motion. Adapt the path.
 Dirt	**Adds Dirt** Use Number and Size settings. Try with variance.
 Hair	**Adds Hair** Use Number and Size settings. Try with variance. hair_generator4 is linked to hair_generator1, and hair_generator3 is linked to hair_generator2.
 Oscilloscope	**Oscilloscope effect** Change the colour by using beam_generator's Diffuse parameter. Use axis to control the beam's amplitude, frequency, and phase.
 Paint_Effect	**Paint effect** Use Number, Lifetime, and Size settings to customize the paint effect. Try changing the texture media (white square, soft cloud, etc.).

Name/Preview	Description/Comments
 Scratch	**Adds scratches** Use axis to control the scratch's position, the amplitude of the horizontal motion of the scratch, and the frequency of the scratch. Hide some generators for fewer scratches, or duplicate generators if you want more scratches.
 Simple_Sun	**Sun effect** Processes quicker than Sun.
 Sun	**Astronomical view of the sun** Slower to process than Simple_Sun.
 Vu_Meter	**Animated vu-meter** Use beamgen Speed, PD_axis Scale_Z, and beam_emitter positionning to modify the number and width of stripes.
 Waveform	**Animated audio signal monitor effect** Use axis to control the beam.

Pyrotechnic

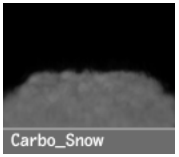
Use the pyrotechnic presets to create fire and flame effects.

Name/Preview	Description/Comments
 Birthday_Sparkles	**Sparkles from a birthday candle**
 Candle_Flame	**Candle flame effect**
 Fire_ring	**Ring of fire**
 Gas_Flame	**Gas flame effect**
 Meteor	**Meteor burning in space**
 Rocket_propulsion	**Rocket engine output**

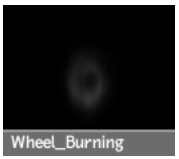
Name/Preview	Description/Comments
	**3D Text effect**

Smoke

These presets allow you to create various smoke effects.

Name/Preview	Description/Comments
	**Snow coming toward camera** Use Size to characterize the snow. Use the Magnitude parameter of the gravity manipulator to set the snow's motion.
	**Dark smoke** Use on a light background. Use Number, Lifetime, Size, and Speed to characterize the smoke. Use the Magnitude parameter of the gravity manipulator to set the smoke's motion.
	**Smoke from a fire** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude parameter of the gravity manipulator to set the smoke's motion. Modify the fire colour using the fire_light_1 and fire_light_2 colour parameters.
	**Smoke produced by a shotgun**

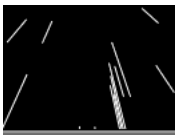
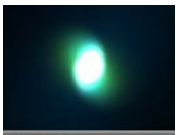
Name/Preview	Description/Comments
 Heavy_Smoke	**Heavy smoke** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude parameter of the gravity manipulator to set the smoke's motion.
 Jet_Engine_Smoke	**Smoky jet engine output** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude parameter of the gravity manipulator to set the smoke's motion.
 Smoke	**Smoke effect** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude parameter of the gravity manipulator to set the smoke's motion.
 Smoke_Lookat	**Smoke pointing to a target** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude parameter of the gravity manipulator to set the smoke's motion.
 Smoke_Track	**Smoke to track** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude parameter of the gravity manipulator to set the smoke's motion.
 Train_Smoke	**Smoke produced by a steam engine in motion** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude parameter of the train_speed manipulator to control the train's speed and its effect on the smoke.

Name/Preview	Description/Comments
 Vapour	**Vapour rising from the ground** Use Number, Lifetime, Size, and Speed settings to characterize the vapour. Try the Rotation_V and Spin_V settings to control the vapour animation
 Wheel_Burning	**Animated burning wheel effect**

Space

Use these presets to create astronomy effects.

Name/Preview	Description/Comments
 Aurora	**Aurora borealis effect** Use Number and Size to characterize the aurora. Use Lifetime to speed up or slow down the animation. Change colours by modifying the left_light & right_light parameters.
 Blue_Star	**Slow moving star** Use Number, Lifetime, Size, and TrailSize settings.
 Blue_Star_2	**Slow moving star** Similar to Blue_Star, but with a bigger hotspot in the middle. Use Number, Lifetime, Size, and TrailSize settings.

Name/Preview	Description/Comments
 Blurry_Sparks	**Emission of blurry sparks** Use Number, Lifetime, Size, and Speed settings to set how the sparks are emitted. Use the transparency, gravity, and size manipulators to specify the behaviour of the falling sparks.
 Eye_Strike	**Moving glows** Try with circ_gradient1_ax and circ_gradient2_ax parameters.
 Light_Speed	**Reaching light speed in space** Use Number, Speed, and TrailSize settings. Use the speed manipulator's power to set the Z acceleration.
 Pulsar	**Pulsating glow** Use Number, Lifetime, Size, Speed, and TrailSize settings.
 Red_Pulse	**Pulsating red star** Use Number and Speed settings to customize the star. Set the star's intensity with transparency magnitude.
 Star	**Glowing animated star** Use Number, Size, and Speed settings to characterize the star. Change colour using the light's Colour settings.

Name/Preview	Description/Comments
	**Flickering stars in the sky** Modify Number at frame 0 to control the amount of stars in the sky.
	**Flying into space** Use Number and Speed settings.