Autodesk® **Flame®** Premium 2013

A Discreet® creative finishing products

20th Anniversary Edition



User Guide

Autodesk®

© 2012 Autodesk, Inc. All rights reserved. Except as otherwise permitted by Autodesk, Inc., this publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose.
Certain materials included in this publication are reprinted with the permission of the copyright holder.
Portions related to MD5 Copyright © 1991-2, RSA Data Security, Inc. Created 1991. All rights reserved. License to copy and use this software is granted provided that it is identified as the "RSA Data Security, Inc. MD5 Message-Digest Algorithm" in all material mentioning or referencing this software or this function. License is also granted to make and use derivative works provided that such works are identified as "derived from the RSA Data Security, Inc. MD5 Message-Digest Algorithm" in all material mentioning or referencing the derived work. RSA Data Security, Inc. makes no representations concerning either the merchantability of this software or the suitability of this software for any particular purpose. It is provided "as is" without express or implied warranty of any kind. These notices must be retained in any copies of any part of this documentation and/or software.
The following are registered trademarks or trademarks of Autodesk, Inc., and/or its subsidiaries and/or affiliates in the USA and other countries: 123D, 3ds Max, Algor, Alias, Alias (swirl design/logo), AliasStudio, ATC, AUGI, AutoCAD, AutoCAD Learning Assistance, AutoCAD LT, AutoCAD Simulator, AutoCAD SQL Extension, AutoCAD SQL Interface, Autodesk, Autodesk Homestyler, Autodesk Intent, Autodesk Inventor, Autodesk MapGuide, Autodesk Streamline, AutoLISP, AutoSketch, AutoSnap, AutoTrack, Backburner, Backdraft, Beast, Beast (design/logo) Built with ObjectARX (design/logo), Burn, Buzzsaw, CAiCE, CFdesign, Civil 3D, Cleaner, Cleaner Central, ClearScale, Colour Warper, Combustion, Communication Specification, Constructware, Content Explorer, Creative Bridge, Dancing Baby (image), DesignCenter, Design Doctor, Designer's Toolkit, DesignKids, DesignProf, DesignServer, DesignStudio, Design Web Format, Discreet, DWF, DWG, DWG (design/logo), DWG Extreme, DWG TrueConvert, DWG TrueView, DWFX, DXF, Ecotect, Evolver, Exposure, Extending the Design Team, Face Robot, FBX, Fempro, Fire, Flame, Flare, Flint, FMDesktop, Freewheel, GDX Driver, Green Building Studio, Heads-up Design, Heidi, Homestyler, HumanIK, IDEA Server, i-drop, Illuminate Labs AB (design/logo), ImageModeler, iMOUT, Incinerator, Inferno, Instructables, Instructables (stylized robot design/logo),Inventor, Inventor LT, Kynapse, Kynogon, LandXplorer, LiquidLight, LiquidLight (design/logo), Lustre, MatchMover, Maya, Mechanical Desktop, MIMI, Moldflow, Moldflow Plastics Advisers, Moldflow Plastics Insight, Moldflow Plastics Xpert, Moondust, MotionBuilder, Movimento, MPA, MPA (design/logo), MPI, MPI (design/logo), MPX, MPX (design/logo), Mudbox, Multi-Master

Editing, Navisworks, ObjectARX, ObjectDBX, Opticore, Pipeplus, Pixlr, Pixlr-o-matic, PolarSnap, PortfolioWall, Powered with Autodesk Technology, Productstream, ProMaterials, RasterDWG, RealDWG, Real-time Roto, Recognize, Render Queue, Retimer, Reveal, Revit, RiverCAD, Robot, Scaleform, Scaleform GFx, Showcase, Show Me, ShowMotion, SketchBook, Smoke, Softimage, Softimage|XSI (design/logo), Sparks, SteeringWheels, Stitcher, Stone, StormNET, Tinkerbox, ToolClip, Topobase, Toxik, TrustedDWG, T-Splines, U-Vis, ViewCube, Visual, Visual LISP, Voice Reality, Volo, Vtour, WaterNetworks, Wire, Wiretap, WiretapCentral, XSI.
Adobe, Flash and Reader are either trademarks or registered trademarks of Adobe Systems Incorporated in the United States and/or countries.
Automatic Duck and the duck logo are trademarks of Automatic Duck, Inc.
Lucida is a registered trademark or trademark of Bigelow & Holmes in the U.S. and other countries.
Python is a registered trademark of Python Software Foundation.
All other brand names, product names or trademarks belong to their respective holders.

Disclaimer

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

Contents

Chapter 1 **What's New in Flame Premium** 1

Chapter 2 **Getting Started with the New Workflow** 13

Chapter 3 **Flame Premium Interface Overview** 15
About the Media Panel 16
Media Icons and Selection Reference 17

Chapter 4 **Managing Projects and Users** 21
Working with Projects 21
Working with Users 28

Chapter 5 **Organizing Media in the Workspace** 31
Changing Your View of the Flame Premium Workspace 35
Generating Clips 35
Displaying Media Metadata 35
Creating and Opening Sequences 37
Working in Thumbnails View 38
Working with Reels 39
Undo and Redo 44

Chapter 6 **Setting Up a Flare Workflow** 45
About Flare 45
Flare Workflow 45
Considerations for Working in a Collaborative Environment 46
Starting Flare 48
Changing Projects and Users 48

Chapter 7 Importing and Exporting Media . . . 51
Importing File-Based Media . . . 51
Exporting Media to Files . . . 52
Supported Media File Formats . . . 64
Importing a Final Cut Pro XML Sequence . . . 72
Importing an AAF Sequence . . . 94
MediaHub Reference: Browsing for Files . . . 101
Modifying Import Options After Import . . . 138
Working with Others . . . 139
Using Subtitles Files . . . 142
Conform Tab Overview . . . 150
Clip Input/Output Using a VTR . . . 180
Clip Output Using Real-Time Deliverables . . . 213

Chapter 8 Archiving in Flame Premium . . . 227
Archiving a Project to a File-based Archive . . . 228
Restoring a Project from a File-based Archive . . . 229
Archiving Clips to a File-based Archive . . . 229
Restoring Material from a File-based Archive . . . 230
Tips for Organizing Archives . . . 230
Configuring a Data Tape Device . . . 231
About VTR Archives . . . 235
Archiving to Multiple Volumes . . . 240
Supplementing Archives with System Drive Backups . . . 240
Restoring Legacy Archives . . . 244
Deleting Content from an Archive . . . 246
MediaHub Reference: Browsing for Archives . . . 246

Chapter 9 Playing Back Media . . . 249
Playback . . . 249
Cue Marks . . . 253
Playback Options . . . 254
Color and Light Levels . . . 257
Preview options . . . 258
Changing Playback Scan Mode Options . . . 259
Enabling the Aspect Ratio Display . . . 259
Setting up the Image Display Viewer in the Tools . . . 259
Displaying the Reference Area . . . 260
Displaying Multiple Views . . . 262
Viewing Settings tab . . . 264

Chapter 10 Editing Clips and Sequences . . . 267
Inserting a Clip on the Timeline . . . 268
Overwriting a Clip on the Timeline . . . 269
Replacing a Clip on the Timeline . . . 270
Other Editing Operations . . . 270
Timeline Editing Settings . . . 271
Trimming Clips . . . 273
About Timeline Tracks and Versions . . . 280
About Timeline Patching . . . 282
Navigating and Searching the Timeline . . . 283
Using Markers on the Timeline . . . 285
About Timeline Containers . . . 285
About Timeline Gaps . . . 286
Playing the Timeline . . . 286
Cutting Clips on the Timeline . . . 287
Swapping Timeline Elements . . . 288

Removing Elements From the Timeline . . . 289
Matching Clips with Their Sources . . . 290
Grouping and Syncing Elements on the Timeline . . . 291
Timeline Reference . . . 293

Chapter 11 Editing on the Desktop . . . 299
About Editing on the Desktop . . . 299
Gestural Editing on the Desktop . . . 299
Ganging Clips on the Desktop . . . 301
Cutting Clips . . . 301
Splicing Clips . . . 303
Inserting Frames and Clips . . . 304
Replacing Frames and Clips . . . 304
Matching Clips with Their Sources . . . 306
Swapping Segments . . . 307
Hard Committing from the Desktop . . . 308
Rendering from the Desktop . . . 308

Chapter 12 Audio . . . 311
Accessing the Audio Desk . . . 311
Importing Audio Files . . . 312
Adding an Audio Effect on a Segment . . . 312
Adjusting the Audio Levels on Part of an Audio Segment . . . 312
Cross-fading Two Audio Segments Together . . . 312
Fading Audio Into or Out of a Clip . . . 313
Scaling the Size of the Waveform . . . 313
Displaying Audio Waveforms . . . 313
Muting Audio Tracks . . . 314
Mapping Audio Tracks to Audio Desk Input Strips . . . 314
Converting Audio from Stereo to Mono or Mono to Stereo . . . 314

Chapter 13 Timeline Transitions . . . 315
Adding Timeline Transitions . . . 316
Copying, Moving and Removing Timeline Transitions . . . 316
Adding Dissolve Timeline Transition . . . 316
Adding Wipe Timeline Transition . . . 317
Adding Axis Timeline Transition . . . 326
Adding Fade Timeline Transition . . . 327

Chapter 14 Timeline FX . . . 329
Modifying Clip Format and Import Options on the Timeline . . . 330
Gaps . . . 330
Modifying Multiple Timeline FX . . . 331
Copying Timeline FX . . . 332
Sliding Timeline FX KeyFrames . . . 332
Muting and Deleting Timeline FX . . . 333
Adding Timewarp Timeline FX . . . 333
Adding Resize Timeline FX . . . 334
Adding Text Timeline FX . . . 334
Adding Colour Correct Timeline FX . . . 335
Adding Sparks Timeline FX . . . 336
Adding Blend Timeline FX . . . 337
Adding Wipe Timeline FX . . . 337
Adding Axis Timeline FX . . . 337
Adding Gate Timeline FX . . . 338
Adding Compression Timeline FX . . . 339
Adding EQ Timeline FX . . . 339

Adding Gain Timeline FX . . . 340
Adding Reverb Send Timeline FX . . . 340
Adding Delay Send Timeline FX . . . 341
Adding Modulation Send Timeline FX . . . 341

Chapter 15 Creating New Sources Using Tools from the Tools Tab . . . 343
Accessing Tools . . . 343

Chapter 16 Procedural Compositing with Batch and Batch FX . . . 345
Accessing Batch . . . 345
Creating Batch FX . . . 346
About the Batch and Batch FX Process Tree . . . 348
Adding Clips to the Process Tree . . . 349
Adding and Connecting Nodes to the Process Tree . . . 364
Selecting Nodes in the Schematic . . . 369
Adding Media to an Action Node . . . 370
Viewing Nodes in Context . . . 373
About Batch Rendering . . . 374
About BFX Output Nodes . . . 382
Saving and Loading Batch or Batch FX Setups . . . 384
Saving Sources with Batch Snapshots . . . 384
Using a Create BFX Clip for Caching or Versioning . . . 385
Using the Timing View to Offset Clips . . . 386
Caching Frames . . . 387
Grouping Nodes . . . 389
Mimicking, Copying, and Duplicating Nodes . . . 393
Customizing the Tools Bin . . . 396
Creating Custom Nodes . . . 400
Batch and Batch FX Preferences . . . 401

Chapter 17 Compositing in 3D Space with Action . . . 405
About Action . . . 405
About Media . . . 441
About Axes . . . 451
About Surfaces . . . 457
About Stereoscopic Workflow in Action . . . 485
About the Perspective Grid . . . 491
About Lights and Lighting Effects . . . 495
About 3D Geometry . . . 531
About 3D Text . . . 550
About Shading and Textures . . . 578
About Atomize . . . 633
About 3D Paths . . . 634
About the Camera . . . 640
About the 3D Camera . . . 657
About the Analyzer . . . 674
About Source Nodes . . . 699
About Masks . . . 706
About Action's Particle System . . . 715

Chapter 18 Chroma Keying and Working with Green Screens . . . 791
Setting Up the Nodes and Media to Pull a Key . . . 792
Setting Up Viewports for Keying . . . 794
About the 3D Keyer . . . 794
Creating a Precise Key in the 3D Keyer . . . 795
Animating Key Elements . . . 807
Mastering the RGB Viewer . . . 809

More Keying Techniques . . . 825
Setting 3D Keyer Preferences . . . 827
Creating and Refining a Key in the Master Keyer . . . 828
Creating a Key by Extracting a Single Colour with the Channel Keyer . . . 837
Creating a Key by Extracting a Range of Colours with the HLS, YUV, RGB and RGBCMYL Keyers . . . 838
Creating a Key by Setting the Luminance . . . 842
Refining Your Key . . . 842
Resetting Your Key . . . 856
Rendering Your Key . . . 856

Chapter 19 Stabilizing and Tracking . . . 859
Accessing the Stabilizer . . . 859
How the Stabilizer Works . . . 860
Working with Trackers . . . 862
Selecting a Stabilizer Method . . . 868
Smoothing Out Camera Motion . . . 877
Tracking . . . 878
Tracking Difficult Shots and Correcting Errors . . . 881
Setup Options . . . 888
Copying Shift Channels . . . 889
Importing and Exporting Data . . . 890

Chapter 20 Masking and Rotoscoping . . . 891
Using Garbage Masks in Batch or Batch FX and the Modular Keyer . . . 892
Accessing the Garbage Mask Menu . . . 893
About Garbage Mask Options . . . 894
Setting Mask Options . . . 896
About Drawing Options . . . 896
Drawing a Mask . . . 897
Adjusting the Number of Points in Freehand Segments . . . 901
Garbage Mask Nodes . . . 902
Changing the Priority Order of Masks . . . 902
Transforming the Mask . . . 903
Manipulating Vertices and Tangents . . . 903
Animating Garbage Masks . . . 906
Applying Motion Blur to Garbage Masks . . . 912
Blurring a Stationary Garbage Mask . . . 914
Rendering Garbage Masks in Interlaced Mode . . . 914
Using Multiple Garbage Masks . . . 915
Refining the Mask . . . 916
Saving Garbage Mask Setups . . . 927
Loading Garbage Mask Setups . . . 928
Preset Garbage Mask Setups . . . 929
Importing and Exporting Raw Setup Files . . . 929
Creating Customized Wipes with Garbage Masks . . . 930

Chapter 21 Colour Correcting . . . 933
About Colour Correcting . . . 933
About the Colour Warper . . . 962

Chapter 22 Painting . . . 1001
Paint Node . . . 1001
Paint Tool . . . 1026

Chapter 23 Text and Titling . . . 1123
Text Workflow . . . 1123

Accessing the Text Tool . . . 1124
Setup and Processing Options . . . 1126
Saving and Loading Files . . . 1128
Adding Text to Clips . . . 1130
Creating Text Effects . . . 1135
Loading Logos . . . 1139
Tabulating Text . . . 1140
Spell Checking . . . 1141
Creating Text Rolls and Text Crawls . . . 1141
Animating Text . . . 1144
Defining Styles . . . 1150
Changing Fonts . . . 1152

Chapter 24 Warping and Morphing . . . 1157

About Distort . . . 1157
About the Warper . . . 1199

Chapter 25 Effects and Tools Reference . . . 1223

2D Histogram . . . 1223
2D Transform . . . 1224
3D Blur . . . 1231
Auto Matte . . . 1238
Auto Stabilize . . . 1239
Average . . . 1244
Blend & Comp . . . 1245
Blur . . . 1249
Bump Displace . . . 1255
Burn-In Letterbox . . . 1257
Burn-In Timecode . . . 1258
Clamp . . . 1260
Color Corrector . . . 1260
Colour Curves . . . 1264
Color Warper . . . 1265
Coloured Frame . . . 1271
Combine . . . 1272
Compound . . . 1272
Damage . . . 1273
Deal . . . 1300
Deform . . . 1301
Degrain . . . 1306
Deinterlace . . . 1307
Denoise . . . 1308
Depth of Field . . . 1312
Difference Matte . . . 1318
Distort . . . 1319
Duplicate . . . 1325
Edge Detect . . . 1325
Exposure . . . 1329
Field Merge . . . 1331
Filter . . . 1331
Flip . . . 1333
Garbage Mask . . . 1334
Glow . . . 1338
Gradient . . . 1346
Interlace . . . 1354
Keyer 3D . . . 1354
Keyer Channel . . . 1359
Keyer HLS . . . 1360

Keyer Luma . . . 1362
Keyer RGB . . . 1363
Keyer RGBCMYL . . . 1365
Keyer YUV . . . 1368
Lens Distort . . . 1370
Logical Operation . . . 1373
LUT Editor . . . 1374
Map Convert . . . 1380
Master Keyer . . . 1383
Matchbox . . . 1386
Matte Curves . . . 1390
Matte Edge . . . 1391
Mono . . . 1396
Motif . . . 1397
Motion Analysis . . . 1401
Motion Blur . . . 1402
Motion Convert . . . 1405
MUX . . . 1406
Optics . . . 1407
Paint Node . . . 1409
Pixel Spread . . . 1412
Pulldown . . . 1416
Quick Composite . . . 1417
Recursive Ops . . . 1418
Regrain . . . 1422
Resize . . . 1424
Sparks . . . 1427
Stabilizer . . . 1429
Stereo . . . 1437
Stylize . . . 1440
Substance Noise and Splatter . . . 1449
Text . . . 1454
Warper . . . 1466

Chapter 26 Animating Keyframes . . . 1471

Keyframe Indicator Reference . . . 1472
Animating Keyframes in the Animation Channel Editor . . . 1472
Advanced Animation: About Expressions . . . 1496

Chapter 27 Modifying Clips with Clip History . . . 1533

About Clip History . . . 1534
Clip History Reference . . . 1536

Chapter 28 Colour Management . . . 1539

Colour Management Workflow . . . 1539
Applying Colour Management to Clips . . . 1539
Applying Colour Management to Displayed Images . . . 1556
Autodesk Colour Transform Types . . . 1561
Colour Management Files and Locations . . . 1562
Supported Colour Transform File Formats . . . 1563
Autodesk CTF File Format Version 1.3 . . . 1564
SynColor Configuration File . . . 1602

Chapter 29 Tangent Element Control Surface . . . 1609

The Modes . . . 1609
Using the Modules . . . 1610
Control Surface Layouts . . . 1612

Troubleshooting 1654

Chapter 30 Flame Premium Keyboard Shortcuts 1657

Determining a Button's Current Keyboard Shortcut 1664
Accessing the Keyboard Shortcut Editor 1664
Searching the Keyboard Shortcut Catalogue 1666
Changing Keyboard Shortcuts 1666
Mapping a Button to a Keyboard Shortcuts 1668
Keyboard Shortcut Editor Menu Settings 1669

Chapter 31 Setting Flame Premium Preferences 1671

Audio Preferences 1671
Backburner Preferences 1672
Broadcast Monitor Preferences 1673
General Preferences 1674
Input Devices Preferences 1676
LUT Preferences 1677
Storage Preferences 1677
Support Preferences 1678
Timeline Preferences 1678
Timeline FX / Batch FX Preferences 1679
User Interface Preferences 1681

Chapter 32 Gateway Clip Description 1683

clip 1683
comment 1684
creationDate 1684
denominator 1685
dropMode 1685
duration 1686
duration 1686
feed 1686
feeds 1687
name 1688
nbTicks 1688
numerator 1688
path 1689
paths 1690
rate 1690
sampleRate 1691
span 1691
spans 1692
storageFormat 1692
track 1693
tracks 1694
trackType 1694
userData 1694
version 1695
versions 1696
Creating a Simple Gateway Clip 1697

Chapter 33 Flame Premium Grading - Lustre 1701

Lustre Workflow 1701
Interoperability Workflows in Lustre 1704
Project Management 1719
Basics 1801
Monitor Calibration 1846

Browsing for Footage 1856
Transcoding 1877
Conform 1892
Editing 1912
Playing, Viewing, and Sorting Shots 1968
Removing Dust 1998
Removing Grain 2008
Repositioning Images 2016
Input LUTs 2026
Colour Grading Basics 2038
Primary Colour Grading 2058
Colour Grading RGB and Hue Curves 2074
Secondary Colour Grading 2083
Creating Lustre Sparks Effects 2156
Animation 2171
Stereoscopy 2206
Rendering 2215
Media Export (Linux Only) 2240
Video Capture and Video Playout 2257
Audio 2306
Working with Wiretap 2315
Hotkeys 2341
XML in Lustre 2362
Tangent Element Control Surface 2377
The Autodesk Control Surface 2420
The Tangent CP100 Control Surface 2484

Index 2507

What's New in Flame Premium

1

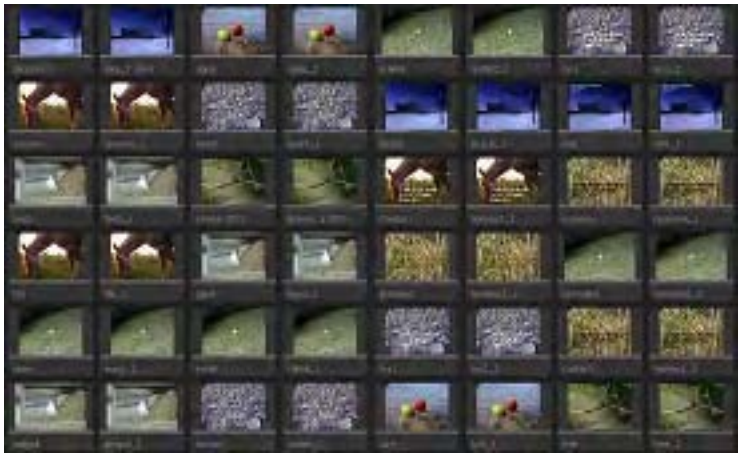
Welcome to Service Pack 3 of 2013 Extension 2, 20th Anniversary Edition.

This service pack includes a number of *fixed bugs*, performance enhancements and feature requests that have come directly from you, including:

- Performance enhancements to the MediaHub (page 1), the Desktop (page 2), and Action (page 8)
- Updates to support for RED (page 2), SonyRAW (page 3), ARRI (page 3), Avid intermediates (page 4), and single-channel DPX files (page 4)
- A new Trim View (page 6) for editorial
- Improvements to Batch (page 7), the timeline (page 6) and player (page 7), many of which come from your feedback
- Updates to Colour Management (page 8)
- Improved feedback (page 9) in the interface
- Updates to Lustre (page 10)

Following is a complete list of all the new features and updates in this release.

MediaHub Performance Improvements



- You can now resize the thumbnails displayed in the MediaHub. The thumbnails are now displayed in 16:9.
- Navigation through folders is more responsive.
- The contents of large folders are displayed faster than before; in some cases, what would take minutes to display now takes seconds.

- Clips are displayed progressively in folders and you can start interacting with them as they appear (for example, you can drag and drop to import).
- The MediaHub continues to load media if you change tabs.
- A new column, File Size, can now be displayed in the List View of the MediaHub. It is hidden by default because file size calculations impact significantly MediaHub performances.
- Clear messages indicate whether the displayed folder is read-only or not.
- When restoring media from an archive, the clips you dragged from the archive to the Media Library now appear in the Media Library. They are greyed out to indicate that they have not been restored yet, and remain so until you click Restore.
- User feedback, in the form of better messaging, has been improved when importing AAF, XML, and EDL sequences, as well as when exporting clips.
- You no longer can rename a folder in the MediaHub by simply double-clicking its name. You must now use the contextual menu.

Desktop Improvements

- Improved performance on Desktop Reels: There's now a preference to disable the Broadcast Monitor in **Preferences ➤ User Interface**. Disabling this option enhances interactivity when manipulating clips and performing gestural edits on Desktop Reels.

  NOTE When this option is disabled, Timeline FX are not automatically rendered and display the message "Unrendered Frame" on the Reels. However, the frame on which the Timeline positioner is parked, is rendered.

- It is now possible to open multiple sequences simultaneously, from the Media panel and the Viewing panel. You can also close multiple sequences directly from the timeline.

Updated Support for RED Import

- Support for REDCODE SDK 4.4
- Support for RED EPIC-M MONOCHROME R3D files
- Updates to DRX processing

  IMPORTANT Changes to the SDK affect how the DRX setting is computed for all R3D clips, and thus can impact the look of come clips. It impacts previously imported clips.

  The clips affected are the ones which have a DRX setting different from 0. To check if this is the case for a clip:

  - In Flame: Open, from the Timeline, the Pre-Processing Editor, and look for DRX under **Basic ➤ Image**.
  - In Lustre: Look for DRX in **Transcode ➤ Colour ➤ Colour Settings**.

  If DRX is not 0, then your media is impacted by the change. How much of an impact is hard to predict, but since DRX (Dynamic Range Extension) is used to reconstruct clipped colour channels, verify for clips with dangerous highlights and for HDRx material.

- Support for DRX processing via RED Rocket

  The DRX setting is now processed by the RED Rocket card if one is present on the Mac running the Wiretap Gateway server used to decode R3D files. The following driver and firmware must be installed on the Mac:

  - RED Rocket driver 1.4.32.0
  - RED Rocket firmware 1.1.16.11 or later
  - RED Rocket Breakout Box firmware 1.0.2.0 or later

Updated Support for SonyRAW Import

- Support for files generated by Sony F5 and F55 lines of cameras, joining the already supported F65.

  NOTE Only the F65 camera supports resolution higher than 4K. For example, selecting 6K for a F5 clip results in a checkerboard, which you can solve by switching to a supported debayering resolution.

- Support for orientation changes (flip/flop).
- Debayering quality now offers 2 settings, Standard and High.
- New debayering resolutions are now available and have been added to the debayering options.
- Debayering options have been renamed to reflect what is used in other applications
  - 0.25K (previously named Sixteenth)
  - 0.5K (previously named Eighth)
  - 1K (previously named Quarter)
  - HD (1920)
  - 2K (previously named Half)
  - QFHD (3480)
  - 4K (previously named Full)
  - 6K
  - UHDTV (7680)
  - 8K

NOTE The colour space options--Rec709 primaries, ACES (daylight illuminant), ACES (tungsten illuminant)--have been updated to solve an issue with black levels. This update impacts previously imported SonyRAW media used in current projects: the black levels will not match. Make sure to update the Format Options from the timeline to reapply.

Updated Support for ARRI Import

IMPORTANT The timecode of some ARRIRAW files was not correctly read in previous releases. This version of Flame Premium fixes this issue. If you use ARRIRAW clips imported in a previous release, loading this content in this new version shows the same visual result but the source timecode will be updated. This also ensures that both the ProRes and the ARRIRAW files have the same timecode, making it easier to offline in ProRes and relink to the raw media.

- Support for ARRIRAW 4.4.
- Support for MXF (DNxHD 145 and 220x as OP1a) files generated by the Alexa camera.
- The debayering settings reflect changes to the ARRIRAW SDK and allow you to use either the original camera image resolution (2880 pixels wide) or the same resolution as the ProRes file generated in camera (2868-pixel wide).
  - SD from 2880px
  - HD from 2880px
  - 2K from 2868px
  - 2K from 2880px
  - Native 2868px
  - Native 2880px

  NOTE Half and Third debayering options are no longer available. This means that if you restore material archived at Half or Third debayering, you will get a checkerboard (Flame) or a red X (Lustre) upon restore. Use the Format Options from the timeline to select a valid debayering setting.

- New debayering modes are now available:
  - Proxy (new): Produces a low-quality debayering.
  - ADA-1 HW (was *Camera*): Visually reproduces the hardware-optimised debayering that is realised in cameras.
  - ADA-2 SW (was *AMC-1*): More efficient debayering than ADA-1.
  - ADA-3 HW (new): Visually matches the hardware-optimised debayering realised in cameras with ARRI SUP 7.0. A more complex debayering than ADA-1, with extended edge and color handling.
  - ADA-3 SW (new): Advanced software debayering provided by the ARRI SDK.

  NOTE The HW refers to the method ARRI cameras use to decode RAW media. It does not mean that the application is performing hardware optimized debayering.

- New Colour Rendering: ACES (Scene Linear) gives you direct access to 16-bit floating point images. You no longer have to apply a colour transformation to get 16-bit floating point images.

Avid Intermediates

Updated support for Avid Intermediate (transcoded) MXF media files.

- Support now includes intermediates encoded with XDCam HD and XDCam EX codecs.
- The following list shows all supported transcoded intermediates generated by Avid Media Composer:
  - AVC-Intra 50
  - AVC-Intra 100
  - DNxHD
  - XDCam EX
  - XDCam HD

- The following is a list of Avid Media Composer intermediate formats not supported by Flame Premium:
  - J2K MXF
  - 1:1 MXF
  - 1:1p 10b MXF
  - Apple ProRes Proxy MXF
  - Apple ProRes LT MXF
  - Apple ProRes MXF
  - Apple ProRes HQ MXF

Support for Single-channel DPX Files

Monochromatic DPX files from the following film scanners have been validated.

- FilmLight Northlight (10 & 16-bit)
- DigitalFilmTechnology SCANITY™ (10 & 16-bit)
- Imagica (8, 10 & 16-bit)

NOTE Lustre supports single channel DPX files only when imported through Wiretap Gateway.

Miscellaneous Media Import Updates

- Audio: WAVE-Extensible format can now be imported, but the spatial information of the tracks is discarded.

- Audio: WAVE-Broadcast support now includes the RF64 standard. This means you can now import and export BWF files larger than 4 GB.
- Video: A clip with an alpha channel can now be imported as: a clip plus a standalone matte clip, a matte container storing both the clip and its matte, or only the clip without any matte. Use the Alpha Channel Processing box in **MediaHub ➤ Browse for Files ➤ General**.
- Video: MXF OpAtom files generated by Avid Media Composer through importing, transcoding, and exporting AAF files are now imported via MediaHub with their original audio channel, instead of defaulting to audio channel 1 as before.

  This means that when browsing these files through MediaHub, you see a single audio channel file (A1) but when imported, the track shows the original channel. This improves file-based conform with media already imported in the Media Library.

Updated SDI Preview with Kona 3G

Monitoring in Flame Premium can now be done through the Kona 3G with the following rasters.

- 1920x1080 @ 50p
- 1920x1080 @ 5994p
- 1920x1080 @ 60p

PCM in QuickTime Export

You can now export QuickTime movies with PCM audio and select an audio encoding profile to define both bit depth and bit ordering. The following profiles are available:

- 16-bit (LE)
- 16-bit (BE)
- 24-bit (LE)
- 24-bit (BE)
- 32-bit fp (LE)
- 32-bit fp (BE)

LE: little endian; BE: big endian. Endianness has no impact on performance, and little endian should work in most cases. Output in big endian mode if you expect compatibility issues with the target application or operating system.

Miscellaneous Media Export Updates

The following Media IO improvements apply to both Flame and Lustre.

- You can now specify Academy Density Exchange as the Transfer Type for DPX files. A new Colour Type box has also been added to the same DPX format, allowing you to specify Colorimetric Encoding information. As before, Transfer Type and Colour Type do not modify the output media: they are metadata fields that can be interpreted by third party applications.
- You can now export an *uncompressed* OpenEXR.
- When exporting a QuickTime in MXF file, the Audio Sampling Rate export is always set to Same As Source.
- You can now use the contextual menu to export a folder. The resulting export maintains the original organization and hierarchy by creating a similar folder structure.
- Codec profiles are now sorted alphabetically.

What's New in Conform

- Conform now benefits from a more accurate timewarp translation (constant and animated) when conforming Apple Final Cut Pro XML and Avid AAF files. Specifically, animated timewarps from Avid Media Composer are more precisely

translated. All unsupported transitions from Media Composer 6.5 AAF are now converted into a Dissolve and a Cut Mark that displays the name of the original transition.

- Apple added support for ARRI Alexa ProRes files in FCP 10.0.8, and now the XML referencing these media files can be conformed in Flame Premium.
- For this release, XML from FCP 10.0.4 and 10.0.5 no longer import in Flame Premium the segments that were Multicam clips in FCP. Apple modified the MultiCam representation in FCP 10.0.6 and it is not possible to support legacy MultiCam content. But XML files from 10.0.6 import just fine: Flame Premium displays MultiCam clips as segments just like before.

New Trim View



A new Trim View, familiar to editors, is available to help you trim video or audio clips (with or without effects), cuts, and transitions on the timeline.

You can now trim using the Trim View, with keyboard shortcuts, or gesturally on the timeline (or any combination of these methods, depending on your editing style). In Trim mode, the Trim View displays the last (outgoing) and first (incoming) frame above the timeline from the two clips you are trimming, allowing you to visualize your trimming operation. In Slip or Slide mode, the Trim View changes to a four frame display, which, in addition to the incoming and outgoing frames of the clip you are slipping or sliding, also displays the outgoing frame of the preceding clip and the incoming frame of the subsequent clip on the timeline.

See Trimming Clips (page 273).

What's New in the Timeline



- Link functionality for timeline groups has been moved from the Timeline Gear menu to its own button (next to the Ripple button). Performing a trim respects selections that are linked.

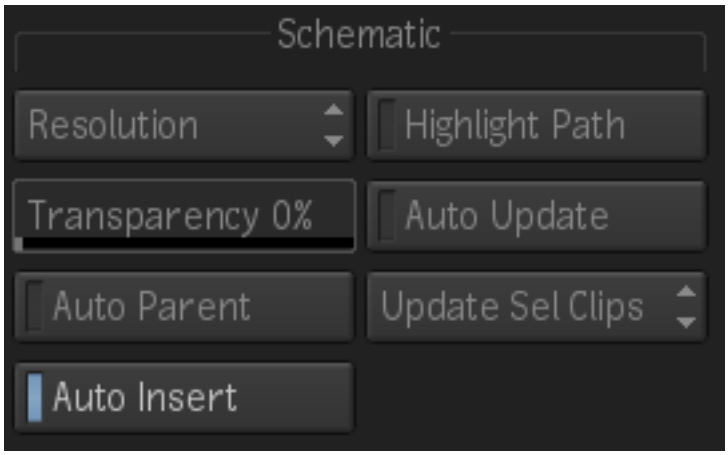
- In the Timeline Options menu, the new Selection Includes Gaps setting allows you to select whether empty gaps are included, when performing a selection on the timeline.
- You can now edit a Gain Animation curve directly on a timeline audio clip, if Show Gain Animation is enabled in the Timeline options menu. In this case, a Gain Timeline FX is automatically applied when you modify the Gain curve.
- You can now drag and drop to copy Format Options (format specific settings such as debayering settings or timecode selection) and Pre-processing Options (resize on import and LUT) between timeline segments.
- While the positioner is over a cut or transition on the timeline, you are now able to set a Mark Out with the keyboard shortcut.

What's New in the Player



- You can now change the behaviour of the Play button. Click and hold the Play button, select the command you want to apply to the Play button from the menu (such as Loop). That behaviour is now assigned to the Play button until you change it again.
- As you drag in the Mark In and Mark Out fields to change values, the image displayed in the Player now updates.
- When working in the Viewing panel, image window viewing settings (such as contrast and image data type) are now persistent by timeline tab type (Source or Sequence), instead of by Player view.

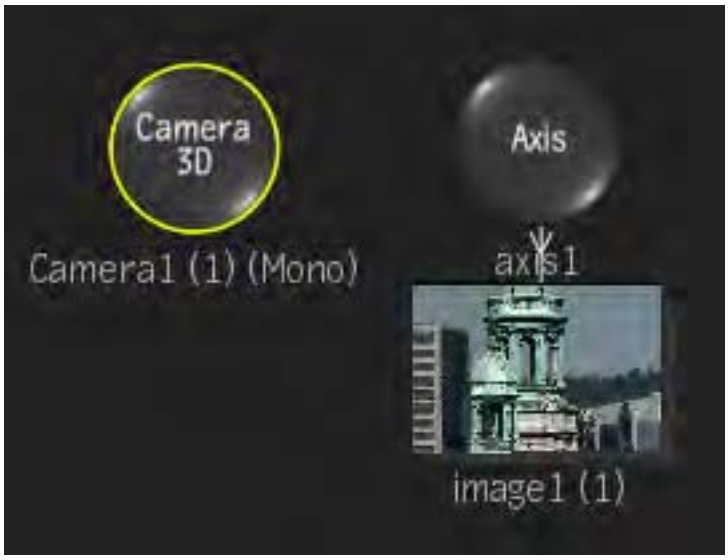
What's New in Batch and Batch FX



- You can now disable or enable the automatic insertion of a node when it is dragged between two connected nodes: Batch or Batch FX Preferences menu, under **Preferences ➤ Schematic ➤ Auto Insert**. When the preference is disabled, you can still press **Shift** to auto insert.
- Use a new preference to set the default Batch FX viewport as 1-Up or 2-Up: **Preferences ➤ Timeline FX / BFX ➤ Viewport Settings**.
- Create and save a Batch setup when you save a Batch Snapshot: Enable Also Save Batch File in **Preferences ➤ Timeline FX / BFX ➤ Batch Snapshot**. When enabled, the setup is saved in the default Batch folder *~/batch/flame* in a snapshots subfolder. Note that these setup files are not managed by Flame Premium. Therefore, if you delete a Snapshot, the corresponding setup file is not deleted automatically.
- The name of the last saved BFX setup is displayed in the Name field when exiting and re-entering the same BFX.
- The Offset field for setting the number of frames by which a Batch or Batch FX clip is offset is now available from the Timing view as well as the Clip settings.

- Render options are now available as a contextual option when right-clicking a clip node (in addition to the Timing View Render box).
- The UI has been cleaned up slightly to provide consistency. For example, the Node Load and Save buttons are now always available when a node is selected in schematic.
- The MUX node includes the Freeze Current Frame button, used to output the image of the currently selected frame for the duration of the clip.
- In addition to adding clips directly to Batch by dragging to the Batch Sources folder in the Workspace Media panel, you can now also drag clips to the Batch header (above the Batch sources folder). In this case, the clip is added to the Batch Sources folder, and you are also automatically switched to the Batch tab.
- Clips added by dragging to the Batch or BFX Sources folder in the Workspace Media panel now appear close to the last selected node in the schematic. If no nodes are selected in the schematic, clips are added in the centre of the schematic.
- You are now able to replace a clip in Batch FX by dragging a source clip from the Workspace Media panel over a clip in the schematic.

What's New in Action



- Flame Premium now supports the import of Alembic (*.abc*) cameras in Action. The Camera FBX node is renamed to Camera 3D node.
- Performance in Action is improved thanks to optimizations in the handling of 3D objects, shaders, and texture transfer speed.
- When accessing the Modular Keyer from Action, if the Action back clip is of a different resolution than the front clip, a Resize node is now added to the Back pipeline.

What's New in Colour Management

- You can import other transform file formats wherever a Colour Transform (.ctf) file may be loaded. Supported formats include:
  - Autodesk legacy 1D LUT: .lut
  - Autodesk legacy 3D LUT: .3dl
  - ASC color decision list: .cdl
  - ASC color correction collection: .ccc (uses the first .cdl in the file)
  - Cinespace: .csp

- ICC monitor profiles: .icc, .icm, .pf (uses the PCS-to-monitor-RGB transform)
- Iridas Cube: .cube
- Iridas itx: .itx
- Iridas Look: .look (partial support)
- Nuke: .vf
- Imageworks 1D LUT: .spi1d
- Imageworks 3D LUT: .spi3d
- Imageworks matrix: .spimtx
- Pandora: .m3d, .mga

- You can now define a Look transform in LUT preferences, and toggle it on and off from the viewer heads-up display. You can use any supported file format. In addition, you can reference the defaultLook alias in your own custom transform files. Note that this works only in Flame.
- The graphics-monitor transforms have been replaced by current-monitor transforms. For example, ACES_to_graphics-monitor.ctf has been replaced with ACES_to_current-monitor.ctf. The current-monitor transforms make use of the currentMonitor alias, which gets resolved to the display profile set in LUT Preferences in Flame for either the Broadcast Monitor Transform or Graphics Monitor Transform depending on which display the image is being drawn to. In addition, you can reference the currentMonitor alias in your own custom transform files.
- You do not need to re-import viewer transforms after changing the Graphics Monitor Transform, Broadcast Monitor Transform, or Default Look Transform in LUT Preferences in Flame.
- New ASC_CDL operator for representing an ASC color decision list in the CTF file format.
- New Log operator for applying a logarithmic or anti-logarithmic function to convert between linear and Cineon-style encodings.
- You can now use the Matrix operator in a custom .ctf file to modify the alpha channel.

New Feedback Mechanisms

- A new console displays messages about the status of tasks being performed by the application. To open, click the arrow button on the lower left corner of the UI.
- An updated Background Tasks view shows status for various tasks, including caching, media import and export, and Burn. To open the Background Task view, open the Console. Then click View All in the Background Tasks section of the Console. From the Background Tasks view, you can restart, delete, suspend, or resume a task using the buttons or from the right-click menu. You can also sort and re-order the columns in the list.
- A dialog box displays the render progress.

Project Conversion and Deletion at Startup

With this version of Flame Premium, you can now convert projects created in Flame 2013 20th Anniversary Edition to the current version, from the Startup screen. You do not have to first archive the project or go through **MediaHub ➤ Projects**. You can also delete old projects.

To convert a project to the current version of Flame Premium:

1. From the Startup screen, set the Versions filter to All Versions.
2. Select the project to convert from the Project box.
3. Click Start.

IMPORTANT The conversion is one way: once you convert the project, you cannot open it again in an older version of Flame Premium.

To delete a project from a previous version:

1. From the Startup screen, set the Versions filter to All Versions.
2. Select the project to convert from the Project box.
3. Click Edit to open the Edit Project window.
4. From the Project Edit box, select Delete Project.
5. Click Delete.
   Flame Premium now deletes the project and all the related files.

Background Processes Detection

- Now Flame Premium detects at launch if the Wiretap background services are running and compatible with the application version. If the background services are not running, or if there is a compatibility mismatch, the application asks you how to proceed.
- If you need to restart your local Wiretap Gateway from the Preferences: **Preferences ➤ Storage ➤ Local Gateway**. You can also find this button in the Media Browser in Lustre.

Miscellaneous Updates

- When Flame Reactor rendering engine is enabled, legacy Keyer setups (Axis Timeline FX) are converted to a Modular Keyer setup.

  NOTE The colour section of the legacy keyers is only carried over to the Modular Keyer if the legacy keyer setup used the Master Keyer. If any other legacy keyer was used (RGB, Luminance, etc.), the colour section is discarded when the setup is loaded. To retain the colour section of these legacy keyer setups, enable Classic Engine when creating your project.

What's New in Lustre

- Grading: It is now possible to copy or move grading parameters between the Input, Output and any of the Secondary stages with the following key strokes:
  - To copy the parameters from one of the grading tabs (Colour/Grading, Colour/Curves or Colour/Secondaries) of the current grading stage (Input, Secondary or Output), to another stage, hold `Ctrl` and click the destination stage (Input, Secondary or Output).
  - To copy the parameters of all the grading tabs (Colour/Grading, Colour/Curves and Colour/Secondaries) of the current grading stage (Input, Secondary or Output), to another stage, hold `Shift` and click the destination stage (Input, Secondary or Output).
  - To move the parameters of all the grading tabs (Colour/Grading, Colour/Curves and Colour/Secondaries) of the current grading stage (Input, Secondary or Output), hold `Shift + Ctrl` and select the destination stage (Input, Secondary or Output). The source stage is reset to the initial values.
  - To copy the parameters as defined by the Selector of the currently selected stage (Colour/Grading, Colour/Curves or Colour/Secondaries), hold `Alt` and select the destination stage (Input, Secondary or Output).
- DVCProHD: New Scale to full HD option when importing MXF P2 & XDCAM HD narrow content (1440x1080) in Image/Transcode/Format panel.
- Cut-based Confidence Check reference: When using a clip from the Library for a Confidence Check, the reference clip, its timing offset and the viewing mode (Single or Dual and Wipe position) are now saved with the cut information. When reloading a cut, you do not need to manually reload the Confidence Check clip and set the proper offset value. Simply clicking the Conf Check button (located in the Editing/Edit menu) displays the confidence check view with the reference clip and the defined offset value.

  NOTE Only source clips can be used as a Confidence Check reference. Sequences are not supported.

- Image Reposition: When using Image Reposition > Flip or Flop, now all geometries in the current shot follow the Flip and Flop orientation. You can also use the Geometries Flip and Flop options to control the orientation of the geometry independently from the image orientation.

Interchange between Lustre and Smoke or Flame (or Flame Premium Grading and Flame Premium)

There is a new behavior when dropping a sequence into an existing Lustre timeline.

When dropping content, coming from a Smoke or Flame project, to a Lustre timeline, the sequence or the clip is no longer added as a new video track within the current timeline. The selected content is now inserted as a layer above the current video layer or track, at the content's record timecode. This functionality is useful when importing revised editorial content from Smoke and Flame into the currently opened Lustre timeline.

NOTE To drop imported content from Wiretap server at its native record timecode, the selected sequence must be dropped directly from the Media Browser to the current Lustre timeline. Dropping the selected sequence to the Media Library first then dropping the sequence from the Media Library to the current Lustre timeline will use the timeline Snap options (Off, Positioner or Transition) rather than its native record timecode.

`Build 2013-05-21`

Getting Started with the New Workflow 2

Welcome to the New Unified Creative Workflow

This topic addresses some of the common questions about the new workflow, describes some of the benefits, and provides tips for navigating the interface.

Video content for this topic available in the Getting Started topic on the *Flame Premium wiki* and in the *offline Help you can install*.

We hope this helps you speed up your transition to the new workflow.

Why did you change the UI so drastically? The old one worked fine, and now I have to retrain my muscle memory!

Unifying the Desktop, Batch and Timeline into one workspace allows us to treat your data more efficiently and provide you with centralized access to functions that were once limited to modules.

Think of the Desktop, the Timeline, and Batch as three different views of your media. Each of these views provides unique ways of visualizing and interacting with your media.

Now that each of these views are accessible via a tab, you can be more productive and creative.

Talk to me about the creative benefits. What benefit is there to having the Timeline and Desktop together?

Together, the Timeline and the Desktop are a very powerful editorial and visualization tool. With the two, you get detailed editorial information, with powerful Desktop visualization and gestural editing tools.

How does the Batch tab fit into the new workflow?

Now that Batch is a different view on your media, rather than a separate module, you can use it interactively with the timeline, the Desktop, and the Media panel.

You have talked about some of the creative benefits, what about productivity benefits?

These are some of the productivity benefits you will see:

- You can unlink and reformat clips right from the Desktop.
- Iterating in Batch is even more efficient than it was with the old Desktop tools because of its integration with the workspace.
- It's more efficient to save your Batch flowgraph and media for future edits with BFX clips than saving and reloading setups.
- It is easier to manage iterations of media from the Media panel. Dragging clips from the Media Library to any location creates a copy automatically. Media in the Media Library is protected from editing by default.

How do I save my work now that the Clip Library is gone?

The Media Library in the Media panel provides a similar workflow as the previous Clip Library. In addition, there are a number of ways that you can save and version your work in the 20th Anniversary Edition.

Where has the Record Area gone?

Although we no longer have the concept of a Record Area in the application, you can replicate some of the Record Area functionality using features in the 20th Anniversary Edition.

How do I navigate quickly between the different areas of the interface?

- Show/Hide the Media panel: Ctrl + Swipe (Shift + ESC also works)
- MediaHub tab: Spacebar + F1
- Conform tab: Spacebar + F2
- Timeline tab: Spacebar + F3
- Batch tab: Spacebar + F4
- Tools tab: Spacebar + F5
- Open the timeline of a clip: double-click the clip.
- Go to the Batch schematic: double-click the Batch label in the Media Library.
- Go to the Desktop: double-click the Desktop label in the Media library.

Flame Premium Interface Overview

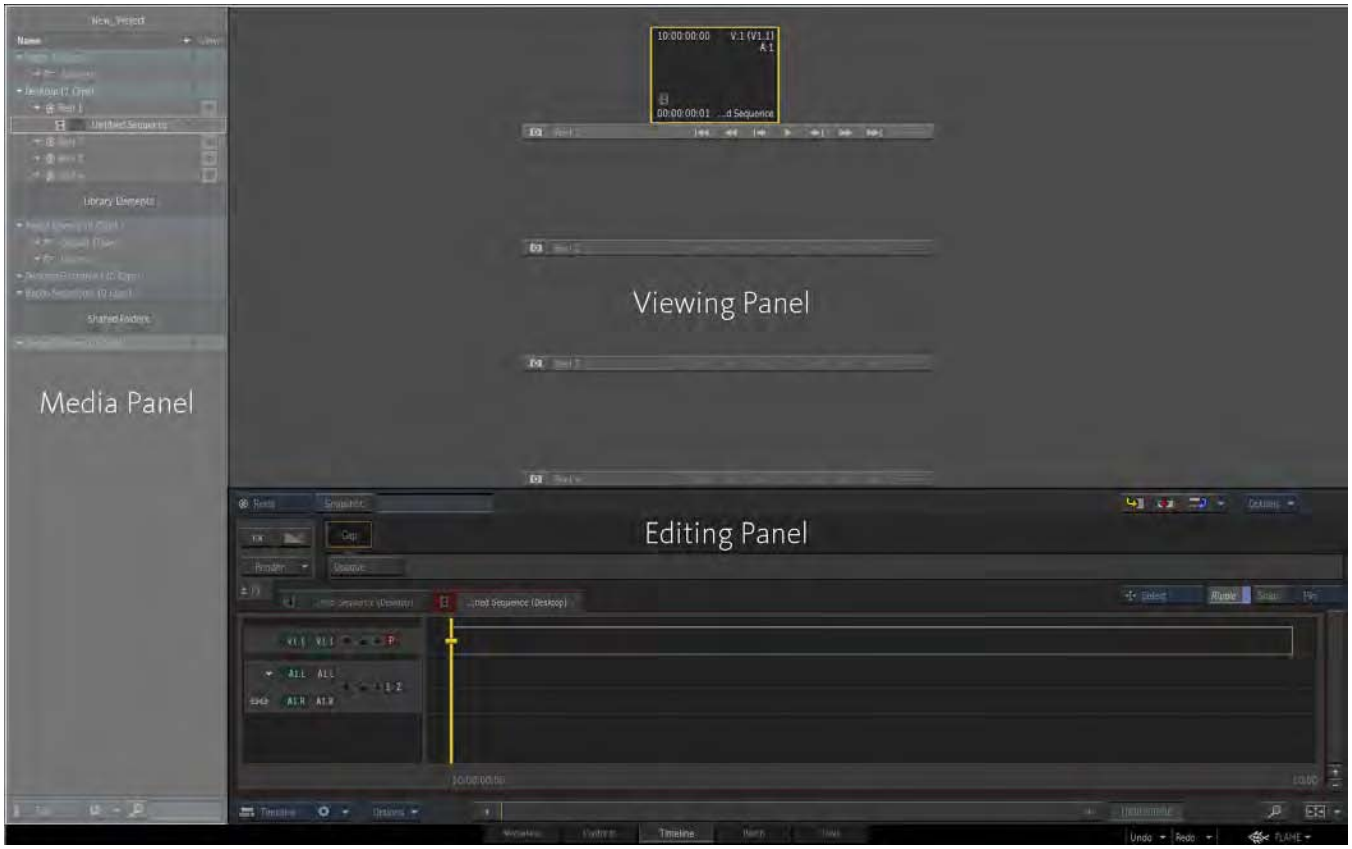
3

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

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

Each panel has its own menu bar with commands specific to the tasks you can perform in that space.

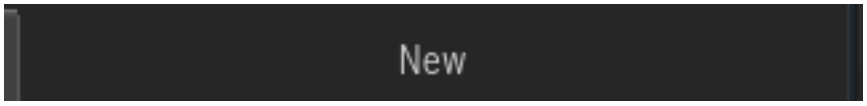
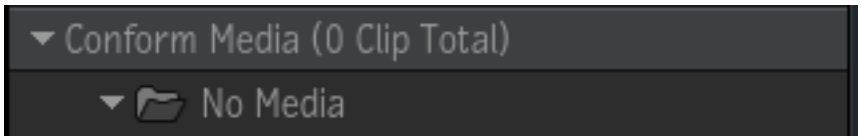
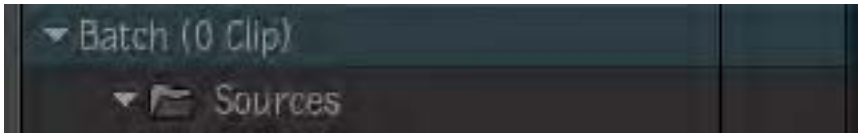
At the bottom of the user interface is a series of five tabs. Each tab gives you access to different views within the application. Regardless of which view you are in, you remain in the same environment.



About the Media Panel

The Media panel is the centralized location where you organize all of your project media.

NOTE When you launch Flame for the first time, the Media panel is not displayed. To display it, from the Media panel View Mode box, deselect Hidden.

Section:	Description:
	**Project Name**: The name of your current project is dis
	**Workspace Name**: The name of your current worksp name, if there are multiple workspaces available for that
	**Conform Media**: Any conform media imported in the Conform Media section. This section is only displayed w
	**Batch Sources**: Any media used in Batch is displayed
	**Desktop Reels**: All of the reels and their media are dis
	**Workspace Library**: All media and folders reside in a create folders to organize your media.

Section:	Description:
	**Desktop Snapshots**: Save Desktop Snapshots to this restore them. Desktop Snapshots include all the settings Desktop.
	**Batch Snapshots**: Save Batch snapshots to this section include all the nodes, settings and media associated wit
	**Shared Libraries**: Drag clips and folders from the Me the content available to other Creative Finishing applica
	**Search Field**: Perform name based searches for media are displayed in alphabetical order. The search results lis panel.

Media Icons and Selection Reference

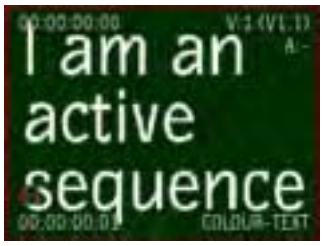
Media Thumbnail Icons

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

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

Media Selection Colour Coding

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

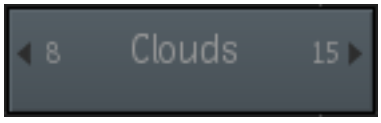
Border:	Selection:
	Unselected clip. No border.
 	Primary/secondary selection. A yellow border is displayed on the primary selection and a grey border on the secondary selection(s). The last selected clip becomes the primary selection.
	Active sequence. A red border is displayed when an active sequence is selected.
	Active source. A green border is displayed on the currently open source when an active sequence is selected.

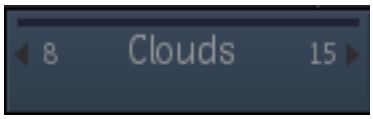
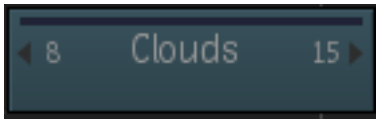
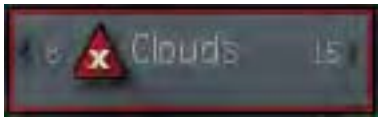
Border:	Selection:
	Last rendered clip. A blue border is displayed on the last rendered clip.
 	Ganged clips. The clip information displayed on the thumbnails of ganged clips turns green when selected. Otherwise, the clip information on ganged clips is yellow.

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

Timeline Colour Coding

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

Element:	State:
	A selected video segment.
	An unselected video segment.
	An audio segment.

Element:	State:
	A container.
	A segment with a BFX applied.
	A segment with unlinked video.

Managing Projects and Users

4

Projects and users define your working environment in Flame Premium.

- Projects define the display environment.
- Projects provide settings for project resolution, aspect ratio, bit depth, scan mode, etc

  You typically create a project for each job you work on.

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

A user is a profile that maintains the following preferences:

- User interface settings
- Pen and tablet preferences
- Keyboard shortcuts

Working with Projects

To create a project:

1. To create a project on start-up, click New from the Project panel.
2. To create a project in the middle of a session, from the Flame menu, select Project and User Settings.
   - The Project and User Settings dialog box appears.
   - From the Project panel, click New.
   - The Create New Project dialog box appears.
3. In the Name field, enter a name for the new project. Names can be up to 120 characters long.
4. If you have multiple volumes on your system, you can select the volume you want to work with from the Volume field. If you only have one volume, the volume name is displayed.

   NOTE Projects are tied to the volume they were created on and cannot be accessed from a different volume. To access the content from a project on a different volume, you can wire the clips and sequences from that project to a project on the current volume.
5. Set the setup directory for the project.
6. Set the setup mode for the project.
7. Set the default resolution for the project.

The default project configuration template is automatically loaded. You can manually select a different project configuration template if you like. This can be useful if your monitor does not natively support the resolution of your project. See About the Project Configuration Template (page 24).

8 Set the aspect ratio for the project.

9 Select the bit depth for the project.

10 Set the scan mode for the project.

11 Select the graphics rendering bit depth for the project. For best results, leave the default option of 16-bit FP Graphics.

12 Select Flame Reactor or Classic Engine for rendering.

- **Flame Reactor**: This is the default option when creating a new project. Use this option to enable rendering optimization. However, when working with content created in a previous releases, final renders may look different. This option may not be available on some configurations.

  NOTE

  - Flame Reactor is not supported on configurations using NVidia Quadro FX4500 and FX5500.
  - It is recommended that you use Burn nodes with GPU processing capabilities when sending remote rendering jobs from a project that has the Render option set to Flame Reactor, to avoid discrepancies.

- **Classic Engine**: Use this option when working with content generated in pre-2013 Extension 1 releases that need to have the same look.

13 Set the Cache and Renders settings.

14 Set the Proxy Settings.

15 When you are satisfied with project settings, click Create.

TIP

- Projects have an autosave feature. The project is automatically saved at the interval set in the preferences. You can manually save your project with the `Alt+S` keyboard shortcut or by selecting Save Project from the Flame menu.
- If you create a project in the middle of a session, you must load it by clicking Load in the Project panel.
- To reset project settings, click Reset. You can edit a project's settings at any time.
- You can exit the Create Project dialog box at any time without creating a project, by clicking Cancel.

To edit a project:

1 From the Flame menu, select Project Management. The Project and User Settings dialog box appears.

2 From the Project box, select the project you want to edit.

3 Click Edit. The Edit Project dialog box appears.

4 Modify the settings.

5 Click Done and then Confirm.

The modifications are applied to your project.

To delete a project:

1 From the Flame menu, select Project And User Settings. The Project and User Settings dialog box appears.

2 From the Project box, select the project you want to delete.

NOTE You cannot delete the current project.

3 Click Edit. The Edit Project dialog box appears.

4 Click the Project Edit box and select Delete Project.

5 Click Done and then Confirm.

You are prompted to confirm the deletion of the clips contained in the project.

6 Click Confirm.

You are prompted to confirm the deletion of the setups contained in the project.

7 Click Confirm.

The project is deleted.

NOTE When you delete a project, all its associated clips and setups are deleted with the project.

NOTE You cannot open a project created in Flame 2013 20th Anniversary EditionFlame Premium without first converting it.

To open a project created in Flame 2013 20th Anniversay Edition:

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

2 Select the project to convert from the Project box.

The version of Flame Premium used to create it appears in brackets.

3 Click Start. Flame Premium converts and opens the project.

IMPORTANT The conversion is one way: once you convert the project, you cannot open it again in an older version of Flame Premium.

To open a project created in Flame Premium before Flame 2013 20th Anniversay Edition, use one of the following options.

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

About the Project Setup Directory

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

When you create a project, the project's setup directory appears in the Setup Directory box.

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

Sharing a Setup Directory with an Existing Project

You can share the project home directory with an existing project, allowing you to share setups between the two projects.

To share the project home directory with an existing project:

1 From the Setup Directory box, select the existing project's home directory that you want to share.

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

Copying Setups from an Existing Project:

To copy setups from an existing project:

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

About the Graphics Rendering Bit Depth

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

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

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

About the Project Configuration Template

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

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

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

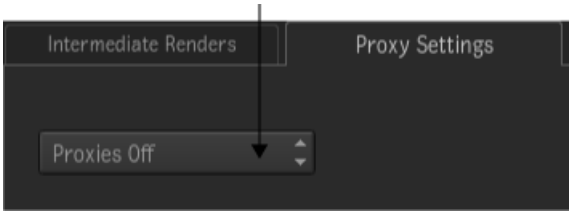
Setting the Proxy Management Options

Proxies are low-resolution copies of high-resolution clips. Using proxies enables you to get a higher level of playback performance and responsiveness, especially when working with large files. Set proxy management options to specify how and when proxies are generated for clips used in the project. The default setting is Proxies Off.

To set proxy management options:

1. From the Proxy Management option box, select an option.

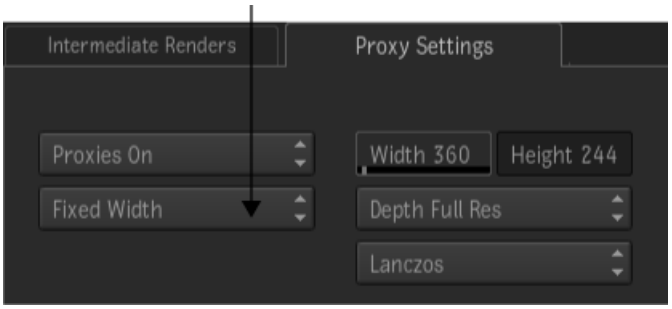
   TIP To change proxy management options for a current project, you must access the Project and User Settings dialog box.



Select:	To display:
Proxies On	Proxies for all clips larger than the proxy width you set for all proxies.
Conditional	Proxies only for clips that meet conditional criteria, for example, clips that exceed a width of 4000 pixels.

2 Specify the proxy size by doing one of the following:

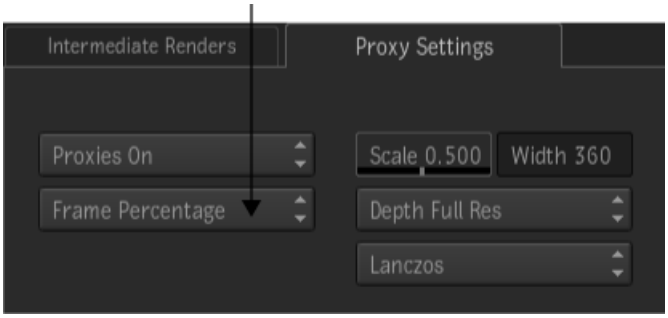
- Select Fixed Width from the Proxy Size box and then enter a value in the Proxy Width field beside the Proxy Size box. For example, enter 720 to create and store proxies 720 pixels wide. Proxy height is determined by the clip's aspect ratio.



NOTE

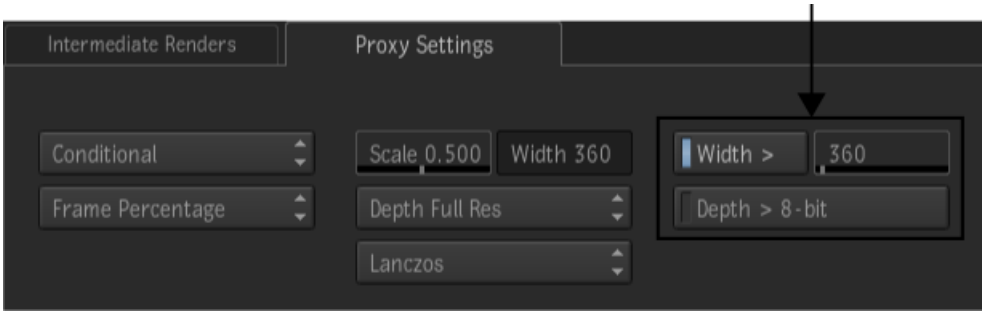
- If the proxy resolution and bit depth settings are the same as the as the full resolution clip, proxies are not generated.
- If the proxy resolution is the same as the full resolution clip, but their bit depth is different, proxies are generated.

- Select Frame Percentage from the Proxy Size box and then enter a value in the Scale field. For example, enter 0.50 to create and store proxies that are 50% of the clip resolution.



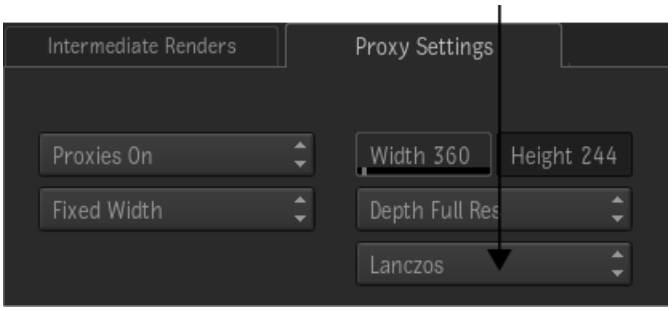
Proxies always have the same aspect ratio as the full-resolution clip. The width of proxies for clips with the default resolution corresponding to the scale value you set is displayed in the Width field.

3 If you selected Proxies Conditional, set your conditional criteria.



- To store proxies for all clips of a certain width and greater, enable Frame Width > and then enter a value in the adjacent field. For example, enter 4000 to store proxies for all clips wider than 4000 pixels.
- To store proxies only for 10-bit, 12-bit, or 12-bit unpacked clips, enable Depth > 8-bit.

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

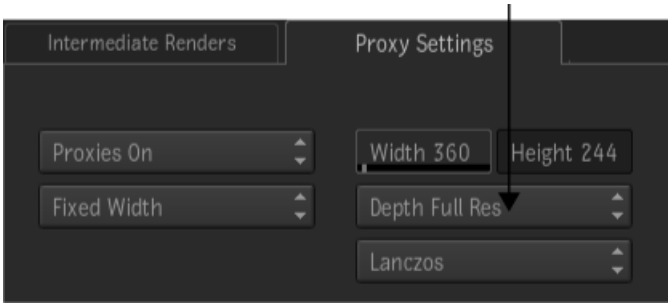


Proxy quality affects rendering and proxy generation duration since lower qualities are faster to calculate. However, the quality does not affect the amount disk space required for proxies.

Select:	**To specify:**
Lanczos	Excellent and sharp results. Recommended for upscale and downscale. Expensive to compute.
Shannon	Excellent and sharp results. Results are sharper than Lanczos in small details. Recommended for upscale and downscale. Expensive to compute.
Gaussian	Medium quality and softer results.
Quadratic	Medium quality and softer results.
Bicubic	High-quality results, but not as sharp as Shannon. Use for both upscale and downscale.
Mitchell	High-quality results, but not as sharp as Shannon. Use for both upscale and downscale.
Triangle	Low quality results that are fast to compute. Use for downscale.
Impulse	Very low quality results that are fast to compute. Use for downscale.

Select:	To specify:
Draft	The lowest possible quality. This is the quality used when proxies are generated automatically following video I/O.

5 From the Proxy Bit Depth box, select the bit depth for proxies.



Select:	To set:
Depth: 8-bit	The proxy bit depth to 8 bits.
Depth: Full Res	The proxy bit depth to be the same bit depth as that of the clip.

Setting Cache and Renders Format

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

When rendering, media is rendered to the format specified in the Cache and Renders tab.

When importing media with Cache Source Media enabled, the media is transcoded to the format specified in the Cache and Renders tab. Media imported with Cache Source Media disabled is not transcoded and keeps its original format.

To set the Cache and Renders format:

1 Click the Cache and Renders tab.

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

NOTE Preferred Formats are all available in RAW and non-RAW flavours. Use RAW flavours in most cases, and non-RAW ones if you plan on exporting OpenEXR in linked publish.

The Format Restrictions for the selected format are displayed in a table to the right of the drop down menu.

Format Restriction:	Description:
Maximum Width	Displays the maximum width of a frame for it to use the Preferred Format. A frame wider than this is written using one of the Alternate Formats.
Maximum Height	Displays the maximum height of a frame for it to use the Preferred Format. Any frame bigger than this is written using one of the Alternate Formats.

Format Restriction:	Description:
Depths	Displays the maximum bit depth of a frame for it to use the Preferred Format. Any frame at a bit depth higher than this is written using one of the Alternate Formats.
Alternate Formats	Displays the fallback formats used to write frames that do not fit within the parameters defined by Maximum Width, Maximum Height, and Maximum Bit Depth. The application follows a specific fallback strategy, based on the bit depth of the frame and the displayed formats: ■ DPX, OpenEXR, RAW: DPX for 8-, 10-, and 12-bit frames. OpenEXR for 16-bit floating point frames, RAW for 12-bit packed ones. ■ DPX, RAW: DPX for 8-, 10-, and 12-bit frames. RAW for 12-bit packed, 16-bit floating point, and higher.

Working with Users

Creating a User

You can create a user:

- On start-up from the Project panel.
- During a session, by selecting Project and User Settings from the Flame menu.

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

To create a user:

1. If you are creating a user on start-up, select New under User on the startup screen. (If you are starting Flame Premium for the first time, New is the only option).
2. If you are creating a user in the middle of a session, from the Flame menu, select Project And User Settings.

   The Project and User Settings dialog box appears.
3. Under User, click New.

   The Create New User Profile dialogue box appears.
4. Under User, click New.
5. In the Name field, enter a name for the new user.
6. Set your Preferences Directory.

   If you are connected to a network with multiple hosts, you can select a remote host to store your user preferences. The default option is Local Host.
7. Select your Keyboard Shortcuts scheme. Options are:
   - Smoke (FCP 7)
   - Flame
   - Smoke Classic
8. If you do not want to copy user preferences from an existing user, select New Prefs and click Create.

   The user is created. The dialog box closes and you can begin working.
9. If you want to copy user preferences from an existing user, select Copy From.

10 From the Files box, select the type of files to copy. Options are:
   - All
   - Shortcuts

   NOTE If the user you want to copy the settings from was created on an older version of the application, you can only copy shortcut preferences.

11 From the Host box, select the host on which the user is saved.
12 From the Version box, select the software version used to create the user.
13 From the User box, select the user from which you want to copy the preferences.
14 Click Create.
   The user is created. The dialog box closes and you can begin working.

   NOTE If you create a user in the middle of a session, you must load it by clicking Load in the User panel.

Editing a User

You can edit a user after it has been created.

To edit a user:

1 From the Flame menu, select Project And User Settings. The Project and User Settings dialog box appears.
2 From the User box, select the user you want to edit.
3 Click Edit. The Edit User Profile dialogue box appears.
4 Modify the appropriate settings.
5 Click Done and Confirm.

The modifications are applied to your user.

Deleting a User

To delete a user:

1 From the Flame menu, select Project And User Settings. The Project and User Settings dialog box appears.
2 From the User box, select the user you want to delete.
3 Click Edit. The Edit User Profile dialogue box appears.
4 Click the User Edit box and select Delete User.
5 Click Done and Confirm.

The user is deleted.

Organizing Media in the Workspace

5

Displaying and Hiding the Media Panel

The first time you run the application, the Media panel is hidden. To display it, from the Media panel View Mode box, deselect Hidden.

The Media Library is the centralized location to which you import all of your media. The Media Library organizes your media into Libraries which contain your clips. Within a Library, you can create as many folder as you like to organize your clips. Libraries can be opened and closed. Closing the Libraries releases them from memory and makes their content inaccessible. You open and close Libraries from the contextual menu.

The Media panel is persistent throughout the application. You can toggle the display of the Media panel by using the `Shift + Esc` keyboard shortcut.

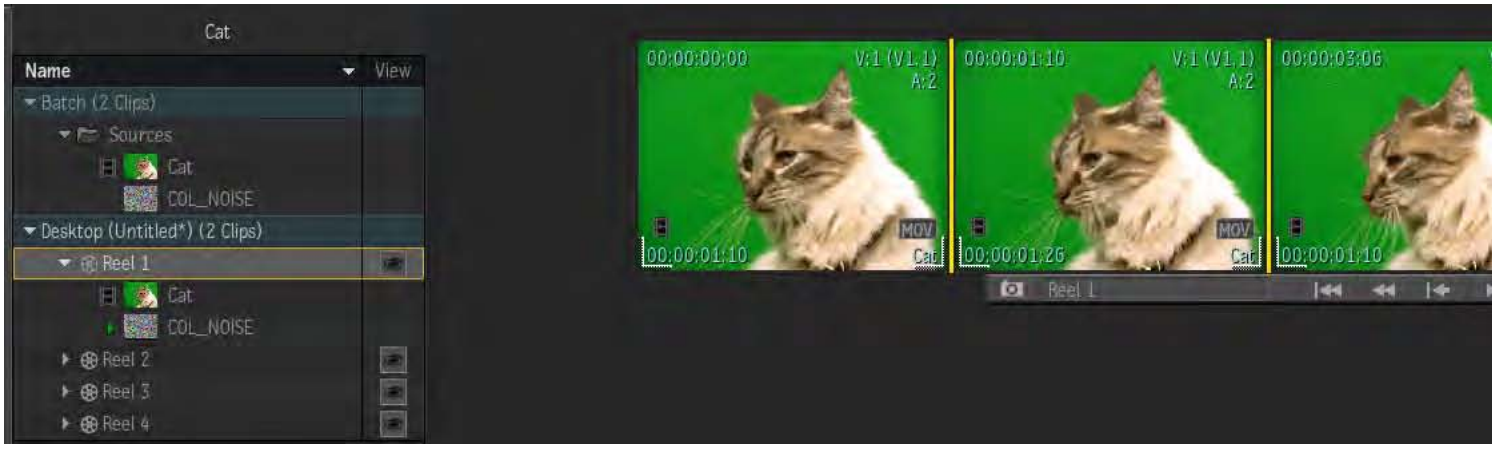
TIP You can view the clips in the Media panel as large thumbnails by `Alt + clicking` them.



Dragging and Dropping media in the Media Panel

From the Media Library you can drag and drop your clips to the different sections of the Media panel, creating copies of the clips for you to work on, leaving the originals intact in the Media Library.

Drag and drop the media from the Workspace Library to the Desktop section of the Media panel to perform gestural edits to the copies on the Desktop Reels.



Drag and drop the media from the Workspace Library to the Batch Sources section of the Media panel to apply procedural effects to the copies in the Batch Sources folder.



Drag and drop the media from the Workspace Library to the Shared Libraries to make the copies of the clips available to other workstations on your local network.



Tips for Working with the Media Panel

Dragging a clip from the Workspace Library to the Batch section header (above the Sources folder) adds the clip to the Batch schematic and automatically opens Batch view.

Dragging a Library or a folder from the Media Library to the Desktop section header (above the Reels list) automatically creates a new Reel named after the Library or folder.

Dragging a Reel from the Desktop section of the Media panel or from the Desktop itself to the Media Library automatically creates a Library named after the Reel.

When in Desktop view, you can display the contents of Libraries and folders in the Media Library by double clicking the Library or folder or by enabling the eye icon to the right of the Library or folder. The view automatically switches to Thumbnail and displays the contents of the Library or folder. You can then drag the clips you want to work with to a Reel in the Media panel and set the view back to Desktop.

Protecting Media from Editing in the Media Library

You can protect clips in the Media Library from editing, by enabling the Protect from Editing button under the General tab in the Preferences. When this preference is enabled, the following operations cannot be performed on clips in the Media Library:

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

Clips on the Desktop and in the Batch Sources area can still be edited with the Protect from Editing preference enabled.

Changing Your View of the Flame Premium Workspace

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

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

- **Desktop Reels**: Displays reels onto which you place clips and sequences. Clips and sequences can be edited and played back from the reels. The contents of reels can also be displayed in Thumbnails view, by enabling the View icon next to reel when in Thumbnails view.
- **Thumbnails**: Displays the clips within the selected Media panel folder or reel as thumbnails.
- **Player**: Displays and plays back the selected clip in a Player.
- **Source - Sequence**: Displays and plays back the selected source clip and sequence in two side-by-side players.
- **Triptych Player**: Displays and plays back one or multiple clips in three side by side players. This can be useful for colour matching, for example.
- **Trim View**: Displays the last (outgoing) and first (incoming) frame above the Timeline from the two clips you are trimming.

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

Generating Clips

You can generate the following generic video and audio clips and use them as would any other media:

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

To generate a clip:

1. In the Viewing panel, right-click and select New.
2. Select the type of clip you want to generate.
3. Set the options in the dialog box that appears.
4. Click Create.

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

Displaying Media Metadata

By default, basic metadata is displayed on each corner of a thumbnail, such as current timecode, video and audio tracks, duration and clip or sequence name. You can also view detailed metadata for each clip or sequence in the Viewing panel.

There are two ways you can display detailed clip information:

To display clip information on a thumbnail:

1. Make sure you are in Thumbnail view.

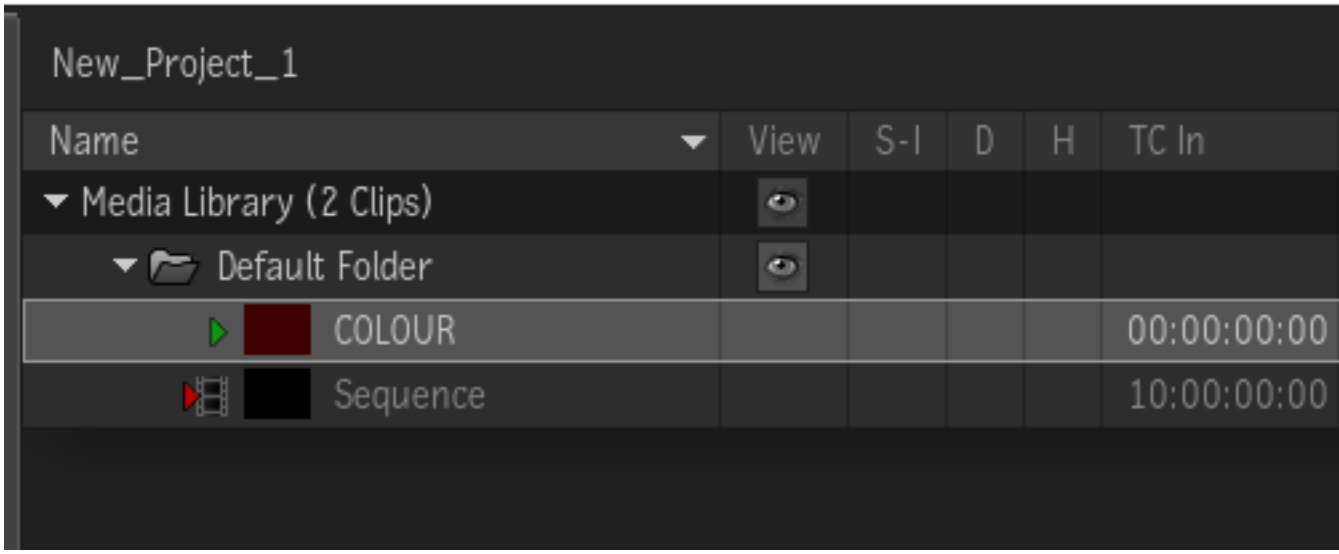
2 Press the `Alt` key and click a thumbnail.



To display detailed clip information in the Media panel:

1 From the View mode box, select Details.





Creating and Opening Sequences

A sequence is an edited clip that can contain multiple sources. When you edit a clip, it becomes a sequence. When you create a new project, an empty sequence is created within the project. You can create new sequences from the Media panel.

Creating an Empty Sequence

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



The New Sequence dialog appears.

2 Specify the settings for your sequence.

3 Press Create.

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

Opening a Source as a Sequence

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

2 From the contextual menu, select Open as Sequence.

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

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

Opening Multiple Sequences Simultaneously

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

2 From the contextual menu, select Open All as Sequences.
All the clips and sequences within the reel, folder or Library are opened as individual sequences.

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

Opening Multiple Clips as One Sequence

1 Create an empty sequence.
2 Select the clips to load and join as one sequence from the Media panel or the Viewing panel.
3 Drag the selected clips to the empty sequence at the desired timecode.

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

Closing All Sequences Simultaneously

1 Mouse over an open sequence tab in the Editing panel.
2 From the contextual menu, select Close All Sequences.

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

Working in Thumbnails View

Organizing the Thumbnails in the Viewing Panel

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

Arranging the Thumbnails

When in Thumbnails view, you can arrange the displayed thumbnails in a number of ways. To arrange the thumbnails, do one of the following:

From the contextual menu, select:	To:
Arrange ➤ Clean Up All	Tile the thumbnails across the Viewing panel.
Arrange ➤ Fit All	Tile the thumbnails across the Viewing panel at the highes
Arrange ➤ Cascade Selection	Arrange the selected thumbnails as cascading stacks.

Displaying a Snap Grid

Use the grid to snap thumbnails into position when moved.

To display a snap grid in the Thumbnails view:

1 Make sure the Viewing panel is set to Thumbnail view.
2 Access the Preferences dialog box by clicking the Flame button at the bottom of the window.
3 Under the User Interface tab, in the Thumbnails view section, enable Snap To Grid.
4 Set your preferred grid size.

5 Set your preferred proxy height.

6 Click Close to close the Preferences dialog box.

The Viewing panel displays a grid of the specified size.

Ganging Clips in the Viewing Panel

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

To gang clips in the Viewing panel:

1 From the View Mode box, select Thumbnail view.

2 In the Viewing panel, locate all the clips or sequences you want to gang, and move their positioners to the timecode you want to lock.

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

3 Hold the `Ctrl` key and select the clips or sequences you want to gang.

4 While the cursor is over one of the clips to be ganged, from the contextual menu, select Gang.

All selected clips are ganged and the clip information turns green.

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

5 Jog the positioner of one of the ganged clips.

All ganged clips or sequences are jogged.

To select all clips in a gang:

1 Select any clip or sequence that is part of a gang.

2 From the contextual menu, select **Gang ➤ Select Gang**.

All ganged clips or sequences are selected.

To remove a clip from a gang:

1 Select the clip(s) or sequence(s) you want to remove from the group.

2 From the contextual menu, select **Gang ➤ Ungang**.

All selected clips or sequences are unganged.

Working with Reels

Organizing the Desktop Reels

A Desktop can contain as many reels as you like. The reels can be displayed in any order within the Desktop. However, a maximum of six reels can be displayed on a Desktop at a time.

To add a reel to a Desktop:

1. From the Media panel, select the current Desktop.
2. From the contextual menu, select New Reel.

The new reel, labeled Untitled Reel, is added to the Desktop. You can rename it by highlighting it and clicking the name field, in the Media panel or by clicking the name field on the reel itself and clicking the camera icon to the left of the reel name to save the change.

NOTE Saving two reels with the same name triggers a prompt to either Cancel, Bump or Replace the snapshot. Selecting Bump creates a new version of the reel and appends a version suffix to the reel name.

To display active reels:

1. From the Media panel, click in the View field to show or hide the selected reel.
   An eye icon appears in the View field next to the active reels.

NOTE A maximum of six reels can be displayed on the Desktop at a time.

To change the order of the reels on the Desktop:

1. Click the Move button to the right of the reel you want to move.



2. Drag and drop the reel to its new position on the Desktop.

Saving and Restoring Desktops with Snapshots

You can save multiple Desktop snapshots to the Media panel and restore them. When you restore a Desktop Snapshot, you restore settings and media associated with the Desktop.

When you make a change to a saved Desktop (for example, deleting a clip from a reel), an asterisk appears next to the Desktop section in the Media panel. This lets you know at a glance that changes have been made to the Desktop since you last saved it.

To save a Desktop Snapshot:

1. From the Viewing panel menu bar, enter a name in the Desktop Name field.
2. Click the Desktop Snapshot button.



A closed snapshot folder is created and can be seen in the Desktop Snapshots area in the Workspace Media panel. If you right-click, and select Open Snapshot, you can expand the reels under the Desktop Snapshot to view the sources saved with the snapshot.

NOTE Saving two snapshots with the same name triggers a prompt to either Cancel, Bump or Replace the snapshot. Selecting Bump creates a new version of the snapshot and appends a version suffix to the snapshot name.

To restore a Desktop:

1 In the Media panel, right-click the Desktop snapshot you want to restore and do one of the following:
- **Replace Desktop** : Clears the current Desktop and replaces it with the Desktop you restored.
- **Append Desktop** : Appends the restored Desktop to the current Desktop.

The selected Desktop is loaded and appears under Desktop, in the Media panel.

You can also drag and drop the Desktop Snapshot to be restored, from the Media panel to the Desktop or simply double-click the Desktop Snapshot to restore it. A prompt pops up asking you to select between the Replace and Append options. Once you make your selection, the selected Desktop is loaded and appears under Desktop, in the Media panel.

Dragging and dropping multiple Desktop Snapshots is also possible. In this case, all of the selected Desktop Snaphots are appended together and all of the media contained in the multiple snapshots are displayed on the Desktop, on the appropriate reels.

Scrubbing Clips, Sequences and Reels

Scrubbing a Clip or Sequence

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



Scrubbing a Reel

Click:	To:
	Go to the previous clip or sequence on the reel.
	Go to the first frame of the clip or sequence currently over the reel's playback controls.
	Scrub the reel backward.

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

To scrub a reel using the cursor:

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

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

Displaying Collapsed, Frames and Storyboard Views

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

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



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



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



To toggle the view for one clip or sequence:

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

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

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

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

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

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

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

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

Revealing a Clip in a Reel

If you have a lot of clips on a reel, it can be difficult to quickly locate a specific clip. To facilitate this, you can perform a Reveal in Reels operation. Reveal in Reels automatically centers the selected clip on its reel, eliminating the need to scroll through the reel to locate your clip.

To reveal a clip in a reel:

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

The selected clip is centered on its reel.

Syncing Reels

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

To sync reels:

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

The selected reels are synced.

Undo and Redo

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

- **Undo combo box**: Click to undo the last operation. You can also click the white arrow to list the last 10 operations performed. Selecting an operation from the list undoes all operations performed after the selected operation (including the selected operation). The undo operations are specific to each tab.

  TIP You can set the Undo levels anywhere between 2 and 50, under the General tab in the Preferences. It is set to 10 by default.

  NOTE While in Batch FX, you have access to all of your undo levels. Once you exit Batch FX and return to the timeline, all operations performed in Batch FX are considered one undo level. This means that even if you performed 7 different operations in Batch FX, when you return to the timeline you can only undo all the Batch FX operations at once, not each individual operation.

- **Redo combo box**: Click to redo the last operation. You can also click the white arrow to list the last 10 undone operations. Selecting one reapplies all undone operations performed before the selected operation. The redo operations are specific to each tab.

Setting Up a Flare Workflow

6

About Flare

Flare is a fully compatible assistant to Flame Premium, featuring the same creative toolset as Batch. All Batch nodes found in Flame Premium are fully supported in Flare. You can extend the capabilities of Flame Premium by performing any Batch task on a Flare system. You can also offload time-consuming tasks such as rotoscoping and particle creation to Flare.

Because the focus of Flare is on the Batch toolset, there are some tasks performed by Flame Premium that cannot be performed by Flare. For example, Flare does not support video I/O or conforming. In addition, Flare supports archiving, but only to or from file archives.

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

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

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

Flare Workflow

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

In the following workflow, steps without cross references are specific to Flare and are detailed in this chapter. Steps with cross references are covered in other chapters of the user's guide and may contain some Flame

Premium functionality that does not apply to Flare. Any Flare-specific considerations are outside the scope of the other chapters.

Flare with Flame Premium workflow:

1. Start the application and connect to a remote host computer. See Starting Flare (page 48)
2. Load the media from the Flame project's shared folder by either dragging it directly to the schematic area or by dragging it to the Flare Workspace Library to make a copy of the media and dragging the copy to the schematic area.

   NOTE Make sure the media was saved to a shared folder in Flame Premium to make them accessible to Flare.

   See Shared Libraries (page 47)
3. Work with the Batch toolset. See Using Batch and Batch FX (page 345).
4. Save your Batch branches as BFX clips by selecting the last node in a branch and selecting Create BFX from the contextual menu.

   A BFX clip contains all of the media and Setup information of the Batch branch it was created from. It is essentially a Batch branch represented as a clip, which can be expanded back into the original branch by selecting Explode BFX Clip from the contextual menu.
5. (Optional) Process by doing one of the following:
   - Render clips from the contextual menu.
   - Render clips using the Render node.
6. Place the resulting media (BFX clips) back into the shared folder, making it available to the Flame workstation.

   TIP It is recommended to create a sub folder within the Flame shared folder named From_Flare, for example, to be able to easily distinguish the initial media from the result media. In order to create a sub folder, you first must acquire write access (lock).

Considerations for Working in a Collaborative Environment

The ability to work in a collaborative environment is integral to the Flare remote connection workflow. From Flare, you can open a Flame Premium project on a remote storage volume and access all of the media stored in a shared folder. Conversely, Flame Premium systems can also access Flare media.

Workspaces

By default, when Flare connects to a remote Flame workstation, it automatically creates its own Flare workspace, associated to its own workstation, and which has access to all the contents of the shared folders in the Flame project.

If there are multiple Flare stations on your network, there may already be existing Flare workspaces for a given Flame project.

- If there is only one existing workspace and that it **is** associated to the current Flare workstation, Flare selects it automatically.
- If there is only one existing workspace and that it **is not** associated to the current Flare workstation, a Workspace Selection button is displayed next to the project name, enabling you to either use the existing Flare workspace or to create a new one.

- If there are many existing workspaces, a Workspace Selection button is displayed next to the project name, enabling you to either use an existing Flare workspace or to create a new one. If you do not click the Workspace Selection button, Flare automatically creates a new workspace for the current project.

NOTE Workspaces are project specific. You can only access the workspaces of the current project.

Shared Libraries

About Shared Libraries

The Flare workflow is based on shared libraries. Any media placed in the shared libraries becomes accessible to remote workstations that connect to the project. Shared libraries enable Flame and Flare to quickly and conveniently share media, allowing you to spend more time on creative tasks and less time managing your media.

Write Access

Placing media in a shared library automatically grants read access to any remote workstations connected to the project. The Flare operator can access the media and start working immediately after connecting to the Flame project. Once finished, the Flare operator needs to save the resulting media back to the shared library. To do this he must acquire write access.

Write access locks the library so that only the user having acquired write access can modify the library contents (i.e. write to the library). Read access is still available to other remote workstations connected to the project. Once the media is saved, the user can disable write access, enabling other remote workstations to obtain write access and modify the content.

To acquire write access:

1. Select the top level shared library you want to acquire write access to in the Media panel.
2. From the contextual menu, select Acquire Write Access.

To disable write access:

1. Select the top level shared library you want to acquire write access to in the Media panel.
2. From the contextual menu, select Release Write Access.

Open / Close

You can open and close shared libraries. Closing shared libraries turns off access to the media within the shared libraries as well as freeing up your system memory.

To open / close shared libraries:

1. Select the top level shared library you want to open or close in the Media panel.
2. From the contextual menu, select Open Library or Close Library.

Refresh

You can manually refresh the content of shared libraries, to make sure you have access to all the shared media.

To manually refresh the content of shared libraries:

1. Select the top level shared library you want to refresh in the Media panel.
2. From the contextual menu, select Refresh.

   The content of the selected shared library is updated.

Starting Flare

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

To start Flare:

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

   The Project Management menu appears.
2. Select a remote host from the Host Computer box. If the remote host has more than one volume, select a volume from the Volume box. Click Open.
3. Select a project from the Project box or create one. See Managing Projects and Users (page 21).
4. Select or create a user:
   - To select a user, select the user name from the User box. You can select a local or remote user by first selecting the appropriate option from the List From box.
   - To create a user, select <New> from the User box. In the Create User menu that appears, you can choose to create a user by copying a user profile.

   If you copy a user profile, note the following:
   - You can copy a user profile from a local or remote system.
   - You can copy all preferences from a user profile in the current version of the application.
   - You can copy only keyboard shortcuts preferences from a user profile in an older version of the application.

     See Managing Projects and Users (page 21).
5. Click Start.

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

To exit Flare:

1. Click the Flare button at the bottom right of the user interface.
2. Select Exit Flare.

Changing Projects and Users

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

You perform these tasks from the Project and User Settings section of the Preferences menu.

You create projects and users in the same way as from the start-up menu.

To access the Project and User Settings:

1. Click the Flare button at the bottom right of the user interface.

2 Select Project and User Settings
The Project Management dialog box appears.

To change projects in a Flare session:

1 Select a project from the Project box.
2 Click Load.
The Confirm dialog box pops up.
3 Click Confirm.
4 Click Close.
The new project is loaded.

To change users in a Flare session:

1 Select a user from the User box.
2 Click Load.

NOTE You can select a local or remote user by first selecting the appropriate option from the List From box.

3 Click Close.
The new user is loaded.

To edit a user profile in a Flare session:

1 Select a user from the User box.

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

2 Click Edit.
3 Modify the user.
4 Click Done.
The Confirm dialog box pops up.
5 Click Confirm.
6 Click Close.
The modifications are applied to the user.

Importing and Exporting Media 7

Importing File-Based Media

TIP To work in a manner similar to offline editing suites, enable **MediaHub ➤ General tab ➤ Cache Source Media**. This creates local, transcoded, and managed versions of your media. To work online, disable Cache Source Media: the clips remain linked to the original media, and are not transcoded.

To import media using the MediaHub:

1. Click the MediaHub tab.
2. Using the MediaHub file browser, navigate to the file to import.
3. Review and edit as necessary the resolution, the bit depth and scan mode, and the LUT options. See Colour Management (page 1539).
4. Click Import.

Importing File-Based Media Tips

- Drag and drop a folder in the Media Library. Flame Premium imports all the media files and folders contained therein. Note that only supported media files and folders are imported: other file types are ignored.
- Drag and drop multiple files in one operation: hold Ctrl to select the files to import before dragging them over to the Media Library.
- Before importing the media, review it using the Previewer.
- Use the Previewer's tabs to display the clip information and additional metadata.
- For large media, use the Previewer to set In and Out points and import only a subclip.
- If the media file to import is located on a network drive, and if you plan on using referenced media instead of project media, make sure that the network connection is at at least 1 GB ethernet to have decent playback.
- If the media file to import is located on a removable media such as a USB drive, and you plan to remove the drive before the end of your project, import with Project Media enabled. This way Flame Premium creates natively managed media out of the original, removing the need for the connected drive.

- A folder in the Media Library is the same as a bin in Avid Media Composer or Apple Final Cut Pro.
- From the Media Library, right-click > Import... to import media to that location.
- The files to import cannot contain any hyphen (-) or square brackets ([ ]).

NOTE Getting clips with checkerboards instead of images? Check if your workstation is still connected to the volume you used to store your media; if not, reconnecting the volume to the workstation should do the trick.

Exporting Media to Files

To export a clip:

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

   In the MediaHub, you can also drag-and-drop the file from the Media panel to a location displayed in the MediaHub browser.
2. Navigate to the clip's destination, using the Media Export window.
3. Select an Export type and a Format Preset.
4. Click Export.

   Flame Premium takes a few instants to prepare the export job. Once that preparation is done, the rest of export happens in the background and frees up Flame Premium for your use.

TIP Select multiple clips (CTRL+Click or Shift+Click) to export multiple clips in one operation. You can also export a folder and its structure. Note that some timings are incompatible with some codecs, such as exporting a 30fps clip to XDCAM HD 422 (which accepts only 25fps source clips). In such cases, clips using incompatible timings are not exported.

NOTE Flame Premium exports OP1A MXF files. Avid only supports OP-Atom MXF files. Use AMA MXF plug-in to bring the files into an Avid application, like you would for QuickTime files.

Publishing A Sequence as an EDL With Media

To publish a sequence as an EDL with transitions but flattened tracks:

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

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

   Flame Premium takes a few instants to prepare the export job. Once that preparation is done, the rest of export happens in the background and frees up Flame Premium for your use.

NOTE An EDL can only describe one video track, so your exported sequence will be flattened.

Publishing a Sequence as a Custom EDL

To publish custom EDL without any media:

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

   In the MediaHub, you can also drag-and-drop the file from the Media panel to a location displayed in the MediaHub browser.
2. Navigate to the EDL destination, using the Media Export window.
3. Set Export Type to Sequence Publish.
4. Set Format Preset to one of the EDL presets.
5. Click Show Advanced Options.
6. Click **Sequence Options ➤ Custom EDL Export**.
7. Set the EDL options as required.
8. Click Export, and then click Save Generated EDL.

About the Custom EDL Export Window

EDL Event Combination box Select an option to indicate how events with the same source timecodes, record timecodes, and tape ID are combined when the EDL is generated.

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

EDL Format box Select the format of the generated EDL. You can save EDLs in any of the following formats:

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

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

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

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

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

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

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

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

Use Delayed Dissolves button When enabled, delayed dissolves are included in the generated EDL.

Default Tape field Enter a tape name to override the default tape ID when saving an EDL. Source clips are assigned tape IDs when loaded using the Input Clip or Import EDL menu. For example, an edit that uses a clip created with the Colour Corrector does not have a tape ID. When the EDL is generated, the clip is given the tape ID in the Default Tape field.

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

Use Tape Name Extension button Enable to add a list to the end of the EDL that shows the relationship between the short tape name (8 characters, maximum) used in the EDL and the long tape name (52 characters, maximum) that you can set in the Import EDL menu.

Media Export Window Overview

Basic Options

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

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

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

Advanced Options button Enable to display the advanced export options. You should not have to edit these settings, unless the pre-configured presets cannot meet your needs.

Create Export Format Preset Dialog Box

Preset Scope box Select Shared to have this new preset available across users and projects. Select This Project Only to make it available to the current project only.

Sequence Options Tab

Sequence Format box Select the format of the sequence. Media Only exports the segments of the sequence as individual clips, but does not export the sequence itself.

Include Video button Enable to include in the published sequence the video tracks information. Required to export video.

Include Audio button Enable to include in the published sequence the audio tracks information. Required to export audio.

Sequence Filename field Displays the sequence filename based on the Pattern field. Each type of exported file has its own filename defined in the relevant tabs. Non-editable.

Sequence Filename Pattern field Displays how to name the exported media files. Build a dynamic naming scheme using Add Token, or characters normally allowed in a file name. Create folder structure using / . The extension for the file format is automatically appended. Frame identifiers are automatically added to image sequence files. Editable.

Add Token box Inserts in the Pattern field a token to build a dynamic filename. Note: date is the current date, formatted as YYYY_MM_DD.

Select:	To insert the token:	Definition:
Clip Name	<name>	The clip's name
Date	<date>	The current date (YYYY_MM_DD)
Workstation	<workstation>	The name of the workstation, as displayed in the Host Computer field in **Flame ➤ Project and User Settings**.
Project	<project>	The name of the project, as displayed in **Flame ➤ Project and User Settings**.
User	<user>	The user name, as displayed in **Flame ➤ Project and User Settings**.
Clip Height	<height>	The clip's height, after resize if applicable.
Clip Width	<width>	The clip's width, after resize if applicable.
Tape/Reel/Source	<tape>	The clip's tape name
Time	<time>	The time, formatted HH:MM:SS.

Export Video button Enable to export the segments of the sequence as files of the type specified in Video Format.

Video Format box Select the type of video file to create. For movie, you select the wrapper and codec in the Movie Options tab. For file sequence, you select the file type in the Video Options tab.

Media Source box Select Use Original Media to export the original source referred by the exported sequence, without any modifications. Select Use Media with FX to export the rendered media.

Video Tracks and Transitions box Select Keep All Tracks to export a clip for each segment of the sequence. Select Flatten Tracks to export commit every transition and flatten the sequence. Select Flatten with Transitions to flatten the sequence and commit every transition except dissolves, creating a single clip.

Include Video Handles button Enable to add head and tail frames to exported segments.

Video Handles field Display the amount of head and tail frames.

Export Audio button Enable to export the audio files the sequence will link to.

Audio Source Selection box Select Use Original Media to export the original source referred by the exported sequence, without any modifications. Select Use Media with FX to export the rendered media.

Audio Track State box Select Flatten Tracks to commit all transitions and create one audio clip per track. Select Flatten Tracks with Transitions to create an audio clip per track but keep live transitions. Select Keep All Tracks to export one audio clip per audio segment.

Include Audio Handles button Enable to add head and tail frames to exported segments.

Audio Handles field Enable to export the audio files the sequence will link to.

Movie Options tab

Container Format box Select the container for the exported file.

YUV Headroom Disable to export a YUV headroom compliant clip (also known as valid- or legal-range). As a general rule, should be enabled when deliverable is for broadcast. Enable to export a standard, full-range, clip.

Include Audio button Enable to include audio tracks within the exported file. Available only Export Type is Movie.

Video Options tab

Video Format field Select the format for the exported media.

Compression box Select the image compression to apply to the exported file. The available compressions depend on the selection in File Format.

Codec Profile box Select a pre-defined video compression codec profile when exporting QuickTime files using the H.264 or MPEG-4 codecs.

LUT Activation button Enable to apply the LUT displayed in Applied LUT to the clip .

LUT Type box Select the type of LUT to apply to the clip.

Applied LUT field Displays the type of conversion LUT applied to the clip, either imported using Import, or edited using Edit.

Import button Use to browse and select a LUT.

LUT Editor Access button Click to open the LUT editor.

Link Original Media button Enable to hard link the published files to the original files if both the original and exported files are located on the same filesystem. If not, the application creates soft links back to the originals. Available for file sequences only.

This option saves disk space on export as files that remain unchanged by the export are not duplicated. Unchanged in this context means they were not rendered not resized in anyway. For example, a file sequence of 20 dpx are imported in Flame Premium. Of those 20, 12 are modified in some fashion. With Link Original Media enabled, of the sequence of 20 dpx, only the modified 12 are actually created at export; the other 8 dpx are just linked to from the export folder.

NOTE You can have Flame Premium copy files instead of creating symbolic links across filesystems by editing the *StandardFSMediaOptions* keyword in the *stone+wire.cfg* configuration file.

Frame Padding field Define the padding of the frame identifiers appended to each file of an image sequences. This ensures that the images are listed and stored in the correct order. Only used with image sequences.

For example, a frame pad of 6 indicates that each frames's file name has its frame identifier padded with a number of zeroes required to make it a 6-digit number: frame 1 is written as 000001, frame 22 as 000022, frame 55555 as 055555, and so on.

Use Timecode button Enable to base the start number of the exported sequence of numbered images on the timecode read from the clip file.

Start Frame field Enter the start number to be used in the exported sequence of numbered image files. Disabled when Use Clip TC Names is enabled.

Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

DPX Transfer Characteristics box Select an option to identify the attributes associated with a particular film or video format, such as resolution, frame rate, or colour space. Available when Video Format is set to DPX.

When exporting DPX files, you can choose a DPX Transfer Characteristic. A DPX Transfer Characteristic is information that is stored in the DPX image file header. It identifies the attributes associated with a particular film or video format, such as resolution, frame rate.

Setting a Transfer Characteristic in no way changes the *image* information stored in the DPX file. The Transfer Characteristic simply indicates the attributes of the DPX file read by another device or application. Some devices or applications may take advantage of this information to improve workflow. For example, selecting Logarithmic can allow a film recorder to adjust its parameters to print film-originated DPX files with the correct densities.

Select:	For:
Z depth homogeneous, Z depth linear, PAL, NTSC, CCIR 601 (525), CCIR 601 (625), CCIR 709-4, SMPTE 274M	Images that you want to identify as one of these types. Although the SMPTE 274M standard defines these DPX Transfer Characteristics, it does not provide usage specifications for them. As a result, these Transfer Characteristics are not generally used in the industry.
Unspecified	Images where the format is not specified.
Logarithmic	Negative film scanners recording status M densities.
Linear	Video images which have built-in gamma correction. This refers to images having a true linear quantization scheme (such as CG-originated material).
Printing Density	Negative film scans which use the SMPTE Printing Density settings. SMPTE Printing Densities use status M density measurements with a higher gain in the red component.
Academy Density Exchange (ADX)	Images at 10- and 16-bit film density encoding, in which colour information is encoded using a logarithmic scale. Usually used in the context of an Image Interchange Format workflow.

DPX Colorimetric Specification box Select an option to identify the colorimetric specifications used to encode the DPX files. Available when Video Format is set to DPX.

Setting a Colorimetric Specification in no way changes the *image* information stored in the DPX file. The Colorimetric Specification simply indicates the attributes of the DPX file read by another device or application. Some devices or applications may take advantage of this information to improve workflow. For example, selecting Academy Density Exchange (ADX) can allow a film recorder to adjust its parameters to print film-originated DPX files with the correct densities.

JPEG Quality field Specifies the degree of quality versus compression. A value of 0 gives the lowest quality (and highest compression), while a value of 100 gives the best quality (but applies no compression). Available when File Format is set to JPEG.

Frame Width field Displays the frame width of the selected clip. By clicking it you activate the field, allowing you to enter the frame width value that you want to use on export.

Frame Height field Displays the frame height of the selected clip. By clicking it you activate the field, allowing you to enter the frame height value that you want to use on export.

Bit Depth box Select a bit depth to be used on export. Some file formats support multiple bit depths. Bit Depth is active only when Resize is enabled.

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

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

Resize Filter box Select the filter option to determine the quality of the interpolated resize result. The Resize Filter box is active only if Fit Method is set to Crop Edges, Fill, or Letterbox.

Select:	To get:
Impulse	Quick, low-quality results.
Triangle	Moderate results with little processing overhead.

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

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

Aspect Ratio field Displays the aspect ratio defined by Aspect Ratio Presets. Editable.

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

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

Select:	To resize:
From Clip	Using the scan mode of the source clip.
Progressive	A frame-based clip to another frame-based clip.
Field 1	A clip by drawing Field 1 followed by Field 2.
Field 2	A clip by drawing Field 2 followed by Field 1.

Audio Options Tab

Audio Format box Select the audio format of the exported audio tracks. If Export Type is Movie, the audio is embedded within the video file. Any other Export Type outputs a separate audio file.

Audio Bit Depth box Select the bit depth of the exported audio file.

Audio Compression field Select the compression for the exported audio file. The available options depends on the Audio Format.

Audio Sample Rate box Select the sample rate of the exported audio.

Audio Mixdown box Select a mixdown to apply to the audio tracks, if any are included with the source clip.

Select:	To mix down:
No Mixdown	Nothing
Mixdown As Is	With the current output strip assignments.
Mixdown To 4 tracks	To four tracks. The output strips are assigned sequentially in fours to the mixed-down channels (where M1 goes to 1, M2 to 2, M3 to 3, M4 to 4, M5 to 1, and so on).
Mixdown To Stereo	To one stereo track. The output strips are assigned sequentially in twos to the mixed-down channels (where M1 goes to 1, M2 to 2, M3 to 1, M4 to 2, and so on).
Mixdown To Mono	To one mono track.

Clip Options Tab

Create Clip in Media Library button Enable to automatically create an unmanaged copy of the exported media in the Media panel, linked to the exported files. Available when performing a Sequence Publish with a File Sequence, without any resize or bit depth change. This is similar to importing back the exported media with Create Source Cache disabled.

Setting Default Format Presets

The default preset is the one that appears first in the Presets drop-down list, knowing that the list is sorted by category, and then alphabetically within each category. So if you have the following presets available to the File Sequence export:

- Project
- Shared
  Alias for grading
  JPEG
  Tiff
- Autodesk
  Alias (8-bit RLE)
  Cineon (10-bit)
  ...

The default preset for File Sequence is *Alias for grading*. A new preset named *DPX for grading* in the Project section will become the new default for File Sequence. The same preset but in the Shared section means that *Alias for grading* remains the default for File Sequence.

You can have one default preset per Export Type.

To set a default format preset:

1 Right-click a clip and select Export.

2 Select the Export Type for which you want to define a default preset.

3 Select the Format Preset to use as the default.

You will create a renamed copy of this preset in the step below.

4 Click Save.

5 In the Save Export Format Preset dialogue box, do the following:

- Rename the preset.
- Set the visibility to Shared or This Project Only.

This creates a renamed copy of the preset. Make sure it name and category makes it appear at the top of the Format Preset drop-down list.

6 Click Save to save the preset.

You have just renamed the original preset and copied it to the Shared or Project section.

7 Click Cancel to close the Media Export dialogue box.

H.264 Codec Export Profiles

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

NOTE The H.264 codec profiles are xml files stored in */usr/discreet/mediaconverter/2013.../profiles/Quicktime/video/H264*.

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

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

Profile	Description	Suggested Clip Resolution	Bit Rate
Baseline_CIF_600Kbits	H264_CIF at, Baseline profile	352x288 or 352x240	600 Kb/s
Baseline_RIM_12Mbits	H264_BASELINE, Baseline profile	1920x1080	12 Mb/s
Baseline_RIM_20Mbits	H264_BASELINE, Baseline profile	1920x1080	20 Mb/s
Baseline_RIM_4Mbits	H264_BASELINE, Baseline profile	1920x1080	4 Mb/s
HDTV_1080i_10Mbits	H264_HDTV_1080i, High profile, interlaced	1920x1080	10 Mb/s
HDTV_720p_8Mbits	H264_HDTV_720p, High profile	1280x720	8 Mb/s
High_1080i_6Mbits	H264_HIGH, High profile, interlaced	1920x1080	6 Mb/s
High_AVC_HD_20Mbits	H264_AVCHD AVCHD, High profile, interlaced	1920x1080	20 Mb/s
High_AVC_Intra_111Mbits	H264_INTRA_CLASS_100 AVC Intra Class 100, High 10 profile, interlaced	1920x1080	111 Mb/s
High_AVC_Intra_54Mbits	H264_INTRA_CLASS_50 AVC Intra Class 50, High 10 profile, interlaced	1440x1080	54 Mb/s
High_Blu_Ray_20Mbits	H264_BD_HDMV Blu-ray HD, High profile, interlaced	1920x1080	20 Mb/s
High_Blu_Ray_8Mbits	H264_BD Blu-ray SD, High profile, interlaced	720x576 or 720x480	8 Mb/s
High_Divx_2Mbits	H264_DIVX DivX+, High profile	1920x1080	2 Mb/s
High_DVD_3Mbits	H264_DVD, High profile, interlaced	720x576 or 720x480	3 Mb/s
High_HD_DVD_20Mbits	H264_HD_DVD, High profile, interlaced	1920x1080	20 Mb/s

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

Exporting to QuickTime ProRes Using a Remote Gateway

You use a remote Gateway running on a Mac to export media to QuickTime ProRes files from a Linux-based Flame.

1. Set **Preferences ➤ Backburner ➤ Backburner Manager box** to a Mac OS X workstation running a Gateway.

   This ensures that the exports are done using that workstation's Gateway. This has the side effect of disabling, for your exports, your access to your local volumes. Flame Premium now considers that you are exporting from the Mac OS X workstation. Switching the Backburner Manager box back to your workstation solves this.
2. Close the Preferences window.
3. Right-click the clip to export and select Export.

Or in the MediaHub, you can drag-and-drop the file from the Media panel to a location displayed in the MediaHub browser.

4 Navigate to the clip's destination, using the Media Export window.

Remember, the navigation is from the perspective of the Mac OS X workstation running the Gateway, not from your Flame Premium.

5 Select an Export type and a Format Preset.

Since you are exporting from a remote Gateway running on a Mac OS X workstation, you now have access to QuickTime ProRes file formats.

6 Click Export.

Flame Premium takes a few instants to prepare the export job. Once that preparation is done, the rest of export happens in the remote Gateway and frees up Flame Premium for your use.

NOTE The Gateway, also known as the Wiretap Gateway, is the component of the Flame Premium application responsible for all import and export operations. You are usually provided with one remote Gateway license with your Flame Premium, allowing you to install one such Gateway on a Mac OS X workstation and thus export to QuickTime ProRes files.

Supported Media File Formats

Tables in this topic:

- Image Sequence (page 64)
- QuickTime (page 65)
- MXF (page 68)
- MPEG 4 (page 69)
- Other Streaming Codecs (page 69)
- Audio File Formats (page 70)

Image Sequence

Format	Extension	Import	Export	Depth
Alias®	.als	yes (page 115)	**yes**	8 bits
ARRIRAW	.ari	yes (page 106)	—	12 bits
	NOTE Always presented as a clip, never as a sequence of RAW images. Supports material shot on camera using ARRI SUP 7 or earlier, and features from version 4.4 of the ARRIRAW SDK.			
Cineon®	.cin	yes (page 115)	**yes**	10 bits
DPX	.dpx	yes (page 112)	**yes**	8, 10, 12, 16 bits and ADX encoding.
DPX - Single channel	.dpx	yes (page 112)	—	See Note.
	NOTE Monochromatic DPX files from the following film scanners have been validated: ■ FilmLight Northlight (10 & 16-bit)			

Format	Extension	Import	Export	Depth
	■ DigitalFilmTechnology SCANITY™ (10 & 16-bit) ■ Imagica (8, 10 & 16-bit) **NOTE** Lustre supports single channel DPX files only when imported through Wiretap Gateway.			
Gateway	.clip	yes (page 114)	—	n/a
HDR	.hdr	yes (page 114)	—	32 bits
JPEG	.jpg	yes (page 115)	**yes**	8 bits
OpenEXR	.exr	yes (page 120)	**yes**	8, 10, 12u, 12, 16 fp, or 32 fp bits
	NOTE Export is 16-bit only.			
Pict (Macintosh®)	.pict	yes (page 115)	**yes**	8 bits
Pixar	.picio	yes (page 115)	**yes**	8 bits
Portable Network Graphics	.png	yes (page 125)	—	8 or 16 bits
	NOTE Supports alpha.			
Precomp	.precomp	yes (page 126)	—	8, 10, 12u, 12, 16 fp, or 32 fp bits
SGI®	.sgi	yes (page 115)	**yes**	8 or 16 bits
Softimage®	.pic	yes (page 115)	**yes**	8 bits
TARGA®	.tga	yes (page 115)	**yes**	8 bits
Tdi/Maya®	.iff	yes (page 115)	—	8 or 16 bits
Tiff	.tif	yes (page 115)	**yes**	8 or 16 bits
Wavefront®	.rla	yes (page 115)	**yes**	8 or 16 bits

QuickTime

Format	Extension	Import	Export	Depth
8-bit Packed YUV 4:2:2	.mov	yes (page 126)	**yes**	
	NOTE Lossy codec. Avoid using for intermediates.			

Format	Extension	Import	Export	Depth
10-bit Packed YUV 4:2:2	.mov	—	**yes**	
	NOTE Lossy codec. Avoid using for intermediates.			
Apple Animation	.mov	yes (page 126)	—	with alpha
Apple Graphics	.mov	yes (page 126)	—	
Apple® Video	.mov	yes (page 126)	—	
Cinepak	.mov	yes (page 126)	—	
Component Y'Cb-Cr 8-bit 4:4:4	.mov	yes (page 126)	—	8-bit planar
Component Y'Cb-CrA 8-bit 4:4:4:4	.mov	yes (page 126)	—	8-bit planar
Component Y'Cb-Cr 10-bit 4:4:4	.mov	yes (page 126)	—	10-bit packed
Component Y'Cb-Cr 10-bit 4:2:2	.mov	yes (page 126)	—	10-bit packed
Component Video	.mov	yes (page 126)	—	8-bit packed
	NOTE 4:2:2 format			
DV 25 NTSC	.mov	yes (page 126)	**yes**	
	NOTE NTSC & PAL Although the specifications allow the DV format to be field 1 or 2, the industry standard is "bottom first". Thus, before exporting to Flame Premium, ensure that the clip is Field 2. Reformat, if necessary.			
DVCPRO 50	.mov	yes (page 126)	**yes**	
	NOTE NTSC & PAL			
DVCPRO HD	.mov	yes (page 126)	**yes**	
DNxHD	.mov	yes (page 126)	**yes**	8 bits: 36, 145, 220 (and variants) 10 bits: 220x (and variants)

Format	Extension	Import	Export	Depth
	NOTE 36, 145, 220, 220x @ export			
H.264	.mov	yes (page 126)	**yes**	
IMX	.mov	yes (page 126)	**yes**	
	NOTE Includes support for IMX 30, 40, and 50.			
MJPEG	.mov	yes (page 126)	**yes**	
	NOTE JPEG compatible			
MPEG-1	.mov	yes (page 126)	—	
MPEG-4	.mov	yes (page 126)	**yes**	
MSMpeg 4v3 (Di-vX)	.mov	yes (page 126)	—	
PhotoJPEG	.mov	yes (page 126)	—	
	NOTE RT PhotoJPEG compatible			
PNG	.mov	yes (page 126)	**yes**	without alpha
PNGA	.mov	yes (page 126)	**yes**	with alpha
ProRes 4444	.mov	yes (page 126)	**yes**	12-bit
	NOTE About QuickTime ProRes Support (page 72)			
ProRes 422 (HQ)	.mov	yes (page 126)	**yes**	10-bit
	NOTE About QuickTime ProRes Support (page 72)			
ProRes 422	.mov	yes (page 126)	**yes**	10-bit
	NOTE About QuickTime ProRes Support (page 72)			
ProRes 422 (LT)	.mov	yes (page 126)	**yes**	10-bit
	NOTE About QuickTime ProRes Support (page 72)			

Format	Extension	Import	Export	Depth
ProRes 422 (Proxy)	.mov	yes (page 126)	**yes**	10-bit
	NOTE About QuickTime ProRes Support (page 72)			
Quicktime Planar RGB	.mov	yes (page 126)	—	
RGB Uncompressed	.mov	yes (page 126)	**yes**	without alpha
RGBA Uncompressed	.mov	yes (page 126)	**yes**	with alpha
TGA	.mov	yes (page 126)	—	
	NOTE TARGA			

MXF

Format	Extension	Import	Export	Depth
AVC-Intra 50	.mxf	yes (page 119)	**yes**	
	NOTE Panasonic P2			
AVC-Intra 100	.mxf	yes (page 119)	**yes**	
	NOTE Panasonic P2			
DNxHD	.mxf	yes (page 119)	**yes**	
	NOTE Includes support for formats: 36, 60, 75, 90, 90x, 110, 110x, 115, 145, 175, 175x, 185, 185x, 220, 220x DNxHD from ARRI ALEXA cameras (145 and 220x as OP1a) are supported. In MXF Op-Atom files generated by Avid Media Composer, audio tracks appear in the MediaHub as a single audio channel file (A1). But once imported, the tracks display the original channels.			
DV 25	.mxf	yes (page 119)	—	
	NOTE Panasonic P2			
DVCPRO	.mxf	yes (page 119)	**yes**	

Format	Extension	Import	Export	Depth
	NOTE Panasonic P2			
DVCPRO 50	.mxf	yes (page 119)	**yes**	
	NOTE Panasonic P2 (PAL & NTSC)			
DVCPRO HD	.mxf	yes (page 119)	**yes**	
	NOTE Panasonic P2. Available in 1080p@25/50, 1080p@24/30/60, 720p@25/50, and 720p@24/30/60.			
SonyRAW	.mxf	yes (page 132)	—	
	NOTE Includes support for Sony F65, F55, and F5 camera outputs.			
XDCAM MPEG-2 IMX	.mxf	yes (page 119)	—	
	NOTE 30, 40, and 50			
XDCAM HD MPEG-2 long-GOP	.mxf	yes (page 119)	**yes**	
	NOTE Import: 4:2:0 and 4:2:2. Export: 4:2:2.			

NOTE Flame Premium exports MXF as OP-1a files (including the timecode) encoded with audio encoded as PCM, 16-Bit or 24-Bit.

MPEG 4

Format	Extension	Import	Export	Depth
H.264	.mp4	yes (page 117)	—	
XDCAM EX	.mp4	yes (page 136)	—	
	NOTE MPEG-2 long-GOP			

Other Streaming Codecs

Format	Extension	Import	Export	Depth
AVCHD	.mts or .m2ts	yes (page 111)	—	
	NOTE Only linear PCM audio is supported. Some cameras can record AC-3 audio, but this format is not supported. Only the video portion of AVCHD media with AC-3 audio content is accessible from Flame Premium.			

Format	Extension	Import	Export	Depth
R3D	.r3d	yes (page 128)	—	RAW
	NOTE RECODE RAW 2 and 3. Supports RED SDK version 4.4, including media from the EPIC MONOCHROME camera.			
	In Flame 2013 20th Anniversay Edition Extension Extension 2 Service Pack3, changes to the RED SDK affect how the DRX setting is computed for all R3D clips, and can impact clips imported before these changes. See the DRX setting in NO LABEL (page 131).			

Audio File Formats

Format	Extension	Import	Export	Depth
AIFF	.aiff	yes	yes	16 or 24
AIFF-C	.aifc	yes	yes	16, 24, 32 (float)
Audio Visual Research	.avr	yes	yes	16
Berkeley/IRCAM/CARL Sound (BISCF)	.bsf	yes	yes	16
MP3	.mp3	yes	yes	16
Nextsnd	.au	yes	yes	16, 24, 32 (float)
WAVE	.wav	yes		16, 24, 32 (float)
WAVE - Broadcast	.wav	yes	yes	16, 24, 32 (float)
	NOTE Includes support for RF64 files (BWF-compatible format that supports files larger than 4 GB).			
WAVE - Extensible	.wav	yes	—	16, 24, 32 (float)
	NOTE Audio tracks are imported as regular audio tracks, without mapping the channels to spatial locations.			

NOTE Files of any sample rate can be imported, but they are all resampled to 48 kHz.

Supported QuickTime Audio

The table below lists the audio codecs supported in .mov files, as audio-video and audio-only files.

Audio CODEC	Import	Export
16-bit PCM	**yes**	**yes**
	NOTE Export supports both big and small endian.	
24-bit PCM	—	**yes**
	NOTE Export supports both big and small endian.	
32-bit floating point PCM	—	**yes**
	NOTE Export supports both big and small endian.	
32-bit float	—	**yes**
A-law 2:1	**yes**	**yes**
ADPCM ima WAV	**yes**	—
Advanced Audio Codec (AAC)	**yes**	—
	NOTE Supported in mp4 and m4v files.	
Apple lossless	**yes**	—
IMA 4:1	**yes**	**yes**
Linear PCM (QT 7)	**yes**	**yes**
MPEG-2 Layer 2 Audio	**yes**	**yes**
MS ADPCM	**yes**	—
Ogg Vorbis (qt4l compatible)	**yes**	—
Ogg Vorbis (qtcomponents compatible)	**yes**	—
QDM2 Audio	**yes**	—
Raw 8-bit audio	**yes**	—

Audio CODEC	Import	Export
Sowt	**yes**	**yes**
	NOTE 16-bit PCM (Little Endian)	
Twos	**yes**	**yes**
	NOTE 16-bit PCM (Big Endian)	
Ulaw	**yes**	**yes**

About QuickTime ProRes Support

On Smoke, which runs on Mac OS X, Apple QuickTime ProRes imports and exports are supported natively.

For all Linux-based application, exports to the Quicktime ProRes format requires connecting to a Wiretap Gateway running on a Mac OS X workstation.

Native ProRes import is supported in the following Linux applications:

- Flame
- Flame Premium
- Flame Premium - Grading
- Flare (when working with a Flame or Flame Premium project)

The following services can also decode clips using ProRes media imported by one of the above products:

- Burn
- Burn Lustre and Lustre Slave Renderer
- Wiretap Central

Other Linux-based products can import ProRes by connecting to a Wiretap Gateway running on a Mac OS X workstation.

Importing a Final Cut Pro XML Sequence

Flame Premium supports both FCP 7 and FCP X file formats.

NOTE To simplify the conform process, create the FCP XML project on the workstation running Flame Premium. And when exporting the FCP XML, save the FCP XML to the root of the media used in that timeline: the media should either be with the FCP XML, or within a folder alongside the FCP XML.

To import a sequence using the MediaHub:

1. Click the MediaHub tab.
2. Review the Sequence Import Options. Pay attention to the following options:
   - Preferred Media: If offline intermediates were used during the offline editing, decide now whether you wish to relink to the original media or to those offline intermediates.

- Search and Import Files: Enable if you want to import the linked media.
- Use Filename: For the best results, enable only this option. Disable Use Timecode, Use Tape, Use UMID, Use Resolution, and Use Framerate.

3 Using the file browser, navigate to the FCP XML sequence to import.

4 Drag the file from the browser to the Media Library.

Flame Premium converts the FCP XML to its timeline format, and imports the linked media using the Search Options as match criteria. The media itself is imported using the option file format options defined in the Format Specific Options tab.

FCP 7 Sequence Import: Supported Transitions and Effects

Sections in this topic:

- Supported Data (page 73)
- Animation Interpolation (page 74)
- Motion (page 75)
- Video Transitions (page 75)
- Video Filters (page 82)
- Video Generators (page 86)
- Audio (page 88)
- Composite Modes (page 90)

Supported Data

Flame Premium allows you to import multi-track compositions from Apple Final Cut Pro (FCP). Flame Premium reads XML exported from FCP (up to version 7.x of FCP, exported as XML version 2.0) and recreates a timeline accordingly.

The following tables describe the data that is output from FCP and input into Flame Premium.

General Data

FCP composition(s) data maps to Flame Premium timeline data.

Final Cut Pro	Flame Premium
Name	Name
Framerate	Framerate
Duration	Duration

Editorial Data

FCP Source media data and Record side data maps to Autodesk clip data.

Final Cut Pro	Flame Premium
Source media data: ■ Tape name ■ Source TC in/out ■ Edge code ■ Log notes ■ Aspect ratio ■ Comments	Source clips: ■ Tape name ■ Source TC in/out ■ Keycode ■ Elements comments ■ Aspect ratio ■ Elements Comments
Record side data: ■ In/Out ■ Transition type ■ Number of video tracks ■ Number of audio tracks ■ Marker ■ In/Out marker	Record clip: ■ Segment ■ Cut/Dissolve/Wipe/Axis ■ Video layers ■ Audio tracks ■ Track marks ■ In/Out marks

Effect Data

FCP transitions map to Flame Premium transitions, while FCP Filter effects and FX Script data map to Flame Premium Timeline effects.

Final Cut Pro	Flame Premium
Filter effects	Timeline effects
Transitions	Transitions

Animation Interpolation

FCP animation interpolation maps to Autodesk interpolation.

Final Cut Pro	Flame Premium
Corner	Linear
Smooth	Hermite

Motion

The following table describes how motion from FCP is mapped to Flame Premium Axis Timeline effect parameters.

Final Cut Pro	Flame Premium
Basic Motion	Axis Axis
Crop	Axis Edges
Distort	Axis Surface
Opacity	Axis Transparency (partially supported)
Drop Shadow	Axis Shadow
Motion Blur	Not supported
Time Remap	Timewarp (see below)

About Time Remap to Timewarp

To make sure the conform is accurate with the creative editorial decisions from Final Cut Pro 7, the conformed timewarp speed value seen in Flame Premium timewarp editor can be slightly different from the one seen in FCP7. But the actual Timewarp effect will be visually similar to the expected result, and be frame and keyframe accurate. There are exceptions:

- Frame blending used with this effect is not translated.
- Negative constant timewarp speeds are not applied to the audio tracks of the segment being timewarped.
- Variable timewarps are not applied to the audio tracks of the segment being timewarped.

Video Transitions

The names of FCP transitions are preserved in Flame Premium and are visible in the timeline.

3D Simulation

Final Cut Pro	Flame Premium
Cross Zoom	Dissolve (partially supported)
Cube Spin	Dissolve (partially supported)
Spin 3D	Axis transition (partially supported)
Spinback 3D	Not supported; replaced by Dissolve
Swing	Axis transition (partially supported)

Final Cut Pro	Flame Premium
Zoom	Axis transition (partially supported)

Dissolve

Final Cut Pro	Flame Premium
Additive Dissolve	Not supported; replaced by Dissolve
Cross Dissolve	Dissolve Additive
Dip to colour Dissolve	Dissolve To/From colour (partially supported)
Dither Dissolve	Not supported; replaced by Dissolve
Fade in/fade out Dissolve	Dissolve To/From Black (partially supported)
Non-Additive Dissolve	Dissolve Non-Additive
Ripple Dissolve	Not supported; replaced by Dissolve

Iris

Final Cut Pro	Flame Premium
Cross Iris	SMPTE 007 (partially supported)
Diamond Iris	SMPTE 102 (partially supported)
Oval Iris	SMPTE 119, 120, or 121 (partially supported)
Point Iris	SMPTE 047 (partially supported)
Rectangle Iris	SMPTE 101 (partially supported)
Star Iris	SMPTE 127, 128, or 129 (partially supported)

Map

Final Cut Pro	Flame Premium
Channel Map	Not supported; replaced by Dissolve
Luminance Map	Not supported; replaced by Dissolve

Page Peel

Final Cut Pro	Flame Premium
Page Peel	Not supported; replaced by Dissolve

QuickTime

Final Cut Pro	Flame Premium
Channel Compositor	Not supported; replaced by Dissolve
Chroma Key	Not supported; replaced by Dissolve
Explode	Not supported; replaced by Dissolve
Gradient Wipe	Not supported; replaced by SMPTE 002 without Softness
Implode	Not supported; replaced by Axis transition
Iris	SMPTE Wipes (partially supported): ■ Rectangle = SMPTE 101 ■ Diamond = SMPTE 102 ■ Triangle = SMPTE 103 ■ Triangle Right = SMPTE 104 ■ Triangle Upside Down = SMPTE 105 ■ Triangle Left = SMPTE 106 ■ Arrowhead = SMPTE 107 ■ Arrowhead Right = SMPTE 108 ■ Arrowhead Upside Down = SMPTE 109 ■ Arrowhead Left = SMPTE 110 ■ Pentagon = SMPTE 111 ■ Pentagon Upside Down = SMPTE 112 ■ Hexagon = SMPTE 113 ■ Hexagon Side= SMPTE 114 ■ Circle = SMPTE 119 ■ Oval = SMPTE 120 ■ Oval Side = SMPTE 121 ■ Cat Eye = SMPTE 122 ■ Cat Eye Side = SMPTE 123 ■ Round Rect = SMPTE 124 ■ Round Rect Side = SMPTE 125 ■ 4 Point Star = SMPTE 127 ■ 5 Point Star = SMPTE 128 ■ 6 Point Star = SMPTE 129

Final Cut Pro	Flame Premium
	■ Heart = SMPTE 130 ■ Keyhole = SMPTE 131
Matrix Wipe	Not supported; replaced by SPMTE 001 with a comment indicating the type of FCP Matrix Wipe that had been at this mark
Push	Not supported; replaced by Dissolve
Radial	SMPTE Wipes (partially supported): ■ Rotating Top = SMPTE 201 ■ Rotating Right = SMPTE 202 ■ Rotating Bottom = SMPTE 203 ■ Rotating Left = SMPTE 204 ■ Rotating Left Bottom = SMPTE 205 ■ Rotating Left Right = SMPTE206 ■ Rotating Quadrant = SMPTE 207 ■ Top to Bottom 180 degree = SMPTE 211 ■ Right to Left 180 degree= SMPTE 212 ■ Top to Bottom 90 degree= SMPTE 213 ■ Right to Left 90 degree = SMPTE 214 ■ Top 180 Degree = SMPTE 221 ■ Right 180 Degree = SMPTE 222 ■ Bottom 180 Degree = SMPTE 223 ■ Left 180 Degree = SMPTE 224 ■ Counter Rotating Top Bottom = SMPTE 225 ■ Counter Rotating Left Right = SMPTE 226 ■ Double Rotating Top Bottom = SMPTE 227 ■ Double Rotating Left Right = SMPTE 228 ■ V Open Top = SMPTE 231 ■ V Open Right = SMPTE 232 ■ V Open Bottom = SMPTE 233 ■ V Open Left = SMPTE 234 ■ V Open Top Bottom = not supported; replaced by SMPTE 001 ■ V Open Left Right = not supported; replaced by SMPTE 001 ■ Rotating Top Left = SMPTE 241 ■ Rotating Bottom Left = SMPTE 242 ■ Rotating Bottom Right = SMPTE 243 ■ Rotating Top Right = SMPTE 244 ■ Rotating Top Left Bottom Right = SMPTE 245 ■ Rotating Bottom Left to Top Right = SMPTE 246 ■ Rotating Top Left Right = SMPTE 251

Final Cut Pro	Flame Premium
	■ Rotating Left Top Bottom = SMPTE 252 ■ Rotating Bottom Left Right = SMPTE 253 ■ Rotating Right Top Bottom = SMPTE 254 ■ Rotating Double Center Right = not supported; replaced by SMPTE 001 ■ Rotating Double Center Top = not supported; replaced by SMPTE 001 ■ Rotating Double Center Top Bottom = not supported; replaced by SMPTE 001) ■ Rotating Double Center Left Right = not supported; replaced by SMPTE 001
Slide	Axis transition
Wipe	SMPTE Wipes (various partially supported): ■ Slide Horizontal = SMPTE 001 ■ Slide Vertical = SMPTE 002 ■ Top Left = SMPTE 003 ■ Top Right = SMPTE 004 ■ Bottom Right = SMPTE 005 ■ Bottom Left = SMPTE 006 ■ Four Corner = SMPTE 007 ■ Four Box = SMPTE 008 ■ Barn Vertical = SMPTE 021 ■ Barn Horizontal = SMPTE 022 ■ Top Center = SMPTE 023 ■ Right Center = SMPTE 024 ■ Bottom Center = SMPTE 025 ■ Left Center = SMPTE 026 ■ Diagonal Left Down = SMPTE 041 ■ Diagonal Right Down = SMPTE 042 ■ Vertical Bow Tie = SMPTE 043 ■ Horizontal Bow Tie = SMPTE 044 ■ Diagonal Left Out = SMPTE 045 ■ Diagonal Right Out = SMPTE 046 ■ Diagonal Cross = SMPTE 047 ■ Diagonal Box = SMPTE 048 ■ Filled V = SMPTE 061 ■ Filled V Right = SMPTE 062 ■ Filled V Bottom = SMPTE 063 ■ Filled V Left = SMPTE =064 ■ Hollow V = SMPTE 065 ■ Hollow V Right = SMPTE 066

Final Cut Pro	Flame Premium
	■ Hollow V Bottom = SMPTE 067 ■ Hollow V Left = SMPTE 068 ■ Vertical Zig Zag = SMPTE 071 ■ Horizontal Zig Zag = SMPTE 072 ■ Vertical Barn Zig Zag = SMPTE 073 ■ Horizontal Barn Zig Zag = SMPTE 074
Zoom	Axis transition (partially supported)

Slide

Final Cut Pro	Flame Premium
Band Slide	Not supported; replaced by Dissolve
Box Slide	Not supported; replaced by Dissolve
Center Split Slide	Not supported; replaced by Dissolve
Multi Spin Slide	Not supported; replaced by Dissolve
Push Slide	Not supported; replaced by Dissolve
Spin Slide	Not supported; replaced by Dissolve
Split Slide	Not supported; replaced by Dissolve
Swap Slide	Not supported; replaced by Dissolve

Stretch

Final Cut Pro	Flame Premium
Cross Stretch	Not supported; replaced by Dissolve
Squeeze	Axis transition (partially supported)
Squeeze and Stretch	Not supported; replaced by Axis Transition
Stretch	Axis transition (partially supported)

Wipe

Final Cut Pro	Flame Premium
Band	Not supported
Center Wipe	SMPTE 021 (partially supported)
Checker Wipe	Not supported
Checkerboard Wipe	Not supported
Clock Wipe	SMPTE 201 (partially supported)
Edge Wipe	SMPTE 001 (partially supported)
Gradient Wipe	Not supported
Inset Wipe	■ Upper Left = SMPTE 003 (partially supported) ■ Top = SMPTE 023 ■ Upper Right = SMPTE 004 ■ Right = SMPTE 024 ■ Lower Right = SMPTE 005 (reverse) ■ Bottom = SMPTE 025 ■ Lower Left = SMPTE 006 (reverse) ■ Left = SMPTE 026
Jaws Wipe	SMPTE 073 (partially supported, FCP's is smaller than Flame Premium's)
Random Edge Wipe	Not supported; replaced by Wipe 001 transition
V Wipe	■ Right = SMPTE 064 (partially supported) ■ Down = SMPTE 61 ■ Left = SMPTE 62 ■ Up = SMPTE 63
Venetian Blind Wipe	Not supported; replaced by SMPTE 001
Wrap Wipe	Not supported; replaced by SMPTE 001
Zig-Zag Wipe	Not supported; replaced by SMPTE 001

Video Filters

Blur

Final Cut Pro	Flame Premium
Gaussian Blur	Axis Surface + Axis (partially supported)
Radial Blur	Not supported
Wind Blur	Not supported
Zoom Blur	Not supported

Border

Final Cut Pro	Flame Premium
Basic Border	Timeline Resize
Bevel	Not supported

Channel

Final Cut Pro	Flame Premium
Arithmetic	Not supported
Channel Blur	Not supported
Channel Offset	Not supported
Colour Offset	Timeline CC (partially supported)
Compound Arithmetic	Not supported
Invert	Timeline CC Basics (partially supported)

Colour Correction

Final Cut Pro	Flame Premium
Broadcast Safe	Not supported
Colour Correction	Not supported; replaced by Timeline CC
Colour Correction 3-way	Not supported; replaced by Timeline CC

Final Cut Pro	Flame Premium
Desaturate Highlights	Not supported
Desaturate Lows	Not supported
RGB Balance	Not supported; replaced by Timeline CC

Distort

Final Cut Pro	Flame Premium
Bumpmap	Not supported
Cylinder	Not supported
Displace	Not supported
Fisheye	Not supported
Pond Ripple	Not supported
Ripple	Not supported
Wave	Not supported
Whirlpool	Not supported

Image Control

Final Cut Pro	Flame Premium
Brightness & Contrast (Bezier)	Not supported; replaced by Timeline CW
Colour Balance	Not supported; replaced by Timeline CC
Desaturate	Timeline CC
Gamma Correction	Not supported; replaced by Timeline CW
Levels	Not supported
Proc Amp	Not supported; replaced by Timeline CC
Sepia	Not supported; replaced by Timeline CC
Tint	Not supported

Key

Final Cut Pro	Flame Premium
Blue and Green screen	Not supported
Chroma Keyer	Not supported; replaced by Timeline Axis
Colour Smoothing 4:1:1	Not supported
Colour Smoothing 4:2:2	Not supported
Colour Key	Not supported; replaced by Timeline Axis
Difference Matte	Not supported
Luma Key	Not supported
Spill Suppressor - Blue	Not supported; replaced by Timeline Axis
Spill Suppressor - Green	Not supported; replaced by Timeline Axis

Matte

Final Cut Pro	Flame Premium
8-Point Garbage Mask	Not supported; replaced by Timeline Axis
Extract	Not supported
4-Point Garbage Mask	Not supported; replaced by Timeline Axis
Image Mask	Not supported; replaced by Timeline Axis
Mask Feather	Not supported
Mask Shape	Timeline Axis (partially supported)
Matte Choker	Not supported; replaced by Timeline Axis
Soft Edges	Not supported; replaced by Timeline Axis
Widescreen	Timeline Resize (partially supported)

Perspective

Final Cut Pro	Flame Premium
Basic 3D	Timeline Axis Axis (partially supported)
Curl	Not supported
Flop	Timeline Axis Axis
Mirror	Not supported
Rotate	Timeline Axis Axis

QuickTime

Final Cut Pro	Flame Premium
Gaussian Blur	Timeline Axis Axis
Brightness/Contrast	Not supported; replaced by Timeline CC
Colour Style	Not supported
Colour Tint	Timeline CC (partially supported)
Colour Sync	Not supported
Edge Detection	Not supported
Emboss	Not supported
General Convolution	Not supported
HSL Balance	Not supported; replaced by Timeline CC
Lens Flare	Not supported
RGB Balance	Not supported; replaced by Timeline CC
Sharpen	Not supported

Sharpen

Final Cut Pro	Flame Premium
Sharpen	Not supported

Final Cut Pro	Flame Premium
Unsharp Mask	Not supported

Stylize

Final Cut Pro	Flame Premium
Anti-Alias	Not supported
Diffuse	Not supported
Emboss	Not supported
Find Edges	Not supported
Posterize	Not supported
Replicate	Not supported
Solarize	Not supported

Video

Final Cut Pro	Flame Premium
Blink	Timeline Axis
De-Interlace	Timeline Resize (partially supported)
Flicker Filter	Not supported
Image Stabilizer	Not supported; replaced by Axis Stabilizer
Stop Motion Blur	Not supported
Strobe	Timewarp Strobe
Timecode Generator	Not supported
Timecode Reader	Not supported
Viewfinder	Not supported

Video Generators

The following tables describe generated effects that are translated into equivalent effects in Flame Premium.

Once imported into Flame Premium, effects created with the FCP video generator use the project's default resolution, regardless of their original resolution in FCP. This matches the behaviour of FCP: XML files with these effects that are reimported into FCP projects with different resolutions similarly inherit the project's resolution.

Video Generator

Final Cut Pro	Flame Premium
Bars and Tone HD108060i	Colour Source SMPTE Bars (partially supported)
Bars and Tone HD720p60	Colour Source SMPTE Bars (partially supported)
Bars and Tone (NTSC)	Colour Source SMPTE Bars (partially supported)
Bars and Tone (PAL)	Colour Source SMPTE Bars (partially supported)
Slug	Black colour Source SMPTE Bars (partially supported)

Matte

Final Cut Pro	Flame Premium
Colour	Colour Source (Fill Colour): ■ R = 0-255 ■ G = 0-255 ■ B = 0-255

Others

Final Cut Pro	Flame Premium
More Bars and Signals	Not supported

Render

Final Cut Pro	Flame Premium
Custom Gradient	Not supported
Gradient	Not supported
Highlight	Not supported
Noise	Colour Source Noise
Particle Noise	Not supported

Shapes

Final Cut Pro	Flame Premium
Circle	Not supported
Oval	Not supported
Rectangle	Not supported
Square	Not supported

Text

Final Cut Pro	Flame Premium
Crawl	Timeline Text (partially supported)
Lower 3rd	Timeline Text (partially supported)
Outline Text	Timeline Text (partially supported)
Scrolling Text	Timeline Text (partially supported)
Text	Timeline Text (partially supported)
Typewriter	Not supported; replaced by Colour Source

Audio

Some FCP audio elements are mapped to Flame Premium equivalents. Audio keyframes are not supported.

Audio Transitions

Final Cut Pro	Flame Premium
Cross Fade (0dB)	Audio fade (partially supported)
Cross Fade (+3dB)	Not supported; replaced by Audio Fade

Audio Controls

Final Cut Pro	Flame Premium
Stereo	Audio Gain (partially supported)

Apple

Final Cut Pro	Flame Premium
AuBandPass	Not supported
AuDelay	Not supported
AuDynamicProcessor	Not supported
AuGraphicsEQ	Not supported
AuHighShelfFilter	Not supported
AuHighPass	Not supported
AuLowPass	Not supported
AuLowShelfFilter	Not supported
AuMultibandCompress	Not supported
AuParametricEQ	Not supported
AuPeakLimiter	Not supported

Final Cut Pro HD

Final Cut Pro	Flame Premium
3 Band Equalizer	Not supported
Band Pass Filter	Not supported
Compressor/Limiter	Not supported
DC Notch	Not supported
Echo	Not supported
Expander/Noise Gate	Not supported
High Pass Filter	Not supported
High Shelf Filter	Not supported
Hum Remover	Not supported

Final Cut Pro	Flame Premium
Low Pass Filter	Not supported
Low Shelf Filter	Not supported
Notch Filter	Not supported
Parametric Equalizer	Not supported
Reverberation	Not supported
Vocal DeEsser	Not supported
Vocal DePopper	Not supported

Composite Modes

Some FCP composite modes are mapped to Flame Premium Axis surface blend modes.

Final Cut Pro	Flame Premium
Modify	■ Add = Add (partially supported) ■ Subtract = Subtract ■ Difference = Negate ■ Multiply = Multiply ■ Screen = Screen ■ Overlay = not supported ■ Hard Light = not supported ■ Soft Light = not supported ■ Darken = Min ■ Lighten = Max ■ Travel Matte - Alpha = not supported ■ Travel Matte - Luma = not supported ■ Normal = not supported

FCP X Sequence Import: Supported Transitions and Effects

Sections in this topic:

- Supported Data (page 91)
- Transitions (page 91)
- Retime Effects (page 92)
- Title Effects (page 93)
- Audio (page 93)

Supported Data

Flame Premium supports Final Cut Pro X (10.0.8) XML export, including ones referencing directly R3D, MXF, and ARRI Alexa ProRes files.

NOTE XML sequences from FCP 10.0.4 and 10.0.5 no longer import correctly in Flame Premium the segments that were Multicam clips in FCP. Starting with FCP 10.0.6, the way MultiCam clips are represented in the XML changed, and it is not possible to support legacy MultiCam content. XML files from 10.0.6 and later import correctly: Flame Premium displays MultiCam clips as regular segments just like before.

Due to the limited information written to the XML file by Final Cut Pro X, the conform in Flame Premium can look different from the sequence created in Final Cut Pro X: Flame Premium cannot set parameters or recreate effects for which there is no information.

To see how the timeline will look in Flame Premium, export in Final Cut Pro X the timeline to XML, and then import it back in Final Cut Pro X.

The following general limitations apply to Final Cut Pro X XML imports in Flame Premium:

- Compounded (nested) clips are not supported. As a workaround, remove any compound clip before exporting the sequence from FCP.
- All transitions and supported effects are reset to default values.
- Only transitions, retime and title effects, and a subset of audio effects are supported. Other clip effects such as Color are not supported, due either to lack of information within the XML, or to the complexity of the effect.

  NOTE When working with .R3D files, Color settings are supported as RMD settings in Flame Premium. See Regarding Support for RED in FCP (page 93).

Transitions

In cases where the video and audio are imported and treated as a single entity in the Final Cut Pro X sequence, applying a video transition affects both video and audio and is translated in Flame Premium as follows:

- Dissolve transition: Flame Premium applies a dissolve to video and audio.
- Wipe transition: Flame Premium applies a matching wipe to video, and a dissolve transition to audio.
- DVE transition: Flame Premium applies a dissolve to video and audio.

Apple Final Cut Pro X Transition	Translation in Flame Premium
Blur (any)	Dissolve
Dissolve	Dissolve except: ■ Cross Dissolve changes to Dissolve (no keyframes) ■ Fade to Colour changes to Fade To/From Black (no keyframes)
Lights	Dissolve
Movements	Dissolve
Objects	Dissolve

Apple Final Cut Pro X Transition	Translation in Flame Premium
Replicator/Clones	Dissolve
Stylized	Dissolve

Apple Final Cut Pro X Wipes	Translation in Flame Premium
Bands	SMPTE Wipe 001 + cue mark
Center	Supported
Checker	SMPTE Wipe 001 + cue mark
Circle	Supported
Clock	Supported
Gradient Image	Dissolve + cue mark
Inset Wipe	Supported
Letter X	Supported
Wipe	Supported

Retime Effects

A clip retime is translated to a Soft Timewarp with matching speed up or down.

NOTE To make sure the conform is accurate with the creative editorial decisions from Final Cut Pro X, the conformed timewarp speed value seen in Flame Premium timewarp editor can be slightly different from the one seen in FCPX. But the actual Timewarp effect will be visually similar to the expected result, and be frame and keyframe accurate.

Final Cut Pro X Retime	Translation in Flame Premium
Slow	Slower than real-time linear timewarp, at same speed as in Final Cut Pro X
Fast	Faster than real-time linear timewarp, at same speed as in Final Cut Pro X
Normal 100%	Not Timewarp
Hold	0% Soft Timewarp, using same frame as the one defined in Final Cut Pro X

Final Cut Pro X Retime	Translation in Flame Premium
Reverse	Reverse speed linear timewarp, at same speed as in Final Cut Pro X
Speed Ramp (to 0% / from 0%)	Animated speed ramp timewarp
Rewind (1x / 2x / 4x)	Animated speed ramp timewarp

Title Effects

Text effects parameters are not saved to the Final Cut Pro X XML, only the text strings are imported.

All text effects are set to:

- Colour: White
- Font: Discreet
- Size: 50
- Alignment: Centered

Audio

Only audio fades are supported. Audio keyframes are not supported.

Flame Premium correctly imports FCP X XML using multi-track audio.

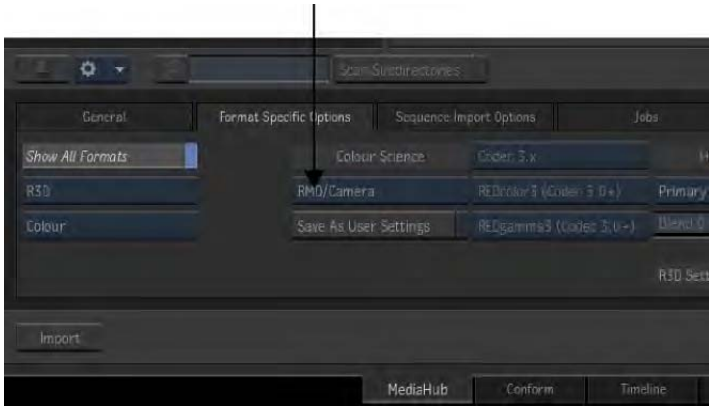
Regarding Support for RED in FCP

NOTE The following only applies when using Final Cut Pro X 10.0.8.

With Final Cut Pro, you can work directly with R3D files, created by RED cameras, instead of using the QuickTime files. This means that when you import a FCP X XML sequence in Flame Premium, you can relink the sequence to the R3D files, providing you with the best image quality.

Another benefit of using the R3D files is the RAW parameters set in Final Cut Pro X are saved by FCP as an RMD file. This file can be read by Flame Premium on import.

To read FCP X Color settings for R3D files, you have to select an RMD option from **MediaHub ➤ Browse for Files ➤ Format Specific Options ➤ Colour Settings box** for R3D files.



Importing an AAF Sequence

NOTE To simplify the conform process, save the AAF to the root of the media used in that timeline: the media should either be with the AAF, or within a folder alongside the AAF.

To import a sequence using the MediaHub:

1. Click the MediaHub tab.
2. Review the Sequence Import Options. Pay attention to the following options:
   - Preferred Media: If offline intermediates were used during the offline editing, decide now whether you wish to relink to the original media or to those offline intermediates.
   - Search and Import Files: Enable if you want to import the linked media.
   - Use Filename: For the best results, enable only this option. Disable Use Timecode, Use Tape, Use UMID, Use Resolution, and Use Framerate.
3. Using the file browser, navigate to the AAF sequence to import.
4. Drag the file from the browser to the Media Library.

   Flame Premium converts the AAF to its timeline format, and imports the linked media using the Search Options as match criteria. The media itself is imported using the option file format options defined in the Format Specific Options tab.

Importing an AAF Sequence: Supported Transitions and Effects

Flame Premium supports intermediates encoded with XDCam HD and XDCam EX codecs.

Supported transcoded intermediates generated by Avid Media Composer:

- AVC-Intra 50
- AVC-Intra 100
- DNxHD
- XDCam EX
- XDCam HD

Unsupported transcoded intermediates generated by Avid Media Composer:

- -J2K MXF
- 1:1 MXF
- 1:1p 10b MXF
- Apple ProRes Proxy MXF
- Apple ProRes LT MXF
- Apple ProRes MXF
- Apple ProRes HQ MXF

NOTE In MXF Op-Atom files generated by Avid Media Composer, audio tracks appear in the MediaHub as a single audio channel file (A1). But once imported, the tracks display the original channels.

Sections in this topic:

- Video and Audio Effects (page 95)

■ Video and Audio Transitions (page 98)

Video and Audio Effects

The following tables describe how effects are supported in Flame Premium.

Blend

Avid	Flame Premium
Picture-in-picture	Supported
Superimpose	Translated to Soft Axis (transparency value is translated)

Film

Avid	Flame Premium
1.66 mask	Supported; bkg is black, horizontal position ignored
1.85 mask	Supported; bkg is black, horizontal position ignored
16:9 mask	Supported; bkg is black, horizontal position ignored
Anamorphic mask	Supported; bkg is black, horizontal position ignored
Mask	Supported; bkg is black, horizontal position ignored

AVX Plugin

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

Image

Avid	Flame Premium
Avid Pan and Zoom	Not supported; replaced by Cue mark
Blur effect	Not supported; replaced by Cue mark
colour Correction	Converted to Soft CC (empty) + cue mark
colour Effect	Converted to Soft CC (empty) + cue mark

Avid	Flame Premium
Flip	Converted to Soft Axis (Flip effect)
Flip-flop	Converted to Soft Axis (Flip-flop effect)
Flop	Converted to Soft Axis (Flop effect)
Mask	Supported; bkg is black, no mask, horizontal position off
Resize	Supported; background is black, no left and right cropping
Scratch removal	Not supported; replaced by Cue mark
Submaster	Converted to Container

Reformat

Avid	Flame Premium
14:9 Letterbox	Not supported; replaced by Soft Axis + Cue mark
16:9 Letterbox	Not supported; replaced by Soft Axis + Cue mark
4:3 Sidebar	Not supported; replaced by Soft Axis + Cue mark
Pan and Scan	Not supported; replaced by Soft Axis + Cue mark

Titles

Avid	Flame Premium
Classic Title	Converted to Soft Text; only text string is available (white Discreet font, size 50)
Marquee Text	Converted to Soft Text; only text string is available (white Discreet font, size 50)

Key

Avid	Flame Premium
Animatte	Not supported; replaced by Cue mark
Luma key	Converted to Soft Axis (empty) + cue mark
Matte key	Converted to Soft Axis (empty) + cue mark

Avid	Flame Premium
RGB keyer	Converted to Soft Axis (empty) + cue mark
Spectra Matte	Converted to Soft Axis (empty) + cue mark

Miscellaneous

NOTE To make sure the conform is accurate with the creative editorial decisions from the Avid editor, the conformed timewarp speed value seen in Flame Premium timewarp editor can be slightly different from the one seen in the Avid application. But the actual Timewarp effect will be visually similar to the expected result, and be frame and keyframe accurate.

Avid	Flame Premium
Timewarp	Converted to Soft TW; recreate the curve type
Motion Effect	Translated to Soft TW (Constant speed, no strobe effect)
3D PIP	Translated to Soft Axis (Position / Scaling, ISO, Softness / Crop) + cue mark
Peel	Not supported; cue mark
Push	Not supported; cue mark
Spin	Not supported; cue mark
Squeeze	Not supported; cue mark
Video gap	Video gap
Video filler	Video gap
Video match frame edit	Match frame

General Audio

Audio keyframes are not supported.

Avid	Flame Premium
Audio level	Audio gain
Audio dissolve	Audio dissolve
Audio fade in	Audio dissolve
Audio fade out	Audio dissolve

Avid	Flame Premium
Audio gap	Audio gap
Audio filler	Audio gap
Audio match frame edit	Match frame splice

Audio Effects

Flame Premium does not support any of the RATS Audio Effects. Every RATS Audio Effect is replaced with a cue mark.

Video and Audio Transitions

The following tables describe how AAF transitions are supported in Flame Premium.

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

Blend

Avid	Flame Premium
Dip to colour	Converted to Dissolve (linear animation)
Dissolve	Converted to Dissolve (linear animation)
Fade from colour	Supported; background is black only, reset manually
Fade to colour	Supported; background is black only, reset manually
Picture-in-picture	Converted to Dissolve + cue mark

Film

Avid	Flame Premium
Film dissolve	Converted to Dissolve (bezier animation)
Film fade	Converted to Dissolve (linear animation)

Box wipe

Avid	Flame Premium
Bottom box*	Converted to SMPTE 025; softness not supported
Bottom left to top right*	Converted to SMPTE 006; softness not supported

Avid	Flame Premium
Bottom right to top left*	Converted to SMPTE 005; softness not supported
Left box*	Converted to SMPTE 026; softness not supported
Right box*	Converted to SMPTE 024; softness not supported
Top box*	Converted to SMPTE 023; softness not supported
Top left to bottom right*	Converted to SMPTE 003; softness not supported
Top right to bottom left*	Converted to SMPTE 004; softness not supported

Edge Wipe

Avid	Flame Premium
Horizontal*	Converted to SMPTE 001; softness not supported
Horz open*	Converted to SMPTE 021; softness not supported
Bottom left diagonal*	Converted to SMPTE 042; animation is inverted, softness not supported
Bottom right diagonal*	Converted to SMPTE 041; animation is inverted, softness not supported
Upper left diagonal*	Converted to SMPTE 041; softness not supported
Upper right diagonal*	Converted to SMPTE 042; softness not supported
Vert open*	Converted to SMPTE 022; softness not supported
Vertical*	Converted to SMPTE 002; softness not supported

Shape Wipe

Avid	Flame Premium
4 corners*	Converted to SMPTE 007; softness not supported
Horizontal bands	Converted to Dissolve + cue mark
Horizontal blinds	Converted to Dissolve + cue mark
Vertical blinds	Converted to Dissolve + cue mark
Center box*	Converted to SMPTE 101; softness not supported

Avid	Flame Premium
Circle*	Converted to SMPTE 119; softness not supported
Ellipse*	Converted to SMPTE 120; softness not supported
Clock*	Converted to SMPTE 201; softness not supported
Diamond*	Converted to SMPTE 102; softness not supported

Sawtooth Wipe

Avid	Flame Premium
Horizontal sawtooth*	Converted to SMPTE 071; softness not supported
Horz open sawtooth*	Converted to SMPTE 073; softness not supported
Vert open sawtooth*	Converted to SMPTE 074; softness not supported
Vertical sawtooth*	Converted to SMPTE 072; softness not supported

Matrix Wipe

Avid	Flame Premium
Grid*	Not supported; replaced by SMPTE 008
One-way row	Not supported; replaced by SMPTE 001
Speckle	Not supported; replaced by SMPTE 001
Spiral	Not supported; replaced by SMPTE 001
Zig-zag	Not supported; replaced by SMPTE 001

Xpress 3D Effect

Avid	Flame Premium
3D ball	Converted to Dissolve (linear) + cue mark
3D page fold	Converted to Dissolve (linear) + cue mark
3D slats	Converted to Dissolve (linear) + cue mark
3D PIP	Translated to Soft Axis (Position / Scaling, ISO, Softness / Crop) + cue mark

Miscellaneous

Avid	Flame Premium
Conceal	Converted to Soft Axis (bottom left to top right); softness not supported
L-Conceal	Converted to Soft Axis (bottom left to top right); softness not supported
Squeeze	Converted to Soft Axis (bottom centered); softness not supported
Peel	Converted to Dissolve + cue mark
Push	Converted to Dissolve + cue mark
Spin	Converted to Dissolve + cue mark
Video gap	Video gap
Video filler	Video gap

About Avid Stereoscopic Sequences

Flame Premium can import Media Composer 6 stereoscopic timelines; an Avid Stereo3D timeline is imported as a stereoscopic timeline with Left and Right tracks, with the following limits.

Avid Stereoscopic Sequence Limitations:

- Segments with multiple nested effects may not be translated as expected.
- The Avid stereoscopic sequence cannot contain S3D master clips with frame compatible contributors (clips with side-by-side or top-bottom stereoscopic media), it can only contain full resolution contributors.

Supported Avid S3D Effects and Transitions:

- S3D Vergence Effect: Translated as a Soft Axis with an equivalent Convergence value.
- S3D Floating Window Effect: Translated as a Soft Axis with a Crop.
- S3D Spatial Alignment Effect: Translated as a Soft Axis with a subset of the original parameters.
- S3D Depth Transition: Not supported.
- Clips with non-stereo effects on top of stereo effects are seen as mono.

MediaHub Reference: Browsing for Files

The MediaHub consists of two panels: the browser and the MediaHub tabs. Use the browser to locate the files to import, or the location where to export your clips. Use the MediaHub tabs to set you media and sequence import options.

- MediaHub Browser (page 102)
- General Tab (page 102)
- Format Specific Options (page 104)

- Sequence Import Options (page 104)
- Jobs (page 105)

MediaHub Browser

The MediaHub file browser displays two sections:

- The Autodesk Network lists other Smoke, Flame, or Lustre workstations on the network, and can be used to import files from those workstations.
- The Local Devices displays the hard disks, external disks, and other storage devices that appear under the Devices category in the Finder. Use the Local Devices to access local storage.

  TIP Network volumes connected to your workstation appear in Local Device.

General Tab

Cache Source Media button Enable to create a managed copy of the media in the application storage; this copy is a transcoded version of the original media, using the Cached Media Compression Format as the transcode target. Disable to link to the media of the imported clip, without transcoding or copying the media.

Enabling Cache Source Media ensures that the application is the sole owner of the media, preventing the media from being modified by an external source. With Cache Source Media disabled, the application decodes the clip on-the-fly and there is no transcoding.

You can always change your mind after importing a clip: right-click the clip and select **Media ➤ Cache Source Media** to transcode the media and copy it to your local storage. Right-click and select **Media ➤ Flush Source Media Cache** to get rid of the transcoded copy and refer back to the original media.

Alpha Channel Processing box Select how alpha channels of RGBA media is processed on import: discard alpha channel information, create a matte container for the clip and its alpha channel, or create an RGB clip and another clip for its matte.

Cached Media Compression Format field Displays the compression format applied to clips imported with Cache Source Media enabled. Defined for the project, in the Cache and Renders tab of the Project dialogue box.

You acces your project's settings in **Flame ➤ Project and User Settings**.

Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

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

Select:	To fit:
Centre/Crop	The source image, centred, over the destination resolution frame. If the source is larger than the destination, it is cropped. If the source is smaller than the destination, it is surrounded by a black border.
Crop Edges	One edge of the source into the destination resolution frame without stretching or squashing the frame. Excess parts of the source frame after resizing are cropped. If the source—after the one edge is resized—is wider than the destination, its overhanging left and right edges are cropped. If the source is taller than the destination, the upper and lower edges are cropped.

Select:	To fit:
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.

Resize Filter box Select the resize filter to apply to the clip; all but the Impulse filter are rendered using the GPU. This box does not appear if you select the Centre/Crop fit.

Select:	For:
Lanczos	Excellent and sharp results. Recommended for upscale and downscale. Expensive to compute.
Shannon	Excellent and sharp results. Results are sharper than Lanczos in small details. Recommended for upscale and downscale. Expensive to compute.
Gaussian	Medium quality and softer results.
Quadratic	Medium quality and softer results.
Bicubic	High-quality results, but not as sharp as Shannon. Use for both upscale and downscale.
Mitchell	High-quality results, but not as sharp as Shannon. Use for both upscale and downscale.
Triangle	Low quality results that are fast to compute. Use for downscale.
Impulse	Very low quality results that are fast to compute. Use for downscale.

Width field Displays the custom width resolution of the clip. Editable.

Height field Displays the custom width resolution of the clip. Editable.

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

Aspect Ratio field Displays the custom render/output aspect ratio. Editable

Frame Depth box Select the render/output frame depth of clips.

Scan Mode box Select the scan mode of clips.

Use LUT button Enable to apply the LUT or colour transform displayed in the Applied LUT field to the clip.

LUT Format box Select the type of LUT or colour transformto apply to the clip.

Applied LUT field Displays the type of conversion LUT applied to the clip, either imported using Import, or edited using Edit.

Conversion LUT Type box Select a basic LUT type, EXR Display, PhotoMap, or Gamma correction.

LUT Editor Access button Click to open the LUT editor.

Format Specific Options

Show All Formats button Disable to display only the format of the file selected in the MediaHub browser. Enable to view or edit import options for any of the available file formats.

File Format box With Show All Formats disabled, File Format displays the format of the file selected in the browser; if more than one file is selected, File Format displays the formats of the selected files. With Show All Formats enabled, select the file format to view or edit its import options.

Options Category box Select the set of options to view or edit.

Sequence Import Options

Media Options

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

Preferred Media button Select which, of either offline intermediates or original sources, to import when both are found by the application. Only used when importing AAF sequences.

Consolidate on Import button Enable to import only the segments of the sources used by the imported sequence. Enabled when Use Timecode is enabled.

Maximum Handles field Displays the maximum number of handles (head and tail) allowed for each event in the sequence. If the sequence also sets the amount of handles, the application imports the sources using the lowest number of handles set between Maximum Handles and the sequence. Editable.

Relative Search Options

Search and Import Files button Enable to locate and import the media listed in the sequence, using the selected Search Criteria options but not the filepaths the sequence might contain. The media found is imported as segments of the sequence. Enabled if Link to Files is enabled.

Directories Up field Use to expand the search to parent directories. The application searches for media to match by going down any folder structure, starting with the directory from where the sequence file is imported.

NOTE When setting the Directories Up field, keep in mind that the application navigates through the whole directory structure. This means that the higher up you go in the folder structure, the longer the conform takes.

Save Sources Separately button Enable to also import each source referred to in the sequence. The application imports the sources in the same folder as the imported sequence, in the Media Library.

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

Use Timecode button Enable to use the source timecode specified in the sequence as a match criteria. Enables Consolidate on Import.

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

Use UMID button Enable to use the starting SMPTE UMID in the sequence as a match criteria. This is only used with MXF files and is ignored in all other cases.

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

Use Framerate button Enable to use the frame rate specified in the sequence as a match criteria. Disable to disregard the frame rate; try slipping the clip and using a timewarp in the timeline after loading the timeline to correct any frame rate discrepancy.

Be careful when using the Use Framerate search option with FCP XML files, especially if the FCP sequence is using multiple frame rates. With Use Framerate enabled, Flame Premium uses the frame rate of the sequence's *edits* as a match criteria to relink to the correct sources: if the frame rate of the considered source is 24fps while the edit is at 30fps, that source is not a potential candidate. Whether or not Flame Premium relinks the considered source to the edit has nothing to do with the frame rate of the *sequence*.

But, if you are importing an FCP sequence, Flame Premium timewarps the *edits* so that they match the frame rate of the imported sequence. For example, an FCP sequence @60fps contains edits @50fps: Flame Premium timewarps the edits to 60fps as it imports the sequence. Whether or not edits are relinked to their sources has nothing to do with this: if Use Framerate is enabled, Flame Premium only relinks the above edits if the sources are matching the *edits' original frame rate*, in this case 50fps.

Jobs

The Jobs tab displays ongoing and completed background processes. Use the Jobs tab to monitor the import and export processes.

Actions box Select from the list an action to perform on the selected job. Note: any frames already processed are retained. For example, if the Status column reads 20 of 44 when you click Abort, you will have 20 of the 44 frames of the clip.

Customizing your File Imports Options

Flame Premium includes pre-defined import options for each available file format. While these presets should answer most of your needs, there are times you might want a different debayering settings for .r3d files, or rename a clip on import.

To customize the import options of a selected file:

1. In the MediaHub, click the Format Options tab.
2. In the file browser, navigate to, and double-click the media file to import.

   The Selected Format field displays the format of the selected file.
3. From the Options box, select a category of options to edit.
4. Edit the options as needed.

   The import options are saved automatically; no need for a manual save. Flame Premium will import media of that format using the new options.
5. Drag and drop the media file from the browser to the Media Library. A new clip is created, using the new import options.

To customize the import options at any time:

1. Click the MediaHub tab.
2. In the MediaHub menu, click the Format Options tab.
3. Enable Show All Formats.
4. From the Selected Format box, select the media format to edit.
5. From the Options box, select the category of options to change.
6. Edit the options as needed.

The import options are saved automatically; no need for a manual save. From now on, media of that format are imported using the options you have set.

ARRIRAW Format Settings - Import

IMPORTANT The timecode of some ARRIRAW files was not correctly read in versions of Flame Premium prior to Flame 2013 20th Anniversary Edition Extension 2 SP3. This issue is now fixed. But be careful if you use ARRIRAW clips that were imported in a previous release. Loading that material in the fixed version shows the same visual results, but the source timecode is automatically updated. This ensures that both the ProRes and the the ARRIRAW files have the same timecode, making it easier to offline in ProRes and relink to the ARRIRAW files.

Metadata Settings

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

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

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

Timecode box Select an option to specify which of the available timecodes to use as ARRIRAW clips can contain multiple timecodes. If the selected timecode is not availablein the selecte clip, it displays 00:00:00 for timecode.

Selection:	ARRIRAW Definition
Master TC	Internally generated timecode.
External LTC	Longitudinal Time Code from an external sync.
External VITC	Vertically Integrated Time Code from an external sync.
Time of Day	Internally generated free run timecode starting at time of day.

Selection:	ARRIRAW Definition
Free Run	Internally generated free run timecode starting at a time defined by the user.
Edge Code	Internally generated timecode frame count increasing with each new media.
Clip Code	Internally generated timecode frame count increasing with each new take.
Record Run	Internally generated record run timecode starting at a value defined by the user.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Resolution Settings

Debayering box Select the resolution of the imported media. ARRIRAW decoding provides for using the original camera image resolution (2880x1620), or the resolution of the ProRes files (2868x1614). SD, HD, and 2K resolutions are not downscaled from the original image, but different debayering resolutions.

NOTE Prior to Flame 2013 20th Anniversary Edition Extension 2 SP3, the debayering options were named Full, Draft, Half... This naming scheme is no longer in use. Additionally, Half and Third cannot be translated into one of the new debayering setting. If you restore material archived at Half or Third debayering, you get a checkerboard (Flame) or a red X (Lustre) upon restore. Open the clip on the Timeline, and use the Format Options from the Timeline to select a valid debayering setting.

Debayering Mode box Select the debayer algorithm to use when importing ARRIRAW files. HW/SW refers to the method ARRI cameras use to decode RAW media. It does not mean that the application is performing hardware optimized debayering. All debayering is done software.

Option:	Description:
ADA-1 HW	Reproduces the hardware optimised debayering that is realised in cameras. Previously named Camera option.
ADA-2 SW	A more efficient debayering algorithm than ADA-1. Previously named AMC-1 option.
ADA-3 HW	Visually matches the hardware optimised debayering realised in cameras with ARRI SUP 7.0: more complex debayering than ADA-1, with extended edge and color handling.
ADA-3 SW	Advanced software debayering provided by the ARRI SDK.

Option:	Description:
Proxy	A low quality debayering mode designed for performance over quality.

Crispness field Set the cutoff steepness value of the downscaler.

Aspect Ratio box Set the aspect ratio used when importing the clips, or have Flame Premium use the information stored in the file header to set automatically the aspect ratio.

Colour Settings

NOTE Some combinations of Source Camera, Colour Space, and Colour Rendering are not supported.

Colour Settings box Select how Flame Premium uses the colour information stored within an ARRIRAW file.

Select:	To import ARRIRAW clips using:
User	The options you set in the Colour, Processing, and Image menus.
Camera	The look created on the ARRI camera and stored in the ARRIRAW file. This option disables the Processing and Image menus, as well as the Colour Rendering box.

Save as User Settings button Enable to make the Colour, Processing, and Image menus editable in the Import History. Save as User Settings is implicitly enabled when the Colour Settings box is set to User.

Look Selection box Select from where Flame Premium reads the Look information, when available.

Version 4 of the ALEXA Software Update Packet enables a *look* creation workflow. In this workflow, a look is created in the ARRI Look Creator application and either embedded in the source files during the shoot, or saved to a look file. This look information is accessible by Flame Premium.

When working with Look information, set the Colour Rendering to Rec709 or DCI P3 option. If the Colour Rendering is set to LogC, Flame Premium displays a checkerboard instead of the media as ARRI Look Creator does not support images encoded in LogC.

NOTE When used, the Look file must be placed in the folder containing the source material and named *look.xml*.

Select:	To:
Do not apply Look	Not apply any Look information.
From File Header	Apply Look information contained in the ARRIRAW file header.
From Look File	Apply Look information contained in an XML Look file located in the same folder as the media to import.
Header or Look File	Apply Look information from the header of the media to import. If none is present, use the information from an XML Look file located in the same folder as the file to import.

Select:	To:
Look File or Header	Apply Look information from an XML Look file located in the same folder as the file to import. If none is present, use the information from the header of the file to import.

Apply Look LUT button Enable to use the LUT stored with the Look information read by Flame Premium from the location determined by the Look Selection box.

Colour Rendering box Select one of the preset combinations of colour spaces and encoding to apply to all imported clips, or select From File Header to read that colour space and encoding information from the file. Select Custom to create your own combination using the Colour Space and Colour Encoding boxes.

ACES (Scene Linear) gives you direct access to 16-bit floating point images, so you do not have to apply a colour transformation to get 16-bit floating point images.

Colour Space box Select the colour space used to read the clips. Available when Colour Rendering is set to Custom. Some combinations of Colour Space and Colour Encoding are invalid and display a checkboard instead of a clip. Change either settings to display a valid clip.

An invalid combination would be ACES + Encoding Video since ACES colour space requires 16-bit fp Scene Linear encoding.

Colour Encoding box Select the value of the output gamma curve applied to imported clips. Available when Colour Rendering is set to Custom. Some combinations of Colour Space and Colour Encoding are invalid and display a checkboard instead of a clip. Change either settings to display a valid clip.

Processing Settings

Processing Version box Select the version of the colour science used to process the file, or use From File Header to have Flame Premium read the version of the colour science from the header of the imported file. Use From File Header unless you are trying to recreate a look based on a specific colour science.

Source Camera box Select the camera that created the file, or have Flame Premium read from the file header the camera used. Leave at Camera from File Header unless you are troubleshooting a file.

Image Settings

Exposure box Select how the file's exposure is set. Select Exposure From File to use the camera settings burnt in the clip.

ISO Select the value of the linear gain operation.

White Balance box Select how the white balance is defined. Select White Balance From File to use the camera settings burnt in the clip.

NOTE To determine if white balance information is already present in a clip, look at *White bal burnt in* in the Metadata tab of the Previewer. If it displays *true*, there is white balance camera settings present in the clip.

Kelvin field Set the perceptual colour temperature of the imported clip. Only available if White Balance is set to Set Temperature.

Green/Magenta Tint field Set the green and magenta tint balance. Only available if White Balance is set to Set Temperature.

RGB Gain fields Set the RGB gains of the imported clip. Only available if White Balance is set to Set RGB White Balance.

Audio File Settings - Import

Metadata Settings

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

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

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

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

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.
Timecode from File Name	Use a numerical filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

AVCHD Format Settings - Import

Metadata Settings

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

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

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from Header	Read the clip name from the header of the imported file.
Clip Name from XML	Use the clip name listed in the XML file accompanying the XDCAM media. (XDCAM only)
Clip Name from Essence	Use the clip name inferred from the directory structure of the P2 or XDCAM clip. (P2 and XDCAM only)

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Include YUV Headroom Enable to keep the YUV headroom information during clip import. Only applies to 10-bit YCbCr-material found in some mp4 and QuickTime files. Disabling this option for clips that use the headroom (usually clips from cameras) results in clips with clamped black and whites. Enabling this

option for clips that do not use the headroom (usually clips from Final Cut Pro) results in an imported clip with lowered contrast.

Aspect Ratio box Select the aspect ratio of the frames used to import the file and override the aspect ratio stored in the file as specified. You might need to change the aspect ratio as many applications erroneously write a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires that you enable **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediHub always displays clips with the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select either Square Pixels or Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Set the ratio used to define the frame or pixel aspect. Affects the frame aspect if the Aspect Ratio box is set to Enter Frame Aspect Ratio. Affects the pixel aspect if it is set to Enter Pixel Aspect Ratio.

DPX Format Settings - Import

Metadata Settings

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

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

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

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

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.

Select:	To:
Timecode from File Name	Use a numerical filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Rate box Select how the framerate is determined. Select Auto to use the framerate specified by the imported clip; if the clip specifies no framerate, it uses the one of the project. Select Select rate to set the framerate using the Framerate box.

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

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Enabled if the Rate box is set to Select rate.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Sequence Detection box Select how Flame Premium displays image sequences. An image sequence consists of image files, such as .psd, named identically except for a numeric suffix.

- myPhotoshopFile_version_001.psd
- myPhotoshopFile_version_002.psd
- myPhotoshopFile_version_003.psd
- myPhotoshopFile_version_004.psd

Select:	To display and process the related images:
Sequence	As a single clip made of all these related images.
Frames	As independent images.

Keycode Settings

NOTE The keycode supplied here is only used for information purporses. In case of a discrepancy between the information supplied in the Metadata menu and the Keycode menu, the former is used to determine the timecode and frame rate used.

Keycode Scan Mode box Select an option to determine how keycode data is applied to the imported clip.

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

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

Film Gauge box Select a film gauge for the keycode.

Gateway Clip Format Settings - Import

Gateway clips do not have any available import options.

HDR Format Settings - Import

Metadata Settings

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

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

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

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

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.

Select:	To:
Timecode from File Name	Use a numerical filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Rate box Select how the framerate is determined. Select Auto to use the framerate specified by the imported clip; if the clip specifies no framerate, it uses the one of the project. Select Select rate to set the framerate using the Framerate box.

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

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Enabled if the Rate box is set to Select rate.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from Header	Read the clip name from the header of the imported file.
Clip Name from XML	Use the clip name listed in the XML file accompanying the XDCAM media. (XDCAM only)
Clip Name from Essence	Use the clip name inferred from the directory structure of the P2 or XDCAM clip. (P2 and XDCAM only)

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Exposure Adjustement field Set the exposure of the HDR file. In units of stops.

Image Sequences Settings - Import

Metadata Settings

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

Select:	To:
Enter Tape	Activate the Tape Name field so that you can manually enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.

Select:	To:
Tape from Filename	Use the name of the imported file as the tape name.
Tape from Directory	Determine the tape name from the detected directory structure. A Level field appears below the Tape Name box. Use this to configure the relative path to the directory from which the tape name can be determined.

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

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

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.
Timecode from File Name	Use a numerical filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Rate box Select how the framerate is determined. Select Auto to use the framerate specified by the imported clip; if the clip specifies no framerate, it uses the one of the project. Select Select rate to set the framerate using the Framerate box.

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

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Enabled if the Rate box is set to Select rate.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Sequence Detection box Select how Flame Premium displays image sequences. An image sequence consists of image files, such as .psd, named identically except for a numeric suffix.

- myPhotshopFile_version_001.psd
- myPhotshopFile_version_002.psd

- myPhotshopFile_version_003.psd
- myPhotshopFile_version_004.psd

Select:	To display and process the related images:
Sequence	As a single clip made of all these related images.
Frames	As independant images.

MP4 Format Settings - Import

Metadata Settings

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

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

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

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

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.
Timecode from File Name	Use a numerical filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Rate box Select how the framerate is determined. Select Auto to use the framerate specified by the imported clip; if the clip specifies no framerate, it uses the one of the project. Select Select rate to set the framerate using the Framerate box.

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

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Enabled if the Rate box is set to Select rate.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from Header	Read the clip name from the header of the imported file.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Include YUV Headroom Enable to keep the YUV headroom information during clip import. Only applies to 10-bit YCbCr-material found in some mp4 and QuickTime files. Disabling this option for clips that use the headroom (usually clips from cameras) results in clips with clamped black and whites. Enabling this option for clips that do not use the headroom (usually clips from Final Cut Pro) results in an imported clip with lowered contrast.

Aspect Ratio box Select the aspect ratio of the frames used to import the file and override the aspect ratio stored in the file as specified. You might need to change the aspect ratio as many applications erroneously write a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires that you enable **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips with the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select either Square Pixels or Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Set the ratio used to define the frame or pixel aspect. Affects the frame aspect if the Aspect Ratio box is set to Enter Frame Aspect Ratio. Affects the pixel aspect if it is set to Enter Pixel Aspect Ratio.

MXF Format Settings - Import

Metadata Settings

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

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

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

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

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.
Timecode from File Name	Use a numerical filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from Header	Read the clip name from the header of the imported file.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Scale to Full HD button Enable Scale to Full HD to have media with a 1280x1080 or 1440x1080 resolution appear in the Player at a standard 1920x1080 resolution. Disable Scale to Full HD to display the media at its native (1280x1080 or 1440x1080) resolution in the Player. Available to P2, XDCAM, and MXF.

Include YUV Headroom Enable to keep the YUV headroom information during clip import. Only applies to 10-bit YCbCr-material found in some mp4 and QuickTime files. Disabling this option for clips that use the headroom (usually clips from cameras) results in clips with clamped black and whites. Enabling this option for clips that do not use the headroom (usually clips from Final Cut Pro) results in an imported clip with lowered contrast.

Aspect Ratio box Select the aspect ratio of the frames used to import the file and override the aspect ratio stored in the file as specified. You might need to change the aspect ratio as many applications erroneously write a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires that you enable **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips with the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select either Square Pixels or Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Set the ratio used to define the frame or pixel aspect. Affects the frame aspect if the Aspect Ratio box is set to Enter Frame Aspect Ratio. Affects the pixel aspect if it is set to Enter Pixel Aspect Ratio.

OpenEXR Format Settings - Import

Metadata Settings

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

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

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

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

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.
Timecode from File Name	Use a numerical filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Rate box Select how the framerate is determined. Select Auto to use the framerate specified by the imported clip; if the clip specifies no framerate, it uses the one of the project. Select Select rate to set the framerate using the Framerate box.

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

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Enabled if the Rate box is set to Select rate.

Clip Settings

Clip Names box Channels are imported as clips in Flame Premium. Select how to name, at import, the clips thus created.

Select:	To:
Filename	Use the filename of the container for all the imported channels.
Channel	Use the channel name as the clip name.
Channel + Filename	Combine the channel name and the filename of the container to form the clip name.
Filename + Channel	Combine the filename of the container and the channel name to form the clip name.

Sequence Detection box Select how Flame Premium displays image sequences. An image sequence consists of image files, such as .psd, named identically except for a numeric suffix.

- myFile_version_001.psd
- myFile_version_002.psd
- myFile_version_003.psd
- myFile_version_004.psd

Select:	To display and process the related images:
Sequence	As a single clip made of all these related images.
Frames	As independent images.

P2 Format Settings - Import

Metadata Settings

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

Select:	To:
Enter Tape Name	Activate the Tape Name field so that you can manually enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape Name from Filename	Use the name of the imported file as the tape name.
Tape Name from Directory	Determine the tape name from the detected directory structure. A Level field appears below the Tape Name box. Use this to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Essence	Use the tape name inferred from the directory structure of the P2 clip.

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

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

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.
Timecode from File Name	Use a numerical filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from Essence	Use the clip name inferred from the directory structure of the P2 clip.

Select:	To:
Clip Name from XML	Use the clip name listed in the XML file accompanying the P2 media.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Scale to Full HD button Enable Scale to Full HD to have media with a 1280x1080 or 1440x1080 resolution appear in the Player at a standard 1920x1080 resolution. Disable Scale to Full HD to display the media at its native (1280x1080 or 1440x1080) resolution in the Player. Available to P2, XDCAM, and MXF.

Include YUV Headroom Enable to keep the YUV headroom information during clip import. Only applies to 10-bit YCbCr-material found in some mp4 and QuickTime files. Disabling this option for clips that use the headroom (usually clips from cameras) results in clips with clamped black and whites. Enabling this option for clips that do not use the headroom (usually clips from Final Cut Pro) results in an imported clip with lowered contrast.

Aspect Ratio box Select the aspect ratio of the frames used to import the file and override the aspect ratio stored in the file as specified. You might need to change the aspect ratio as many applications erroneously write a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires that you enable **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips with the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select either Square Pixels or Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Set the ratio used to define the frame or pixel aspect. Affects the frame aspect if the Aspect Ratio box is set to Enter Frame Aspect Ratio. Affects the pixel aspect if it is set to Enter Pixel Aspect Ratio.

Browsing Settings

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

Photoshop Format Settings - Import

Metadata Settings

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

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

Select:	To:
Tape from File Header	Read the tape name from the header of the imported file.

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

Clip Settings

Clip Names box Layers are imported as clips in Flame Premium. Select how to name, at import, the clips thus created.

Select (PSD):	To:
Filename	Use the filename of the container for all the imported layers.
Layer	Use the layer name as the clip name.
Layer + Filename	Combine the layer name and the filename of the container to form the clip name.
Filename + Layer	Combine the filename of the container and the layer name to form the clip name.

Sequence Detection box Select how Flame Premium displays image sequences. An image sequence consists of image files, such as .psd, named identically except for a numeric suffix.

- myFile_version_001.psd
- myFile_version_002.psd
- myFile_version_003.psd
- myFile_version_004.psd

Select:	To display and process the related images:
Sequence	As a single clip made of all these related images.
Frames	As independent images.

Layers Settings

Resolution box Select how Flame Premium sets the resolution of .psd clips.

Select:	To:
Background	Import the layers using the same resolution as the background layer.
Native	Import the layers at their original resolution.

PNG Format Settings - Import

Metadata Settings

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

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

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

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

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.
Timecode from File Name	Use a numerical filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Rate box Select how the framerate is determined. Select Auto to use the framerate specified by the imported clip; if the clip specifies no framerate, it uses the one of the project. Select Select rate to set the framerate using the Framerate box.

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

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Enabled if the Rate box is set to Select rate.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.

Select:	To:
Clip Name from File Name	Use the name of the imported file as the clip name.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Sequence Detection box Select how Flame Premium displays image sequences. An image sequence consists of image files, such as .psd, named identically except for a numeric suffix.

- myFile_version_001.psd
- myFile_version_002.psd
- myFile_version_003.psd
- myFile_version_004.psd

Select:	To display and process the related images:
Sequence	As a single clip made of all these related images.
Frames	As independent images.

Precomp Format Settings - Import

Precomp clips do not have any available import options.

QuickTime Format Settings - Import

Metadata Settings

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

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

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

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

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.
Timecode from File Name	Use a numerical filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Rate box Select how the framerate is determined. Select Auto to use the framerate specified by the imported clip; if the clip specifies no framerate, it uses the one of the project. Select Select rate to set the framerate using the Framerate box.

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

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Enabled if the Rate box is set to Select rate.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from Header	Read the clip name from the header of the imported file.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Include YUV Headroom Enable to keep the YUV headroom information during clip import. Only applies to 10-bit YCbCr-material found in some mp4 and QuickTime files. Disabling this option for clips that use the headroom (usually clips from cameras) results in clips with clamped black and whites. Enabling this option for clips that do not use the headroom (usually clips from Final Cut Pro) results in an imported clip with lowered contrast.

Aspect Ratio box Select the aspect ratio of the frames used to import the file and override the aspect ratio stored in the file as specified. You might need to change the aspect ratio as many applications erroneously write a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires that you enable **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips with the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select either Square Pixels or Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Set the ratio used to define the frame or pixel aspect. Affects the frame aspect if the Aspect Ratio box is set to Enter Frame Aspect Ratio. Affects the pixel aspect if it is set to Enter Pixel Aspect Ratio.

R3D Format Settings - Import

Metadata Settings

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

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

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

Orientation box Select to flip (vertical) or flop (horizontal) the media of the imported clip. Camera uses the orientation transformation defined during the shoot; Disable Flip keeps the orientation of the image as it was shot.

Timecode box Select which timecode to use as RED camera records both edge code (run record) and time-of-day (free run) timecodes during a shoot. Or Use Primary to use the timecode defined during the shoot.

Debayering Settings

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

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

Bit Depth box Set the bit depth of the imported image. If you set Debayering > HDRx box to High Dynamic Range, HDRx footage is always imported as a 16-bit floating point clip, even if Bit Depth is set to 12bit Integer.

Detail box Select the level of detail extraction required.

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

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

Colour Settings

Colour Settings box Select how Flame Premium uses the colour information stored within a R3D file.

Select:	To have:
User	Flame Premium import RED clips using the options you set in the Image, Gain, and Curve menus.
Camera	Flame Premium import RED clips using the look created on the RED camera and stored in the RED clip. Disables the Image, Gain, and Curve menu options.
RMD/Camera	Flame Premium import RED clips using the RMD look created in REDCINE-X, or fall back on the camera settings if there is no RMD settings. Disables the Image, Gain, and Curve menu options.
RSX Only	Flame Premium import RED clips using the RSX look created in RED Alert!. The RSX file of a clip must reside in the same directory as the R3D file of that clip. Disables the Image, Gain, and Curve menu options. With this option selected, only clips with an RSX profile can be imported. Clips without an RSX profile appear to be missing media.
RSX or RMD/Camera	Flame Premium import RED clips using the RSX look. If a clip has no RSX file, Flame Premium imports it using the RMD look created in REDCINE-X, or fall back on the camera settings if there is no RMD settings. Disables the Image, Gain, and Curve menu options.
RSX or User	Flame Premium imports RED clips using the RSX look. If a clip has no RSX file, Flame Premium imports it using the options you set in the Image, Gain, and Curve menus. Enables the Image, Gain, and Curve menu options.

Save as User Settings button Enable to make the Colour, Image, Advanced, and Curve menus editable in the Import History. Save as User Settings is implicitly enabled when the Colour Settings box is set to *User*, or *RSX or User*.

Colour Science box Set the version of the RED codec to use. Using the version 3 of the codec gives you access to the FLUT and the Shadow options in the Image menu, as well as version 3-only colour spaces and gamma curves.

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

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

Gamma Curve box Set the value of the output gamma curve that is applied to the imported clips. If you select Scene Linear, the Bit Depth of imported clips are always set to 16-bit floating point. If you set Debayering > HDRx box to High Dynamic Range, HDRx footage is always imported as Scene Linear, even if Gamma Curve is set to something else.

HDRx Settings box Select which track of a RED HDRx file to import, or how to merge the two tracks together. A RED HDRx media file is made of two tracks, track A (the main exposure) and track X (the highlight protection exposure). Use HDRx Settings to set how you want to use those two tracks.

Select:	**To have:**
Primary Exposure	Only the main exposure (A track).
Highlight Exposure	Only the highlight protection exposure (X track).
Blend Exposures	A single clip resulting from blending together the Primary and Highlight tracks. Blend Exposures behaves similarly to the option of the same name found in REDCINE-X by RED. Use the Blend field to set the blend value to use.
Magic Motion	A single clip resulting from the merge of the Primary and Highlight tracks using an algorithm provided and developed by RED. This algorithm tries to match and blend together the motion blurs of the two tracks.
High Dynamic Range	A single clip resulting from the merge of the Primary and Highlight tracks using an HDR merging algorithm developed by Autodesk. It converts the two exposures into a single 16-bit float image. Importing an HDRx clip as High Dynamic Range forces the Bit Depth of the imported clip to 16-bit floating point, and the Gamma Curve to Scene Linear. Use the Blend, Highlight Threshold, and Exposure Offset fields to refine the blend.
RMD	A clip using blending options read from the RMD file. If there is no RMD file, the Flame Premium loads the main exposure (A track).

Blend field Set how to blend the two exposures of a RED HDRx clip.

With HDRx Settings set to Blend Exposures, this field behaves like the blend in REDCINE-X; the blend also attenuates the artifacts created by the scene operation, which is a blend of the two exposures: -1 shows only the Highlight (X frame), 1 only the Primary (A Frame), and 0 a 50-50 mix.

With HDRx Settings set to High Dynamic Range, Blend also attenuates the artifacts created by scene motions; set to 1 unless you are troubleshooting motion artifacts.

Highlight Threshold field Set the threshold when pixels from the Highlight exposure are used instead of the over-exposed pixels from the Primary exposure. Only available when HDRx Settings box is set to High Dynamic Range. Set Highlight Threshold last because import options such as ISO or FLUT lighten or darken the image. Too high and you get clipping (often including a magenta-coloured cast), too low and the midtones and shadows have noise leaking in from the Highlight track.

Exposure Offset field Set how much greater the Primary exposure was when compared to the Highlight exposure, in units of stops. Only available when HDRx Settings box is set to High Dynamic Range. The Exposure Offset should be set to match the setting on the camera for how many stops separate the Primary and Highlight tracks. This is typically 2 or 3 stops.

Offset From File button Enable to use the Exposure Offset read from the R3D file. Enabled when HDRx Settings is set to High Dynamic Range.

Image Settings

ISO Select the value of the linear gain operation.

Saturation Set the saturation value.

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

IMPORTANT In Flame 2013 20th Anniversay Edition Extension Extension 2 Service Pack3, changes to the RED SDK 4.4 affect how the DRX setting is computed for all R3D clips, and can impact clips imported before these changes.

- Changes to DRX processing

  The clips affected are the ones which have a DRX setting different from 0. To check if this is the case for a clip:

  - In Flame: Open, from the Timeline, the Pre-Processing Editor, and look for DRX under **Basic ➤ Image**.
  - In Lustre: Look for DRX in **Transcode ➤ Colour ➤ Colour Settings**.

  If DRX is not 0, then your media is impacted by the change. How much of an impact is hard to predict, but since DRX (Dynamic Range Extension) is used to reconstruct clipped colour channels, watch out especially for clips with dangerous highlights and for HDRx material.

- Support for DRX processing via RED Rocket

  The DRX setting is now processed by the RED Rocket card if one is present on the Mac running the Wiretap Gateway server used to decode R3D files. The following driver and firmwares must be installed on the Mac:

  - RED Rocket driver 1.4.32.0
  - RED Rocket firmware 1.1.16.11 or later
  - RED Rocket Breakout Box firmware 1.0.2.0 or later

Tint Set the tint value.

FLUT Set the FLUT to refine of the ISO level. As FLUT units are in stops, a +1 FLUT value is the same as doubling the ISO.

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

Brightness Set the brightness value.

Contrast Set the contrast value.

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

Shadow Set the Shadow level.

Advanced Colour Settings

Use the Advanced Colour menu to set the RGB Lift, Gamma, and Gain of RED clips.

Use the Legacy Gain options if you import clips shot with a RED codec that uses a pre-3.0 Colour Science.

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

Colour Curves Settings

Use this menu to set the RGB and Luma curves for RED clips.

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

SonyMXF Format Settings - Import

You can import Sony F5, F55, and F65 RAW MXF files in Flame Premium with a default Scene Linear transformation applied to convert the 16-bit media to 16-bit half-float.

Metadata Settings

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

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

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

Orientation box Select to flip (vertical) or flop (horizontal) the media of the imported clip. Camera uses the orientation defined on the camera during the shoot. Disable Flip to use the true orientation of the image, as it was shot.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.

Resolution Settings

Debayering box Select the resolution at which the media is imported. Higher resolutions require more processing and impact real-time playback. Other resolutions offer better performance at the cost of lower image resolution.

Only the F65 supports resolutions higher than 4K. Using a higher setting for material shot by other cameras prevents the media from being displayed: an error message also appears in the Previewer (*Cannot initiliaze codec*). Select 4K or lower to display the media correctly.

Debayering options have been renamed to reflect what is used in other applications. The table below highlights the changes.

Option in Flame 2013 20th Anniversary Edition Extension 2 SP3:	Old option:
0.25K	Sixteenth
0.5K	Eighth
1K	Quarter
HD (1920)	
2K	Half
QFHD (3480)	
4K	Full
6K	
UHDTV (7680)	
8K	

Quality box Select one of two qualities for the decoding of SonyRAW footage. While debayering resolution result in lower resolution clips, Quality affects the quality of the fully debayered pixels. Depending on your system configuration, Standard provides better decoding performance at the cost of lower image quality compared to High quality. The actual image degradation depends on the footage being decoded, but because

of the Bayer pattern, expect to see differences in the red and blue channels. The Quality box is not available for some of the Debayering resolutions.

Colour Settings

Colour Encoding box Select Scene Linear to apply a colour transformation to convert the 16-bit media to 16-bit half-float. Select Native to disable this transformation and work with the RAW media.

NOTE The colour space options —Rec709 primaries, ACES (daylight illuminant), ACES (tungsten illuminant)— were updated in Flame 2013 20th Anniversary Edition Extension 2 SP3 to solve an issue with black levels. This impacts SonyRAW media imported before the update: the black levels will not match SonyRAW media imported after the update. To fix this, view the older clips on the Timeline, and in the Format Options Editor, select **Basic ➤ Colour**, and then set the correct colour space.

XDCAM Format Settings - Import

Metadata Settings

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

Select:	To:
Enter Tape	Activate the Tape Name field so that you can manually enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape from Filename	Use the name of the imported file as the tape name.
Tape from Directory	Determine the tape name from the detected directory structure. A Level field appears below the Tape Name box. Use this to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Essence	Use the tape name inferred from the directory structure of the XDCAM clip.

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

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

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

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.
Timecode from File Name	Use a numerical filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from XML	Use the clip name listed in the XML file accompanying the XDCAM media.
Clip Name from Essence	Use the clip name inferred from the directory structure of the XDCAM clip.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Scale to Full HD button Enable Scale to Full HD to have media with a 1280x1080 or 1440x1080 resolution appear in the Player at a standard 1920x1080 resolution. Disable Scale to Full HD to display the media at its native (1280x1080 or 1440x1080) resolution in the Player. Available to P2, XDCAM, and MXF.

Include YUV Headroom Enable to keep the YUV headroom information during clip import. Only applies to 10-bit YCbCr-material found in some mp4 and QuickTime files. Disabling this option for clips that use the headroom (usually clips from cameras) results in clips with clamped black and whites. Enabling this option for clips that do not use the headroom (usually clips from Final Cut Pro) results in an imported clip with lowered contrast.

Aspect Ratio box Select the aspect ratio of the frames used to import the file and override the aspect ratio stored in the file as specified. You might need to change the aspect ratio as many applications erroneously write a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires that you enable **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips with the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select either Square Pixels or Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Set the ratio used to define the frame or pixel aspect. Affects the frame aspect if the Aspect Ratio box is set to Enter Frame Aspect Ratio. Affects the pixel aspect if it is set to Enter Pixel Aspect Ratio.

Browsing Settings

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

XDCAM EX Format Settings - Import

About XDCAM EX

Support for Sony XDCam EX media includes support for clips recorded as multiple segments on single SxS card (split clips) or multiple cards (span clips). Flame Premium imports as a single clip a serie of split & span segments.

NOTE Clips spanning two card only become available in Flame Premium when browsing the card that contains the last segment.

To ensure that Flame Premium is able to see the clips organisation, copy the full folder structure from the SxS cards to a single folder on your local storage.

For example, with two SxS cards (named Card1 and Card2), create the following structure:

- storage
  - folder
    - Card1
    - Card2

IMPORTANT Do not modify the file structure inside the cards folders. For example, copying only the media files (.mp4) instead of the whole card structure will prevent Flame Premium from recreating the correct clip structure.

AAF Conform

You can conform AAF files referencing XDCam EX content inFlame Premium. Make sure to import the XDCam EX clips in Avid products through AMA (the Avid Media Architecture). If you prefer to work with transcoded media in Avid products, but still want to conform with the original media fils, enable in Flame Premium the Use Original Media option when importing the AAF file. That way, the conform will use the original media, instead of the transcoded MXF generated by the Avid products.

Metadata Settings

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

Select:	To:
Enter Tape	Activate the Tape Name field so that you can manually enter the tape name. When selecting multiple files for import, this tape name is used for all imported files.
Tape from Filename	Use the name of the imported file as the tape name.
Tape from Directory	Determine the tape name from the detected directory structure. A Level field appears below the Tape Name box. Use this to configure the relative path to the directory from which the tape name can be determined.
Tape Name from Essence	Use the tape name inferred from the directory structure of the XDCAM EX clip.

Level field Select an option to set the relative file path to the directory from which the tape name can be determined. Although it is available for all file types, the Level field is specifically designed to work with the directory structures that are output by film scanners. A typical file structure would look like:

./<tape>/<resolution>/clip.######.dpx. In this case, selecting Up 2 Levels in the Level field identifies the directory that corresponds to the tape name (*./<tape>*). Enabled if Tape Name is set to Tape From Directory.

Tape Name field Set the name to use as the tape name of the imported clip. Enabled if Tape Name is set to Enter Tape.

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

Select:	To:
Timecode from Header	Set the source timecode of the imported clip based on the timecode information in the image file header.
Timecode from File Name	Use a numerical filename (for instance, *100000.dpx*) and translate it into timecode for the resulting clip (based on the selected framerate). This is useful when working with files that do not have embedded timecode.

Rate box Select how the framerate is determined. Select Auto to use the framerate specified by the imported clip; if the clip specifies no framerate, it uses the one of the project. Select Select rate to set the framerate using the Framerate box.

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

Drop Frame button Enable to have use a drop frame mode. Disable to use non-drop frame mode. Enabled if the Rate box is set to Select rate.

Clip Settings

Clip Name box Select how the clip is named when the file is imported.

Select:	To:
Enter Clip Name	Activate the Clip Name field so that you can manually enter the clip name. When selecting multiple files for import, this name is used for all imported files.
Clip Name from File Name	Use the name of the imported file as the clip name.
Clip Name from Essence	Use the clip name inferred from the directory structure of the XDCAM EX clip.

Clip Name field Enter the name to use when importing the clip. Enabled if Name is set to Enter Clip Name.

Image Settings

Include YUV Headroom Enable to keep the YUV headroom information during clip import. Only applies to 10-bit YCbCr-material found in some mp4 and QuickTime files. Disabling this option for clips that use the headroom (usually clips from cameras) results in clips with clamped black and whites. Enabling this option for clips that do not use the headroom (usually clips from Final Cut Pro) results in an imported clip with lowered contrast.

Aspect Ratio box Select the aspect ratio of the frames used to import the file and override the aspect ratio stored in the file as specified. You might need to change the aspect ratio as many applications erroneously write a pixel aspect ratio of 1 (a square pixel), even for formats that have non-square pixels (NTSC, PAL).

Viewing the clip in the Player with the specified aspect ratio requires that you enable **Options ➤ Show Viewing Settings ➤ Use Ratio**. The Previewer in the MediaHub always displays clips with the specified aspect ratio.

Note that the application internally uses a frame ratio, not a pixel ratio. To specify a pixel ratio, you must select either Square Pixels or Enter Pixel Aspect Ratio; Aspect Ratio from Resolution and Aspect Ratio from Header are interpreted as frame ratios.

Aspect Ratio field Set the ratio used to define the frame or pixel aspect. Affects the frame aspect if the Aspect Ratio box is set to Enter Frame Aspect Ratio. Affects the pixel aspect if it is set to Enter Pixel Aspect Ratio.

Browsing Settings

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

About Pixel Aspect Ratio

Some video and film formats use rectangular instead of square pixels. This explains why NTSC and PAL video formats have a different frame dimension in terms of pixels, but are ultimately broadcast to the same 4:3 aspect ratio screen.

- NTSC pixels are narrower than they are high, which allows for the 720x486 (1.481) aspect ratio frame to fit into a 4:3 (1.333) aspect ratio broadcast screen.
- PAL pixels are wider than they are high, which allows for 720x576 (1.25) aspect ratio frame to fit into a 4:3 (1.333) aspect ratio broadcast screen.

When you capture video or import film frames that use rectangular pixels into Flame Premium, the pixels become square because computer graphics work only with square pixels. This is why the NTSC and PAL frames appear as though they are different size—displayed with square pixels, they are.

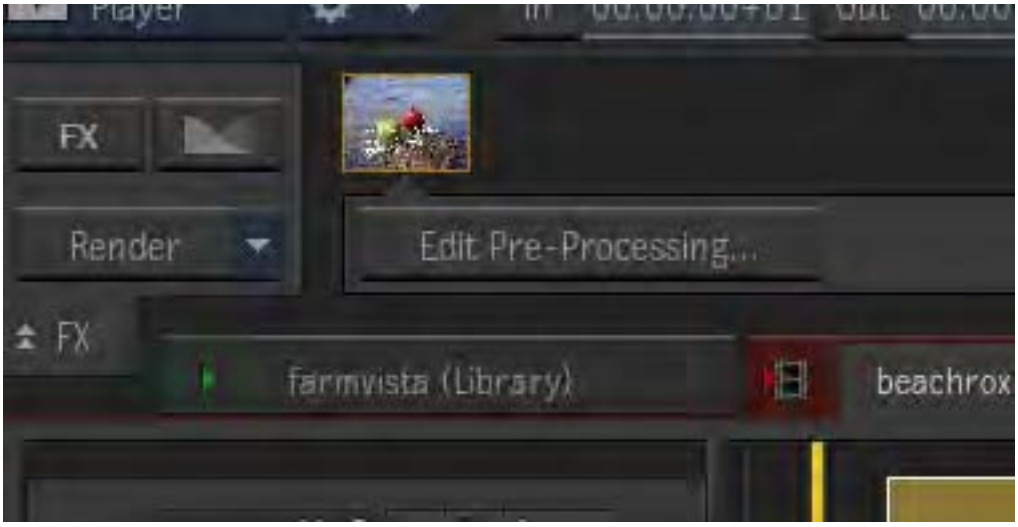
You can set the image window to display clips originating from rectangular pixel formats using filters to simulate the use of rectangular pixels. This provides a display that corresponds with the delivery format.

Using the aspect ratio display affects system performance, so if you are using the aspect ratio filter and notice a performance slowdown, you can disable the option and return to square-pixel display.

NOTE Many HD video formats such as 1920x1080 and 1280x720 use square pixels, so enabling this option has no effect. One HD video format that does use "slightly" rectangular pixels is 1920x1035.

Modifying Import Options After Import

1. Select the Timeline tab.
2. In the timeline, select the segment or clip to edit.
3. In the FX Processing Pipeline, select the thumbnail of the clip and click Edit Pre-Processing.



Colour Transforms and Import History

When creating the color transform in the color transform builder at importing the media, the transform selections can be of type Autodesk, Project, Shared, or Import. When editing the import options of the clip, the Color Transform Builder partially reflects the transform list created during the import:

- Muted transforms in the initial builder do not show up.
- Transforms of the Autodesk category are preserved.
- Transforms from the Project and Shared categories are re-branded as Import.

This happens because Flame Premium does not preserve the complete state of the colour transform builder, but rather a reference transform that creates the desired result.

Working with Others

Flame Premium offers you two methods to work in collaboration with other users of Flame or Smoke.

The first one is the Projects browser in the MediaHub. Use it to transfer clips between your project and another one. This other project can be a local, such as a previous project you worked on, or a remote project, such as a a sequence being put together in Smoke. Or use Projects to transfer clips from a project created with an older version of the software. But if you want to work on the same shots, clips, sequences, use Shared Folders.

The second method is Shared Folders. Shared folders integrate directly in the MediaHub panel, and are best used when you are working with other users on the same project. Flare Workflow (page 45) has more about working with Workspaces, and even if it targets mostly a Flame-Flare workflow, it applies to any setup where Shared Folders are used.

Import History and Lustre

When importing media files and using the Cache Source Media, Flame Premium bakes in import options like resize, LUT, and format specific options like R3D settings. Of course, you always have access to the import history, and edit it if need be. Just use the Edit Pre-Processing button in the Timeline FX ribbon.

But Lustre does not have access to the import history for clips imported in Flame Premium with an enabled Cache Source Media option. This means that if you import in Lustre a Flame Premium-created sequence

that uses such clips, you do not have access to their import history. Flushing the media cache prior to importing the clip in Lustre restores access to import history.

To restore access to the import history for Lustre:

1. From Flame Premium, right-click the material to grade in Lustre.
2. Select **Media ➤ Flush Source Media Cache**.
   You can now edit the import history of the clip from Lustre.

Sharing Clips Between Users

Clips placed in a special folder, called Shared Folder, becomes accessible to remote workstations that connect to the project. Shared folders enable different Creative Finishing workstations to quickly and conveniently share media.

Placing media in a shared folder automatically grants read access to any workstation connected to the project. The remote operator can then access the media and start working immediately after connecting to the project. Once finished, the remote operator needs to save the resulting media back to the shared folder. To do this he must acquire write access.

Write access locks the folder so that only the user having acquired write access can modify the folder contents (i.e. write to the folder). Read access is still available to other remote workstations connected to the project. Once the media is saved, the user can disable write access, enabling other remote workstations to obtain write access and modify the content.

NOTE Clips stored in Shared Folders are archived when you archive a project.

To acquire write access:

1. Select the top level shared folder you want to acquire write access to in the Media panel.
2. From the contextual menu, select Acquire Write Access.

To disable write access:

1. Select the top level shared folder you want to acquire write access to in the Media panel.
2. From the contextual menu, select Release Write Access.

To enable or disable shared folders:

1. Select the top level shared folder you want to enable or disable in the Media panel.
2. From the contextual menu, select Enable or Disable.

To manually refresh the content of shared folders:

1. Select the top level shared folder you want to refresh in the Media panel.
2. From the contextual menu, select Refresh.
   The content of the shared folders is updated.

Transferring Material Between Projects

Elements you can transfer when browsing for Projects:

- Clips
- Folders
- Batch Snapshots
- Reels
- Desktop Snapshots

You can only transfer an item to the same type of location, so a Batch Snapshot to the Batch Snapshots section of the Media panel. One exception is Reels, which can be transferred to the Media Library; this makes it easier to transfer media from a Flame to a Smoke, since Smoke does not have Reels.

To transfer material between your project and one located on another workstation:

1. Select **MediaHub ➤ Browse for Projects**.
2. In the Autodesk Network list, double-click the workstation where the project is located.
   This displays the list of projects hosted by that workstation.
3. Double-click the project to open it.
   This opens the project and displays its Workspaces.
4. Navigate the project to find what you want to transfer.
5. Drag-and-drop from the Projects browser to the Media panel. Or from the Media panel to the Projects browser: you can import or export between projects.

To transfer material between local projects:

1. Select **MediaHub ➤ Browse for Projects**.
2. In the Local Projects list, double-click the project to open it.
   This opens the project and displays its Workspaces.
3. Navigate the project to find what you want to transfer.
4. Drag-and-drop from the Projects browser to the Media panel. Or from the Media panel to the Projects browser: you can import or export between projects.

NOTE Proxies are only transferred when both projects have identical proxy settings, as defined in the Projects settings window.

Troubleshooting

Read Only (In use by...)

Appears in one of two cases:

- When the workspace of the project you are browsing is opened and used by the user listed in the Read Only message.
- When someone else is browsing that workspace with the button Exclusive Write Access enabled.

You can still copy items from a Read Only workspace. If you absolutely need to write to the workspace, you will need to locate the user and ask him to switch to another project.

Owned by...

The workspace you are trying to write to is owned by another Flame, Smoke, or Flare. Enable Administrator mode to write to the workspace.

Unmanaged media contained in selection. Cache media on Wire?

When transferring media from a remote project, you can get the following message: "Unmanaged media contained in selection. Cache media on Wire?"

This means that the clip you are importing from the remote project has no media cached. Answer Yes to cache the media during the transfer of the clip. Answer No to only keep a reference to the media.

In fact, answering Yes is identical to importing media with the Cache Source Media enabled, and answering No is identical to importing media without Cache Source Media.

MediaHub Reference: Browsing for Projects

Exclusive Write Access button Enable to get exclusive write permission to the Workspace you are browsing, preventing others from modifying the contents of that Workspace. Other users can still read and copy clips of that Workspace.

Administrator Mode button Enable to modify workspaces created by others. You do not need to enable this option only to read clips from other projects.

This can be used by the Flame artist to clean up workspaces created by Flare collaborators.

Workstation Processing box Select which workstation processes the transferred media using to the Wire options set; has an impact on the overall performance of the selected workstation.

Background Wire button Enable to perform the Wire tasks in the background so that you can continue working in the application. Disable to perform the Wire tasks in the foreground, which is faster but prevents you from using the application.

Using Subtitles Files

With Flame Premium, you can load a subtitle file to quickly add subtitles to a sequence. This is the most effective method as it allows a third party to create subtitles without requiring access to a Flame Premium.

The format of this file is described in Subtitle XML Elements (page 144), with a sample provided in Subtitle XML Sample (page 148).

Other methods for subtitling include creating Text Timeline FX on the sequence's segments, or adding a new layer to create the subtitles in Gap effects.

Importing Subtitles to a Timeline Sequence

The procedure below adds subtitles from an imported xml file to an opened sequence.

1. From the MediaHub tab, import the subtitle file to the Media Library. There are no Specific Import Options to set.

   The subtitle XML file is imported as a single-track clip, where the subtitles are editable Text Timeline FX applied to Gap effects.

   NOTE The subtitle clip is renamed to the name defined by the `<name>` element in the file. It also takes the timing and resolution defined for the project.

2 From the Timeline tab, `right-click` the sequence that requires subtitles and select Open or Open as a Sequence.

3 Add a track to the opened sequence.

4 Drag and drop the subtitle clip to the new top track.

Each text element appears as a editable Text Timeline FX, at the timecode and length specified in the xml file.

TIP You can add multiple subtitle clips to a sequence, then mute all but the track you want to process. With this method you can quickly version a clip with different captions.

Defining Subtitles Looks Using Text Setup Files

A text setup file can be referenced from the subtitle XML file. The text setup file overrides the font style and position parameters indicated in the current `<title>` element.

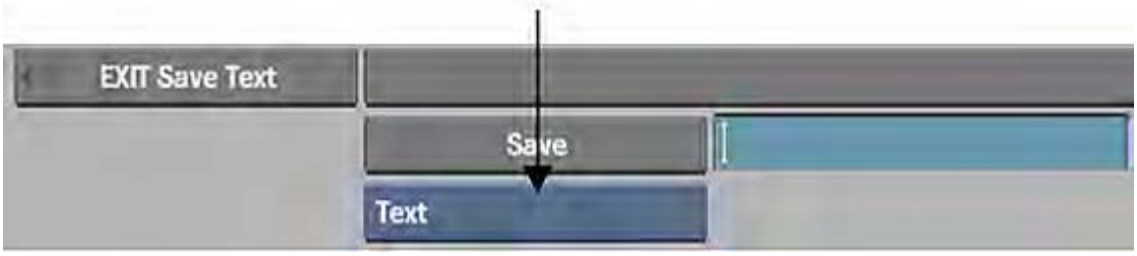
Using a text setup file allows you to apply your preferred text setup to the subtitles in the file, or even have a different text setup for each subtitle.

To create the text setup file:

1 In a Text tool, create a text setup.

2 Click Save.

3 In the Save menu, select Selected Layers from the Save box.



4 Choose the directory into which you want to save the file.

5 Type a name for the file and press `Enter` or click the Save button.

The file is saved and you return to the Text tool.

To associate the text setup file to each subtitle element:

1 Open the subtitle XML file in a text editor.

TIP Consider using a text editor with syntax coloring for XML elements. It makes for easier navigation of files.

2 Within each `<title>` element, indicate the path to the text setup file in the `<setup>` element.

The parameters in the text setup file override the following text parameters within the `<title>` element of the subtitle: font, size, font colour, vertical, and horizontal.

Once you import the subtitle file in Flame Premium, you can see the impact of the text setups on the imported subtitles. Changes to the text setup do not propagate to an imported subtitle file: if you modify the saved text setup after importing a subtitle file, you need to re-import that same file to update the subtitles referencing that setup.

Search for subtitles that are not properly linked to a setup file:

1. Place the positioner on the subtitle track.
2. Click the magnifying glass icon at the bottom right side of the timeline.

   The Find and Select in Timeline window appears.
3. Enable
   - Gaps
   - Every Criteria Below
   - Comments. Also enter in the Comments field *Unable* (make sure to enter the asterisk wildcards).
4. Click Select on Current Track.

   The Find and Select in Timeline windows closes, and subtitles not properly linked to a text setup are highlighted. You need to fix the subtitles file to use the correct text setup path, and then import the file again.

Subtitle XML Elements

Subtitle XML files are text files that use the following elements. Use the following syntax to work with Subtitle XML files external to the Flame Premium workstation.

In Flame Premium-compliant subtitle XML file, the <!DOCTYPE> is `subtitle`.

‹name›‹/name›

Description	An element that defines the name of the Subtitle XML file. This name appears as the name of the clip created when importing the subtitle XML file.
Attributes	none
Sample	`<name>French version: Day at the Races</name>`

‹rate›‹/rate›

Description	A format element that describes the framerate of the Subtitle XML clip that appears in Flame Premium after importing the Subtitle XML file. The frame rate must be that of the target sequence or the subtitles might slip timecode.
Attributes	none
Sample	`<rate>59.94 NDF</rate>`

‹resolution›‹/resolution›

Description	A unique element that wraps around all of the attributes determining the format of the clip when importing a Subtitle XML file. This clip can then be resized within Flame Premium like any other clip, if necessary. If any of these values differ from the target sequence, Flame Premium will apply Resize Timeline FX to the subtitles.

Mandatory children	**<width></width>** The width of the imported Subtitle XML clip. An integer. **<height></height>** The height of the imported Subtitle XML clip. An integer. **<depth></depth>** The bit depth of the imported Subtitle XML clip. **<aspect></aspect>** The aspect ratio of the imported Subtitle XML clip. **<scanformat></scanformat>** The scanformat of the imported Subtitle XML clip.

Sample	`<resolution>`
	`  <width>1920</width>`
	`  <height>1080</height>`
	`  <depth>8</depth>`
	`  <aspect>1.778</aspect>`
	`  <scanformat>default</scanformat>`
	`</resolution>`

‹timecode›‹/timecode›

Description	An element that defines the timecode in and out points during which the subtitles occur. All the titles in the Subtitle XML file will occur during the timecode indicated here, with each title also having its own start and end point indicated within the title elements.
Mandatory children	**<start></start>** The start timecode of the subtitle sequence. **<end></end>** The end timecode of the subtitle sequence.

Sample	`<timecode>`
	`<start>01:00:00;00</start>`
	`<end>01:00:08;00</end>`
	`</timecode>`

‹video›‹/video›

Description	The <video> element wraps around all of the <title> elements, which in turn contain the text and attributes of individual subtitles.
Mandatory children	**<title></title>** The <title> element wraps around each subtitle, with its associated start and end time, text, and font style and position parameters.

Sample	`<video>`
	`<title>`
	`  <start>01:00:01;00</start>`
	`  <end>01:00:02;00</end>`
	`  <text>This is a subtitle string</text>`
	`  <font>Garamond</font>`
	`  <size>20</size>`
	`  <fontcolor>`
	`  <alpha>255</alpha>`

```
        <red>255</red>
        <green>255</green>
        <blue>255</blue>
      </fontcolor>
      <vertical>0</vertical>
      <horizontal>0</horizontal>
     <setup>/usr/discreet/project/test/text/paz.ttg</setup>
   </title>
 </video>
```

<title></title>

Each <title></title> element defines a single subtitle which will appear as a Text Timeline FX applied to a gap once imported into Flame Premium, and added to a sequence.

The font style, colour, size, and position parameters are overridden if a valid text setup file is referenced within the <title> element.

<start></start>

Description	The start timecode of the current subtitle
Attributes	none
Sample	`<start>01:00:01;00</start>`

<end></end>

Description	The end timecode of the current subtitle
Attributes	none
Sample	`<end>01:00:02;00</end>`

<text></text>

Description	The text of the current subtitle To insert a carriage return, add <
> (without the brackets) to the text.
Attributes	none
Sample	`<text>This is a
subtitle string</text>`

****

Description	The font of the current subtitle. This value is overridden if a text setup file is referenced in the <setup> element.

Attributes	none
Sample	`<font>Garamond</font>`

<size></size>

Description	The font size of the current subtitle. This value is overridden if a text setup file is referenced in the <setup> element.
Attributes	none
Sample	`<size>20</size>`

<fontcolor></fontcolor>

Description	This element wraps around the font colour elements (<alpha>, <red>, <blue>, <green>) of the current subtitle. These values are overridden if a valid text setup file is referenced in the <setup> element.
Mandatory children	**<alpha></alpha>** The <alpha> colour component of the font of the current subtitle. **<red></red>** The <red> colour component of the font of the current subtitle. **<green></green>** The <green> colour component of the font of the current subtitle. **<blue></blue>** The <blue> colour component of the font of the current subtitle.

Sample	`<fontcolor>`
	`  <alpha>255</alpha>`
	`  <red>255</red>`
	`  <green>255</green>`
	`  <blue>255</blue>`
	`</fontcolor>`

<vertical></vertical>

Description	The vertical text position of the current subtitle. This value is overridden if a text setup file is referenced in the <setup> element.
Attributes	none
Sample	`<vertical>0</vertical>`

<horizontal></horizontal>

Description	The horizontal text position of the current subtitle. This value is overridden if a text setup file is referenced in the <setup> element.
Attributes	none
Sample	`<horizontal>0</horizontal>`

<setup></setup>

Description	The <setup> element can point to a text setup file from which font style and position parameters will be used. The parameters from the text setup file override the formatting elements of the current <title> element. Leave empty when not using a text setup file.
Attributes	none
Sample	`<setup>/usr/discreet/project/test/text/paz.ttg</setup>`

Subtitle XML Sample

Use the following XML sample as a starting point for your own subtitle XML files.

The following is an example of Subtitle XML which creates three Text soft effects applied to gaps at these timecodes:

- 01:00:01;00 until 01:00:02;00
- 01:00:03;00 until 01:00:04;00
- 01:00:05;00 until 01:00:06;02

```
<?xml version="1.0" encoding="UTF-8"?>
<!DOCTYPE subtitle>
<subtitle version="1">
  <name>SampleSubtitles</name>
  <!-- Rate should be one of : -->
  <!-- -->
  <!-- * '23.976'-->
  <!-- * '24'-->
  <!-- * '25'-->
  <!-- * '29.97 DF'-->
  <!-- * '29.97 NDF' -->
  <!-- * '30'-->
  <!-- * '59.94 DF'-->
  <!-- * '59.94 NDF' -->
  <rate>24</rate>
  <!-- Set width and height to default -->
  <!-- if you want project default res -->
  <!-- depth should be : -->
  <!-- -->
```

```
<!-- * '8', '10' or '12'.-->
<!-- * default : for project default depth -->
<!-- aspect should be : -->
<!---->
<!-- * '1.77777'-->
<!-- * '1.333'-->
<!-- * default-->
<!-- scanformat should be : -->
<!---->
<!-- * 'f1' -->
<!-- * 'f2' -->
<!-- * default-->
<resolution>
  <width>1920</width>
  <height>1080</height>
  <depth>8</depth>
  <aspect>1.778</aspect>
  <scanformat>default</scanformat>
</resolution>
<timecode>
  <start>01:00:00;00</start>
  <end>01:00:08;00</end>
</timecode>
<video>
  <title>
    <start>01:00:01;00</start>
    <end>01:00:02;00</end>
    <text>This is a subtitle string</text>
    <font>Garamond</font>
    <size>20</size>
    <fontcolor>
      <alpha>255</alpha>
      <red>255</red>
      <green>255</green>
      <blue>255</blue>
    </fontcolor>
    <vertical>0</vertical>
    <horizontal>0</horizontal>
    <setup>/usr/discreet/project/test/text/paz.ttg</setup>
  </title>
  <title>
    <start>01:00:03;00</start>
    <end>01:00:04;00</end>
    <text>This is another subtitle string</text>
    <font>Garamond</font>
    <size>20</size>
    <fontcolor>
      <alpha>255</alpha>
      <red>255</red>
      <green>255</green>
      <blue>255</blue>
    </fontcolor>
    <vertical>100</vertical>
    <horizontal>100</horizontal>
    <setup>/usr/discreet/project/test/text/paz.ttg</setup>
  </title>
```

```
    <title>
      <start>01:00:05;00</start>
      <end>01:00:06;02</end>
      <text>This is a third subtitle string</text>
      <font>Garamond</font>
      <size>20</size>
      <fontcolor>
        <alpha>255</alpha>
        <red>255</red>
        <green>255</green>
        <blue>255</blue>
      </fontcolor>
      <vertical>-100</vertical>
      <horizontal>-200</horizontal>
      <setup>/usr/discreet/project/test/text/paz.ttg</setup>
    </title>
  </video>
</subtitle>
```

Conform Tab Overview

Use the Conform tab to troubleshoot imported timelines and to relink segments to sources.

Event List Displays the events that make up the sequence displayed in the Timeline section. Clicking an event in the list also selects it in the Timeline. You can sort the list using by clicking the column headers. Linked events are usually displayed, but if you are only interested in unlinked events, you can hide the linked events from the Display Options box. If the Event list displays the message All Sources Linked, the selected sequence is already conformed.

Every column, other than Status, Matches, and Media, is a potential match criteria used to find the correct source from the Potential Matches, displayed at the bottom of the Media panel. The data displayed in the Event list is always one of the following colours:

Red This information is used as a match criteria, and the Potential Matches contain no source that matches this criteria.

Yellow This information is used as a match criteria, and the Potential Matches contain more than one source that matches this criteria.

White This information is used as a match criteria, and the Potential Matches contain only once source that matches this criteria.

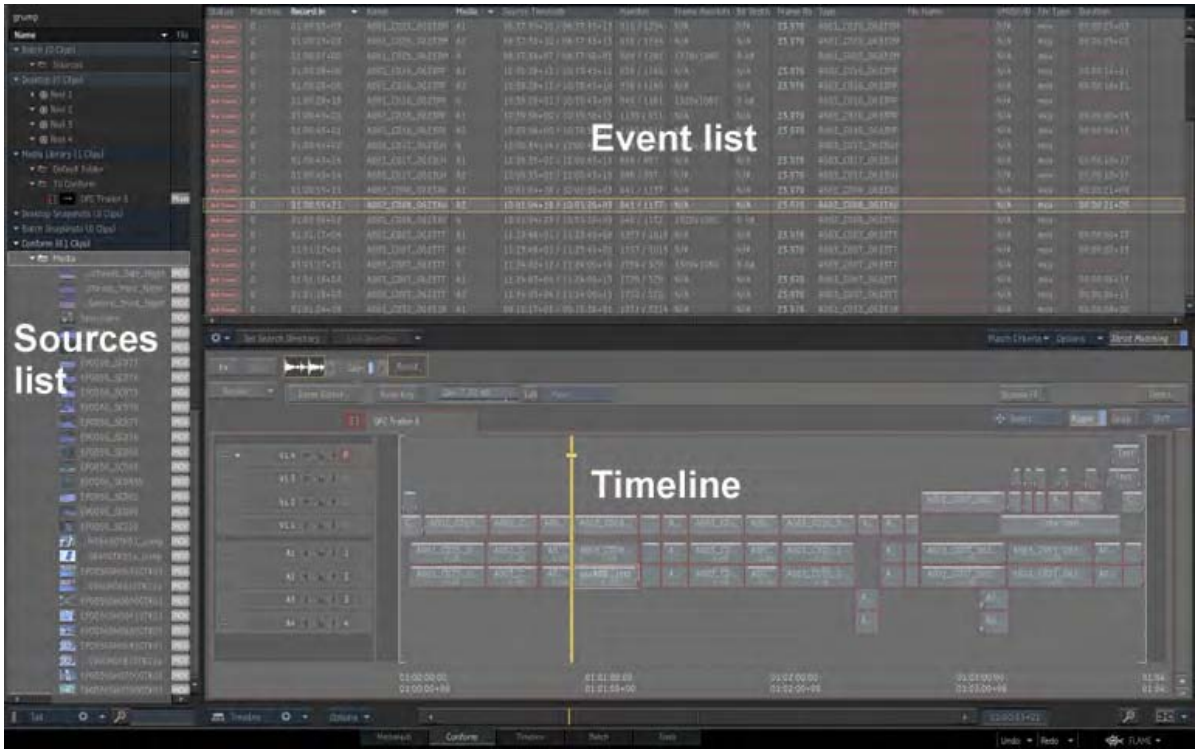
Grey This information is not used as a match criteria.

Potential Matches The Potential Matches display all the information about the sources found when using Set Search Directory or Set as Conform Search Location, from the contextual menu. From all the potential sources, Flame Premium uses the criteria you set using the Match Criteria box to filter in potential matches. From there, you can select the right source and link it to the selected event.

NOTE If you select an event with no matches, it shows the full list, in order to allow for force linking.

Disable Display Options > Filter Potential Matches to show all the sources.

Timeline The Timeline displays an open sequence, with controls similar to the ones found in the Timeline tab. Unlinked events are boxed in red.



Conform Gear menu Use to import an AAF, an XML, or an EDL. If connected to a VTR, select Capture... to send the selected events to the VTR queue and enter the VTR Recapture module.

Linking combo box Select Link Selected to link a selected source to the selected event. Select Link Matched Sources to relink every event marked as Match to its found source.

Match Criteria box Select from the list the criteria used to match sources to events. In the Event list, grey indicates unused criterion, white is matched criteria, yellow indicates multiple sources match the criteria, and red indicates no match was found in the Conform Media section.

Display Options box Select display options for Source and Event lists.

Strict Matching button Enable to force exact matching between source and event. When disabled, matching occurs when source and event have in common a string of at least 7 characters. Only applies to Name and File Name criteria.

Preview button Enable to display the Preview panel. The Source Player displays the source selected in the Conform Media section of the Media Panel.

Hide Linked button Enable to filter out, from the Event list, the events that are relinked.

Set Search Location button Displays a browser to locate the source files, to relink to the events from the Event list.

Adding Sources for Relink

TIP To use clips from the Workspace or Workspace Library as conform sources, drag-and-drop them to the Conform Media folder. Or right-click a folder, a library, or a reel and select Set as Conform Search Location.

To locate the directory that contains the sources:

1 In the Conform tab, click Set Search Directory.

2 Using the Set Directory window, select the directory where the sources are located.

3 Click Set.

Flame Premium clears previous sources from the Conform Media folder, and then displays all the media files from the selected location, including any media files located in sub-directories.

Notes

- Flame Premium only loads files it can use in a conform, so do not worry about non-media files being loaded to the Conform Media folder.
- Use the options found in the General and Format Specific Options tabs of the Set Directory window to modify the characteristics of the source files found: edit their resolution, bit depth, debayering settings, etc. Keep in mind that this will either facilitate or hamper the relink, depending on the settings and match criteria you set during the conform.
- When you load sources using the Set Search Location button, Flame Premium does not import the media files at that time: it creates only a list of references to the clips. This means that:
  - Even if proxies are enabled for the project, Flame Premium does not create proxies for media files loaded in the Conform Media folder. Proxies are only generated when you link an event to its source.
  - Flame Premium does not cache source media loaded to the Conform Media folder until you actually link an event to its source.

Conforming an EDL

To conform an EDL using file-based media:

1 In the Conform tab, right-click the Event list and select Load new EDL.

2 From the window that appears, locate and select the EDL to import, and then click Load.

The EDL is loaded and opened as a sequence in the timeline view of the Conform tab.

3 Click Set Search Location to locate the directory where Flame Premium can find the media files you want to use for sources.

You now have two lists: the Event lists, displaying all the events making up the sequence, and the Conform Media lists displaying all the source media Flame Premium found at the location you set using the Set Search Directory button.

4 With Options > Filter Potential Matches selected, click an event to view, in the Sources list, the sources that Flame Premium determined to be a potential match.

5 What you do next depends on the Status column of each event.

Match Select Link Found Sources from the Linking combo box. Link Found Sources links all events with the Match status to their unique sources. Match indicates that Flame Premium found only one source from the Sources list that fit the selected Match Criteria.

Multiple Matches Select the event and the source to relink, and then select Link Selected from the Linking combo box. If there are too many sources to choose from, you can set additional criteria using the Match Criteria drop-down menu; by trial and error you reduce the number of filtered sources. Make sure Options > Filter Potential Matches is enabled, to display only the filtered souces.

No Match Found Either add more sources to the Media folder or modify the criteria selected in the Match Criteria drop-down box. Not Found indicates that either the criteria are too restrictive for Flame Premium to find a match, or that the source is simply not there.

Unlinked Add sources to the Media folder. Unlinked indicates that the event is not linked to any source. It appears only when no sources are available in the Media folder, and the Conform section of the Media panel displays 0 clips available.

Linked Don't do anything: the event is already linked to an event. A Linked event only appears if Options > Hide Linked is not selected.

6 Once all the events are linked to sources, you can leave the Conform tab.

About the Load EDL Menu

Use the Load EDL menu to set a variety of options when loading EDLs.

Exit Load EDL button Exits the Load EDL menu.

EDL Type box Select an EDL type to load. Available options are: ALE, FLX, ATN, TLC, EDM, R23, ETL, and OMF.

File Extension field Displays the file extension for the associated file type. Click to enter a new one.

EDL Conversion button Enable to convert the EDL to another frame rate.

2:3 Removal Mode button Enable to remove pulldown when loading the EDL.

Varicam button Enable if the EDL is used to capture material shot with Varicam.

Fix TW Match Frames button Enable to fix match frame errors. When you import an EDL that contains dissolves or timewarps, a match frame error may occur. This can cause an unwanted cut at the point where the timewarp begins in your EDL. Match frame errors occur when the in point of the second edit in a dissolve is not the same timecode as the out point of the previous shot. This feature is enabled by default.

EDL Frame Rate box Displays the framerate of the current EDL. Click to select another one.

Conversion option box Select the option that corresponds to the type of conversion that you want to apply to the EDL. The conversion scripts that appear in this list depend on the frame rate of the EDL you are loading.

Frame Code Mode box Select the option (29.97 fps or 30 fps) that corresponds to the EDLs that you are loading. If you are loading multiple EDLs, the same Frame Code Mode is used for all of them.

Varicam Frame Rate box Select the Varicam frame rate.

Editing an EDL

Once you import an EDL, you can edit any value except the event number. This includes:

- The tape name for single or multiple events
- The audio patching information
- The transition duration and speed value of dissolves
- A cut to a dissolve or wipe
- The source and record timecodes of events

These tasks are described in the following sections. You can open multiple EDLs and copy and paste events between them. You can also use Auto Edit mode to quickly make changes to the entries you specify.

About the Import EDL Menu

Use the Import EDL menu to create, import, edit, auto-capture, assemble, and save EDLs.

EDL List

The EDL list is divided into 12 columns.

Event # The event number, followed by a capture indicator.

Tape The name of the tape containing the source clip.

A. Patch The Audio Patch information.

Tracks The track for the edit (shown as a combination of: V, 1, 2, 3, 4, 5, 6, 7, 8).

Transition The type of transition between the clips: C for cuts, D <duration> for dissolves, and W <wipe code> <duration> for SMPTE wipes.

Source In The starting timecode of the element in the source clip.

Source Out The ending timecode of the element in the source clip. (`Ctrl`-click the heading to view Source duration.)

Keycode In The starting keycode of the element in the source clip.

Keycode Out The ending keycode of the element in the source clip.

Speed The speed at which the source clip is timewarped in the edit. The value is preceded by a negative sign if the effect is a reverse. The column is blank if the speed value is 100% (no timewarp).

Record In The starting timecode of the element in the result clip.

Record Out The ending timecode of the element in the result clip. (`Ctrl`-click the heading to view Record duration.)

Operations Group

The Operations group contains the commands that allow you to import an EDL.



(a) File Name field

File Name box Select a previously loaded EDL file to display in the EDL work area.

Load button Opens the Load EDL menu where you set options for loading EDLs.

Save button Opens the Save EDL menu where you set options for saving EDLs.

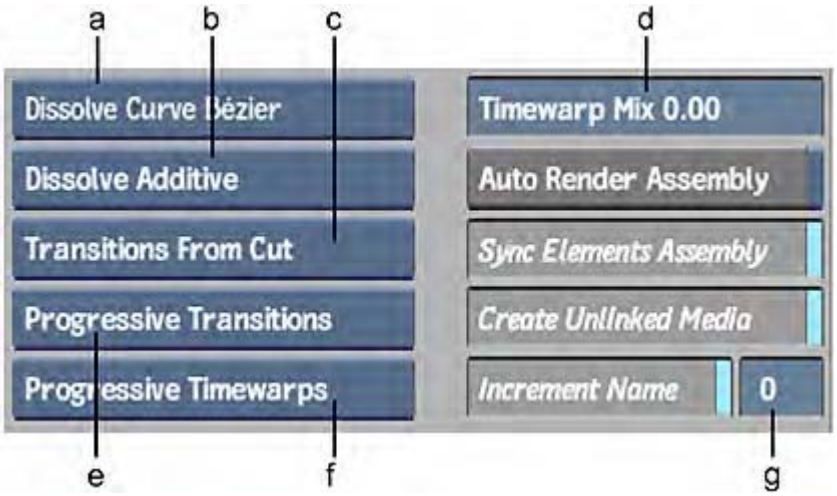
Capture button Opens the Auto-Capture menu where you capture media from a VTR using an EDL.

Close box Select Close to close the currently displayed EDL file, or Close All to close all EDL files.

Assemble box Select Assemble to combine the captured or imported material into a clip on a destination library reel. Select M-Assemble to combine multiple EDLs into a multitrack clip.

Assembly Options Menu

The Assembly Options menu contains options that you can set that affect EDL assembly.



(a) Dissolve Curves box (b) Dissolve Type box (c) Transitions Alignment box (d) Timewarp Mix field (e) Transition Render Option box (f) Timewarp Render Option box (g) Increment field

Dissolve Curves box Select the type of interpolation to use for all dissolves.

Select:	To:
Dissolve Curves Linear	Create dissolves with linear interpolation curves.
Dissolve Curves Bézier	Create dissolves with Bézier interpolation curves.

Dissolve Type box Select a rendering option for dissolves during EDL assembly.

Select:	To:
Dissolve Additive	Render dissolves as additive.
Dissolve Non Additive	Render dissolves as non additive.
Dissolve Inv Non Additive	Render dissolves as inverse non additive.

Transitions Alignment box Select an option for the placement of transitions in assembled EDLs.

Select:	To:
Transitions From Cut	Align transitions after the cut.
Transitions Centred	Centre transitions on the cut.
Transitions Up To Cut	Align transitions before the cut.

Transition Render Option box Select the rendering mode for transitions in assembled EDLs.

Select:	To:
Interlaced Transitions	Override the Preferences settings and render transitions in Interlaced mode.
Progressive Transitions	Override the Preferences settings and render transitions in Progressive mode.

Timewarp Render Option box Select an option for the rendering mode for timewarps in assembled EDL.

Select:	To:
Progressive TW	Render timewarps in Progressive mode.
Interlaced TW	Render timewarps in Interlaced mode.

Timewarp Mix field Enter the mix value for assembled timewarps. You can enter any value from 0.00 to 1000.00.

Auto Render Assembly button Enable to render transitions and timewarps when you assemble the final clip.

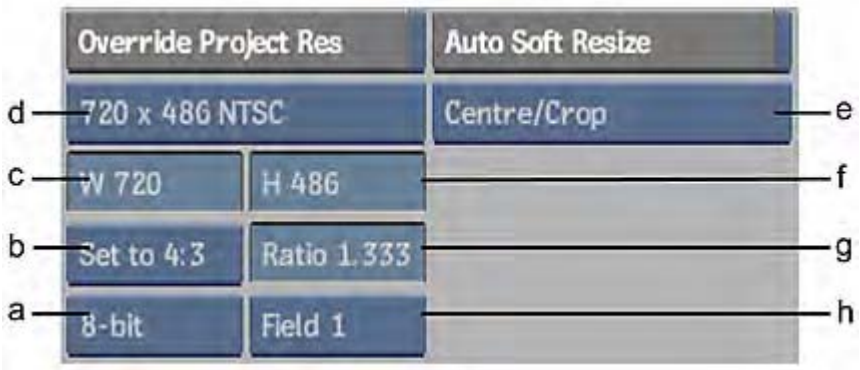
Sync Elements Assembly button Enable to automatically create Edit Sync groups for video and audio tracks that share the same timecode.

Create Unlinked Media button Enable to assemble an EDL, even if you do not have all the sources and, later, use the Recapture/Relink media feature to bring the missing media into your work-in-progress timeline clip.

When you disable this option, you get virtual sources with no indication of source timecode and tape name for the events when no recapture media exists for an event.

Increment Name button Enable to increment the assembled clip name (when assembling the same clip multiple times).

Increment field Enter a number by which the name is incremented. Active only when the Increment Name button is enabled.



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

Override Project Resolution button Enable to assemble pre-captured material of a different resolution than your project's default resolution. Additional controls become active that you use to specify the resolution options for assembly.

During assembly, the system searches the specified library reel for matching media. For media to match an event during EDL assembly, all the specified parameters must be the same (including frame rate, resolution, bit depth, and so on). When a match occurs for an event, the event is marked as captured.

By default, Override Project Resolution is disabled and the project's default resolution is used when matching EDL events to pre-captured media.

NOTE If an EDL refers to a clip that has the same tape name and timecode but is of a different resolution than the default resolution or that of the first matched event, the clip will not be marked as captured. For example, if you are assembling an NTSC EDL and the EDL refers to a captured HD clip with the same tape name and timecode, the HD clip will not be marked as captured.

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

Auto Soft Resize button Enable to apply a soft resize when assembling any clip that was captured at a resolution that differs from the native project resolution. Options become active that allow you to specify how the soft resize should be applied.

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

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

Frame Width field Displays the frame width of the selected resolution preset. If Resolution Presets is set to Custom then this field is active, allowing you to enter the frame width value that you want to use.

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

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

Frame Height field Displays the frame height of the selected resolution preset. If Resolution Presets is set to Custom then this field is active, allowing you to enter the frame height value that you want to use.

Aspect Ratio field Displays the aspect ratio of the imported clip. When Aspect Ratio Presets is set to Custom, this field becomes active so that you can enter a custom frame aspect ratio.

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

Select:	To:
Progressive	Scan a frame-based clip with no interlacing.
Field 1	Scan Field 1 first, followed by Field 2.
Field 2	Scan Field 2 first, followed by Field 1.

Preferences Menu

Use the options in the Preferences menu to specify your EDL preferences.



Beep on Assembly Complete button Enable to hear an audible tone when the EDL is assembled.

Allow Zero-Length Transitions button Enable to allow zero-length wipes and dissolves. If this option is disabled and you change a transition duration to zero, the transition automatically becomes a cut.

Keep Cursor In View Area button Enable to keep the cursor from leaving the window when you scroll an EDL. When you scroll an EDL, the cursor will stop at the first or last event on the EDL page.

Compare Tape Names Search button Disable to ignore tape names during assembly.

Clip Name Comments Capture button If your events contain clip names in their comments, enable this option to use those names during capture. If this option is disabled, the event number is used as the captured shot's name.

Use Hotkey For Add Selection button Use to change the functionality of selecting events. When this option is disabled, you add to your selection range by clicking the events. When this option is enabled, you must hold the `Ctrl` key or the pen button while clicking events to add them to your selection range.

Auto Disable Slip Tape button By default, the Slip Tape option remains enabled after you slip sources. Enable this option to disable the Slip Tape option automatically after you slip sources.

Clip Name Comments Assembly button If your events contain clip names in their comments, enable this option to use those names during assembly. If this option is disabled, the EDL title is used as the assembled clip's name.

Keep System Comments button Use the Keep System Comments option to protect system comments. When enabled, you can only delete user comments.



(a) Max TW Capture Handles field

Import As Soft Import button Enable to soft-import files with a file-based EDL that are located on a standard filesystem. If you disable this option, the files are hard-imported.

Import Background Proxies button Enable to allow proxies to be generated in the background.

Include VTR Preroll button Enable to capture the events in one pass and store them as individual clips in the clip library. By default, if events in an EDL are separated by a shorter duration than the VTR preroll and more than 10 frames, they are captured in a single pass.

Capture Proxy Only button Use this option in HD projects to auto-capture proxies only when conforming an EDL. This option does not appear in SD projects.

Max TW Capture Handles field You can set the maximum number of handles to be captured for source clips that are part of timewarped events. On timewarped events, the capture handles value is equal to the capture handles value multiplied by the speed of the timewarp. For example, for an EDL event with an increased speed of 500% (in NTSC), if the capture handles are set to 30 frames, 150 frames are captured as handles (30 frames multiplied by 5).

Standard handles may not be enough in the case of high-speed timewarps.

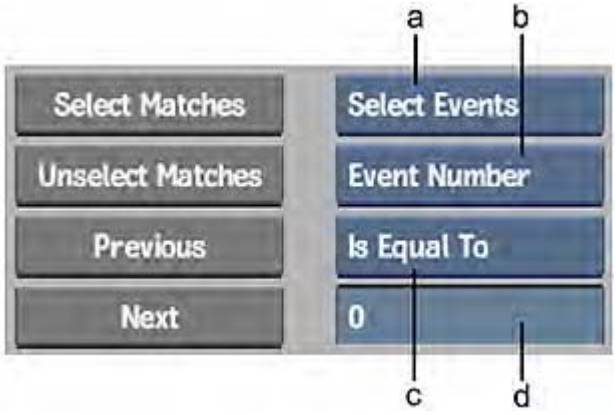
Capture Creates One Clip Per Event button Enable to create a clip in the clip library for every event in the EDL. When disabled, a single clip is created in the library for each group of events captured during the same pass. You can assemble the montage regardless of which option you chose to store EDL events in the clip library.

To reduce the size of clip libraries, you should disable this feature. However, if you need to reference back to individual shots, they may be easier to find if you enable the option and generate a clip for every event.

Auto Save After Each Tape button When this option is enabled, the system performs an AutoSave between each tape when auto-capturing.

Search Events Menu

Use the Search Events menu to search entries for text or timecode in an EDL. When you search the EDL, you need to specify the item you want to search, the filter type you want to use, and the criteria for which you want to search.



(a) Select Item box (b) Search Item box (c) Filter Type box (d) Search Criteria field

Select Matches button Highlight all items that match the search criteria.

Unselect Matches button Remove the highlight from all items that match the search criteria.

Select Item box Select the option that you want to search for.

Select:	To highlight:
Select Events	All events that match the search criteria.
Select Edits	All edits that match the search criteria.

Search Item box Specify the item that you want to search for in the EDL.

Select:	To search for:
Event Number	Event numbers.
Tape	Tape names.
Transition Length	A specific transition duration.
Source In	A source in timecode.
Source Out	A source out timecode.
Speed	A specific timewarp speed.
Record In	A record in timecode.
Record Out	A record out timecode.
Comment	A comment or text within a comment.
Track	A track.

Previous button Move the cursor to the previous item that matches the search criteria.

Next button Move the cursor to the next item that matches the search criteria.

Filter Type box Specify the filter type. The filter type will differ depending on the item you are searching. For numerical searches, select one of the following options from the Filter Type box.

Select:	To find:
Is Equal to	All event numbers that match your search criteria.
Is Not Equal to	All event numbers that do not match your search criteria.
Is Less Than	All event numbers that are less than your search criteria.
Is Greater Than	All event numbers that are greater than your search criteria.

For alphabetical searches, select one of the following options from the Filter Type box.

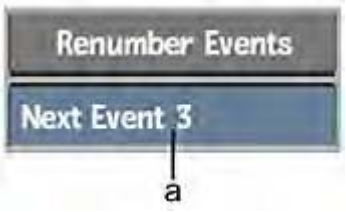
Select:	To search:
Matches	For all tapes that match your search criteria.
Does Not Match	For all tapes that do not match your search criteria.
Contains	For all tapes that contain your search criteria.
Does Not Contain	For all tapes that do not contain your search criteria.
Is Less Than	For all tapes lexicographically less than your search criteria.
Is Greater Than	For all tapes lexicographically greater than your search criteria.

Alphabetical searches are performed on Tape and Comment items by default. To perform a numeric search on these items, enable Treat As Numeric. To match results by case, enable Case Sensitive.

Search Criteria field Enter the search criteria.

Renumber Events Menu

Use the options in the Renumber Events menu to specify the value by which events are renumbered.



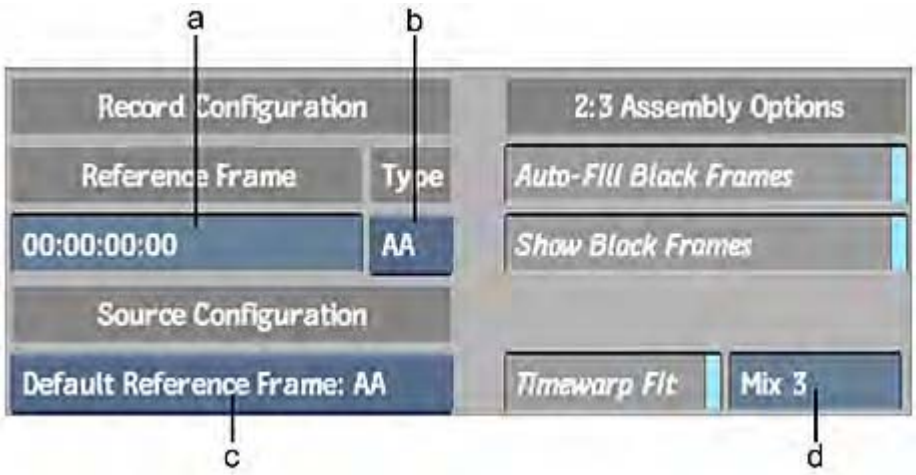
(a) Next Event field

Renumber Events button Renumbers the selected events according to the value specified in the Next Event field.

Next Event field Enter a value for the next event.

2:3 Pulldown Options Menu

Use the 2:3 Pulldown Options menu to set the parameters for 2:3 pulldown insertion upon import.



(a) Reference Frame field (b) Reference Frame Type box (c) Default Reference Frame box (d) Mix field

Reference Frame field Enter the timecode for the reference frame. This is only required for clips stored as files, and not for material on tapes.

Reference Frame Type box Set the reference frame type: AA or BB, corresponding to the timecode value entered in the Reference Frame field.

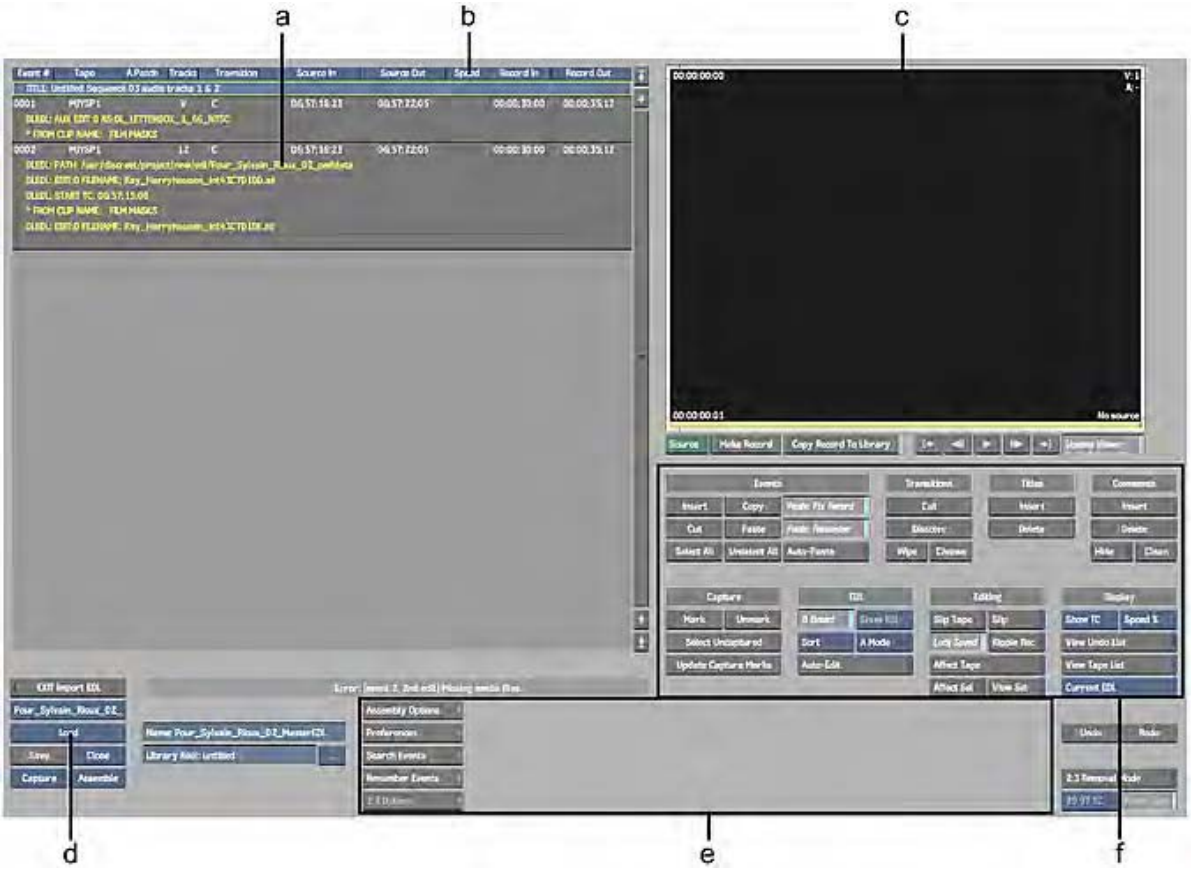
Auto-Fill Black Frames button Enable to fill black frames with the last frame of the outgoing event. By default, gaps occuring between EDL events are filled with black frames.

Default Reference Frame box Select AA or BB.

Show Black Frames button Enable to highlight the places in the currently loaded EDLs where gaps of black frames occur.

Timewarp Fit button Enable to automatically apply a timewarp to material on tapes referenced by EDLs where 2:3 pulldown removal was disabled (for example, when a tape contains native 30 fps interlaced video material), so that it can be easily integrated with 24 fps material.

Mix field Enter the number of frames to ensure that match frames on timewarped elements are maintained. This field is active when Timewarp Fit is enabled.



(a) EDL work area (b) Work area column headers (c) EDL Player (d) Load button (e) Import EDL menus (f) EDL editing tools

About EDL Editing Commands

Events Group

Use the commands in the Events group to cut or copy single or multiple events and paste them to a new location in any open EDL. You can also use special paste functions and selection tools for events.



Insert Event button Inserts a new event into the EDL.

Cut Event button Cuts the selected event. May be used with Paste Event button.

Copy Event button Copies the selected event. May be used with Paste Event button.

Paste Event button Pastes the previously cut or copied event to the selected location.

Paste: Fix Record button Enable to change Record In of the first pasted event to start at Record Out of the previous event. All pasted events are then rippled by the same amount.

Paste: Renumber button Enable to automatically renumber all events according to the Next Event value in the Renumber Events menu. If this option is disabled, the pasted events retain their original event numbers.

Select All button Click to select all events for capture.

Unselect All button Click to unselect all events.

Auto-Paste button Enable to quickly cut events from multiple EDLs and automatically paste them to a single EDL. For a selected EDL, when clicking the Paste button, any events that are cut or copied are automatically pasted to the first EDL.

Comments Group

You can add or delete comments from events using the commands in the Comments group of the EDL menu. Comments can contain up to 256 alphanumeric characters. You can also toggle comments on or off.

EDL comments become part of the source clip when assembling the EDL. Multiple comment lines are merged into one timeline comment line.



Insert Comment button Add a comment to the event selected in the EDL work area. You can add multiple comments to an event. To edit a comment, double-click it and enter the new comment.

Delete Comment button Delete a comment selected in the EDL work area.

Hide button When enabled, will hide all comments in the EDL work area.

Clean button Removes asterisks from comments.

Titles Group

You can add or delete a title from an EDL using commands in the Titles group of the Import EDL menu.



Insert Title button Enter a new title in the Titles area of the EDL work area. You can add multiple titles. To edit a title, double-click it and enter the new title.

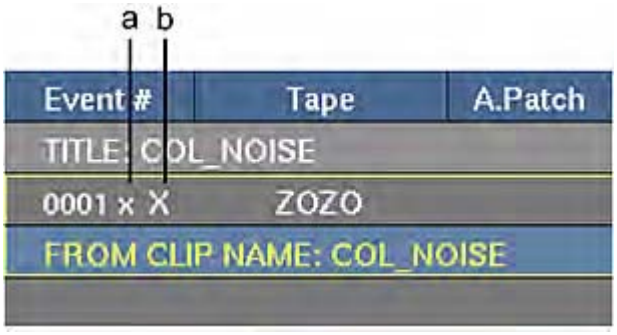
Delete Title button Delete a title in the EDL work area.

Capture Group

You can mark events for capture or select only the uncaptured ones for recapture.



Mark button Click to mark the selected events for recapture with an "X". This allows you to keep track of events that you want to recapture. Events that have already been captured are marked with a lower case x. Therefore, when you mark a captured event for recapture, a lower and upper case x appear (x X) to the right of the event number.



(a) Capture mark (b) Recapture mark

Unmark button Click to remove a recapture mark from the selected events.

Select Uncaptured button Click to select all uncaptured events in the EDL. The next time you click Auto-Capture, only the selected events are captured.

Update Capture Marks button Click to update the list of captured events.

Transitions Group

Use the commands in the Transitions group to change any transition to a cut, dissolve, or standard SMPTE wipe.



Cut button Changes the transition type to a cut for the selected transition, or range of transitions.

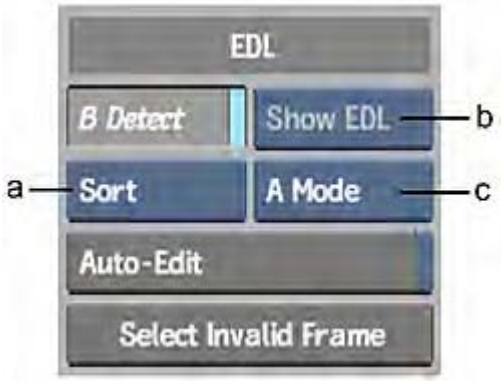
Dissolve button Changes the transition type to a dissolve for the selected transition, or range of transitions.

Wipe button Changes the transition type to a wipe for the selected transition, or range of transitions. The wipe type is set to SMPTE 001.

Choose button Opens the Choose Wipe menu from which you can select from a variety of standard SMPTE wipe types.

EDL Group

You can use any of these options in the EDL group when using EDLs to capture media.



(a) Sort box (b) Show Timecode box (c) Sort Mode box

BRoll Detect button Enable to detect BRolls. BRolls appear as the original tape. You are not prompted for the tape containing the BRoll when you capture the clips. When disabled, BRolls are indicated with a "B" following the tape name in the Tape entry.

Show Timecode box Select the type of timecode to display.

Select:	To:
Show EDL	Display EDL timecode.
Show VC	Display Varicam timecode.

Sort box Click Sort to sort the current EDL, or click Sort All to sort all the EDLs. EDLs are sorted according to the sort order specified in the Sort Mode box. You can change the way an EDL is sorted.

Auto-Edit button Enable to quickly modify only the entries in the selected columns of the EDL. Click the column headers of the columns that you want to auto-edit. Starting with the first entry that you edit, press `Enter` to move automatically to the next entry. If no further entries exist, a new event is added.

Sort Mode box Select the appropriate sort order.

Select:	To sort the EDL:
A Mode	By Record In timecode. If you want to view the EDL in the order of the final assembly, sort the EDL in A Mode.
B Mode	By Tape and Record In timecode.
C Mode	By tape number and Source In timecode. If you want to view the EDL in the order the clips are captured, sort the EDL in C Mode. EDLs are always captured in C Mode, regardless of the sort mode you selected.

Select:	To sort the EDL:
S Mode	By Source In timecode, regardless of tape number. This sort mode is useful for multi-camera real-time EDLs.
by Event#	By event number.
by Tracks	By tracks. In this sort mode, video tracks are placed at the bottom of the list, and the highest audio track is placed at the top of the list.
by Uncaptured	By placing all uncaptured events at the top of the list.

Select Invalid Frame button Click to select a frames of 0 length.

Editing Group

When you modify an entry in an EDL, several options control how other entries are affected. Make sure these options are either enabled or disabled, depending on how you want to edit the EDL.



Slip Tape button Enable to slip all source clips on the tape, or all record clips in the EDL list. All the in and out points of all source or record clips change without affecting their duration.

Lock Speed button Enable to edit the timecode without changing the speed value for the event.

Slip button Enable to slip a source or record clip in a single event. The in and out points of the selected clip change without affecting its duration.

Ripple Rec button Enable to move all Record entries, following an edited or pasted Record Out entry, forward or backward accordingly. This is useful when you want to edit the Record Out of an event without overwriting or creating a gap between the following shots.

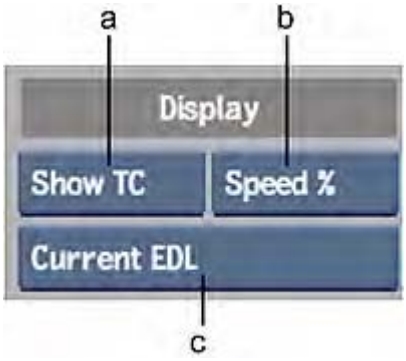
Affect Sel button Enable to affect only the selected events.

Affect Tape button Enable to affect every relevant entry on the same tape when you modify a single entry.

View Sel button Enable to view only the selected events.

Display Group

The Display group contains options for modifying the display of events in the EDL list.



(a) Show TC/KC box (b) Speed box (c) EDL Display box

Show TC/KC box Select whether to display keycode or timecode.

Select:	**To:**
Show TC	Display timecode.
Show KC	Display keycode.

Speed box Select a mode to display the speed change for timewarps.

Select:	**To:**
Speed %	Display the speed change as a percentage.
Speed FPS	Display the speed change in frames per second.

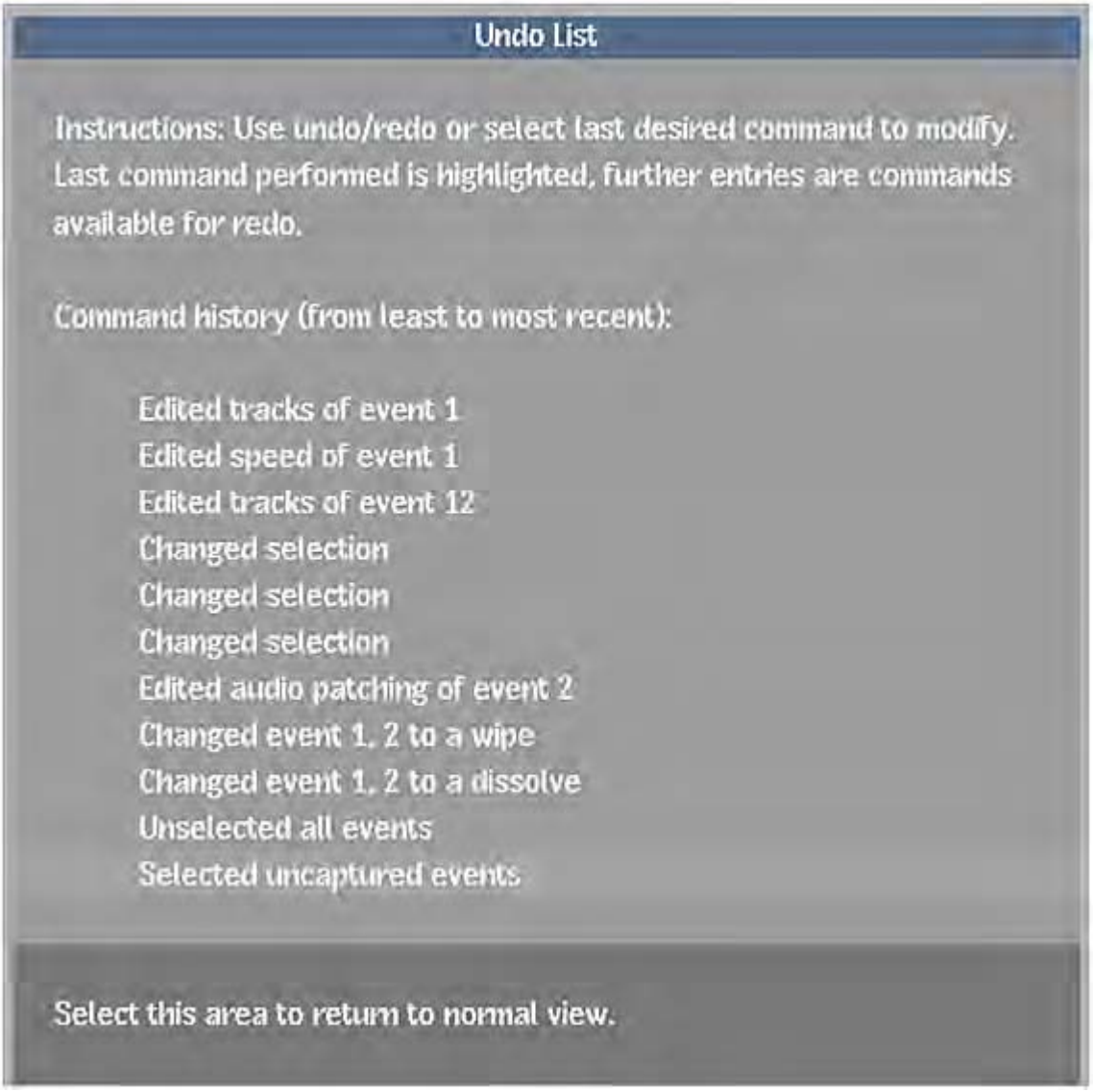
EDL Display box Select a display mode for EDLs, Tape List view, or Undo List view.

Select:	**To:**
EDL	Display timecode.
Undo List	View all modifications (up to the number of undo levels specified in the Preferences menu) that you made in the current session, and undo a single or series of commands. Click the item in the undo list that you want to undo. All operations are undone up to and including that modification. All operations prior to the highlighted modification are redone.
Tape List	Display the Tape List view, showing the last selected EDL.
Tape List (All EDLs)	Display the Tape List view, showing all selected EDLs.

Undo List View

Use the Undo List to view all modifications (up to the number of undo levels specified in the Preferences menu)that you made in the current session and undo a single or series of commands.

Click the item in the Undo List that you want to undo. All operations are undone up to and including that modification. All operations prior to the highlighted modification are redone.

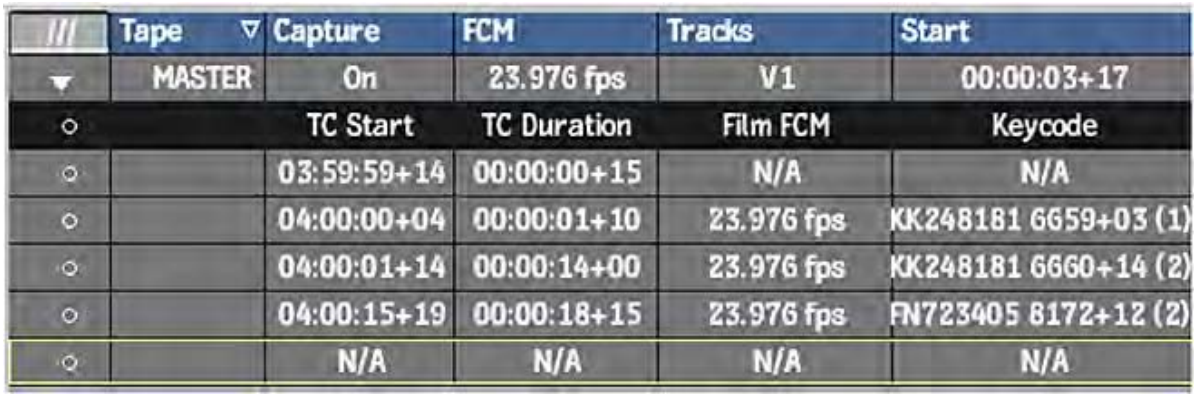


Tape List View

You can use the tape list to modify information pertaining to events for a given tape.

As well, each tape's entry can be expanded in Tape List view to show keycode and 2:3 pulldown information associated with the source timecode.

Click the column header to sort the tape list in descending order using that column. Click the column heading again to sort the tape list in ascending order.



///	Tape	Capture	FCM	Tracks	Start
▼	MASTER	On	23.976 fps	V1	00:00:03+17
○		TC Start	TC Duration	Film FCM	Keycode
○		03:59:59+14	00:00:00+15	N/A	N/A
○		04:00:00+04	00:00:01+10	23.976 fps	KK248181 6659+03 (1)
○		04:00:01+14	00:00:14+00	23.976 fps	KK248181 6660+14 (2)
○		04:00:15+19	00:00:18+15	23.976 fps	FN723405 8172+12 (2)
○		N/A	N/A	N/A	N/A

The work area of the Tape List view is divided into nine columns, and two sections for each tape.

The upper fields of each tape entry contain the following columns.

Tape The name of the tape that contains the source footage. To change the tape name, click and enter a new name.

Capture The capture flag for the tape. When set to On, the tape is captured during an auto-capture session. To set the capture flag for a given tape to On or Off, click the item under the Capture column and drag left or right to set it On or Off.

FCM The frame code mode for the tape. If the frame code mode is 29.97, you can drag over the item in this column to switch between DF/NDF modes.

Tracks The number and type of tracks that will be captured for the tape.

Start The starting timecode for the first event to be captured on the given tape.

Capture Time The duration of the material to capture for the given tape.

Telecine Indicates the telecine log associated with the tape.

Final Telecine Indicates the final telecine log associated with the tape.

Events The number of events to capture for the given tape. You cannot modify this value in Tape List view.

The lower fields of each tape entry contain the following columns.

TC Start Start timecode for each keycode sequence.

TC Duration Duration of time for the keycode sequence.

Film FCM Film frame code mode indicating the speed of the telecine when it scanned the film frames. Change the film FCM by clicking the Keycode field to access the calculator.

Keycode Keycode for the first frame of the sequence.

Removal Whether to apply 2:3 pulldown to the sequence.

Ref Frame Timecode of the reference frame.

Ref Type Type of reference frame.

DF Ref Frame Timecode of the drop frame reference frame.

EDL View

An EDL contains events, edits, and entries. An event is a complete element in the EDL such as a shot, dissolve, or wipe. An edit is an individual component that makes up an event, such as an outgoing shot or an incoming shot. An entry is an individual value for any variable in the edit, such as dissolve length, Source In, or Speed.



(a) Edits (b) EDL events (c) EDL entries

Every event has a different event number. Some events, such as dissolves and wipes, consist of two edits. The two edits in a dissolve (outgoing and incoming shots) have the same event number.

Editing a Tape Name

You can change a single event's tape name, or the tape name for all events on the same tape. This is useful if the events you want to capture are on a different tape than that listed in the EDL.

Tape names can be up to 56 characters long. The EDL Editor displays only the first eight characters, but the clips maintain the original tape name.

To edit a tape name:

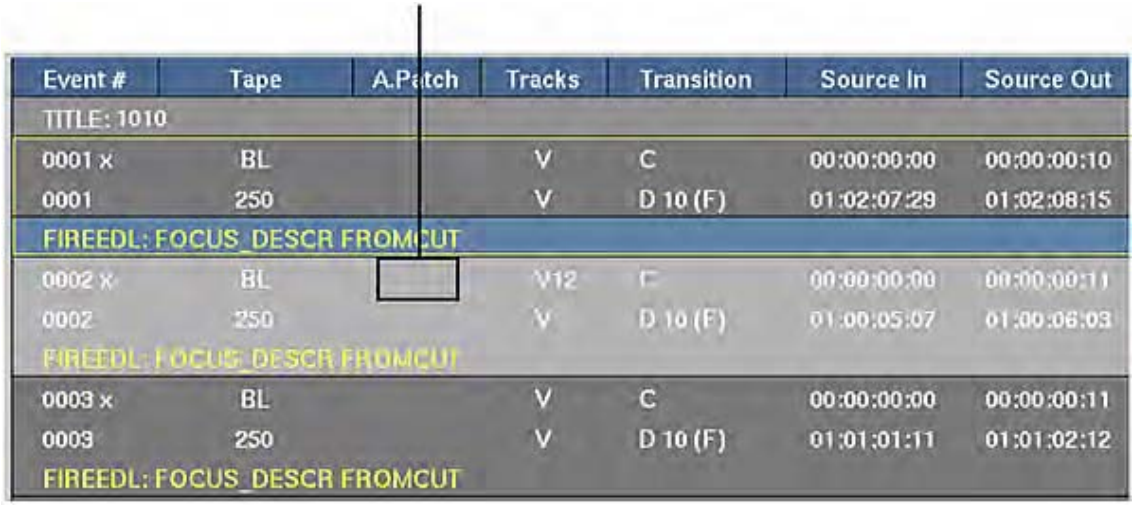
1. In the Import EDL menu, do one of the following:
   - To change the tape name for all events on the same tape, enable Affect Tape.
   - To change only the selected tape name, disable Affect Tape.
2. Click the Tape entry in the event you want to edit and enter the new name:
   - If the Affect Tape option is enabled, the tape name for all events on the same tape changes automatically.
   - If the Affect Tape option is disabled, only the tape name for the selected event changes.
   - If the Affect Selection option is enabled, only the tape names for selected events change.

Audio Patching

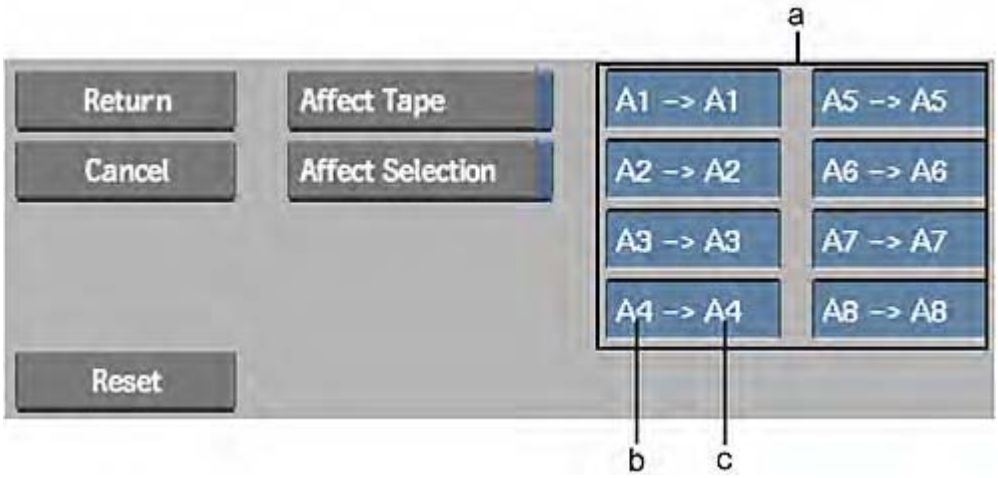
If you want to capture audio from a different track than that specified in the EDL, you can change the audio patching for a single event, selected events, or all events on the same tape. For example, if an edit specifies the audio tracks as 1 and 2, but you want to capture the audio from tracks 3 and 4, use the Audio Patching menu to reroute the audio during auto-capture.

To change audio patching:

1. In the EDL work area, select the events that you want to modify.
2. Click an Audio Patch field.



The Audio Patching menu appears.



(a) Audio Patch boxes (b) Source audio track (c) Destination audio track

You can edit the source audio tracks. Each one of the eight available source audio tracks can be patched to one of the eight available destination audio tracks. You can patch the same source audio track to multiple destination audio tracks.

3 Enable or disable the Affect Tape and Affect Selection buttons.

4 Reroute the audio in the Audio Patch boxes by doing one of the following:

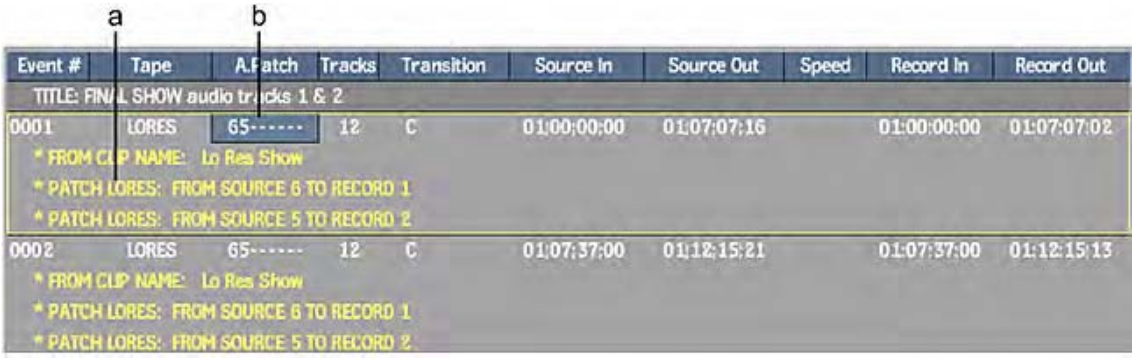
- Drag in an Audio Patch box to cycle through source audio tracks one through eight.
- Click an Audio Patch box to access the calculator, then type a value one through eight indicating the source audio track.

5 Click Return.

To exit the Audio Patching menu without changing the patching information, click Cancel.

The source tracks that have been patched to different audio tracks than the defaults are indicated in the affected event's Audio Patch Field. Dashes in the Audio Patch Field indicate the track is patched to its default. Comments are also added which indicate the name of audio tracks and their patching information.

When clips are captured, audio tracks are patched as indicated.



(a) Comments indicate patching information (b) Source 1 and 2 patched to destinations 5 and 6

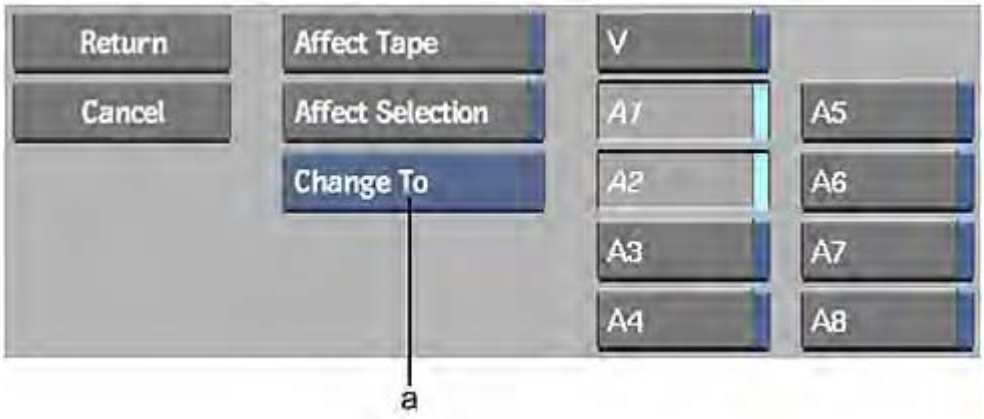
Editing Tracks

You can edit the tracks of an event in the EDL. Editing tracks is useful if you want to modify the tracks captured when you auto-capture or assemble the EDL.

For example, if the Track entry for an event is V1 and you want to capture only the video track (V), change the Track entry to V.

To edit the tracks for an event:

1 In the EDL work area, select the Track entries in the event that you want to edit from the EDL.
The Track menu appears.



(a) Track Edit Mode box

2 Enable the tracks that you want to edit.

3 Enable or disable the Affect Tape and Affect Selection buttons.

4 From the Track Edit Mode box, select the mode that you want to use.

Select:	To:
Change To	Replace tracks in the event with tracks you specified in the Track menu.
Add	Add the tracks you specified in the Track menu to the tracks in the event.

Select:	To:
Filter	Keep only the tracks you specified in the Track menu. For example, if an event is "V12" and you specify "V1," audio track 2 is filtered out, and the resulting event has "V1."
Flip	Reverse the status of tracks you specified in the Track menu. For example, if an event is "V23" and you specify "V34," the resulting event has "V24."

5 Click Return to confirm the change or Cancel to exit back to the EDL module without making any modifications.

Editing Transitions in EDLs

You can change any transition to a cut, dissolve, or SMPTE wipe. If the effect of an event is a dissolve, you can edit its duration (Transition Duration). If the effect of an event is a wipe, you can also select the type of wipe in the SMPTE Wipe library.

You can add or modify transitions based on a selection range.

You can also change a cut to a dissolve by clicking the Transition entry of the cut. You can change a dissolve to a cut by setting the Transition Duration to 0, as long as the Allow Zero-Length Transitions option is disabled in the EDL preferences.

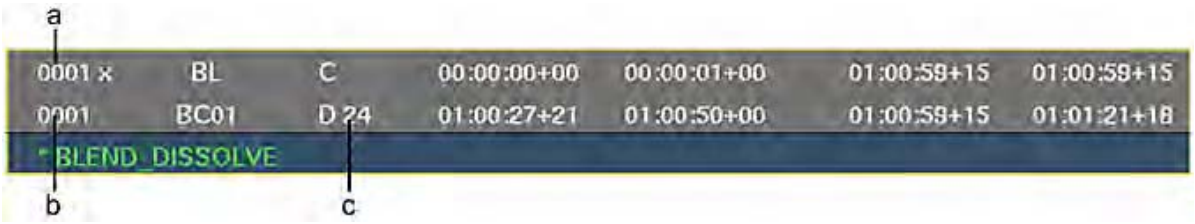
Dissolves in EDLs

Dissolves are represented in the EDL by two consecutive edits with the same event number. The first edit represents the outgoing shot of the dissolve and the second edit represents the incoming shot of the dissolve. The dissolve is listed in the Transition column of the second edit as D *n* (where *n* is the transition duration).

The Transition duration appears in the Transition entry of the second edit in the dissolve. The Start Location for the dissolve is also indicated in the Transition entry.

If the EDL was exported from Flame Premium, the dissolve can be Centred, From Cut, or Up To Cut. The start location for the dissolve appears beside the Transition duration.

The following illustration is a typical dissolve in an EDL.



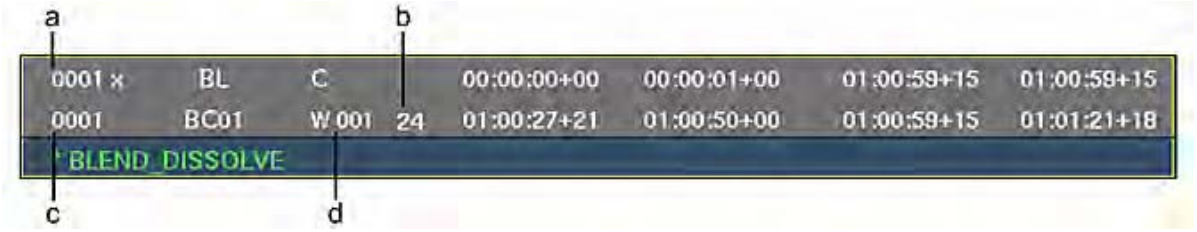
(a) Outgoing shot (b) Incoming shot (c) Transition duration

If you change the Transition duration of a dissolve, the Source Out of the incoming shot changes by the same duration automatically.

Wipes in EDLs

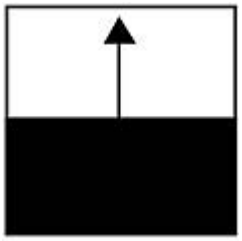
Wipes are represented in the EDL by two consecutive edits with the same event number. The first edit represents the outgoing shot of the wipe and the second edit represents the incoming shot of the wipe. The wipe is listed in the Transition column of the second edit as W *n m* (where *n* is the wipe code and *m* is the wipe duration).

The SMPTE wipe number appears in the Transition entry of the second edit in the wipe. The Start Location for the wipe is also indicated in the Transition entry. A wipe can be Centred, From Cut, or Up To Cut. The start location for the wipe appears beside the Transition duration. The following illustration is a typical wipe in an EDL.

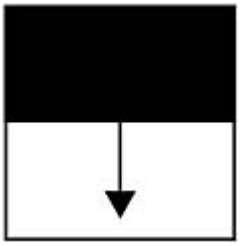


(a) Outgoing shot (b) Transition duration (c) Incoming shot (d) SMPTE wipe number

If you add 500 to the SMPTE wipe number, the wipe is inverted during assembly. For example:



Wipe 002



Wipe 502

If you change the Transition duration of a wipe, the Source Out of the incoming shot automatically changes by the same duration.

Editing Timecode

If you want to capture different material than what appears in the EDL, you can slip or trim the edits to specify the correct material for your final assembly. You can change any source or record timecode in the EDL.

You can also edit timecode and keycode in the Tape List.

To edit timecode:

1. If you want to edit source timecode, and keycode in and out values are displayed, toggle the Show Timecode/Show Keycode box to Timecode.
2. Do one of the following:
   - Click the source or record timecode entry you want to modify and drag right to increase the value or drag left to decrease the value. You can use hotkeys to change the values in varying increments.

Use:	To:
`Shift`-drag	Jump by seconds.
`Ctrl`-drag	Jump by minutes.

Use:	To:
Alt-drag	Jump by hours.

- Double-click the source or record timecode you want to modify to get the calculator, and then enter a new timecode. If the source or record timecode is already selected, click it again to get the calculator.

When you edit source timecode, the corresponding keycode changes to reflect the new frames to be captured. Toggle the Show Timecode/Show Keycode button to see the edited keycode in and out values. You cannot edit the keycode for events in the EDL work area.

NOTE Other entries in the same edit affected by your modifications become highlighted as you edit an entry.

Editing Tape List Timecode and Keycode

In Tape List view, you can slip all the start timecode values for a given tape. This is useful when you need to modify the start timecode of a tape that is used in multiple EDLs, because you edit the value only once.

You can also slip keycode and its associated start timecode and duration. This is similarly useful to alter keycode values correlated to the timecode for given sequences on the tapes.

To slip the start timecode value for a given tape:

1. In the Import EDL menu, from the Display group, enable View Tape List.
2. Do one of the following:
   - Click the start timecode entry you want to modify and drag right to increase the value or drag left to decrease the value. You can use hotkeys to change the values in varying increments.

Use:	To:
Shift-drag	Jump by seconds.
Ctrl-drag	Jump by minutes.
Alt-drag	Jump by hours.

   - Click the start timecode entry you want to modify to get the calculator, then enter a new timecode. See inferno flamesmoke.

To slip the keycode for a given tape:

1. In the Import EDL menu, from the Display group, enable View Tape List.
2. If necessary, click the triangle on the left of the tape entry to expand its contents.



Sequences of timecode on the tape are associated to keycode values. If no keycode is available from the telecine log(s) you loaded, N/A (not available) appears for the sequence.



(a) Timecode start and duration defines sequence, with corresponding keycode (b) Timecode without corresponding keycode

3 To edit the keycode for a sequence, do one of the following:

- Drag the keycode entry you want to modify to the right to increase the value or to the left to decrease the value.
- Click the keycode entry you want to modify to get the keycode calculator, and then enter a new keycode.

  The keycode values associated to the timecode sequence (defined by the values in TC Start and TC Duration fields) are altered.

  If you change the frame rate in the keycode calculator, it is reflected in the Film FCM field.



Editing and Creating Timewarps

You can edit an existing timewarp or create a timewarp in your EDL. When you edit a timecode entry of an event with the Lock Speed option disabled, no other entries are affected. This results in a speed change because the source duration differs from the record duration.

For example, if you increase the Source In entry of an event by 5 seconds, the record clip of the same event is 5 seconds shorter than the source clip. The result clip is timewarped when assembled. No other entries are affected.

Timewarps are represented in the Speed column of the EDL. If the Speed value is blank, the event is not timewarped. If there is a Speed value, the event is timewarped.



(a) Speed value

To edit or create a timewarp:

1 In the Import EDL menu, from the Editing group, disable Lock Speed.

2 In the EDL, click the source or record timecode entry for the event that you want to edit and enter the new value.

The value of the timewarp appears in the Speed entry. You may also click directly in the Speed box to change the entry. If the event is already a timewarp, editing any timecode of that event changes its speed value.

You can view the speed value in either percentage or frames per second. Select the View mode in the EDL Preferences menu.

Conforming an EDL with VTR Recapture

NOTE The following requires that the workstation is connected to a VTR.

1. In the Conform tab, right-click the Event list and select Load New EDL.
2. From the window that appears, locate and select the EDL to import, and then click Load.

   The EDL is loaded and opened as a sequence in the timeline view of the Conform tab.
3. Right-click the event list, and select Capture All From VTR.

   The VTR module opens and cues the VTR.
4. Proceed with the recapture of the cued events.
5. Once back to the Conform tab, continue with the conform process as you would with a file-based sequence.

Conforming an Avid AAF

1. In the Conform tab, right-click the Event list and select Load new FCP XML/AAF.

   Or if you already imported the AAF using the MediaHub, open it on the Timeline and switch to the Conform tab.
2. From the window that appears, locate and select the AAF to import, and then click Load.

   NOTE When conforming file based media, Flame Premium automatically searches for and links the matching media. The AAF is then loaded and opened as a sequence in the timeline view of the Conform tab, whether or not all your sources are fuond and relinked. This is the default behavior of the application. You can change this behavior by disabling Link to Media Files and Search and Import Files the in the Sequence Import Options, from the Media Import window. If Flame Premium does not find all of your associated media, proceed as follows.
3. Click Set Search Directory to locate the directory where Flame Premium can find the media files you want to use for sources.

   You now have two lists: the Event lists, displaying all the events making up the sequence, and the Sources lists, displaying all the source media Flame Premium found at the location you set using the Set Search Directory button.
4. With Options > Filter Potential Matches selected, click an event to view, in the Sources list, the sources that Flame Premium determined to be a potential match.
5. What you do next depends on the Status column of each event.

   Match Select Link Found Sources from the Linking combo box. Link Found Sources links all events with the Match status to their unique sources. Match indicates that Flame Premium found only one source from the Sources list that fit the selected Match Criteria.

   Multiple Matches Select the event and the source to relink, and then select Link Selected from the Linking combo box. If there are too many sources to choose from, you can set additional criteria using the Match Criteria drop-down menu; by trial and error you reduce the number of filtered sources. Make sure Options > Filter Potential Matches is enabled, to display only the filtered souces.

No Match Found Either add more sources to the Media folder or modify the criteria selected in the Match Criteria drop-down box. Not Found indicates that either the criteria are too restrictive for Flame Premium to find a match, or that the source is simply not there.

Unlinked Add sources to the Media folder. Unlinked indicates that the event is not linked to any source. It appears only when no sources are available in the Media folder, and the Conform section of the Media panel displays 0 clips available.

Linked Don't do anything: the event is already linked to an event. A Linked event only appears if Options > Hide Linked is not selected.

6 Once there all the events are linked to sources, you can leave the Conform tab.

Conforming an Final Cut Pro XML

1 In the Conform tab, right-click the Event list and select Load new FCP XML/AAF.

Or if you already imported the XML using the MediaHub, open it on the Timeline and switch to the Conform tab.

2 From the window that appears, locate and select the XML to import, and then click Load.

The XML is loaded and opened as a sequence in the timeline view of the Conform tab.

3 Click Set Search Directory to locate the directory where Flame Premium can find the media files you want to use for sources.

You now have two lists: the Event lists, displaying all the events making up the sequence, and the Sources lists, displaying all the source media Flame Premium found at the location you set using the Set Search Directory button.

4 With Options > Filter Potential Matches selected, click an event to view, in the Sources list, the sources that Flame Premium determined to be a potential match.

5 What you do next depends on the Status column of each event.

Match Select Link Found Sources from the Linking combo box. Link Found Sources links all events with the Match status to their unique sources. Match indicates that Flame Premium found only one source from the Sources list that fit the selected Match Criteria.

Multiple Matches Select the event and the source to relink, and then select Link Selected from the Linking combo box. If there are too many sources to choose from, you can set additional criteria using the Match Criteria drop-down menu; by trial and error you reduce the number of filtered sources. Make sure Options > Filter Potential Matches is enabled, to display only the filtered souces.

No Match Found Either add more sources to the Media folder or modify the criteria selected in the Match Criteria drop-down box. Not Found indicates that either the criteria are too restrictive for Flame Premium to find a match, or that the source is simply not there.

Unlinked Add sources to the Media folder. Unlinked indicates that the event is not linked to any source. It appears only when no sources are available in the Media folder, and the Conform section of the Media panel displays 0 clips available.

Linked Don't do anything: the event is already linked to an event. A Linked event only appears if Options > Hide Linked is not selected.

6 Once there all the events are linked to sources, you can leave the Conform tab.

Clip Input/Output Using a VTR

Flame Premium allows you to perform numerous VTR-based clip input and output operations. You can capture individual clips or frames, or log clips for capture using an EDL. Similarly, the application allows you to output single clips or frames, as well as multiple clips to a VTR device.

Clip input and output general workflow:

1. Make sure all hardware devices involved in the clip input and output process are properly configured. If not, the VTR Input and VTR Output menu options are disabled.
2. Edit the software initialization configuration file to ensure the proper devices and settings are initialized on application start-up.
3. Do one of the following:
   - Input clips.
   - Output clips.

If you are using an NVIDIA graphics card with and SDI daughter card, Flame Premium also supports real-time operations for some deliverables.

Configuring Hardware For Clip Input and Output Using a VTR

When preparing for a VTR session, confirm the following:

- Audio and video outputs of the VTR are connected to the audio and video inputs of the Flame Premium system. Audio and video inputs of the VTR are connected to the audio and video outputs of the Flame Premium system.
- The VTR is connected, using an RS-422 video I/O control cable, to a serial port of your workstation to enable remote control of the device.
- A video sync signal is connected to the sync input of the video device to ensure frame-accurate capture. If there is a separate audio device, an audio sync signal must be connected to it as well.

Configuring Software For Clip Input and Output Using a VTR

Software configuration in preparation for clip input and output session involves editing the software initialisation configuration file.

When you are preparing for a clip input and output session, consult the software initialization configuration file to confirm the following keyword sections are uncommented and properly configured:

- VIDEO KEYWORD

  You do not need to specify the default video I/O timing. The initial timing corresponds to the default resolution and frame rate of the project that you select on start-up. If you select a film or custom resolution for your project on start-up, video I/O timing corresponds to the resolution that was selected as the preferred resolution when the application was installed.

- VTR KEYWORD

  You must assign a specific VTR timing, such as NTSC, PAL, or 1920x1080_5994i. Multi-format video input and output capability of your system allows you to modify the VTR timing, without exiting the project, to any timing supported by your system's video board.

  If you plan to perform input and output operations using a live video signal from a broadcast feed or from a device that does not support remote control functionality using RS-422 interface, such as a VCR or camcorder, uncomment the corresponding Live video keyword at the bottom of the SD DECKS section.

NOTE When working in HD projects and capturing material from a Varicam (variable frame rate 720p material), ensure that the following VTR are enabled: *VTR DVCProHD 720 59p, SERIAL1, 1280x720_5994P* and *VTR DVCProHD 720 60p, SERIAL1, 1280x720_60p*.

Inputting Clips From a VTR

Once you have set up your hardware devices and configured the software initialization file, you are ready to perform clip input and output. The VTR Input module contains various controls to help you perform operations, such as capturing a single frame, capturing a single clip, capturing and logging multiple clips.

Accessing the VTR Input Module

1. Select a folder from the Media Library. This is where the captured clips will be created.
2. Open the Tools tab.
3. In the Utilities tab, select Clip VTR Input.

   The VTR Input module appears.

If VTR Input is not available, connect your workstation to a VTR, and configure Flame Premium to use the VTR.

Capturing Single Clips

You can capture a single clip from a VTR. You can capture on the fly with your in and out points determined by mouse clicks during playback, or be more precise by setting in and out points in the corresponding fields.

To capture a single clip:

1. Cue up the tape to the frame at which you want to begin capturing.
2. Do one of the following:
   - From the Capture Method box, select Start On Pen/Stop On Pen.
   - Enter clip input in and out points in the In and Out fields. See Setting Input and Output In and Out Points (page 199).
3. To begin capturing, click Process.

   The timecode field turns green, indicating that capture is in progress. Depending on your hardware configuration, the preview window may go black during capture.
4. To end the capture at any time, click the cursor anywhere over the preview window.

   Depending on your project proxy settings, a post-process may occur, generating proxies for each captured frame. You can see a notification on the process that is taking place in the message bar.

   Once all capture-related processes are complete, the clip is saved to the Workspace location that you selected before you entered the VTR Input module.

Capturing Single Frames

A single frame capture, or a frame grab, is the simplest clip input operation. This is useful for extracting snapshot frames for a preview or promotion piece.

To capture a single frame:

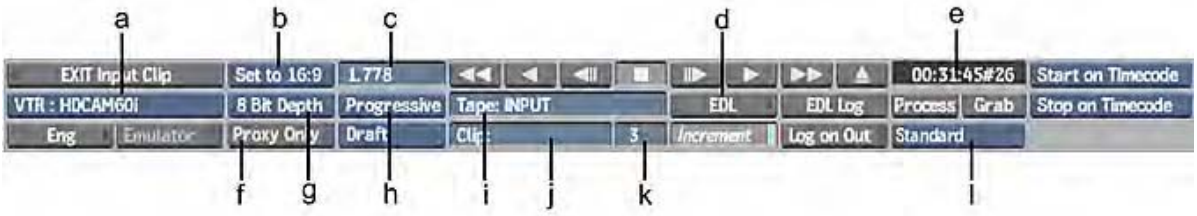
1. Cue up to the frame you want to capture using the VTR Transport controls or by scrubbing the preview window.
2. Click Grab.



   The captured frame is saved in the Workspace.
3. Click EXIT Input Clip to view the captured frame.
4. If you want to compare the captured frame against the frame in the preview window, load the clip into the VTR Input module and enable split-screen view. See Monitoring Video During Clip Output (page 199).

VTR Input Menu Options

The basic VTR Input controls are described as follows.



(a) Device Name box (b) Aspect Ratio box (c) Aspect Ratio field (d) EDL button (e) Current Timecode field (f) Proxy Quality box (g) Bit Depth box (h) Scan mode box (i) Tape Name field (j) Clip Name field (k) Increment Name field (l) Tape Type box

Device Name box Select the VTR and timing combination to use to capture clips. The available VTR are defined in the software configuration file using the VTR keyword.The available VTR are defined in the Smoke Setup application.

Engineering button Opens the engineering menu.

Aspect Ratio box Select the aspect ratio of the clip to capture.

Aspect Ratio field Displays the aspect ratio of the clip to capture. Editable.

Bit Depth box Select the bit depth used to capture the clip.

Scan Mode box Selects how to flag the captured clip: Progressive, Field 1 dominant, or Field 2 dominant. The flag is only there as a reminder; for example, setting Field 2 does not prevent you from de-interlacing on Field 1. In most cases, select the option that matches the format you are capturing.

Proxy Only button Enable to capture proxies only. In this case, the high-resolution media is captured, proxies are generated, but the high-resolution media is discarded. As a result, much storage space is required.

Only available if the current project is configured to use proxies. If your proxies are set to be generated as a post process in the Engineering menu, you can see an estimated time required for proxies generation in the message bar. You can abort this process at any time by clicking anywhere on the screen. When you click the screen, a message appears asking you to confirm the operation. Only the captured media that corresponds to completed proxy generation is preserved. Any captured frames for which no proxies have been generated are purged.

Proxy Quality box Proxies generated during capture are always of draft quality. Using the Proxy Quality box, you can set the default quality for proxy generation after capture. Results vary depending on the type of clips involved, so it is a good idea to try different settings. Only available if the current project is configured to use proxies.

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

Emulator button Enable to have the application emulate a VTR.

Capture button Starts the capture.

Grab button Grabs the current frame.

Input Type box Select the type of footage found on the tape. Use Standard for regular capture. 2X can only be used with HDCAM SR. Dual Image is for stereo footage recorded side-by-side on the tape; use Slice to create a single Stereoscopic clip on capture, but with half the horizontal resolution; use Scale to create a single Stereoscopic clip, but resized to full horizontal resolution. With Scale, use Engineering > Dual Image Resizing Filter to select the quality of the resize.

EDL Log button Logs the clip to the EDL.

EDL button Opens the EDL menu where you can capture and edit EDLs.

Log On Out button Enable to log EDL events every time you enter an out point.

Tape Name field Displays the name of the tape from which you are capturing. This name is important for EDL assembly and media recapture procedures. Editable.

Clip Name field Displays the name to use for the clip to capture. Editable.

Increment field Displays the number automatically appended to the clip name. Active when Increment is enabled.

Increment button Enable to append numerical increments to the clip name automatically. For example, enter “My_Clip” in the Clip Name field and then enable Increment Name. The first clip you capture is named “My_Clip-1”, the second is named “My_Clip-2”, and so on.

Start Mode box Determines the start mode for clip input.

Select:	To:
Start On Pen	Capture starting from the currently displayed frame on the tape. Click Process to activate the start-on-pen trigger.
Start On Timecode	Capture starting from the timecode you enter in the In field. Click Process to start the clip input process.

Stop Mode box Determines the stop mode for clip input.

Select:	To capture until:
Stop On Pen	You click anywhere on the screen.
Stop After Frames	A specific number of frames have been captured. When you select this option, a field appears in which you enter the number of frames you want to capture.
Stop On Timecode	A timecode on the tape has been reached (entered in the Out Timecode field).

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

The following controls are found on the right side of the menu (not shown in preceding illustration).

In Timecode field Displays the timecode on the tape at which point the clip input begins. Editable.

Out Timecode field Displays the timecode on the tape at which point the clip input process ends. Editable.

Duration field Displays the duration, in timecode, between the clip in point and out point. Editable.

Tape EE button Click to toggles E-to-E on and off. When lit, indicates that the VTR is in E-to-E mode (electronic to electronic). This means that the VTR output is showing its input signal.

Standby button When lit, the VTR is in standby mode. Click to toggle between standby modes.

Cue In button Cues the VTR to the value of the In Timecode field.

In button Sets the In Timecode field to the current VTR timecode.

Cue Out button Cues the VTR to the value of the Out Timecode field.

Out button Sets the Out Timecode field to the current VTR timecode.

Toggle Audio Tracks button Switch between audio banks of 4 audio tracks.The actual number of tracks depends on your capture device.

Video Track button Enable to capture the video track.

VTR Status display Indicates the current status of the VTR.

Selecting a VTR Device For Input

The VTR devices available depend on the VTR keyword lines uncommented in the software initialisation configuration file. Make sure the VTR you select is appropriately connected using the corresponding audio, video, and RS-422 connections.

In certain situations, Flame Premium pre-selects a VTR with appropriate timing for you.

To select the VTR device for input:

1. From the Device Name box, select an option corresponding to your VTR device.



2. Make sure that the selected device is in Remote mode.

Naming Tapes and Clips

Proper media management is an essential part of the clip input and output process. One of the elements of efficient media management is methodical and consistent naming of the clips you capture and the tapes from which they originate. Before capturing a clip, assign a name to the tape and the resulting clip. This assignment makes it easier to organize your clips, trace them back to the source tape, and, when necessary, recapture them.

To name a tape and clip:

1. In the Tape field, enter the tape name.

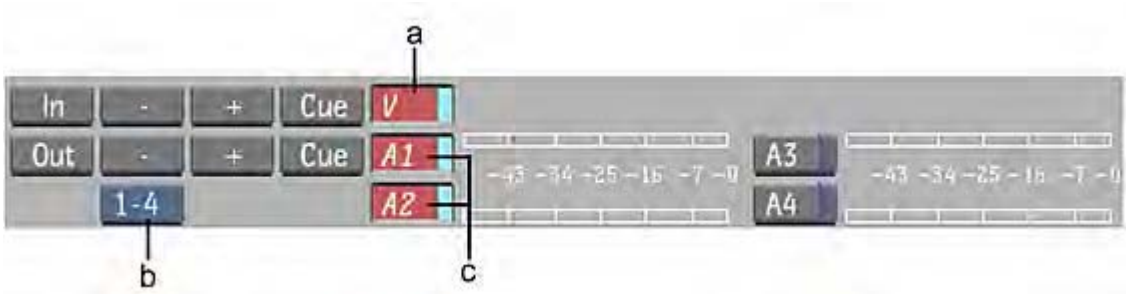
   The tape name is stored with the clip. In the Workspace, you can `Alt`-click the clip to view its information, including the tape name.
2. In the Clip field, enter a clip name.
3. Enable Increment to use the same name for multiple clips captured in sequence. This appends a numerical tag at the end of each clip so you can differentiate them.
4. To change the number of the numerical tag for the next captured clip, enter a number in the Clip Index field. By default, the increment begins at 1.

Selecting Tracks and Channels to Capture

Whether you are inputting single frames, single clips, multiple clips or conforming EDLs, you must select the tracks and channels you want to capture from the VTR.

To select the tracks and channels to capture:

1. To capture the video track, enable V.
2. To switch between audio tracks banks, use the Toggle Audio Tracks button.
3. To capture audio channels, enable the corresponding channel selection buttons.



(a) Video track enabled for capture (b) Toggle Audio Tracks button (c) Audio channels 1 and 2 enabled for capture

Audio meters display the gain level being captured. You cannot change this level during capture. You can modify the gain of a clip once capture is complete.

HDCAM SR Double-Speed and Stereo Tape Capture

Using an HDCAM SR, you can capture material from specially formatted double-speed and stereoscopic tapes.

Double-speed tapes allows you to capture material twice as fast. Stereoscopic tapes essentially stores in an interlaced timing two progressive clips; a 60i (50i) "clip" contains two 30p(25p) clips.

This feature does have the following limitations:

- To use this feature, you must use specially formatted tapes. If you insert a regular tape in the HDCAM SR and try to capture it as double-speed or stereoscopic material, the capture fails.
- Audio monitoring is not available during capture.
- When capturing stereo tapes, only audio channels 1 through 8 are available.

To capture material recorded at double-speed:

1. Ensure that the HDCAM SR is connected to the AJA card using a dual-link.
2. Set the HDCAM SR VTR to DBL 422.
3. From the Device Name box, select the HDCAM SR VTR.
4. From the Input Type box, select 2x-DOUBLE.



NOTE If the player displays the clip with some colour bias, it is because the player falsely interprets the 4:2:2 signal from the VTR as a 4:4:4 signal. This does not impact the capture; the stereoscopic clip will be captured without that bias. To remove this bias, go to the Engineering menu and set the Input Connection box to Serial 1 4:2:2.

5. Capture the clip. See Capturing Single Clips (page 181).

To capture material recorded on stereoscopic tapes:

1. Ensure that the HDCAM SR is connected to the AJA card using a dual-link.
2. Set the HDCAM SR VTR to the stereoscopic setting.
3. From the VTR Input menu, select the HDCAM SR VTR from the Device Name box.

4 From the Input Type box, select 2x-STEREO.



NOTE If the player displays the clip with some colour bias, it is because the player falsely interprets the 4:2:2 signal from the VTR as a 4:4:4 signal. This does not impact the capture; the stereoscopic clip will be captured without that bias. To remove this bias, go to the Engineering menu and set the Input Connection box to Serial 1 4:2:2.

5 Capture the clip. See Capturing Single Clips (page 181).

The stereoscopic material is captured as a single, regular stereoscopic clip, with two layers, one for each eye.

Outputting Clips To a VTR

Once you have clips that are ready for output to tape, use the VTR Output module to perform this operation. Like capturing, you can output single clips or multiple clips, and enable a split view to preview the clip you want to output alongside the media on the tape simultaneously.

Another way to output multiple clips is to generate an EDL and then output the clips using the EDL.

Most of the procedures in this section assume that you have already blacked your tape. However, if necessary, you can output in assemble mode, which allows you to perform clip output to a tape that was only partially blacked. See Outputting Clips in Assemble Mode (page 198).

Accessing the VTR Output Module

1 Open the Tools tab.
2 In the Utilities tab, select Clip VTR Output.
3 Select the clip to output from the Media Library, `Ctrl+click` to select multiple clips. You can also select a folder if you wish to output multiple clips in one session.

The VTR Output module appears.

If VTR Output is not available, connect your workstation to a VTR, and configure Flame Premium to use the VTR.

VTR Output Menu Options

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

When one of the Output, Audio, or Engineering tabs is selected, both the large and small formats are available. To toggle between the large and small formats, Control-swipe the bottom of the screen.

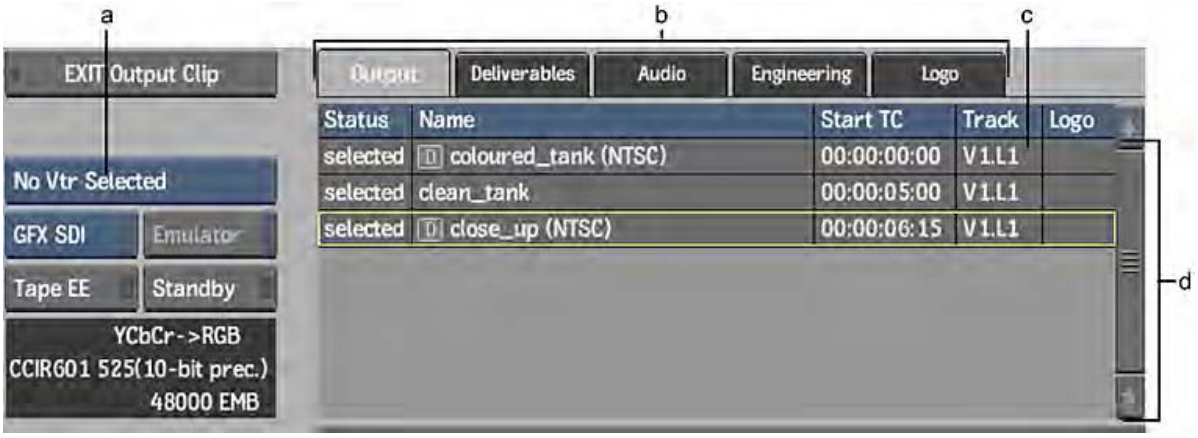
When the Deliverables or Logo tab is selected, only the large format is available, but additional Pan and Zoom controls are available.

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

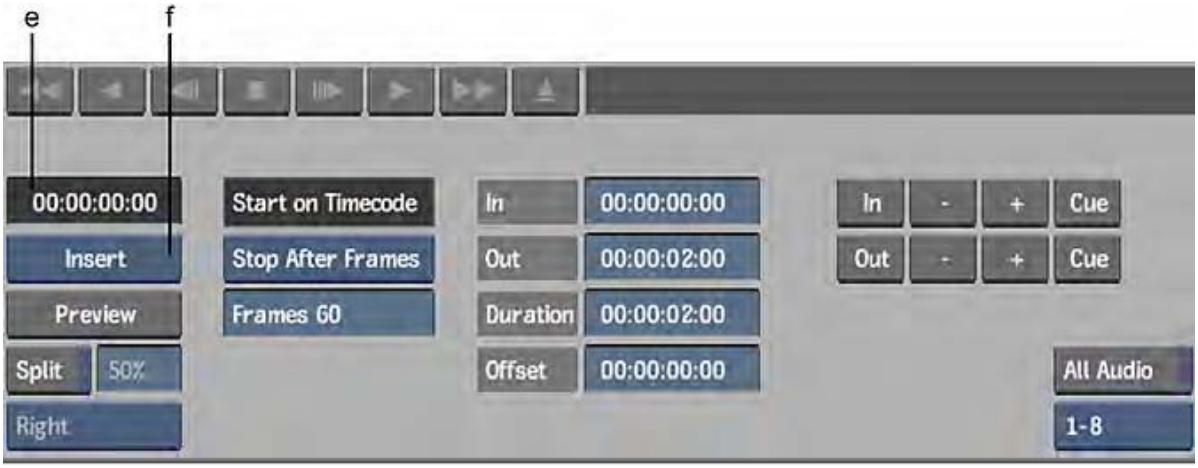
Small VTR Output module (left portion):



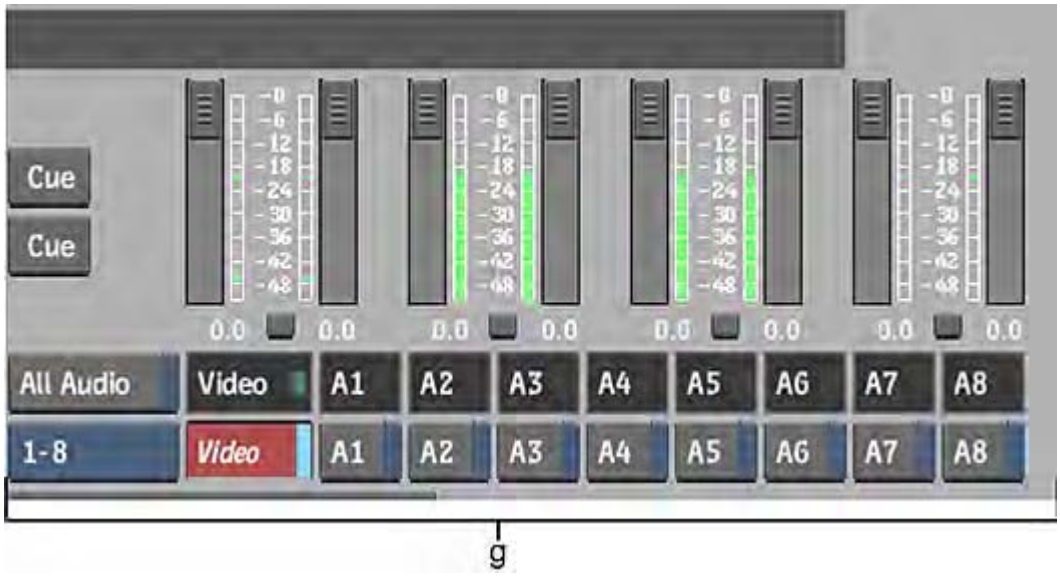
Large VTR Output module (broken into three parts):



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



(e) Current Timecode field (f) Output box



(g) Audio controls

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

Device Name box Displays the options for each uncommented VTR keyword line in the software initialisation configuration file. Select the option corresponding to the VTR to which you want to output clips.

Navigation tabs Switch between different VTR Output tabs.

Select:	To:
Output	Configure the output settings described in this section. This is the default menu.
Deliverables	Set up Real-Time Deliverables on output, such as Letterboxes or LUTs.
Audio	Set Audio preferences. Changes are reflected in the Audio section of the Preferences menu, and vice versa.
Engineering	View the VTR Output module Engineering menu.
Logo	Set up a logo on output.

NOTE The Deliverables and Logo tabs are only available if the workstation uses an NVIDIA Quadro FX 5600 SDI graphics card.

Output list Displays information about the clip selected for output. If there are multiple clips, you can sort them by clicking the column headings. This changes the output sequence order. You can also edit the Timecode field in this list.

If the list includes Deliverables and you do not have the hardware required by Real-Time Deliverables, the Deliverables are greyed out.

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

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

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

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

VTR Status display Indicates the current status of the VTR.

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

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

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

Split View button Enable to simultaneously monitor the clip selected for output and the contents of the tape.

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

Stop Mode box Determines the stop mode for clip output.

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

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

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

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

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

In/Out Point controls Use to enter, adjust, and cue the in and out points.

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

Audio Channel buttons Enable and control audio output signals.

Selecting a VTR Device For Output

The VTR devices available depend on the VTR keyword lines uncommented in the software initialisation configuration file. Make sure the VTR you select is appropriately connected using the corresponding audio, video, and RS-422 connections.

Flame Premium pre-selects a VTR with appropriate timing when entering the Output clip module.

To select the VTR device for output:

1. From the Device Name box, select a VTR device.
2. Make sure that the selected device is in Remote mode.

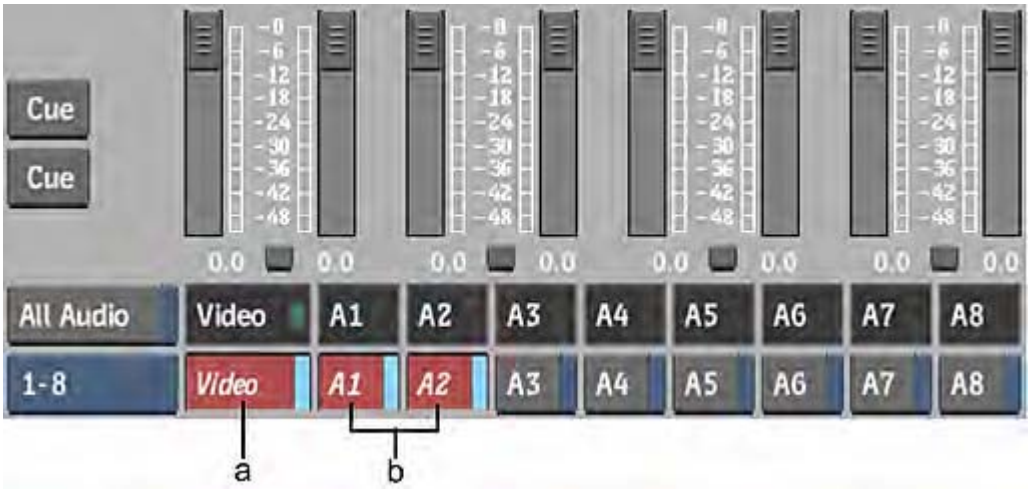
Outputting a Single Clip

To output a clip to a VTR, load a clip into the VTR Output module and then set the in and out points for clip output. If necessary, enable the options to offset the start frame for output and to apply a letterbox overlay. See VTR Output Menu Options (page 187).

If the duration marked by the in and out points that you set for output is greater than the duration of the clip you are outputting, the last frame of the clip is repeated to fill the gap between the end of the clip and the out point of clip output. For example, if you set an in point at 01:00:00:00 and an out point at 01:01:00:00, the clip output process has a duration of one minute. If the clip you are outputting is 30 seconds long, the last 30 seconds of the output result on tape are filled with the last frame of the clip you are outputting.

To output a single clip to a VTR:

1. Load the clip that you want to output into the VTR Output module. See Accessing the VTR Output Module (page 187).
2. Make sure that the correct video track and audio channel buttons are enabled so that the corresponding video track and audio channels are output to tape.



(a) Video track enabled for output (b) Audio channels 1 and 2 enabled for output

3. Enable or disable All Audio in the Clip Output menu (to the left of the audio controls).

   When All Audio is enabled, all audio channels are sent to the audio converter, not just the ones you enabled in the previous step.

   This means you can monitor all audio, even though you are only recording one or two tracks to the VTR. You can also route the other audio channels to a mixer or any other device capable of capturing an audio signal.
4. Make sure the appropriate VTR is selected in the Device Name box. See Selecting a VTR Device For Output (page 190).

   The preview window displays the contents of the tape currently in the selected VTR.
5. To preview the clip before outputting, click Preview.
6. To preview the clip you are outputting against the contents on the tape, enable Split View. See Monitoring Video During Clip Output (page 199).

7 To output the selected clip starting at any frame other than the first one, enter the start timecode in the Start Offset field.

8 Set the in and out points for clip output. See Setting Input and Output In and Out Points (page 199).

9 To output the clip to the VTR, select Insert from the Output box.

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

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

10 After the transfer is complete, verify that it was successful by playing the transferred clip. To do so, cue to the in timecode and click the Play button in the VTR Transport controls.

11 When you are done, click EXIT.

Outputting Multiple Clips

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

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

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

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

To output multiple clips to VTR:

1 Load the clips that you want to output into the VTR Output module. See Accessing the VTR Output Module (page 187).

2 If the clips have conflicting timings, you are prompted to select the timing you want to use.

Clips that do not match this timing are discarded.

3 To ouput a clip, its status field must have the *selected* indicator. Click the Status field to alternate between the *selected* indicator and an empty field.

4 Set output options for each clip.

5 Optional: Click the column headers to sort the list. This changes the order of the output sequence.

6 To output the clip to the VTR, select Insert from the Output box.

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

- Pending: the clip is waiting to be output to tape.
- Output: the clip is being output to tape.

- Done: the clip has been ouput to tape.

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

8 When finished, click EXIT.

When outputting multiple clips with the small VTR Output module, note the following differences:

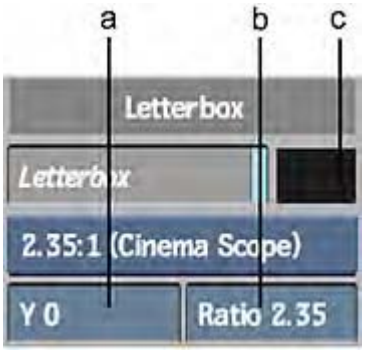
- Use the Previous and Next buttons to navigate the clip list.
- Once output options have been set for each clip, make sure you are viewing the first clip you want to output.
- When you select Insert in the Output box, the clip you are viewing, and all subsequent clips, are output to tape. Clips that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

Outputting Clips With a Letterbox Overlay

You can apply a letterbox overlay on output. The letterbox does not affect the clip itself; it is applied on the fly as part of the real time output process.

To output clips with a letterbox overlay:

1 In the Output Clip menu, click the Deliverables tab to see the Letterbox controls.



(a) Letterbox Offset field (b) Letterbox aspect ratio field (c) Colour selector box

2 Enable Letterbox.

3 Set the letterbox options:

- In the Aspect Ratio field, enter a value to set the aspect ratio of the letterbox overlay. Alternatively, select a preset from the Aspect Ratio box.
- In the Letterbox Offset field, enter a value to shift the letterbox overlay up or down, as required.
- Use the Colour Selector box to set the colour of the letterbox overlay.

Outputting Clips with a Logo Overlay

You can add a logo overlay on output, in real time. This feature is available only if your workstation is equipped with an NVIDIA Quadro FX 5600 SDI graphics card.

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

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

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



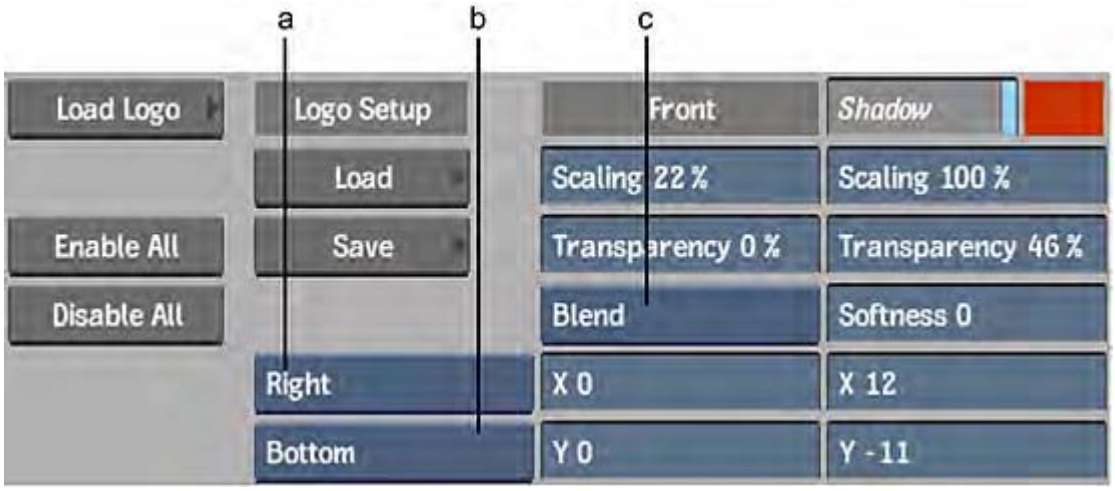
If you do not see GFX SDI as an available option in the Graphics Card box, it is either not installed, or it is not set up or configured properly.

About the Logo Controls

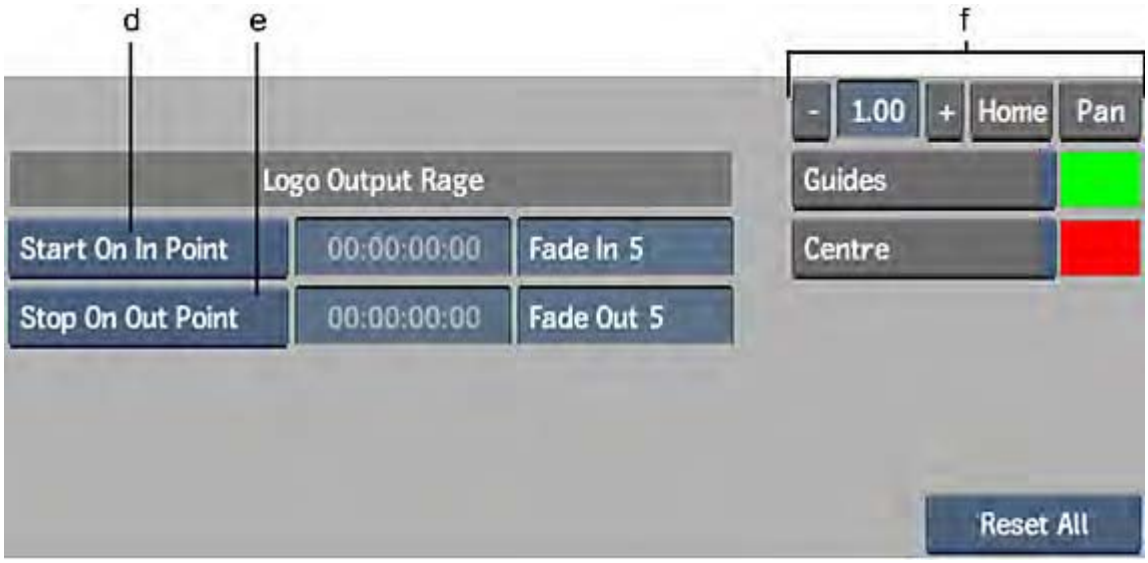
The logo controls are found in the VTR Output module, under the Logo tab.

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





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



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

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

Load Logo Opens the Clip Select menu where you can select logo clips.

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

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

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

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

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

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

Select:	To:
Add Trans	Compensate for the soft or anti-aliased edge on an object in a front and matte clip media, with transparency.

Select:	To:
Blend	Punch the matte through the front. This blends the edge of the front clip and adds additional softness to the media.

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

Zoom controls The small VTR Output module is not available from the Logo tab. Use the Zoom controls to view oversized clips.

Reset All Use to reset the Logo controls to default.

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

Inserting Logos for Real-Time Output

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

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

Logo settings are persistent, and previous logo settings are loaded when you open the VTR Output module.

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

To insert a logo in real time:

1. Select clips from the Media List and select File > VTR Output...
2. In the VTR Output module, use the Logo column to indicate which clips are to be output with the logo.
3. Click the Logo tab to access the logo controls.
4. Select one or two clips to use as logo:
   - If you select one clip, it is used as the front clip and a luminance key is applied to it. If the clip has multiple frames, only the first frame is used.
   - If you select two clips, the first is used as the front and the second as the matte.

5 Use the logo controls to position and configure the appearance of your logo, and to define in and out points. See About the Logo Controls (page 194).

6 Output the clip.

Outputting Clips in Assemble Mode

If you do not have time to black an entire tape, you can black a small portion of the tape and output clips to the tape in assemble mode. In this mode, the timecode on the tape is generated by the VTR as part of the recording process. Make sure that timecode regeneration is properly configured on the VTR. The standing recommendation is blacking one minute at the beginning of the tape.

If outputting a clip in assemble mode, extend the duration of the output to avoid problems when you need to recapture the segment. For example, this can be done by adding 5 to 10 seconds of black at the beginning and at the end of the clip, before entering the output clip module.

To output a clip in assemble mode:

1 On the VTR, set the TC generator switch to Internal and Regen to make the VTR generate the timecode. Refer to your VTR manual for instructions on setting the TC generator.

TIP If you have just added black to your tape, the VTR should already have these settings.

2 Set clip output options. See Outputting a Single Clip (page 191).

3 With the clip you want to output in the VTR Output module, set the clip output in and out points. See Setting Input and Output In and Out Points (page 199).

4 If black was not added at the end of the clip, make sure the out point exceeds the duration of the clip by five to ten seconds.

5 Select the Assemble option from the Output box.

6 Click Assemble to output the clip in assemble mode.

HDCAM SR Stereo Tape Output

Using an HDCAM SR, you can output material to specially formatted stereoscopic tapes. Stereoscopic tapes essentially stores in an interlaced timing two progressive clips; a 60i (50i) “clip” contains two 30p(25p) clips.

To output stereoscopic material to stereoscopic tape:

1 Ensure that the HDCAM SR is connected to the output device using a dual-link.

IMPORTANT If you use an NVIDIA SDI card to output the stereoscopic material, the NVIDIA card downconverts the material to 8-bit.

2 Set the HDCAM SR VTR to the stereoscopic setting and insert a stereoscopic tape.

3 In the application, enter the VTR Output module with a stereoscopic clip. The presence of an **S** in the bottom right corner of a proxy indicates a stereoscopic clip.

NOTE You can output only clips at 23, 24, 50 or 59 fps, with a 1080i or 1080PsF resolution.

4 Output the clip. See Outputting a Single Clip (page 191).

The player in the VTR Output module displays only the left-eye layer, but the application outputs both right- and left-eye tracks, in a 4:2:2 colour space.

Monitoring Video During Clip Output

When you output a clip to a tape that already has media on it, you can enable a split view to compare a clip with a portion of tape, in much the same way as split view allows to compare two video segments in the player. This can be especially useful if you are outputting a clip to seamlessly overwrite a section on the tape.

To use the split view preview option:

1 In the Output tab of the VTR Output module, enable Split.



The Split View controls are enabled and you can view the clip that is currently set for output alongside the content on the tape.

2 Set Split View preview options:
- In the Split field, enter a value to set the relative position of the split.
- From the Current Clip Display box, select an option to set the location of the current clip display.
- From the Split Orientation box, select Horizontal Split or Vertical Split to set the Split View orientation.

3 With Split View enabled, you can scrub the VTR to locate the frame that matches the frame of the current clip.

4 Set the In point to the timecode of the frame on tape that corresponds to the first frame of the clip.

5 Cue to the In point. The first frame of the clip and matching frame on tape are displayed in Split View.

6 Click Play to play the tape and the clip simultaneously in Split View.

7 Click Stop, and disable Split View when you are finished.

Setting Input and Output In and Out Points

You can enter timecode values directly in the In, Out, and Dur fields to set in and out points for clip input and output. However, there are other methods of setting the in and out points that are more convenient if you are scrubbing the tape in the VTR to locate reference frames for clip input and output. For example, if you output a clip to overwrite a segment on a tape, you scrub the tape to locate the frame that matches the first frame of the clip you are outputting.

You can also enter in and out points, as well as duration values based on the timecode value of a clip selected for input. Simply click Input Clip, then while holding down the left `Ctrl` key, select a clip and a destination reel.

Besides the described methods of entering timecodes in the In, Out, and Dur fields, you can modify clip in and out points or cue the VTR to these points using the following controls on the right side of the Input or Output menu.



In button Click to set the In field to the VTR's current timecode.

Out button Click to set the Out field to the VTR's current timecode.

+ and - buttons Click to frame-step the In or Out timecodes forward or backward. If Stop After Frames is selected from the Stop Mode box, both the in and out points move because the duration between them is locked.

Cue buttons Click to cue the VTR to the in or out point.

Inputting and Outputting a Live Video Signal

In the configuration file, if you uncomment the `live NTSC` or `live PAL` VTR lines, you can capture a live video signal or crash-record a clip using the pen to start and stop the clip input or output process.

On input, use the Live NTSC or PAL option to capture directly from a camera, VCR, or any other device that does not support remote control via RS-422. You can also feed a live video signal from satellite or any other broadcast source.

On output, use the Live Video option to output clips to a device that does not support remote control via RS-422.

To input a live video signal:

1. Enter the VTR Input module.
2. From the VTR Device box, select Live NTSC or Live PAL.

   The incoming live video signal appears in the preview window. The Start Mode box is unavailable. With Live Video capture, you must use Start On Pen mode. You can use Stop On Pen or Stop On Frames to end the capture.

   NOTE When Stop On Pen is selected as the preferred capture stop mode, the out point and the duration timecode fields are updated to reflect the longest possible duration corresponding to the full capacity of a framestore. The capture stops either when you click anywhere on the screen or the timecode indicated is reached, meaning that the framestore is full.

3. Set input options. For example, enter the clip name and enable the video tracks and audio channels that you want to capture. See Inputting Clips From a VTR (page 181).

4 Make sure you are receiving the live video signal.

5 If you are capturing from a device that does not support remote control, press Play (allowing enough time to click Process in Flame Premium).

6 To begin capturing, click Process.

After a moment, you are prompted to click anywhere to begin the output process.

7 Click anywhere on the screen.

8 To end capturing in Stop On Pen mode, click anywhere on the screen.

To output a live video signal:

1 Load a clip into the VTR Output module.

2 From the VTR Device box, select Live NTSC or Live PAL.

The Start Mode box is unavailable. With Live Video output, you must use Start On Pen mode. You can use Stop On Pen or Stop On Frames to end the output.

3 Set output options. For example, enter the clip name and enable the video tracks and audio channels that you want to capture. See Outputting Clips To a VTR (page 187).

4 If you are outputting to a device, click Record, or take any action required to enable the device and capture the signal being output from Flame Premium.

5 To begin capturing, click Process.

After a moment, you are prompted to click anywhere to begin the output process.

6 Click anywhere on the screen.

7 To end capturing in Stop On Pen mode, click anywhere on the screen.

Supported Timings and Resolutions for Video I/O

In addition to 2K broadcast and SD timings, the following HD formats are supported.

Format	Resolution	Scanning	Supported Frame Rates
720p	1280 x 720	Progressive	50, 59.94, and 60 Fps, Hz
1080i	1920 x 1080	Interlaced	25, 29.97, and 30 Fps, Hz
1080p	1920 x 1080	Progressive	23.976, 24, 25, 29.97, 30, 50, 59.94, and 60 Fps, Hz
1080PsF	1920 x 1080	Progressive	23.976, 24, 25, 29.97, and 30 Fps, Hz

Generating Proxies from VTR Input

With sufficient processing power, proxies are generated in real time. Otherwise, this occurs as a post-process. To achieve better performance, or to use higher quality proxy types, you might want to perform clip input without proxy generation, then edit project settings to generate proxies overnight.

The following guidelines refer to working on projects set to generate proxies.

You may create projects with proxy management options set to generate proxies for HD clips. If your hardware configuration supports on-the-fly proxy generation, your clips are captured and proxies are generated as

part of the real-time input process. Otherwise, you may be required to wait for proxy generation once the capture process itself is complete.

If you are capturing many clips one-at-a-time, you may want to turn off proxy generation for you project. That way, you can capture your clips without having to wait for proxy generation.

Once you have completed the capture process, turn proxy generation back on. If you have captured many clips, this process could take a long time. You may want to do this when you can leave the workstation unattended, for example, overnight.

Capturing Material with Variable Frame Rate

To perform frame-accurate video input and output using Panasonic® variable frame rate VTR decks, you must use the following firmware versions:

- AJ-HD3700H D5-HD deck:

  SYSCON: 1.04 or later

  AV: 0.21 or later

  FRONT: 0.13 or later

- AJ-HD1200A DVCPRO HD deck:

  SYSIF: 1.30 or later

  AVDV: 1.39 or later

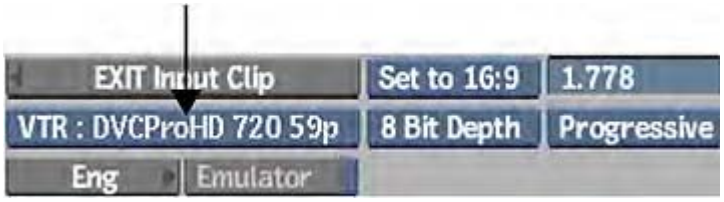
  SERVO: 1.22 or later

Regular video input and output is not affected by a firmware version.

NOTE Refer to your Panasonic documentation for information on verifying the firmware version.

To capture material from a Varicam device:

1. Ensure your devices are properly connected and the corresponding VTR keywords are enabled in the software initialization configuration file. See Configuring Software For Clip Input and Output Using a VTR (page 180).
2. Open the VTR Input module. See Accessing the VTR Input Module (page 181).
3. From the Device Name box, select a video device, supporting variable frame rate (for example, one of the following options: VTR DVCProHD 720 59p or VTR DVCProHD 720 60p).



4. Enter the Engineering menu and enable Varicam (on the right side).



5 Make sure the Video Input Delay is set to zero or a positive value (negative video input delay is not supported on Varicam capture).

6 From the Timecode Mode box, select the required timecode.



7 Exit the Engineering menu and click Process.

Flame Premium removes the redundant frames so that the frame rate of the captured clip corresponds to the frame rate of the project. You can monitor the process using the VTR Status display. The Current Frame Rate field is updated depending on the frame rate of the captured material.



(a) Current Frame Rate field

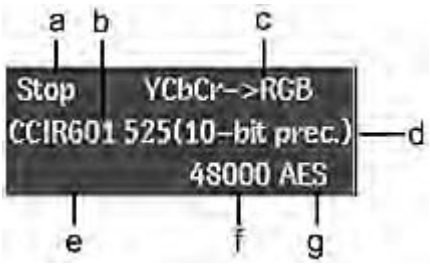
NOTE When performing operations in Varicam mode, audio/video synchronization largely depends on frame rate compatibility and can be guaranteed only if the following conditions are observed: (1) Material shot at 24 fps and captured into a 24 fps project (2) Material shot at 30 fps and captured into a 30 fps project (3) Material shot at 60 fps and captured into a 60 fps project.

Controlling a VTR

The RS-422 serial connection between the VTR and your workstation enables full remote control of the device from Flame Premium. Connected VTR devices send their current status to Flame Premium, and this status is displayed in the VTR Input and VTR Output modules.

Verifying the VTR Status

The following illustration shows a VTR status display.



(a) VTR Status (b) VTR Timing (c) Colour Space (d) Bit Precision (e) Control Mode (f) Audio Sample Rate (g) Audio Input/Output Type

Other VTR status values include.

VTR field:	Indicates:
VTR Status	Play, Play Lock, Stop, Jog, Fast Forward (FFW), Rewind (RWD), 3x, 4x, 8x, 15x, 30x. When the field is blank the VTR is disengaged.

VTR field:	Indicates:
VTR Timing	The video timing currently configured on the video device.
Colour Space	The colour space conversion method for both input and output of clips.
Control Mode (LOCAL, REMOTE, or REC INHIBIT)	Whether the VTR is in Local or Remote control mode. Local appears in yellow when the VTR is in Local mode. In Local mode, the VTR cannot be controlled from Flame Premium. You can switch the VTR between Local and Remote mode only on the VTR. Rec Inhibit implies REMOTE mode, but you cannot output to the tape.
Audio Sample Rate	The audio sampling rate when you input or output from the VTR.
Audio Input/Output Type	The source audio interface type when inputting and the destination audio interface type when outputting.
Bit Precision	The accuracy used (SDI bit depth) when transferring data. This cannot be changed.

Using the VTR Transport Controls

The current frame on the cassette in the VTR appears in the image window and the Current Timecode field (the uppermost timecode field in the Clip Input and Output menus) displays the timecode of the current frame.



(a) Current Timecode field

Use the following VTR Transport controls to play the VTR.

NOTE The VTR must be in Remote mode to use the VTR Transport controls.

Click:	To:	Hotkey:
	Rewind the tape.	`Home`
	Play the tape backward.	-
	Move backward one frame. Hold down to slowly jog backward.	`left arrow key` (`down arrow key` for -5 frames)
	Stop the tape.	`spacebar`

Click:	To:	Hotkey:
	Move forward one frame. Hold down to slowly jog forward.	`right arrow key` (`up arrow key` for +5 frames)
	Play the tape forward.	`Enter`
	Fast-forward the tape.	`End`
	Eject the tape.	none

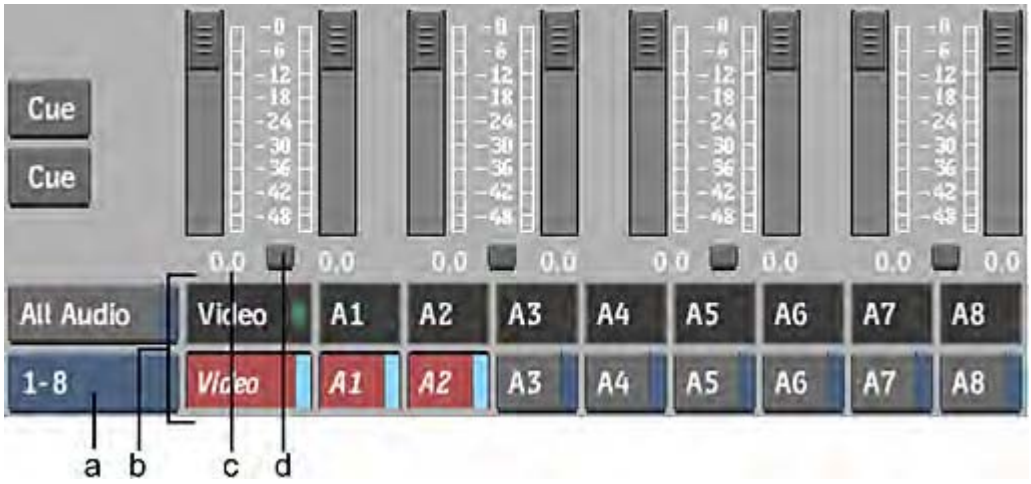
You can also shuttle the VTR by dragging the cursor in the image window. Place the cursor on the image, press the pen or mouse button, and drag the cursor to the right to shuttle forward and to the left to shuttle backward. The farther away you move the cursor horizontally from the middle of the image, the faster the speed.

You can also change the reverse or forward playback speed by pressing the `Up` or `DownarrowKeys` on your keyboard. Each press reduces or increases the speed by two.

Audio Gain Adjustment Settings

Audio gain adjustment is a part of the clip output process only. The clips you are outputting are unaffected by audio gain adjustments made using the VTR Output module.

On output, you can adjust the audio gain, for example, to restore the levels you had monitored on capture.



(a) Audio Tracks Toggle button (b) Channel Selection buttons and indicators (c) Audio Level fields (d) Fader Lock buttons

All Audio button When enabled, outputs to the audio monitor every audio channels. When disabled, outputs only the enabled audio channels to the audio monitor. The All Audio button has no impact on the audio tracks recorded by the VTR.

Audio Tracks Toggle button Toggles the Channel Selection buttons and indicators between audio tracks 1-8 and 9-16.

Channel Selection buttons and indicators Controls and displays which audio channels are recorded by the VTR. The black boxes with the green LEDs indicate video tracks and audio channels that are part of the clip that you want to output. The red buttons indicate the tracks and channels the VTR records on output.

Audio Level fields Displays the audio gain, in decibels. Adjust using the faders. In the small VTR Output module, increase or decrease the gain by dragging left or right on the fields. By default, audio gain is 0 db.

Fader Lock buttons When enabled, locks the faders for the corresponding pair of audio channels together.

Adjusting Audio Gain on Clip Output

To adjust the audio gain on output clip:

1. Enable the Fader Lock buttons (so that they are light grey) if you want to apply the identical value to pairs of audio channels.
2. Slide the faders to adjust the audio gain before you start processing. Use the All Audio button to monitor all the audio tracks that are output, regardless of what audio tracks the VTR records.
3. In the VTR Output module, enable Output All Audio.
4. Select or deselect channels for output by clicking the Channel Selection button for each channel as needed.

   NOTE In a multiple clip selection, channel selection is independent for each clip but the gain levels set with the faders are the same for all clips.

5. Process the clip.

   Selected channels are output.

Setting Video Input and Output Engineering Menu Controls

Clip input and output engineering options include video I/O settings such as pre-roll, post-roll, play delay, colour space conversion, and settings that define the process by which YUV video material on a tape is converted to the RGB format used by Flame Premium, and vice-versa.

Each VTR device is associated with a set of default engineering settings that are specified in the software initialisation configuration file, in the VTR KEYWORD section.

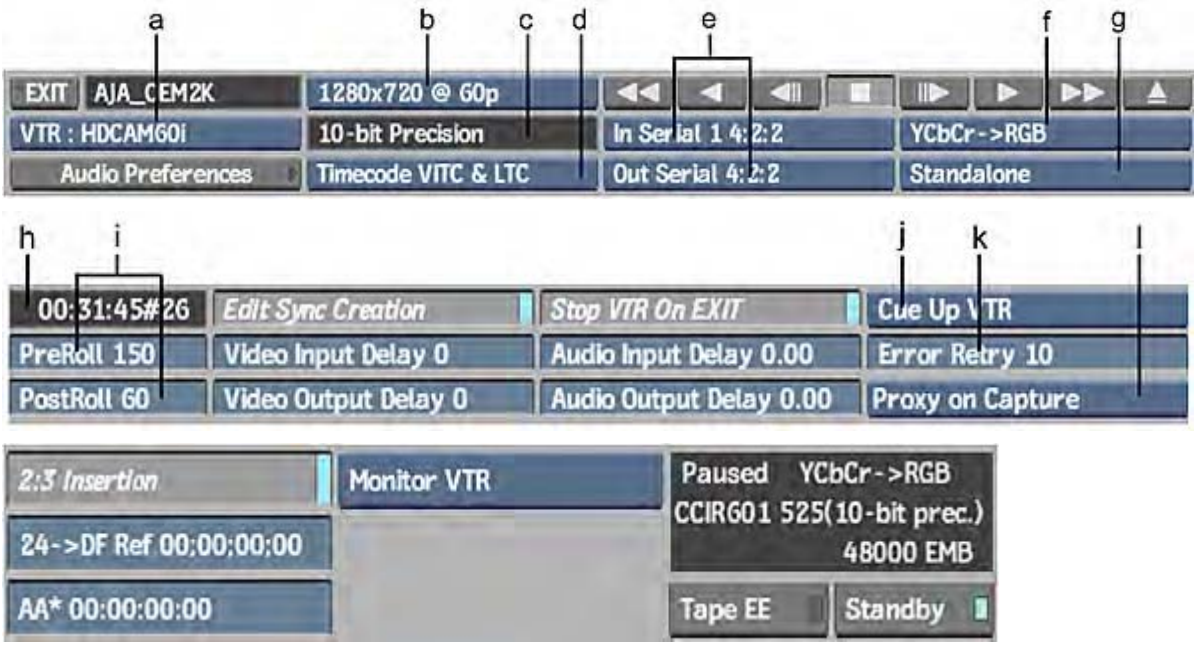
When you start Flame Premium and set the video I/O timing for your project, enabled VTR devices are initialized, and the settings in the Engineering menu are populated accordingly. However, if necessary, you can modify these settings on a session-to-session basis.

To open the Engineering menu:

1. From the VTR Input module, click Eng, or from the VTR Output module, click the Engineering tab, to open the Engineering menu.

Engineering Menu Options

Consult the following illustration (broken into three parts) and explanations of the options in the Engineering menu. These illustrations are of the VTR Input Engineering menu. The VTR Clip Engineering menu contains a subset of these controls.



(a) Device Name box (b) Video I/O Timing box (c) Precision box (d) Timecode Source box (e) Input and Output Connection boxes (f) Colour Space box (g) Output Sync box (h) VTR Current Timecode field (i) PreRoll/PostRoll fields (j) Cue Up box (k) Error Retry field (l) Proxy Box

Device name box The Device Name box differentiates between 3G and non-3G capable VTR and provides an option for each enabled VTR in the software initialisation configuration file. To modify settings for a specific VTR device, select the corresponding option from this box.

Video I/O Timing box Provides video timing options (resolution and frame rate) corresponding to different video formats supported by the video input/output board of the system.

Precision box Determines the video interface precision, or SDI bit depth used. This value cannot be changed.

Audio Preferences button Click to view the Audio Preferences menu.

Timecode Source box Determines which type of timecode is obtained from the VTR device. Timecode VITC & LTC is the default, and recommended option. You should only have to switch to Timecode VITC or Timecode LTC if one of the timecode tracks is corrupted.

Select:	To obtain:
Timecode VITC	Vertical interval timecode (VITC).
Timecode LTC	Longitudinal timecode (LTC).
Timecode VITC & LTC	Both types of timecode. At normal playback speed, Flame Premium obtains LTC, but switched to VITC when the tape is rewinding, fast-forwarding, or otherwise moving at a non-playback speed.

Input and Output Connection boxes These boxes determine the connection by which the video signal is transferred. This box is automatically set to reflect the selected VTR.

- In Serial 1 3G / Out serial 1 3G: The video signal is transferred through one SDI 3G link. This is the only available option when you use a 3G VTR. The VTR Status field indicates any conversion that might happen because of the sampling used, 4:2:2 (conversion YCbCr <-> RGB) or 4:4:4 (no conversion RGB <-> RGB).

- In serial Dual 444 / Out serial Dual 444: The video signal is transferred through dual SDI links; one video field is sent through one SDI cable, the other field through the other cable.
- In Serial 1 4:2:2 / Out serial 422: Traditional VTR, single-SDI connection.

Colour Space box Determines the YCrCb colour space conversion method.

Select:	To perform clip I/O with:
YCrCb->RGB	A standard YCrCb-RGB conversion process that clips superblack and superwhite luma (Y). Use this option for typical clip I/O processes with VTR devices.
YCrCb->RGB + Headroom	A YCrCb-RGB conversion process that preserves superblack and superwhite colour information. Use this option when inputting or outputting greyscale mattes or other clips where preserving extremes in the luma channel is required.
No Conversion	This is available when using dual link for RGB input and output. Video black and white levels on the SDI stream are mapped to black and white values in RGB on the framestore. Use this option in conjunction with 4:2:2 input and output connections to input and output 4:4:4 video using dual-links (4:2:2 and 0:2:2).
No Conversion + Headroom	Also available when using dual link for RGB input and output. This mode uses all levels available and preserves all but a few RGB values. Use this option with the 4:4:4 input and output connections to input clips from and output clips to a Telecine.

Output Sync box Determines the output sync reference source. The reference signal may originate from several different sources. Select the source you are using from this box according to the following table.

Source type	Available on:	Description:
House	All systems	A centralized analogue reference signal, originating from a sync generator, sent to the genlock port on the video board or VBOB.
Digital 1 and Digital 2	Most HP® 8400s and all HP 8600s and 9400s	Same as Digital, except you can choose between two inputs: Digital 1 or Digital 2. On the HP 8400 with the AJA SD (OEM-LH) video board, only Digital 1 is available.
Standalone	All systems	The reference signal generated internally by the Flame Premium workstation.

PreRoll field Indicates the pre-roll, in frames.

PostRoll field Indicates the post-roll, in frames.

Video Input Delay field Indicates the video delay on input, in frames. If this value is incorrect, the result clip when you click Frame Grab in the VTR Input module does not match the frame you see in the preview window.

Video Output Delay field Indicates the video delay on output, in frames. If this value is incorrect, the clip you output does not get recorded to the proper place on the tape.

Audio Input Delay field Indicates the video delay on input, in frames.

Audio Output Delay field Indicates the video delay on output, in frames.

Stop VTR on EXIT button When enabled, sends a stop command to the VTR when you exit the Input Clip, Output Clip, Auto-Capture, or Archiving menu. For example, if the VTR is playing a clip, or if it is cueing to an in point, the transport operation in-progress is interrupted.

Cue Up box Determines the speed of the cueing process.

Select:	To cue up the VTR:
Cue Up VTR	Using the internal cueing algorithm of the VTR.
Cue Up Fast Forward	Using Flame Premium. Use this option if Cue Up VTR is too slow for far cue points, such as on the betacam SP.

Error Retry field Indicates the number of times Flame Premium retries failed input or output processes.

2:3 Removal/Insertion button Enables automatic, real-time 2:3 removal on output and insertion on input. This is only available when the VTR is set to 29.97i or 59.94i.

Proxy box Determines proxy management when inputting clips.

Select:	To generate:
Proxy in Post	Proxies as a post-processing step
Proxy on Capture	Proxies during capture

Generating proxy during capture is the quickest method. Depending on hardware configuration of your system, capture may be performed in real time with playback. Some extra required processing, however, may prevent the graphics board from updating the image window and broadcast monitor in real time, so you may not be able to view the clip being played as it is captured.

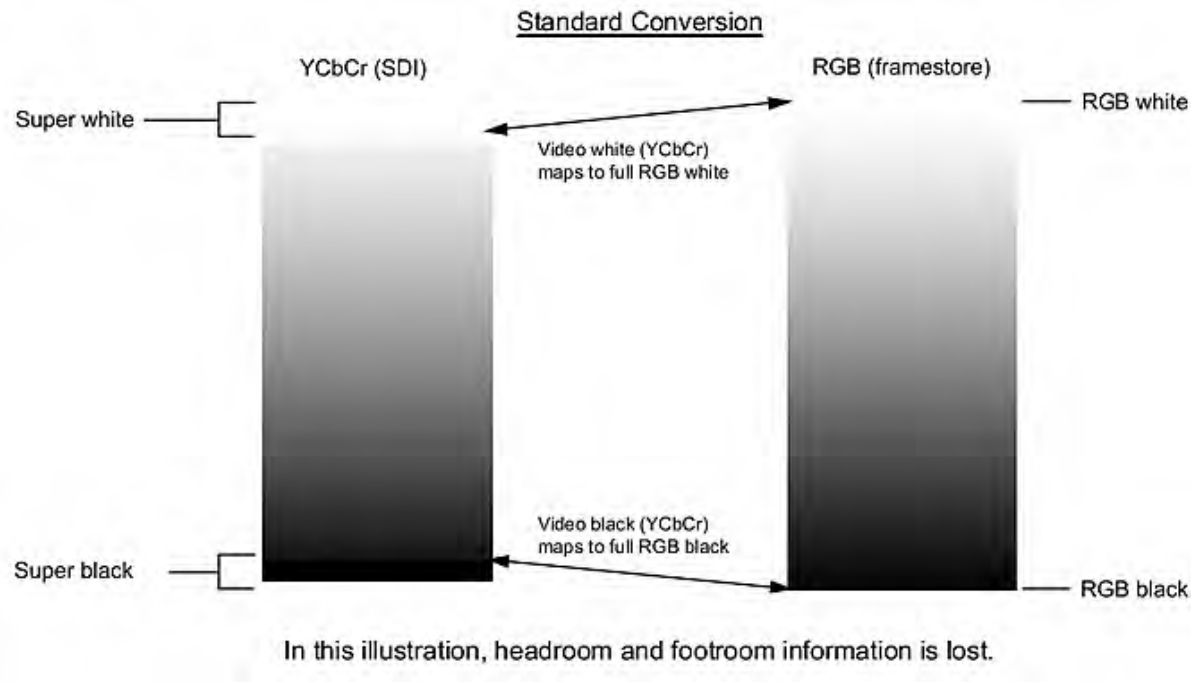
Monitor box Set this to Monitor VTR or Monitor Output. In Monitor VTR mode, the signal coming back from the VTR is displayed, and video may be appear to be late compared to the audio. In Monitor Output mode, the signal being output to tape is displayed, and the video and audio should be in sync.

Dual Image Resizing Filter box Select the resize algorithm used to scale back to full resolution clips captured using the Dual Image - Scale resolution. Ordered from lowest quality (Triangle) to highest (Lanczos). On some lower-end workstation, you might need to select a lower quality algorithm to capture every frame.

Inputting and Outputting with Headroom

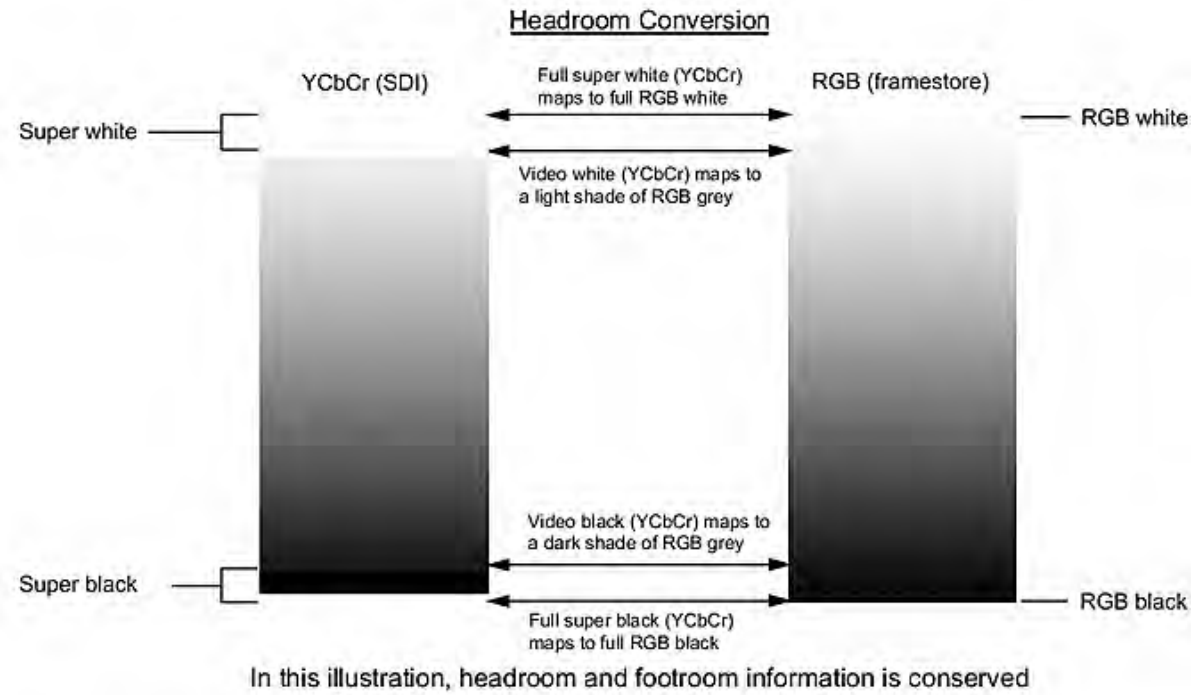
With 4:2:2 serial digital interface (SDI) input and output connections, the colour components of video signals are Y (luma), Cb (blue colour difference), and Cr (red colour difference). For standard video signals in 10 bits, black has a luma value of 64 and white has a luma value of 940. When performing standard captures to RGB values on Flame Premium systems, black YCbCr (64,512,512) maps to RGB (0,0,0) and white YCbCr (940,512,512) maps to RGB (1023,1023,1023). SDI values 0 to 3 and 1020 to 1023 are reserved values for synchronization purposes.

Some cameras record shadow details below the video luma black value of 64 and white detail above the video luma white value of 940. These details are called super blacks and super whites. These are also referred to as “headroom” and “footroom.” Under normal circumstances, headroom is not converted during capture, and the super black and super white details are lost, as described in the following illustration:



In this illustration, headroom and footroom information is lost.

Using the YCbCr<->RGB with headroom colour space option during input on Flame Premium systems, it is possible to capture these super black and super white values. In this case, video black YCbCr (64,512,512) maps to RGB (64,64,64) in 10 bits, and video white YCbCr (940,512,512) maps to RGB (940,940,940). YCbCr Luma values between 4 and 64 and between 940 and 1019 are converted to RGB on capture. While this gives you the advantage of being able to capture super black and super white values, it also means that video black will not map to full black in RGB, and video white will not map to full white in RGB. The following image illustrates this:



In this illustration, headroom and footroom information is conserved

For 4:4:4 RGB standard input and output, a similar explanation applies. On capture using the No Conversion setting, RGB 4:4:4 SDI video black at value (64,64,64) maps to RGB (0,0,0), and RGB 4:4:4 SDI video white at value (940,940,940) maps to RGB (1023,1023,1023).

With the No Conversion with Headroom option, the maximum possible SDI value range is used. RGB 4:4:4 SDI values ranging from (4,4,4) to (1019,1019,1019) map to the same RGB range on capture.

Enabling Colour Space Conversion on Clip Input

Although the preceding descriptions relate to capture from SDI to the framestore, the reverse applies when outputting from the framestore to SDI.

To enable colour space conversion on clip input:

1. In the VTR Input module, click Engineering.
   The Engineering menu appears.



(a) Input Connection box (b) Colour Space box

2. From the Input Connection box, select the input connection.
3. From the Colour Space box, select YCbCr->RGB+Headroom.
4. Capture the material.
   Material is captured with headroom and footroom.

To enable colour space conversion on clip output:

1. Open the clip library with the clip(s) you want to output.
2. Click Output Clip and prepare output clip settings. See Outputting Clips To a VTR (page 187).
3. In the VTR Output module, click Engineering.
4. In the Engineering menu, select the output connection from the Output Connection box.
5. From the Colour Space box, select YCbCr->RGB+Headroom.
6. Output the material.
   Material is output with headroom and footroom.

Capturing 60i Material with 2:3 Pulldown Removal

Using the VTR Input module, you can capture 30 fps film-based material and convert it to 24 fps material in real-time as you capture. The procedure for enabling automatic 2:3 removal differs depending on whether your tape uses non-drop or drop 60i timecode.

To capture 60i material with 2:3 pulldown removal:

1. Set up your capture.
2. In the VTR Input module, click Eng.

The Engineering menu appears. It contains options for removing 2:3 pulldown in real time while capturing.



(a) 2:3 Removal button (b) 24>DF Reference field (c) AA Reference field

3 Enable 2:3 Removal.

4 In the AA Reference field, enter a timecode that corresponds with an AA frame on the tape that is currently in the VTR.

TIP To determine an AA frame, frame-step (`Right arrow`) the tape and look closely at each frame. In 2:3 pulldown, two jitter frames with field artefacts are followed by three normal frames. The second frame after the two jitter frames is an AA frame.

5 When you capture material from a tape that uses 30 fps drop-frame timecode with 2:3 removal enabled, a slight discrepancy between timecodes is introduced due to the reference timecode used to determine the 30 drop frame to 24 timecode conversion. To optimize correspondence between the timecodes, enter the timecode in the 24 >DF Reference field that matches the start reel timecode for the tape.

TIP The start clip timecode can be deduced from the hour-mark of the timecode used by your clips. For example, 1;00;00;00 drop-frame would require a 1;00;00;00 value in the 24p>DF reference field to get a clip starting at 01;00;00+00.

6 Make sure the project's field dominance is set to field1. If the project's field dominance is set to field 2, you must switch the field dominance back to field 1.

7 Click EXIT to return to the VTR Input module.

8 Using the In and Out timecode fields, set the in and out points for the capture session.



(a) In and Out timecode fields (b) 2:3 In and Out pulldown display

With the in and out timecodes set, the corresponding 2:3 pulldown frame is indicated in the In and Out 2:3 pulldown display. Use the 2:3 Pulldown display to determine the 24 fps result in and out frames.

2:3 Pulldown Frame	24 fps Result Frame
In/Out AA	In/Out A
In/Out BB	In/Out B
In/Out DD	In/Out D

2:3 Pulldown Frame	24 fps Result Frame
In BC	In C
In CD	In D
Out CD	Out C

9 Click Process to capture the material with 2:3 pulldown removal.

Configuring Dual-Serial Link I/O on Linux Workstations

Flame Premium supports dual-serial link input and output connections for 4:4:4 clip I/O on Linux® workstations. Configure the type of I/O connection (single- or dual-link serial) using the Input Connection and Output Connection boxes in the Engineering menu. See Setting Video Input and Output Engineering Menu Controls (page 206). When you select an input or output connection for clip I/O, you usually also select the colour space conversion method using the Colour Space box on the Engineering menu. The conversion methods that appear in this box depend on the type of I/O connection used. The following table shows the colour space conversion methods that are available for single-link and for dual-link I/O connections on Linux workstations.

Colour Space Conversion Method	Supported for Single-Link Serial (4:2:2) I/O	Supported for Dual-Link Serial (4:4:4) I/O
YCbCR -> RGB	Yes	No
YCbCR -> RGB + Headroom	Yes	No
No Conversion	No	Yes
No Conversion + Headroom	No	Yes

Clip Output Using Real-Time Deliverables

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
- Frame rate change with deceleration or acceleration with 2:3 insertion

- Audio timewarp
- Letterbox overlay

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

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

Make sure to follow the hardware requirements described in Hardware Requirements for Real-Time Deliverables (page 214).

What Is a Deliverable?

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

A Deliverable is a virtual clip that is associated with the source clip from that point on, in the library or on the Desktop. A clip can have multiple Deliverables, and each Deliverable that you create is itself treated like a clip—a modified version of the original clip. As with clips, you can enter the Output Clip menu and output a Deliverable to tape. See Managing Deliverables (page 221).

Hardware Requirements for Real-Time Deliverables

To follow the Real-Time Deliverables procedures, you must be using an NVIDIA Quadro FX 5600, FX 5800, or FX 6000 SDI graphics card. The configuration for using these cards is described in the *Hardware Setup Guide* for your workstation.

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

```
VideoPreviewDevice 1920x1080@23976psf, nvidia, 1, 1920, 1080, 23976psf, DTVsync
```

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

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

To select the appropriate SDI card in the Output Clip menu:

1. Open the Output Clip menu as described in Outputting Clips To a VTR (page 187).
2. From the Graphics Card box, select GFX SDI.



When GFX SDI is selected, the installed SDI card is automatically used for video output.

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

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

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

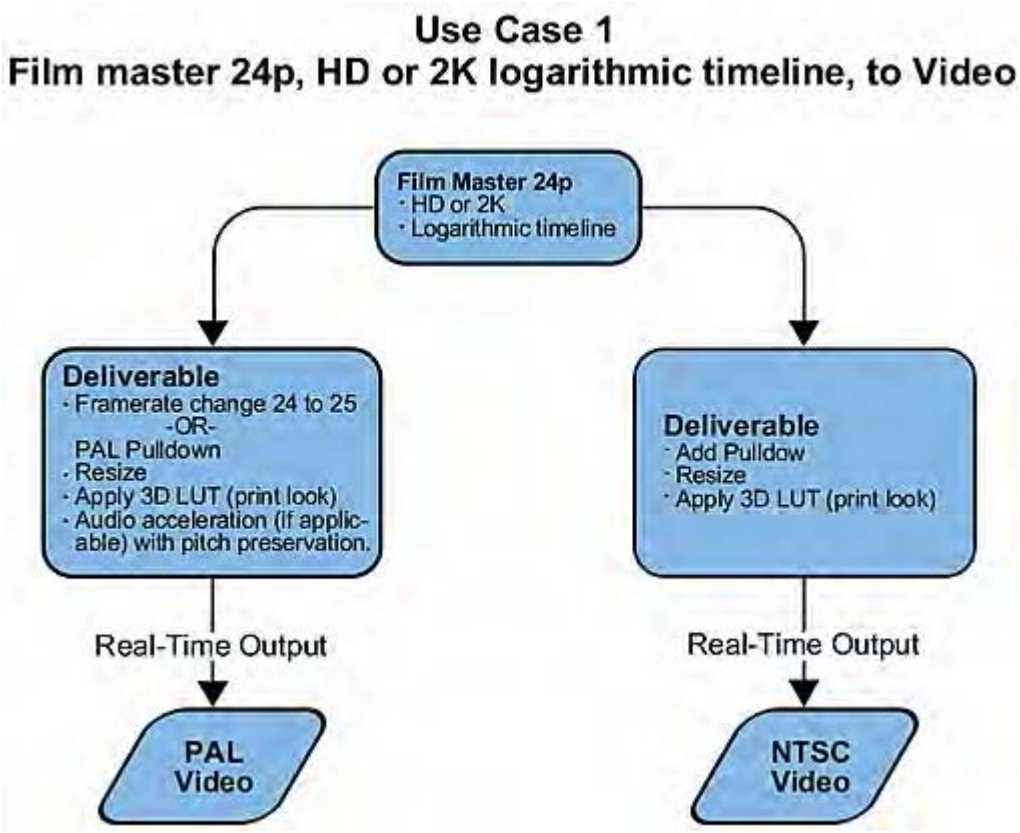
To install or configure an FX 5600, FX 5800, or FX 6000 SDI, see one of the hardware guides previously cited in this topic.

Real-Time Deliverables Use Cases

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

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

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



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

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

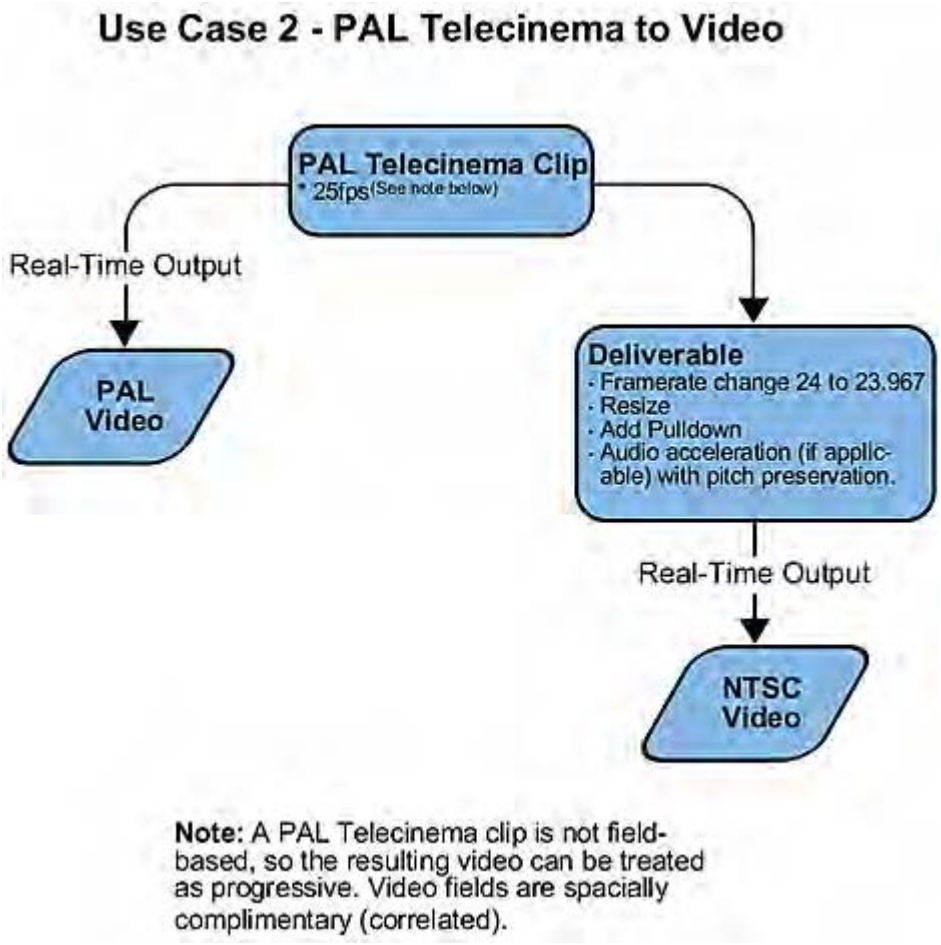
To output to NTSC video, you would create a Deliverable that does the following:

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

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

PAL Telecinema to Video

In this example, NTSC and PAL video are output, but the source material is a PAL Telecinema clip. Because a PAL Telecinema clip is not field-based, the resulting video can be treated as progressive. Its video fields are spatially complementary.



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

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

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

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

Accessing the Real-Time Deliverables Panel

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

To access the Real-Time Deliverables panel:

1. Select the clip you want to work on and open the Player.
2. From the Options box, select Show Real-Time Deliverables.
   The Real-Time Deliverables panel will open to the right of the Player.

Most of the controls in the Real-Time Deliverables panel are common to various parts of the software, and their functions are explained in their respective sections. The following does not describe how to use the controls, but rather how to manage the results using Deliverables and output them to tape in real time. The settings you create in the Real-Time Deliverables panel can be previewed in the Player and are applied in real time during clip output.

To configure real-time operations, see Managing Deliverables (page 221).

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

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

Resize controls Displays values set in the Resize editor, accessible with the Editor button. You can edit the values directly in the Real-Time Deliverables panel. The available destination resolutions and timings depend on the format of the source material.

Audio controls Displays the audio speed. To preserve audio pitch, enable Preserve Pitch.

LUT controls (3D and 1D) Enables or disables the use of LUTs. You can specify a 3D LUT to use, and access the LUT editor.

Stereo controls Enables stereoscopic output settings, if a stereo track is detected. You can select a Stereo Mode option and a corresponding Method option.

Select:	To:
Anaglyph	Output the left eye and right eye as a red/cyan clip. You can reduce ghosting (Dubois), remove RGB values (Mono) or create custom anaglyph results.
Interlaced	Output the left eye and right eye as an interlaced RGB clip. You can output the left eye as field 1 or field 2.
Dual Image - Scale	Output the left eye and right eye so that they are adjacent to each other in the same clip, either in a left and right or top and bottom orientation.
Dual Image - Splice	Ouput the left eye and right eye so that they are side by side in the same clip. Only available when each eye track's horizontal resolution is half that of the output resolution (2 tracks at 960 for a 1920 output).
Dual Output	Output the left and right eye separately to dual streams of a tape.

Letterbox controls Enables or disables a letterbox overlay.

Start TC field Set the start timecode of the Deliverable. The start timecode is used during output, it indicates where on the tape the clip is recorded.

Previewing and Processing Material for Real-Time Output

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

Previewing Deliverables

The real-time operations you create in the Real-Time Deliverables panel can be previewed in the Player.

To preview the result of a Deliverable in the Player:

1 From the Options box, select Show Real-Time Deliverables.
The Real-Time Deliverables panel will open to the right of the Player.

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

You can also preview real-time operations in the Clip Output menu. See Outputting Deliverables (page 219).

Processing Source Clips and Deliverables

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

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

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

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

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

If the GPU Benchmark button indicates that real-time output is impossible, click Process in the resize and frame rate group of the Real-Time Deliverables panel.

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

Outputting Deliverables

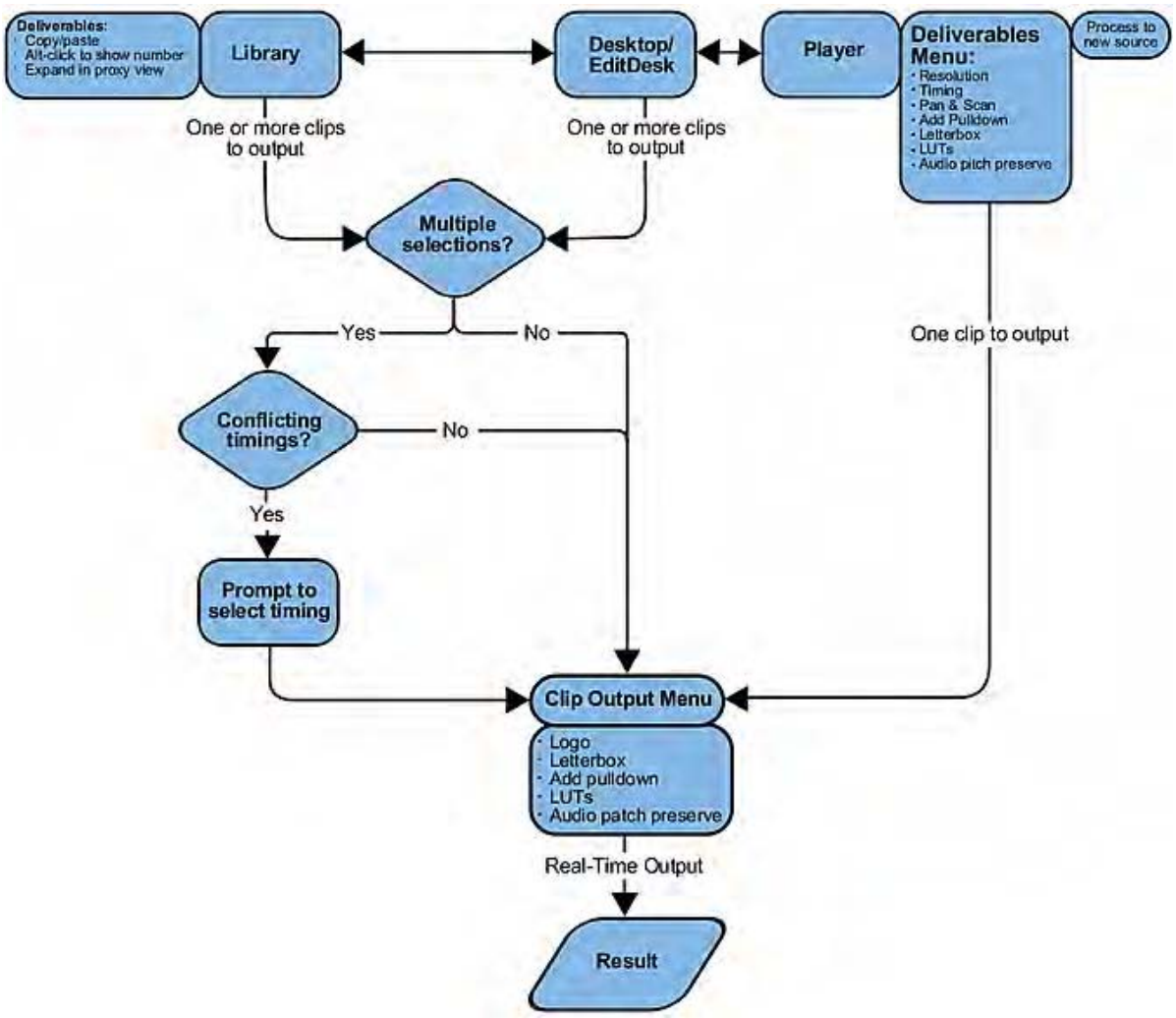
The workflow for outputting Deliverables is like the standard clip output workflow described in Outputting Clips To a VTR (page 187).

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

From the Real-Time Deliverables panel, click the Output Clip button to enter the Output Clip menu with the current Deliverable.

If you enter the Output Clip menu from the library with multiple items (clips, Deliverables, or both), the resulting behaviour is the same as when outputting multiple clips. If there are conflicting timings, you are prompted to select the timing you want to use, and items that do not match this timing are discarded. See Outputting Multiple Clips (page 192).

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



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

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

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

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

1. From the Player, create a Deliverable. See Accessing the Real-Time Deliverables Panel (page 217).
2. Do one of the following:
   - From the Real-Time Deliverables panel, access the Output Clip menu.
   - From the library or Desktop, select the Deliverable or multiple clips and Deliverables, and access the Output Clip menu. If you selected multiple items and they have conflicting timings, you are prompted to select which timing you want to use.
3. In the Output Clip menu, make adjustments to real-time settings if necessary.
4. To preview real-time operations before outputting, use the Player controls in the Real-Time Deliverables panel.
5. Output the material to tape.

Managing Deliverables

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

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

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

Creating and Modifying Deliverables

From the Real-Time Deliverables panel, you can create, rename, and delete Deliverables.

To create a Deliverable:

1. Access the Real-Time Deliverables panel, as described in Accessing the Real-Time Deliverables Panel (page 217).
2. From the Deliverable box, select <New Deliverable>.
3. Type a name for the Deliverable and press `Enter`.

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

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

- The settings of the previous Deliverable are cleared from the Real-Time Deliverables panel.
- The new Deliverable name is added to the list in the Deliverables box.

To rename a Deliverable:

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

To delete a Deliverable:

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

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

Exporting and Loading Deliverables as Templates

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

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

When you load a Deliverable template:

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

Viewing Deliverables in the Desktop

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

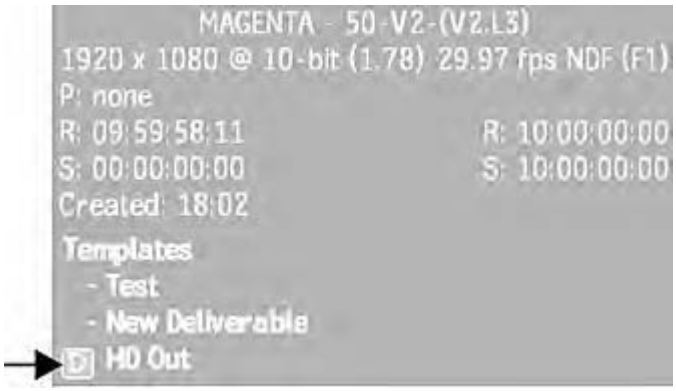
There are two types of "D" overlays, each indicating the selection in the Playback Resolution box:

Grey "D" The Player displays the Deliverable. See Previewing Deliverables (page 219).

Black "D" The Player displays Proxies or Full Res.

To open the Real-Time Deliverables panel from the clip proxy on the Desktop, double-click the "D" overlay.

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



The list of Deliverables is included in the clip information. A "D" appears beside the active Deliverable (the Deliverable most recently selected in the Real-Time Deliverables panel).

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

Transferring Deliverables to Other Systems

When a clip is transferred to an Autodesk Visual Effects and Finishing system using Wire, its Deliverables are also transferred.

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

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

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

Real-Time Deliverables Timing Specifications

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

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

Destination timings 23.967psf through 25p:

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

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

Destination timings NTSC through 60p:

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

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

Archiving in Flame Premium

8

Archiving in Flame Premium saves your media and project setups on external storage devices or in a filesystem. This frees up space for new projects. It is also a convenient way to store your projects offline in a fully restorable form.

A project archive includes all of a project's Media panel content, including the Media library, Batch and Desktop snapshots, as well as all of a project's workspaces.

Or you can archive individual clips from the Media panel.

Choosing a Medium for Archiving

Choosing a medium or device for your archiving needs largely depends on your technical resources and overall needs. There are certain advantages and disadvantages to using each medium/device.

Flame can read and write archives from the following devices: file systems, VTRs, and tape drives.

Smoke can read and write archives from filesystems. While Smoke can read VTR archives, it cannot write to them.

Filesystem Archive

A filesystem archive is an archive stored on a hard disk drive, such as external USB/FireWire® (IEEE 1394) hard drives offers, or networked storage such as a SAN. The device can use any of the formats supported by your workstation, usually ext2 or ext3 for Linux, and HFS+ for Mac. NTFS is not supported.

Using a filesystem to archive your material provides the quickest method of archiving and restoring your material, and offers full support for mixed-resolution projects.

Tape Drives

NOTE Smoke cannot read or write data tape archives.

A tape archive is an archive written directly to a device such as an LTO tape device. Tape provides fast and reliable read and write performance. And, contrary to VTR archives, tape archives can be of any bit depth.

However, data archives (tape archives) can only be restored to a Flame Premium workstation and are unreadable by other applications. Also, magnetic tape is a delicate media and is greatly affected by environmental conditions that cause its deterioration. The procedure of archiving to a tape device is similar to archiving to file.

Flame Premium only supports fibre channel tape devices connected to ports 1 or 4 of a four-port fibre channel adapter, and which the vendor confirms that:

- The device driver is compliant with standard UNIX tape device calls.
- The specific version of your operating system and kernel is supported.

The initialization file (*init.cfg*) for your Flame Premium contains examples of the ClipMgtDevice Tape keyword to help you set up the appropriate block size value for your tape device and define a text label to identify the device in the Archiving module. Please refer to the documentation from your archiving device vendor for guidelines on the actual block size to use.

VTRs

NOTE Smoke only reads VTR archives.

You can use a VTR to archive your material. However, they do have limitations. As a long-term archiving medium, VTR tapes are subject to physical deterioration and format obsolescence. Also, the following clips cannot be archived to a VTR:

- Clips with a colour depth higher than 10-bit
- Clips referencing media, either directly or through clip history, with a resolution that does not match that of the VTR

You can use the following VTRs for archiving your material in Flame Premium:

- Uncompressed SD VTRs such as D-1 or lightly compressed SD VTRs such as Digital Betacam
- Uncompressed HD VTRs or lightly compressed HD VTRs such as Sony HDCAM SR or Panasonic HD D5
- Compressed HD VTRs such as Sony HDCAM (using "HDCAM" rather than "VTR" as the archive device type)

Archiving a Project to a File-based Archive

Archiving a project to a file archive (as opposed to a VTR tape archive) consists of opening an existing archive file, or creating a new archive file, on storage such as an external hard drive.

To archive a project to a new archive:

1. In the MediaHub, enable Browse for Archives.
2. In the Local Devices list, navigate to the location where you want to store the archive.
   This can be a local or remote location.
3. Click New Archive.
4. In the New Archive Creation, enter the required information and click Create.
   Flame Premium creates and opens the archive. You can now archive the project.
5. Set as required the options displayed in the Archive Options tab.
6. Click Archive Project.
   Flame Premium starts archiving the project. The actual duration of the archive process depends on the size of the project and on the options you enabled or disabled. To cancel the process, click anywhere on the screen; you cannot use the application for anything else while Flame Premium archives your project.
7. Click Close Archive to finalize the archive.

To add a project to an existing archive:

1. In the MediaHub, enable Browse for Archives.
2. In the Archives list, double-click the archive to open.
3. Set as required the options displayed in the Archive Options tab.
4. Click Archive Project.

Flame Premium starts archiving the project. The actual duration of the archive process depends on the size of the project and on the options you enabled or disabled. You can cancel the process at any time by clicking anywhere on the screen; you cannot use the application for anything else while Flame Premium archives your project.

5 Click Close Archive to finalize the archive.

Restoring a Project from a File-based Archive

You can only perform the complete project restoration described below if the project to restore *does not* already exist on the workstation. You can see a list of all the projects in **MediaHub ➤ Projects ➤ Local Projects**.

You can delete projects from **Flame Premium ➤ Project and User Settings**.

And even if the project exists on your workstation, you can always restore manually selected clips from the project archive.

To restore a project from a file archive:

1 In the MediaHub, enable Browse for Archives.

2 From the Archives list, double-click the archive to open it.

3 Navigate the archive, and select the project to restore. Projects are usually created at the root of an archive file.

4 Click Restore Project.

Flame Premium starts restoring the project. The actual duration of the archive process depends on the size of the project. You can cancel the process at any time by clicking anywhere on the screen: you cannot use the application for anything else while Flame Premium restores your project. Once restored, you can access the project through **Flame Premium ➤ Project and User Settings**.

Archiving Clips to a File-based Archive

Archiving clips to a file archive (as opposed to a VTR tape archive) consists of opening an existing archive file, or creating a new archive file, on storage such as an external hard drive.

To create a new archive and archive material:

1 In the MediaHub, enable Browse for Archive.

2 In the Local Devices list, navigate to the location where you want to store the archive.

3 Click New Archive, and enter the required information.

4 Click Create.

Flame Premium creates and opens the archive, ready for you to use.

5 Set as required the Archive Options.

6 Drag and drop clips and folders to archive from the Media panel to the archive.

NOTE What you add to the archive appears greyed out in the archive, indicating that it is yet to be committed to the archive: ready to be archived, but not yet copied to the archive. The Pending Archive folder can also be used as a summary of what is being archived.

7 Click Archive to start the archiving process.

The actual duration of the archive process depends on the size of the material being archived and on the options you enabled. You cannot use the application for anything else while Flame Premium archives the material. Cancel the process at any time by clicking anywhere on the screen.

To add material to an existing archive:

1. In the MediaHub, enable Browse for Archive.
2. In the Archives list, double-click the archive to use for archiving.
3. Drag and drop clips and folders to archive from the Media panel to the archive.

   NOTE What you add to the archive appears greyed out in the archive, indicating that it is yet to be committed to the archive: ready to be archived, but not yet copied to the archive. The Pending Archive folder can also be used as a summary of what is being archived.

4. Click Archive to start the archiving process.

   The actual duration of the archive process depends on the size of the material being archived and on the options you enabled. You cannot use the application for anything else while Flame Premium archives the material. Cancel the process at any time by clicking anywhere on the screen.

Restoring Material from a File-based Archive

To restore material from an archive:

1. In the MediaHub, enable Browse for Archive.
2. In the Archives list, double-click the archive that contains the material to restore.
3. Drag and drop clips and folders from the archive to the Media panel.

   NOTE Material you drag and drop does not appear in the Media panel until you click Restore. Use the Pending Restore folder to view what will get restored.

4. Click Restore.

   The length of the restoration process depends on the size of the material being restored. You cannot use the application for anything else while Flame Premium restores the material. Cancel the process at any time by clicking anywhere in the MediaHub.

Tips for Organizing Archives

Although the way you organize your archives is a matter of personal preference, the following guidelines may be helpful:

- After you have processed a clip, remove any material that you are no longer using before you create your archive.
- Try to keep your libraries clean at all times. Remove clips that are no longer being used or earlier unused versions of your work.
- Use a consistent and clear naming technique so that you can easily find your material when you want to create or open an archive.
- Consolidate clips before archiving in order to save space.

Configuring a Data Tape Device

Before you start archiving to a data tape device, configure both the hardware and the software.

Software Configuration

Before you start Flame Premium to archive, edit the following keyword in the software initialization configuration file.

- CLIPMGTDEVICE: Define as many tape devices as you need; only those detected at start are actually listed in the Archives browser.

NOTE The software initialization configuration file is located here: */usr/discreet/Flame Premium_VERSION/cfg/init.cfg*.

Hardware Configuration

Setting the Default Block Size for New Tape Archives

In **MediaHub ➤ Browse for Archives ➤ Archive Options** when you format a new tape archive, the default block size matches the value in the Block Size field.

Keep track of the block size value for each tape in case you need to reset the block size at a later time.

To verify the block size of your tape device:

1. In a command shell, log in as root.
2. Verify the block size of your tape device by typing:

   `mt -f /dev/st <archive device number> status`

Example:

```
mt -f /dev/st0 status
```

This command provides feedback similar to the following:

```
SCSI 2 tape drive:

File number=0, block number=0, partition=0.

Tape block size 65536 bytes.

Density code 0x40 (DLT1 40 GB, or Ultrium).

Soft error count since last status=0

General status bits on (41010000):

BOT ONLINE IM_REP_EN
```

Determining the Device Number for the Archive Device

If you receive an error message when verifying the block size, you may not be using the correct archive device number. Look in the software initialization configuration file to determine the correct number.

To determine the archive device number:

1. In a terminal, type:

   dlcfg
2. Look for the line that begins with the following:

   `ClipMgtDeviceTape, /dev/st<archive device number>`

   The archive device number should be listed on this line. If it is not, or appears to be incorrect, consult the administrator for your system.

Resetting the Block Size for an Existing Tape Archive

If you rebooted the machine or reformatted another archive with a different block size, you must reset the block size for your archive in the command shell.

To reset the block size of your tape:

1. In the command shell, log in as root.
2. Set the correct block size by typing:

   mt -f /dev/st`<archive device number>` **defblksize**`<block size of your tape device>`

   Example:

   mt -f /dev/st0 defblksize 4096

   The block size for your tape is reset and you can continue to archive to this tape.

Archiving a Project to a Tape Device Archive

Archiving a project to a tape device archive consists of opening an existing tape archive, or creating a tape archive using a tape device such as a DLT.

To archive a project to a blank tape:

1. Load the blank to the tape device.
2. In the MediaHub, enable Browse for Archives.
3. In the Local Devices list, double-click the tape device to open it.
4. Click Format to create a new archive.

   WARNING Formatting the tape erases all data already present on it. If you want to add to an existing archive, click Open, and follow the procedure described below.
5. In the New Archive Creation window, enter the required information and click Create.

   Flame Premium creates and opens the archive in the MediaHub. You can now archive the project.
6. Set as required the options displayed in the Archive Options tab.
7. Click Archive Project.

   Flame Premium starts archiving the project. The actual duration of the archive process depends on the size of the project and on the options you enabled or disabled. To cancel the process, click anywhere on the screen; you cannot use the application for anything else while Flame Premium archives your project.
8. Click Close Archive to finalize the archive.

To add a project to a tape already containing an archive:

1. Load the tape to the tape device.
2. In the MediaHub, enable Browse for Archives.
3. In the Local Devices list, double-click the tape device to open it.
4. Click Open to open archive.

   WARNING Do not select Format, as this would erase all data on the tape.

5. Click Archive Project.

   Flame Premium starts archiving the project. The actual duration of the archive process depends on the size of the project and on the options you enabled or disabled. You can cancel the process at any time by clicking anywhere on the screen; you cannot use the application for anything else while Flame Premium archives your project.
6. Click Close Archive to finalize the archive.

Restoring a Project from a Tape Device Archive

You can only perform the complete project restoration described below if the project to restore *does not* already exist on the workstation. You can see a list of all the projects in **MediaHub ➤ Projects ➤ Local Projects**.

You can delete projects from **Flame Premium ➤ Project and User Settings**.

And even if the project exists on your workstation, you can always restore manually selected clips from the project archive.

To restore a project from a tape archive:

1. Load the tape with the archive to the tape device.
2. In the MediaHub, enable Browse for Archives.
3. In the Local Devices list, double-click the tape device to open it and select Open.

   WARNING Do not select Format, as this would erase all data on the tape.

4. Navigate the archive, and select the project to restore. Projects are created at the root of an archive tape.
5. Click Restore Project.

   Flame Premium starts restoring the project. The actual duration of the archive process depends on the size of the project. You can cancel the process at any time by clicking anywhere on the screen: you cannot use the application for anything else while Flame Premium restores your project. Once restored, you can switch to the restored project using **Flame Premium ➤ Project and User Settings**.

To restore project setups from a tape archive:

1. Load the tape with the archive to the tape device.
2. In the MediaHub, enable Browse for Archives.
3. In the Local Devices list, double-click the tape device and click Open.

   WARNING Do not select Format, as this would erase all data on the tape.

4. Navigate the archive, and open the project that contains the setups to restore.
5. Locate and select the Project Setups folder.

6 Click Restore Setups.

Flame Premium starts restoring the project. The actual duration of the archive process depends on the size of the project. You can cancel the process at any time by clicking anywhere on the screen: you cannot use the application for anything else while Flame Premium restores your project. Once restored, you can switch to the restored project using **Flame Premium ➤ Project and User Settings**.

Archiving Clips to a Tape Device Archive

Archiving material to a file archive (as opposed to a VTR tape archive) consists of opening an existing archive file, or creating a new archive file, on a DLT tape device.

To archive a project to a blank tape:

1 Load the blank to the tape device.
2 In the MediaHub, enable Browse for Archives.
3 In the Local Devices list, double-click the tape device to open it.
4 Click Format to create a new archive.

WARNING Formatting the tape erases all data already present on it. If you want to add to an existing archive, click Open, and follow the procedure described below.

5 In the New Archive Creation window, enter the required information and click Create.

Flame Premium creates and opens the archive in the MediaHub. You can now archive the project.

6 Drag and drop clips and folders to archive from the Media panel to the archive.

NOTE What you add to the archive appears greyed out in the archive, indicating that it is yet to be committed to the archive: ready to be archived, but not yet copied to the archive. The Pending Archive folder can also be used as a summary of what is being archived.

7 Click Archive to start the archiving process.

The actual duration of the archive process depends on the size of the material being archived. You cannot use the application for anything else while Flame Premium archives the material. Cancel the process at any time by clicking anywhere on the screen.

To add clips to a tape already containing an archive:

1 Load the tape to the tape device.
2 In the MediaHub, enable Browse for Archives.
3 In the Local Devices list, double-click the tape device to open it.
4 Click Open to open archive.

WARNING Do not select Format, as this would erase all data on the tape.

5 Drag and drop clips and folders to archive from the Media panel to the archive.

NOTE What you add to the archive appears greyed out in the archive, indicating that it is yet to be committed to the archive: ready to be archived, but not yet copied to the archive. The Pending Archive folder can also be used as a summary of what is being archived.

6 Click Archive to start the archiving process.

The actual duration of the archive process depends on the size of the material being archived. You cannot use the application for anything else while Flame Premium archives the material. Cancel the process at any time by clicking anywhere on the screen.

Restoring Material from a Tape Device Archive

To restore material from an archive on a tape device:

1. Load the tape containg the archive to the tape device.
2. In the MediaHub, enable Browse for Archives.
3. In the Local Devices list, double-click the tape device and click Open.

   WARNING Do not select Format, as this would erase all data on the tape.

4. Drag and drop clips and folders from the archive to the Media panel.

   NOTE Material you drag and drop does not appear in the Media panel until you click Restore. Use the Pending Restore folder to view what will get restored.

5. Click Restore.

   The length of the restoration process depends on the size of the material being restored. You cannot use the application for anything else while Flame Premium restores the material. Cancel the process at any time by clicking anywhere in the MediaHub.

About VTR Archives

NOTE Smoke can only read VTR archives, it cannot create nor write to VTR archives.

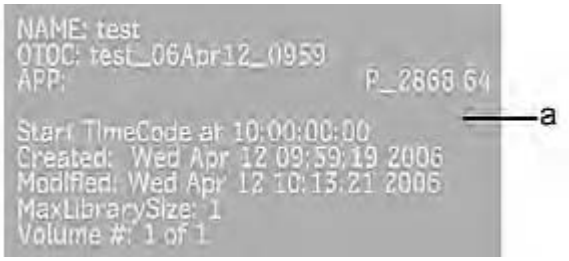
You can create a VTR archive for clips of any resolution supported by your video board and VTR device. To archive and restore from a VTR archive, your clips FPS, resolution and bit depth need to match your VTR tape device settings.

When you change projects without exiting Flame Premium, the video timing selected at start-up applies to the project you are switching to.

Header Information Slate

All VTR archives contain a Header Info Slate, which is a single frame providing information about the archive. Located in the archive before the table of contents, the Header Info Slate provides information such as:

- Name of the archive
- Name of the Online Table of Contents (OTOC)
- Name of the workstation where the archive was created and name and version of the Autodesk application used for archive creation
- Start timecode
- Creation date and time
- Modification date and time
- Minimum size of the clip library buffer (in MB) required to restore the archive



(a) Header Info Slate example

Configuring a VTR for Archiving

Before you start archiving to a VTR, configure both the hardware and the software.

Also make sure that the video hardware and application match the FPS, resolution, and bit depth of the clips, or project, you are archiving.

Hardware Configuration

- Your VTR device is properly connected to video and audio sources and RS-422.
- The sync is properly set up.

Software Configuration

Before you start Flame Premium to archive, edit the following keywords according to the software initialization configuration file. You'll need to restart Flame Premium for these changes to take effect.

NOTE The software initialization configuration file is located here: */usr/discreet/Flame Premium_VERSION/cfg/init.cfg.*

- VTR section: Enable every VTR that you plan to use, either as an archive device or as a regular VTR.
- VIDEO: Enter the video card used by the workstation.
- CLIPMGTDEVICE: Make sure there is a defined VTR unless you plan on using an HDCAM VTR to restore archives, in which case you would select HDCAM. For HDCAM SR, you must select VTR. Note that CLIPMGTDEVICE can define more than just VTR, it can also list File and Tape: they are not mutually exclusive.

  NOTE You can only restore an HDCAM-created archive by using an HDCAM VTR.

Creating a New VTR Archive

Before archiving to VTR, do the following:

1. Prepare striped tapes for each resolution.

   The tapes must be striped continuous non-drop-frame timecode.
2. If the tapes already have material on them, figure out the start timecode for the archive.
3. Determine if your archive requires multiple tapes.
4. Make sure your VTR is in TC (Timecode) timer mode.

   If you are working with an HD board and you selected the 1920x1080 video timing option in the Video Timing box when you started your project, you should specify the exact 1080 line video timing (including framerate) that you are using in the 1080 Video Timing box in the Engineering menu.

To create a VTR archive:

1. Insert a striped tape with continuous non-drop-frame timecode in the VTR.
2. In Flame, select **MediaHub ➤ Browse for Archives**.
3. Select the VTR from the list of Local Devices.
4. Optional: Click Engineering and set Engineering options, and then return to the MediaHub.

   NOTE When you open an archive in read/write mode, play delay and pre- and postroll values are verified and set in the auto test run before archiving begins. Therefore, you do not need to set these options.

5. Click New Archive and enter the required information.

   NOTE If you create the archive on a tape that already contains material, make sure you define the length of the archive as the space you want the archive to use, not the length of the tape.

6. Click Format to ready the tape to receive the contents you will archive.

   The new archive is created on the VTR tape and opens automatically in the MediaHub.
7. Drag and drop clips from the Media Panel to the archive, or click Archive Project to archive the whole project.
8. Click Close Archive to commit the clips and files to tape.

TIP Write down the archive name and creation date on the tape label. This will help you to locate the appropriate table of contents file if you need to restore your archive.

About the Listed VTR

Only those VTRs with a timing that matches the video timing are available for use in the MediaHub. When you want to archive or restore clips of multiple resolutions in one project, enable the VTR lines in the VTR keyword of the configuration file for all the resolutions you use on the project. This way, each time you change the video timing for archiving, the compatible VTR is available in MediaHub.

When you archive to a VTR, only clips with a resolution compatible with the selected VTR device are archived. If you select a set of clips of differing resolutions, only the compatible clips are archived. Clips that are not compatible are greyed out in the library.

VTR Archives and Mixed Resolutions

Generally, archiving to data tape or filesystem is easier for archiving mixed-resolution projects, as clips of any resolution can be archived on the same medium.

While it is possible to archive mixed resolutions projects to a VTR, it will require different tapes with timings matching each segment resolution. Alternatively, you can use the Unlink tool to unlink media from clips of any resolution, archive unlinked clips, and then relink the media at restore time.

NOTE Before starting to archive a mixed-resolution project to a VTR, make sure you have enough tapes matching each resolution used in your project.

Archiving Clips Containing Sources with Different Bit Depths

If you want to archive a clip that contains sources with different bit depths, archive the clip to file or data tape. To archive the clip to a VTR, you must convert all sources to the same bit depth (less than 12).

8- and 10-bit VTR Archiving

A VTR archive can contain 8- or 10-bit clips—although all clips in the archive must have the same bit depth. When archiving 8-bit material to 8- or 10-bit VTRs, the clip metadata remains in an 8-bit format.

You can archive 10-bit material to 8- or 10-bit VTRs. However, when archiving 10-bit material to an 8-bit VTR, you reduce the quality of the archived material. A warning message appears notifying you of the downgrade. 10-bit clips archived to 8-bit format are flagged as such in the archive. Similarly, when archiving 12-bit material to 8- and 10-bit VTRs, a warning message appears, and the 12-bit clips are flagged in the archive. You cannot restore downgraded clips to their original bit depth.

Before archiving to any device in Flame Premium, you can perform archive size estimation. Before proceeding, make sure you have enough tapes ready. You can perform size estimation before archiving to a filesystem to make sure you have enough space on disk.

About Using an HDCAM SR for Archiving

The HDCAM SR device supports 4:2:2 and 4:4:4 archiving. The options required in the Engineering menu are as follows.

Input and Output Connection	Colourspace
In Serial 4:2:2 / Out Serial 4:2:2	YCrCb-->RGB
In Serial Dual 4:4:4 / Out serial Dual 4:4:4	No Conversion

When you switch between 4:2:2 and 4:4:4 options, the Colourspace option changes automatically to ensure the proper settings.

Archiving to a VTR Tape Containing Material

You can create a VTR archive on a tape that already contains material or add material to an archive on a tape. For example, on a 60-minute tape, you could allocate 20 minutes to an archive by setting its start timecode at 01:00:00:00 and defining the length of the archive as 20 minutes in the Length field. You can then add any material, starting after 01:20:00:00.

Restoring Material from a VTR Archive

NOTE When you open a VTR archive in read/write mode, Flame Premium performs an auto-test. The auto-test involves checking whether it can write, read, and verify the integrity of the archived material. This auto-test also scans binary and play/record delay information to verify the quality and accuracy of the video signal.

1. Insert the archive tape in the VTR.
2. In Flame Premium, select **MediaHub ➤ Browse for Arcives**.
3. Select the VTR from the list of Local Devices.
4. Select the start timecode by doing one of the following:
   - Select Autostart if you selected this option when you created the archive (the archive rewinds the tape to find the correct start timecode).
   - Select Manual Start and enter the appropriate timecode if you selected this option when you created the archive.

5 Do one of the following:

- Click Open Archive. Select Open Read/Write or Open Read Only from the Open Mode box.

  NOTE When you try to open a VTR archive in read/write mode, Flame Premium checks for machine errors (for example, dirty heads). If a problem is detected, the archive does not open. If the tape is write-protected, the archive opens in read-only mode.

- Click the Open Using OTOC button and select the OTOC in the file browser. You are given the option to read slates from the tape. Answer 'Y' if you wish to see clip proxies.

The Flame Premium cues the VTR. Once it's done reading the archive, displays its contents in the MediaHub.

6 Restore clips one by one using drag and drop, or click Restore Project to restore a full project.

7 Click Close Archive once you are done.

VTR Archiving Tips

Troubleshooting

The following tips may help avoid some common VTR archiving problems:

- Do not attempt to archive material captured with Headroom as the Output mode. Such material cannot be archived.
- Check the scan mode of Flame Premium and make it consistent with your VTR and VCP settings. Use the F1 Scan mode in the Engineering menu. You cannot create an archive on a VTR when the Scan Mode option in the Engineering menu is set to F2.
- Archives must be written to tapes striped with non-drop-frame timecode. However, media with drop-frame timecode can be written to non-drop-frame timecode tapes, as long as the framerate of the timecodes match.

Tips for Better VTR Archiving

Use the following tips to get better results when archiving to a VTR tape:

- Make sure the input from the computer to the VTR is connected properly. If it is not, yet the output from the VTR to the computer is correct, everything will appear normal when archiving and the monitor will show your material being saved. However, black or random noise is actually being saved to tape.
- Turn the video breakout box on *before* powering up the computer.
- On the video breakout box, make sure that your house sync is connected to the Sync Genlock and *not* to the Sync port in the GBRA/YUVA component. The sync should be connected to the Genlock input of the computer and be terminated with a 75-ohm terminator.
- Set the system reference of the VTR to External Analog.
- Track the VTR to get the best RF (Reference) value and error-rate level. If your VTR has an auto-tracking feature, disable it and track manually, or place it on “one shot.”
- Check the following Engineering menu settings:
  - Timecode type: Make sure the VITC & LTC timecodes are the same. If they are not, set timecode type to LTC.
  - Input format and sync: Make sure that input format and sync match the device.
  - Output format and sync: “House” is the recommended output sync.
  - The required Scan Mode option is F1.

Archiving to Multiple Volumes

If the items you select for archiving exceed the capacity of the archiving device, Flame Premium splits the archive across multiple devices of the same type. The splitting of archives is seamless and virtually transparent to the archiving process.

The procedure for creating a multiple tape archive (an archive that is too large for a single tape) is almost the same as for creating an archive on a single tape. If the entries selected for archiving exceed the capacity of the tape, a message appears indicating that the archive requires multiple tapes. The archiving process begins for the first tape. When the first tape is full, you are prompted to insert a second tape. Insert and format this tape and continue with the archiving. This process continues for as many tapes as are required.

Opening Multi-Tape Archives

When creating large archives, you can use multiple volumes to store one archive. By splitting the contents of one archive onto separate tapes, you can keep similar types of material together in the same archive.

You can open material on any tape in a multiple tape archive but you need the last tape in the multiple tape archive sequence. The last tape contains necessary header information for locating the material you want to restore.

To open material from a multi-tape archive:

1. Insert the last tape in the VTR or tape device.
2. In the MediaHub, browse for Archives.
3. Select the VTR or tape device from the list of Local Devices.
4. Click Open Archive.
5. Follow the instruction.
   You can now restore a project or any part of the archived material.

Supplementing Archives with System Drive Backups

The recommended backup strategy for your Flame Premium workstation is to archive project media and setups to an archive that can be saved to a remote file server or storage device. Do this as often as necessary to protect your media from storage or system failure.

The archiving approach, however, does not protect the data contained on your workstation's system drive. The system drive should not contain any media data, but it does include important project management data. In the event of hardware failure without a system drive backup, you will need to recreate this project data for each archive you restore. This can be a time-consuming and frustrating process, and is a significant issue in large SAN environments, where large quantities of media files are referenced from a shared standard FS volume. In such environments, consider using the data backup method of your choice to save key system drive information. This will facilitate restoration of multiple project archives.

Even with a system drive backup, you will still need to reinstall your software and restore archived projects in the event of a system drive failure.

There are several methods and commercial tools available for backing up system drives. Choose the method and tool that is right for you. Autodesk does not recommend any particular method or tool, and only presents

the directories of note that you should consider backing up. Depending on your requirements and available tools, you may decide to back up your full system drive or only the following critical files or directories.

File/directory	Purpose	Consequence of failure if system drive is not backed up
/usr/discreet/clip	Contains all clip metadata.	Metadata will have to be recreated for each valid archive that is restored.
/usr/discreet/project	Contains full project structures, including settings such as names, resolutions, user information, and so on.	Project settings will have to be recreated for each valid archive that is restored.
/usr/discreet/sw/swdb	Contains the standard FS links database.	Media on a shared SAN volume will have to be re-imported.
/usr/discreet/sw	Contains all Stone® and Wire software and configuration files, including the preceding directory, swdb.	Storage and Wire settings will have to be reconfigured after installation.
/usr/discreet	Contains your product software, as well as all other directories described in this table.	See consequences for all previous items. Back up this directory if you want to be sure to restore all critical data, and do not mind including some superfluous files (such as old unused versions of the application.)

Browsing an Archive Offline

Each archive includes a header file, which is located at the beginning of the archive and contains the metadata necessary for properly restoring the archive. When you create an archive, two versions of the same table of contents are created: one as an ASCII file, and another as an HTML one. Created by default in `/usr/discreet/archive`, and displayed in the list of Archives in **MediaHub ➤ Browse for Archives**, they can be used in cases where the header information becomes corrupted.

You can view the table of contents with the ASCII or HTML TOC.

To view the contents of an archive using its HTML TOC:

1. Open **MediaHub ➤ Browse for Archives**.
2. Select the archive you want to view from the the Archives list.
3. Click View Content.

   A Web browser opens the HTML TOC, allowing you to browse the contents of the archive without actually opeing it.

To view contents of an archive using its ASCII TOC:

1. Open the ASCII version of the table of contents using a text editor to view the contents of an archive without opening it. ASCII TOC filenames have the following format:

<archive name>_<creation date>.atoc

Code	Meaning
P	Project
S	Project setups
W	Workspace or Shared folder
X	Desktop snapshots
L	Folder
R	Reel
C	Clip
E	Source clip
M	Media library
B	Batch snapshots
F	Snapshot folder
D	Desktop

Online Tables of Contents

The table of contents lists the contents of the archive, as well as information such as the order in which clips are assembled on the archive, clip IDs, transitions, and timecodes. When you restore an archive with the table of contents, Flame Premium uses this information to restore the material.

A copy of the table of contents is saved in the filesystem. This copy is referred to as the Online Table of Contents (OTOC). You can open an archive in read-only mode using the OTOC. In read-only mode, you can load but not save or delete entries from the archive. With VTR archives, you can open an archive in read-write mode using the OTOC, and save and delete entries from the archive.

The OTOC is useful for:

- Recovering material in an archive when the table of contents on the medium is corrupted.
- Viewing the contents of an archive without opening it, which is often faster than actually opening the archive.

When an OTOC is created, Flame Premium creates an ASCII text copy of it (ATOC) as well as an HTML and XML copy. Use the ASCII and HTML copies to view the contents of an archive without opening it. For example, use them to view the contents of a VTR archive without connecting to the VTR.

Use the XML TOC to easily populate a database with information about your archives.

The OTOC, ATOC, HTML, and XML table of contents are saved by default to `/usr/discreet/archive`. Or define the location by adding the following token to the `init.cfg` file:

ArchiveLibrary`<directory>`

where `<directory>` is the location for storing your archives.

The OTOC is updated each time you close the archive.

Deleting Tables of Contents

To prevent the loss of important data, Flame Premium never overwrites the previously created tables of contents. If the archive becomes obsolete or if you are sure that older versions of an archive OTOCs are obsolete, you can delete tables of contents. The following name formats are used for the tables of contents.

Type	Format
Online (two files)	`<archive name>_<creation date>`.otoc `<archive name>_<creation date>`.otocx
ASCII	`<archive name>_<creation date>`.atoc
XML	`<archive name>_<creation_date>`.xml
HTML (two or more files)	`<archive name>_<creation_date>`.html `<archive name>_<creation_date>`/*

where `<archive name>` is the name that you type in the Name field when you create the archive.

Recovering a Corrupted Archive

If the table of contents of an archive on tape is corrupted, Flame Premium may not be able to read it. If this happens, open the archive using the OTOC.

Once you open the archive, you can restore the entire contents of the archive and save it to a new tape. Do this, for example, if the tape has been damaged.

For VTR archives, you can also overwrite the corrupted table of contents by saving a single frame to the archive. If the table of contents is successfully overwritten, you can open the archive using the Open button.

To restore the archive:

1. Open the archive.
   If the table of contents is corrupted, the following message appears:
   `CLIP MGT: Warning! Cannot read archive header. Select online TOC?`
2. Click Confirm to open the OTOC.
   The file browser appears.
3. Select the OTOC for the archive you are trying to open and click Confirm.
4. If the archive is on a VTR, a prompt asks if you wish to retrieve the slates (proxies).
   After a moment, the archive appears.

NOTE When you open an archive with the OTOC, proxies only appear if the archive is on a VTR, and you answered Yes to the prompt. For all other types of archives, proxies appear as black images.

5 Select the entire contents of the archive. Make sure you have sufficient space on your storage for the contents of the archive.

6 Click Restore.

7 When the restore is complete, click Close.

The restored clips appear in the selected clip library.

8 Re-archive the material onto a new tape.

VTR Only: To overwrite a corrupted table of contents:

1 Open the archive.

If the table of contents is corrupted, the following message appears:

```
CLIP MGT: Warning! Cannot read archive header. Select online TOC?
```

2 Click Confirm to open the OTOC.

The file browser appears.

3 Select the OTOC for the archive you are trying to open and click Confirm.

A prompt asks if you wish to retrieve the slates (proxies).

4 Click Y to see proxies in the archive or N to have no proxy images.

After a moment, the archive appears.

5 From the Library Type box, select a clip library.

6 Select a reel entry by `Alt`-clicking the reel and then click Archive.

NOTE When you select a reel, no media is appended to the archive but the header is still forced to update.

7 Click Close to close the archive.

8 Click Open to open the archive.

If the archive still does not open using the table of contents on the tape, restore the entire archive.

Restoring Legacy Archives

In earlier versions of Inferno, Flame, and Smoke, archives did not include all the metadata that the current version needs to restore archives with the correct scan mode and aspect ratio. The Default Resolutions menu makes it possible to assign the scan mode and aspect ratio preferences to older archives. The information you need to assign depends on the software version in which the archive was created. Assign the information using the Default Resolutions menu. Use the following table to determine which information you need to assign to your old archive.

Archived clips prior to:	Information you must assign:
Smoke 6.0, Fire® 6.0, Inferno 5.5, Flame 8.5, Flint 8.5, and Backdraft 5.5	Scan mode
Smoke 5.2, Fire 5.2, Inferno 5.0, Flame 8.0, Flint 8.0, and Backdraft 5.0	Scan mode and aspect ratio

The parameters listed in the Default Resolutions menu act as default values for all listed resolutions. For example, if you want to change the default values for PAL clips, you must change the values for the PAL

entry, which is the entry that has a width value of 720 and a height value of 576. If you are working with a PAL archive created in an earlier version, the aspect ratio would have automatically been set to 1.3333, but you may want to restore all your PAL archives with 16:9 aspect ratio. You can change this setting by changing the aspect ratio to 16:9 for the PAL entry.

NOTE You should only restore an old clip at its original resolution. If you restore versions of the same clip at different resolutions, you may encounter problems with the clip.

To add a default resolution:

1 In the MediaHub, browse for Archives and click Default Resolutions.

The Default Resolutions list and menu are displayed.





(a) Add Resolution box (b) Apply Changes button (c) Width and Height fields



(d) Aspect Ratio field (e) Aspect Ratio box (f) Scan Mode box (g) Framerate box

2 Add new preferences to correspond to a specific clip resolution contained in the archive that you plan to open.

3 Click Add Resolution.

4 Click Apply Changes.

The new archive preference is added to the list. The next time you restore an archive missing either Scan mode or aspect ratio, Flame Premium knows how to process that material.

TIP When you decide to open a VTR legacy archive, set the Open Mode box to Open Read Only to keep you from mixing new content with the old.

Restoring Pre-2013 Extension 1 Flame Projects

On Flame Reactor configurations, using Project Restore restores every project as a Flame Reactor-enabled project, even if that project did not use Flame Reactor originally (as is that case with every pre-2013 extension 1 projects). Due to differences in rendering engines, you might want to the restored project to use the Classic Engine instead of the newer Flame Reactor.

To switch back to the old Classic Engine:

1 Quit Flame and restart.

2 From the Project box, select the restored project.

3 Click the Edit Project button.

4 From the Rendering box, select Classic Engine, and then click Done.

5 Click Start.

Deleting Content from an Archive

To ensure data integrity and prevent archive corruption, you cannot delete entries from an archive.

MediaHub Reference: Browsing for Archives

Archives Buttons

With No Opened Archive

New Archive button Creates a new archive in the location displayed in the file browser. Opens the New Archive Creation dialogue box.

View Content button Displays basic information about the contents of the selected archive, in a web browser. This does not open the archive.

Open Archive button Opens the selected archive or device.

With an Opened Archive

Close Archive button Closes the archive and returns to the file browser.

Archive Project button Adds the current project to the archive; this includes every Workspace of the project, Shared Folders, as well as Batch and Desktop; also includes all the setups.

Clear Pending Tasks button Clears the Pending Archive or Pending Restore folders. This does not delete any material in the archive or in the Media panel, it only clears out the queue.

Archive/Restore button Commits the Pending folders, either to archive or restore material clips. Nothing is archived or restored until you click this button.

The folder Pending Restore displays all that will be restored. The folder Pending Archive displays all that will be archived. Only one operation is permitted at any given time: you either restore or archive, you cannot do both simultaneously.

Archive/Restore Setups button Click to archive or restore all of the application's modules setups. You must be at the root of the archive to archive setups. You must select a Projects Setup folder to restore setups.

File-based Archive Options

Linked Archive Option box Select how archive paths are treated when restoring an archive. Use Archived Path is best used when you are restoring the archive on the workstation used to create the archive. Select Convert to Local Path in other cases.

Archive Verification box Verify Source Media checks the integrity of the source media before adding it to the archive. In case of missing source media, the archive proceeds without the clip. Verify Archived Data checks the integrity of the data as it is being written to the archive: archiving stops as soon as it detects an error.

Verify Source Media validates all the media, which can be time consuming.

Media Options box Select Cache Source Media to copy the media to the archive, using the Cache and Renders settings defined at the creation of the project; this ensures that your archive is self-contained, but requires the caching of everything that was not cached at import. Select Archive Links Only to only archive paths to the media; this option requires less space, but does not create a self-contained archive.

Cache Source Media The media is automatically written to the archive and becomes managed. This ensures that material will not be overwritten accidentally in the shared storage when you restore the media. When you restore the media, the clips will reside on the local storage as managed frames.

Archive Links Only Only the path to the media is archived. You can later restore the clip in the original form as unmanaged media, provided the media resides at the original paths saved with the archive. If the source clip is not found at restore time, it is displayed as a checkerboard pattern image. Archiving files using the linked option significantly reduces the amount of space required for your archived clips. It also alleviates bandwidth when archiving to a VTR. Using Archive Links Only, you can archive both video and audio files.

Default Resolutions button Legacy option. Opens the Default Resolutions List menu, used by the application to restore legacy archives from Smoke 6.0 and Flame 8.5 (or earlier).

These versions did not store resolutions and associated pixel aspect ratios, and the Default Resolutions table fills that gap.

Tape Device Options

Block Size box Select Auto-block to use the pre-set block size. Select manual to specify a block size, in bits. Refer to tape device documentation to set optimal value.

VTR Device Options

Auto/Manual Start box Select Auto Start to have the application rewind the tape and then go forward to try and discover the location of an archive. Select Manual to enter the start timecode and get to the archive faster.

Timecode field Displays the start timecode. Editable.

Rewind On Close button Rewinds the tape after when the archive is closed.

Open Mode box Select whether to open in read only mode or in read/write mode.

New Archive Creation Options

Limit File Size button Disable to create an open-ended, one-segment archive file. Enable to partition the archive into manageable segments of the size specified by the Segment Size box. If the archive ends up being larger than what you entered in the Segment Size field, its content is split across multiple archive segments.

Segment Size box Select a pre-defined segment size, or define a custom one. Make sure to use a segment size small enough to fit in the specified destination. Enabled when Limit File Size button is enabled.

Playing Back Media 9

Playback

Quick Access to the Players

You can quickly access the players by the following shortcuts:

- Esc: Player for selected clip.
- Ctrl-Esc: Fullscreen player.
- Ctrl-1: Player view.
- Ctrl-2: Source-Sequence player.
- Ctrl-3: Triptych player.

The shortcut opens the players from the Workspace panel, the Thumbnail view, the Reels view, the Source-Sequence view, and the Tryptich player. When re-applied, the shortcuts return to the previously selected view.

NOTE Use of Esc does not bring the user to the Player when used from the Fullscreen player.

Playing Back Clips in the Full Screen Player

To play back clips in the full screen player:

1. Click the Timeline tab.
2. Do one of the following to select the Player:
   - Click the Layout button and select Player.
   - If you've enabled Layout Selection Overlay in the preferences, swipe the right swipe bar, then select Player.
   - Double-click on the clip in the media library.

   The clip appears in the Player viewer.
3. Click the Full Screen button.

The player displays as a full screen.

To return to the previous display size, click the Full Screen button.

NOTE While in the Full Screen Player, the Player Option Box lists Show Play Reel. This is the only viewer that provides this option. Selecting this option will open up a scrolling clip list of available clips under the control bar.

Jogging and Panning a Clip

You can pan and zoom in the player, tryptich and source sequence viewport. By selecting the Viewing Mode button you can select the Zoom or Pan pointers.

- The Zoom pointer increases the zoom of the clip with a left-to-right motion, and will decrease the zoom in a right-to-left motion.
- The Pan pointer lets you grab the clip and move it around the viewer.

NOTE The pan and zoom values of the players are kept until a clip from a different resolution is loaded; you can change the clip selection without losing the pan and zoom values as long as the clip resolution is maintained. If changing to a clip of a different resolution, the clip will be resized into the viewer.

To jog through a clip:

1. You can jog through a clip by doing one of the following:
   - Drag the positioner along the timebar.

     When you jog through an edit sequence (a series of clips edited together), you can constrain the jogging to the current clip in the sequence by holding down Command while jogging.
   - Use the arrow keys on your keyboard.

     By default, pressing the left arrow and right arrow keys steps frame-by-frame.
   - Place the pointer in the viewer, above the control bar.

     The cursor will change to indicate it can jog/shuttle the clip. The further from the center of the clip the cursor is placed, the faster the shuttling.

Switching the Playback Focus between the Source and Sequence Clips

You can switch the focus between the source and sequence viewers by clicking on either of the Source-Sequence viewers, by selecting a track on the timeline, or by selecting the green (source) or red (sequence) tabs above the Player, Source-Sequence, or Triptych viewers.

Clicking the Lock icon that appears when the pointer is hovering over the source-or-sequence tab will lock the focus on that tab. Clicking again unlocks the viewer.

NOTE Changing the focus through the timeline will also switch the focus between the Source and Sequence viewers. Changing the focus through the Source-Sequence viewer, however, will not change the focus in the timeline. In other words, the player doesn't drive the timeline, but the timeline drives the player.

To switch the playback focus between the source and sequence clips in the Source-Sequence Player:

1. Click the Timeline tab.

2 Do one of the following to select the Source-Sequence Player:
   - Click the Layout button and select Source-Sequence Player.
   - If you've enabled swipe in the preferences, swipe the right swipe bar, then select Source-Sequence Player.
3 Switch viewers by clicking in either the Source or Sequence viewer, or on the green (Source) or red (Sequence) tab.

   The viewer with the focus has a white outline.

Comparing Three Shots in the Triptych Player

The triptych player provides three players, side-by-side, which you can use to simultaneously view the same clip in different contexts. For example, you can set the middle player to play the clip, the left player to show the clip's incoming frame of the current element in the timeline, and the right player to show the clip's outgoing frame.

To compare three shots in the triptych player:

1 Click the Timeline tab.
2 Do one of the following to select the Triptych Player:
   - Click the Layout button and select Triptych Player.
   - If you've enabled swipe in the preferences, swipe the right swipe bar, then select Triptych Player.
3 Select a clip from the Media Library.

   The clip appears in the triptych player in all three viewers.

The default positions for the positioner is the start of the clip for the first viewer, the middle of the clip for the middle viewer, and the end of the clip for the last viewer. You can scrub each of the viewers independently of each other.

Play Back Speed Indicators

As you play clips, the positioner in the image window time bar changes colours if frames are dropped. The colours indicate the following:

Colour:	Indicates:
Yellow	Real-time playback with no dropped frames.
Orange	Real-time playback with no dropped frames of soft-imported clips.
Green/Blue	Alternating dropped frames. Green indicates the first and subsequent alternating dropped frame.
Red/White	Alternating occurrences of dropped audio. Red indicates the first occurrence.

Comparing Tracks in Split Screen

Use the split screen to compare two tracks, such as an online and offline version. Generally this is used to compare a rough cut to the final assembly. By merging both tracks into a single image, you can compare the two for editing synchronization, color matching, etc.

The split screen compares versions as well as video tracks.

To compare tracks in split screen:

1. Click the Timeline tab.
2. Set the Primary and Secondary tracks. (page 283)
3. In the View Area menu bar, select Options.
4. Select Show Viewing Settings.
5. Select the split you will use to compare the two clips in the Compare Mode/Stereo section.

   Depending on the split, you will have different attributes available to change the partition between the two clips. Modify the values in these attributes to provide the best comparison.

   You can now compare the two tracks on the split-screen.

Creating a Subclip in the Player

You can create subclips anywhere that you have access to the source clip (i.e. Thumbnail, Player, Triptych and Source-Sequence viewers).

NOTE Remember you will need handles for trimming and effects that you apply later. It is recommended to cue the tape a couple of seconds before the desired footage starts.

To create a subclip in the player:

1. Click the Timeline tab.
2. Set the positioner to the in point.
3. Click the In button.

   The start of the sub-clip is marked.
4. Set the positioner to the end point.
5. Click the Out button.

   The end of the sub-clip is marked.

   The section of the clip that is between the In and Out marks is the sub-clip.
6. From the Clip menu, select Create Subclip.

   The subclip is created as a new clip in the Media library. It has the original name and append ("_Subclip_") with an incremented number, starting with ("001").

Accessing Clip Information

Any viewer contains the following information located at the bottom right corner of the viewer:

- Name of the viewer OR name of the clip,
- Resolution and aspect ratio of the clip,
- LUT information of the clip (if LUT has been set).

Also, by alt-clicking on a clip proxy you will see additional clip information detail

Cue Marks

Adding Cue Marks

Cue marks are displayed in the player timebar, as well as the timeline.

To add a cue mark:

1. In the timebar, do one of the following:
   - Drag the yellow positioner to the desired frame, or
   - Click on the desired frame on the time bar.
2. Click the Cue Mark button.

   A cue mark is placed in the player timebar, as well as the timeline, to indicate the marked frame. You can change the colour of a cue mark by clicking the colour pot.

To delete a cue mark:

1. Click the Timeline tab.
2. Select the specific cue mark to be deleted.
3. From the Timeline Gear menu, select **Marks ➤ Delete Selected Markers**.

Adding In and Out Marks

In and out marks are displayed in the player timebar, as well as the timeline.

To add in and out marks in the player:

1. In the timebar, do one of the following:
   - Drag the yellow positioner to the desired frame, or
   - Click on the desired frame on the timebar.
2. Click the In or Out button.

   TIP Press `Ctrl` and click the In or Out button to delete the mark.

Navigating between Marks in the Clip

To navigate between marks in the clip:

1. Click the Timeline tab.
2. From the Timeline Gear menu, select a Go To option:
   - In Mark or Out Mark
   - For a cue mark, select the mark's label from the Mark list.

   The positioner is set to the mark.

Changing the Color of Cue Marks in the Clip

You can set the color of the current Player cue mark to one of your preference.

NOTE To cancel the color selection, click on the Close (x) button in the Color Picker window.

The Color Picker window will close and the color swath will revert to the previous color.

To change the color of cue marks:

1. Click the Timeline tab.
2. On the View Area menu bar, click on the color swatch to the right of the Mark button.
   A Color Picker window appears.
3. Do one of the following:
   - Click the Pick button.
     The cursor changes to a color picker, which you can use to select a color from one of the three color columns.
   - Drag the colour sliders.
4. Click Ok to apply the color to this and any future cue marks.
   The color swatch changes to reflect the color selected.

Playback Options

Setting up the Image Display Viewer in the Tools

To set up the image display viewer in the tools:

1. Click the View button.
2. Modify the display attributes.
   The viewer updates the display with your changes.

Setting up the Image Display Viewer in the Player

You can change the display of an image based on the exposure and contrast settings, as well as the type of image data you are working with. By default, all images are displayed in RGB mode with a transformation for video images. You can apply transformations to the image to display an optimal view of logarithmic and linear images.

By modifying the display attributes in Viewing Settings, you can change the way the clip is displayed in the player. Settings are persistent throughout different Player modes, depending on the timeline tab type (Source or Sequence).

NOTE In the case of the Triptych Player, the middle image window shares this persistency; while the left and right image windows each can have their own settings.

To set up the image display viewer in the player:

1. Click the Timeline tab.

2 Select the Player.
3 In the View Area menu bar, select Options.
4 Select Show Viewing Settings.
5 Modify the display attributes.
The viewer updates the display with your changes.

Viewing Image Overlays in the Tools

To view image overlays in the Tools:

1 Click the Grid button.
2 Set your preference for overlay attributes.
The new overlay attributes are set.

Viewing Image Overlays in the Player

To view image overlays in the player:

1 Click the Timeline tab.
2 Select the Player.
3 In the View Area menu bar, select Options.
4 Select Show Overlays.
5 Set your preference for overlay attributes.
The new overlay attributes are set.

Modifying Exposure and Contrast in the Image Display

You can modify the overall brightness of images displayed on the workstation or broadcast monitor, as well as the contrast between light and dark colours. Note that these settings affect only the display, and do not modify the underlying colour values of the clips.



To modify exposure and contrast interactively in an image window

1 Click in an image window to make it active.
2 Do any of the following:
- Press and hold Shift+E while clicking and dragging the mouse to modify the exposure offset (overall brightness). Dragging to the right increases values and dragging to the left decreases them.
- Press and hold Shift+C while clicking and dragging the mouse to modify the contrast between light and dark colours.
- Click Reset to restore the default values.

To modify exposure and contrast using the Image Display settings

1. Access the Image Display Viewer settings.
2. Do either of the following:
   - Edit the Exposure slider to modify overall brightness.
   - Edit the Contrast slider to modify the contrast between light and dark colours.

NOTE If you have applied a colour transform to the display, then you cannot modify the exposure and contrast unless there is a dynamic ExposureContrast operator in the CTF file.

Enabling Aspect Ratio Display in the Player

When you work with clips that have a resolution that does not use square pixels, such as NTSC or PAL, you can adjust the display so that it appears with the correct aspect ratio. By default, the adjusted image is sent to the broadcast monitor. You can switch the adjusted display from the broadcast monitor to the Player.

To enable the aspect ratio display in the Player:

1. Click the Timeline tab.
2. In the View Area menu bar, select Options.
3. Select Show Viewing Settings.
4. Enable the Use Ratio button.

   NOTE The Use Ratio button is only active if you are viewing a clip with a resolution that does not use square pixels.

   The original aspect ratio of the clip is applied to the clip.

Changing Playback Scan Mode Options in the Player

Use the playback scan mode option to define how you want the clip to play and display. By default, everything will be automatically set to the correct scan mode.

To set playback mode in the Player:

1. Click the Timeline tab.
2. Select the Player.
3. In the View Area menu bar, select Options.
4. Select Edit Viewing Settings.
5. Click on the Scan Mode button.

   NOTE The default choice is the clip's native scan mode.

6. Adjust the display to the desired scan mode format.
7. Click Done.
   The player displays the clip in that scan mode.

Displaying a Clip in the Broadcast Monitor

To set broadcast monitoring preferences:

1. Click the Timeline tab.
2. In the View Area menu bar, select Options.
3. Select Show Viewing Settings.
4. Enter the preferences for the broadcast monitor in the Broadcast Monitor, Broadcast Multiview and Broadcast LUT boxes.

   Your preferences will be set for broadcast monitoring.

The broadcast monitor outputs a complete image or a selected viewport.

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 you select multiple clips, the broadcast monitor displays the first clip in a selection.

When the Show Selected Item option is selected, you can manage additional preferences.

Color and Light Levels

Accessing the Vectorscope

To access the Vectorscope:

1. Click the Timeline tab.
2. Select the Player.
3. In the View Area menu bar, select Options.
4. Select Show Vectorscope.

   The Vectorscope is displayed.

Changing the Settings in the Vectorscope

To change the settings in the Vectorscope:

1. Click the Timeline tab.
2. Select the Player.
3. In the View Area menu bar, select Options.
4. Select Show Vectorscope.
5. Click the Vectorscope Display button.
6. Select Vectorscope Settings.

   The Vectorscope Settings window opens.

Scaling the Vectorscope

To scale the Vectorscope:

1. Click the Timeline tab.
2. Select the Player.
3. In the View Area menu bar, select Options.
4. Select Show Vectorscope.
5. Click the Vectorscope Display button.
6. Select Vectorscope > Colour 2D.
7. Shift + drag in the Vectorscope.
   The scale in the Vectorscope scales to a new size.

Preview options

To access the preview options below, select the Options box in the View menu bar.

Colour:	Indicates:
Preview Effects	Render all timeline effects on-the-fly. If the timeline effects cannot be rendered in real time, frames are dropped.
View Effects in Real-Time	Render only those timeline effects that can be rendered in real time.
Hide Effects	Displays the source clips without any rendered effects.
Full Resolution	Play back the full-resolution clip.
Proxy Resolution	Play back using the clip's proxies.
Deliverables	Play back using the current Real-Time Deliverable.

Preview Effects

Preview FX mode processes all timeline effects to create a preview for the current frame. It is useful for viewing timeline effects (stacked or not) in context. Every time you use the frame advance controls or jog the positioner, the image in the Player is updated with all the processed timeline effects. This allows you to make changes to the vertical edit and immediately see your changes. However, when you play the version or clip, the functionality temporarily returns to Realtime mode (until play is stopped). If you create complex timeline effects and find that unrendered black sources appear in some or all your timeline effects when playing in Realtime mode, switch to Preview FX mode to immediately see the results of your changes on the current frame.

Preview FX mode is also useful if you have many unrendered timeline effects, or transitions that use an unrendered timeline effect as an outgoing or incoming clip. Use this mode while you make changes to complex vertical edits, and return to Realtime mode to view the rendered result.

If you are working in an HD project, you should enable the Proxy-Size Player option before using Preview FX mode for faster display of the image. If you have a complicated vertical edit with many timeline effects, you may notice that the Player slows down when jogging or advancing frames. This is normal. Preview FX

mode processes the entire vertical edit for the current frame as you jog, which may take more time as you add more timeline effects. It is recommended to use this mode when changing the timing of a layer, such as slipping or sliding, so that you can immediately see the result for the current frame, and then use Realtime mode to view the processed result.

NOTE In Preview FX mode, the Axis, Blend, Text, and Wipe timeline effects are unavailable for preview in the broadcast monitor.

View Effects in Real-Time

Realtime mode plays clips and versions in real time, allowing you to view your timeline effects. It is recommended that you do most of your timeline work in Realtime mode. However, if your clip has more than one timeline effect, the Player may drop frames or display black frames during playback. In this case, process the timeline effects to get real-time playback. This is true especially if you have stacked timeline effects on many layers. Also, when playing a version with multiple tracks containing unprocessed timeline effects in Realtime mode, you only see the focus track (with its timeline effects).

Changing Playback Scan Mode Options

To set playback mode in the Player:

1. Click the View button.
2. Click on the Scan Mode button.

   NOTE The default choice is the clip's native scan mode.

3. Select a scan mode.
4. Click Done.
   The player displays the clip in that scan mode.

Enabling the Aspect Ratio Display

To enable the aspect ratio display:

1. Click the View button.
2. Enable the Use Ratio button.

   NOTE The Use Ratio button is only active if you are viewing a clip with a resolution that does not use square pixels.

   The original aspect ratio of the clip is applied to the clip.

Setting up the Image Display Viewer in the Tools

To set up the image display viewer in the tools:

1. Click the View button.
2. Modify the display attributes.
   The viewer updates the display with your changes.

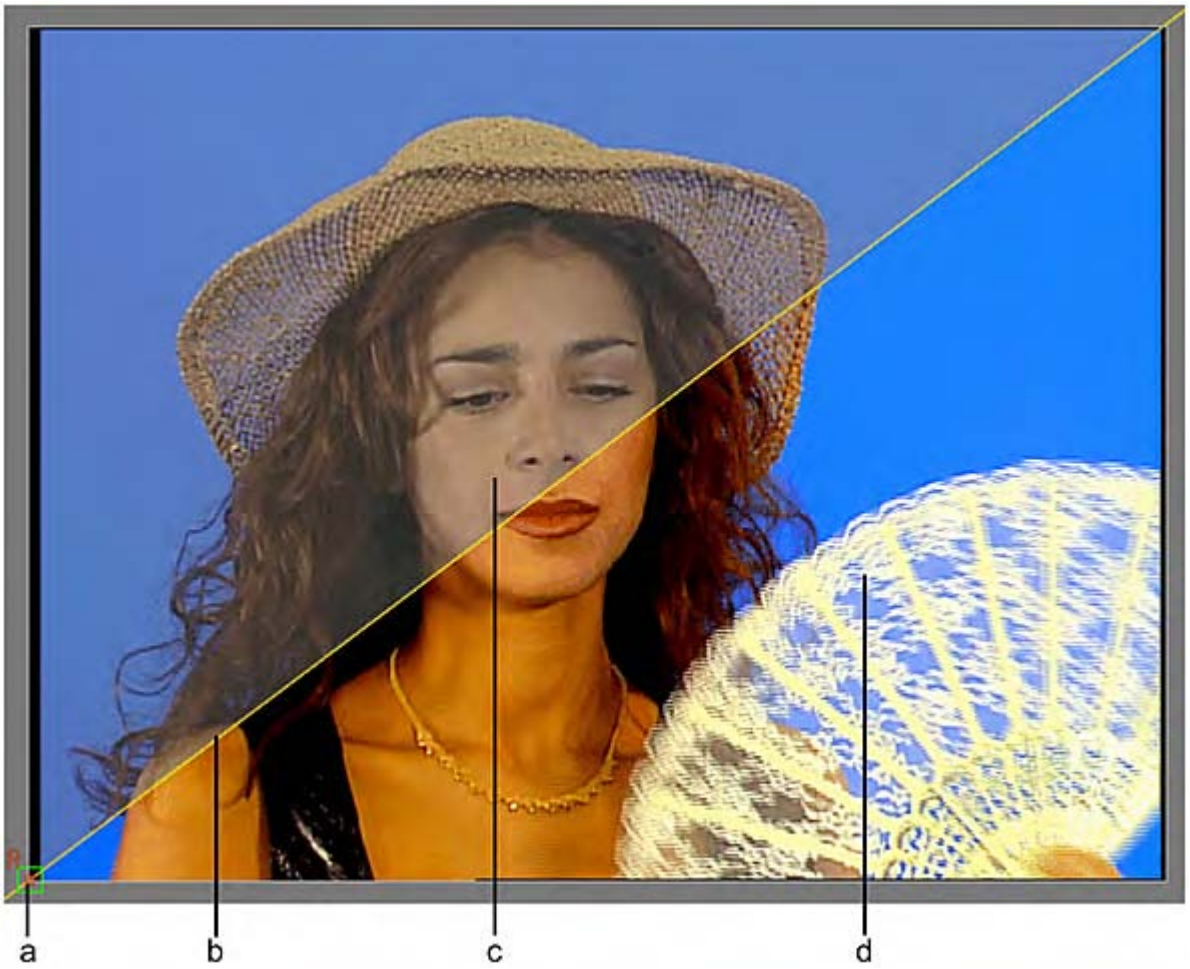
Displaying the Reference Area

In the image window of the tools that support multiple viewports, you can view the reference area with the currently displayed clip. A split bar divides the view between the current clip display (above the bar) and the reference area. The reference area can either display one of the current tool's working clips (for example, front, back, or matte), or one of the reference frames that you grab on-the-fly.

Using the Split Bar

To turn on the split bar, enable Compare in the Reference Buffer section of the View menu (also available below the viewports in Batch or Batch FX). When the split bar is on, by default it lies along the lower edge of the image window.

The split bar has a pivot point (indicated by a small box) around which the split bar can be rotated, or from which the split bar can be moved. The letter R next to the pivot box indicates the side of the bar where the reference clip is displayed.



(a) Pivot point **(b)** Split bar **(c)** Current display **(d)** Reference display
(a) Pivot point (b) Split bar (c) Current display (d) Reference display

Rotate and move the split bar using these techniques:

- To rotate the split bar around its pivot point, drag the split bar.
- To move the split bar, drag the pivot point. You can also `Ctrl+Alt`-click the image to place the pivot point under the cursor, and then drag over the image to move the split bar.

- To restore the split bar to its default position, `Ctrl`-click the pivot point.

Showing and Hiding the Split Bar

When the split bar is on, you can show or hide it in the image window.

To show or hide the split bar:

1. `Ctrl`-click the split bar.

   When the split bar is hidden, it is still active. The letter R is displayed indicating the location of the reference clip.

Loading a Clip into the Reference Area

You can display one of the tool's clip views in the reference area. For example, turn on the split bar to show the reference area, and then display the matte of a Colour Corrector clip while working on its result.

To load a clip into the reference area:

1. Select an option from the Reference box.



Storing Reference Frames in the Reference Buffer

You can grab any number of reference frames in the reference area, limited only by framestore space.

To store reference frames in the reference area:

1. Display the frame you want to grab in the viewport.
2. In the Reference Buffer section of the View menu of a tool, click Grab.

   The current frame appears in the reference area.
3. To grab another frame, display it in the viewport (or make the changes to the current frame) and then do one of the following:
   - Press `Ctrl+G` to replace the current reference frame.
   - Press `Ctrl+Shift+G` to append the new frame to all existing reference frames.

Cycling Through Multiple Reference Frames

When you work with several frames stored in the reference buffer, you can cycle their display in the reference area.

To cycle through multiple reference frames:

1. With the split bar on and the reference area showing, click the Prev and Next buttons in the View menu.

To cycle through multiple reference frames using the reference overlay:

1. Press and hold `Ctrl+NUM5` to display the reference overlay.
2. While holding `Ctrl`, press `NUM4` and `NUM6` to cycle backwards and forwards through all stored reference frames.

Displaying Multiple Views

You can display up to four viewports at a time in the image window of most tools. Multiple viewports are convenient for setting channel values, working in Schematic view, and previewing your results all at the same time without having to switch views.

NOTE The Resize tool supports two viewports; other tools support up to four viewports.

You can apply a different 3D LUT to each viewport.

To view multiple viewports:

1. In a tool that supports multiple viewports, click View.
2. Select an option from the Viewport Layout box.



Select:	To view:
1-up	A single viewport (`Alt+1`).
2-up	Two viewports, side-by-side (`Alt+2`).
3-up split	Three viewports, two side-by-side, and one on top (`Alt+3`).
3-up	Three viewports, side-by-side (`Alt+3`).
4-up	Four viewports, two up and two down (`Alt+4`).

To display views in multiple viewports:

1. Do one of the following:
   - Place the cursor over the applicable viewport and press its associated keyboard shortcut.
   - Click a viewport to select it and then select an option from the View box.

Displaying Widgets in Selected Viewports

If you are working in multiple viewports, you can display widgets such as icons, crop boxes, and 3D scope in all viewports or in a selected viewport.

To display widgets in selected viewports:

1. In a tool that supports multiple views, click View.
2. From the Widget Display box, select Widget Sel to display widgets in the selected viewport, or select Widget All to display widgets in all viewports.



Changing the Multiview Layout

When you select a multiple viewport layout, a default layout appears, and each viewport is set to display a default view. You can change both the default layout and the views in each viewport to suit your needs.

To change the default viewport layout:

1. Do one or more of the following:
   - Hold the cursor over the lines dividing the image window and `Ctrl`-drag. The current zoom/pan settings automatically change so the frame matches its viewport's width.
   - Select the viewport to make it current (a yellow border indicates the current viewport) and then select a view option (for example, Front) or press a hotkey (for example, `F1`).
2. To restore the default settings, select an option from the Reset box.



Select:	To:
Reset Layout	Restore the default layout for the current viewport layout option. You can also `Alt`-click a viewport border.
Reset All	Restore the default layout for all viewport layout options.

Synchronizing Current Frame Display Across All Viewports

By default, when you scrub or jog a clip, only the current viewport (a yellow border indicates the current viewport) is updated. The other viewports continue to display the last frame at which they were parked until you release the cursor. However, you can set all viewports to be updated in sync with the current viewport.

To synchronize the current frame display across all viewports:

1. Display the View menu.
2. Select an option from the Viewport Update box.



Select:	To:
Update All	Update all viewports to display the same frame as the current viewport.
Update Sel	Update only the current viewport. Other viewports continue to display the last frame at which they were parked only once you release the cursor.

Viewing Settings tab

Compare Mode /Stereo

Primary Video Track field Displays the track and layer assigned as the Primary track in the Player. Editable.

Secondary Video Track field Displays the track and layer assigned as the Secondary track in the Player. Editable.

Preview Setup box Select which tracks to display in the Player. If displaying both tracks, select a split screen option.

Image

Use Ratio button Enable to simulate rectangular pixel display that corresponds to the delivery format.

Filtering button Enable to set the filtering.

Scan Mode button Adjusts the display to the desired scan mode format.

Playback

Play Source box Select the source of the playback clip (disk or memory).

Playback Options box Select a play-from-disk playback option.

Preroll field Displays the number of preroll frames when playing a clip. Editable.

Postroll field Displays the number of postroll frames when playing a clip. Editable.

Colour Mode box Select the colourspace and bit rate for playback. Select Clip Format (the default) to use the original clip settings.

In Timecode field Displays the timecode at which to begin playback. Editable.

Out Timecode field Displays the timecode at which to end playback. Editable.

Reload From Current button Click to reload the buffer with the current Colour Mode settings.

Exposure & Contrast

Exposure field Sets the exposure that is used to transform image display in the image window. Editable.

Contrast field Sets the contrast that is used to transform image display in the image window. Editable.

Enable Look Transform Toggles the dynamic look of a colour transform on and off. Available only when the selected viewing transform has dynamic look operators defined. Editable.

Channels

Red Channel button Display the red channel of the clip.

Green Channel button Display the green channel of the clip.

Blue Channel button Display the blue channel of the clip.

Exclusive button Enable to display an exclusive greyscale representation of the selected colour channel.

Image Data Type

Image Data Type box Select the type of image data that you are displaying in the image window.

If you are not using a 3D LUT, then logathmic and linear data are converted to video for display.

If you are using a 3D LUT, then linear and video data are first converted to logarithmic before being processed by the LUT for display.

If you are using a colour transform for display, then the image data type is ignored and the transform is applied directly.

Apply All button Enable to apply the transformation for the current viewport to all viewports.

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

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

The color management settings that are appropriate for displaying RGB images is not always appropriate for displaying mattes. Matte mode defaults to Video, which is best for viewing mattes.

1D LUT Display

Use 1D LUT button Enable to select and use a gamma or 1D LUT.

1D LUT List box Select a preloaded gamma or 1D LUT.

3D LUT & Colour Transform Display

Use 3D LUT button Enable to select and apply a 3D LUT or colour transform to the image.

3D LUT box Select a preloaded 3D LUT or colour transform.

Editing Clips and Sequences

10

Adding Your First Clips to the Timeline

The following procedures list a few methods to add clips quickly to a timeline after starting Flame Premium for the first time. These methods are certainly not the only way to edit, but should give you a start in familiarizing yourself with editing in Flame Premium.

When first starting Flame Premium with a new project, you can select a workspace with a Source-Sequence player and then switch to the Timeline tab to see an empty sequence on the timeline. Once you have loaded some media into the Media Library using the MediaHub, you are ready to add clips to the timeline. You can also create new sequences from the Workspace Media panel, or right-click a clip and select Open as Sequence.

To insert a first clip on the timeline:

1. Select a clip in the Media panel to display it in the source player.
2. Drag the positioner in the timebar to the frame where you want your clip to start, then click In.
3. Drag the positioner in the timebar to the frame where you want your clip to end, then click Out.
4. Click the Insert button.



The clip is added to the timeline. Extra frames outside of your selected In and Out points remain as handles.

To insert a second clip on the timeline:

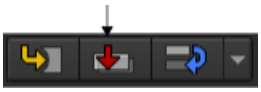
1. Select another clip in the Media panel to display it in the source player viewer.
2. Drag the positioner in the timebar to the frame where you want your clip to start, then click In.
3. Drag the positioner in the timebar to the frame where you want your clip to end, then click Out.
4. Click the Insert button.



The clip is added to the timeline starting at the positioner location. If you hadn't moved the positioner after inserting the first clip, the positioner was located at the last frame.

To overwrite a clip on the timeline:

1. Move the timeline positioner to the first frame of the second clip that you inserted on the timeline.
2. Select another clip in the Media panel to display it in the source player.
3. Drag the positioner in the timebar to the frame where you want your clip to start, then click In.
4. Drag the positioner in the timebar to the frame where you want your clip to end, then click Out.
5. Click the Overwrite button.



Depending where the positioner was placed and the length of the second and third clips, existing material is overwritten, but the overall length of the timeline sequence does not change.

Continue adding clips to your sequence, as needed. At this point, you are ready to fine-tune your edit sequence by trimming clips, adding tracks for compositing, adding effects to your timeline clips, and using other Flame Premium timeline and editing tools.

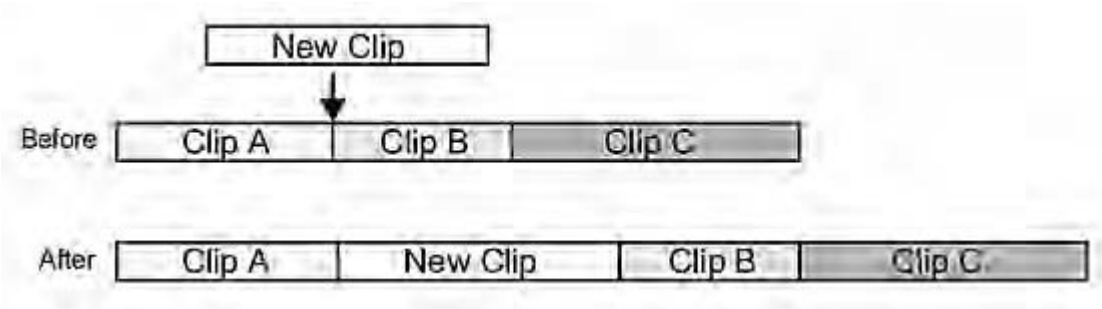
Inserting a Clip on the Timeline

To insert a clip to the timeline using the Insert button:

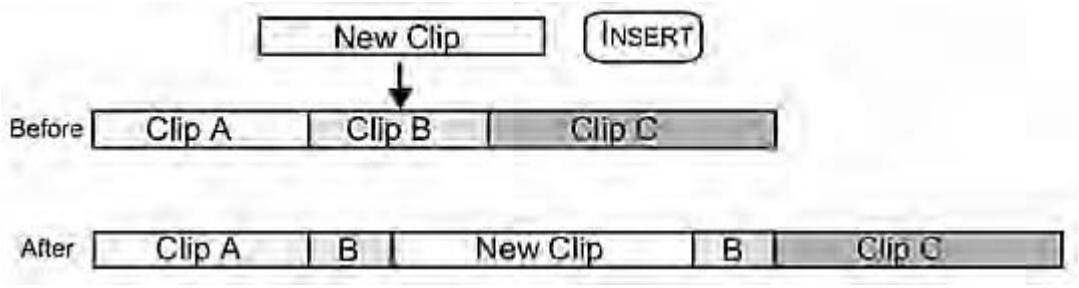
1. Move the timeline positioner to the frame that you want to use as the beginning of the insert.
2. From the workspace, select the clip that you want to insert.
3. Add In and Out points, if needed.
4. Click the Insert button.



Depending where the positioner is placed, the clip is inserted, and the overall length of the timeline sequence is changed. The following example shows a New Clip inserted at the transition between Clip A and Clip B. Everything after the transition is moved ahead in order to make room for New Clip.



This example demonstrates the result of inserting New Clip at a frame in the middle of Clip B. Every frame after the edit point is moved ahead to accommodate New Clip.



NOTE You can also perform three-point or four-point insert edits with In and Out points on the source and timeline clips.

To insert a clip to the timeline gesturally:

1. Enable Ripple.
2. From the Workspace Media panel or Viewing panel, select the clip that you want to insert.
3. Add In and Out points, if needed.
4. Drag the clip towards the timeline.

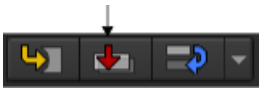
   As you hover over the timeline with your clip, you see a phantom visual guide as to the space on the track or tracks your clip will take when dropped.
5. Release the clip on the timeline.

TIP You can also drag and drop a clip to the timeline from the source tab of the Player.

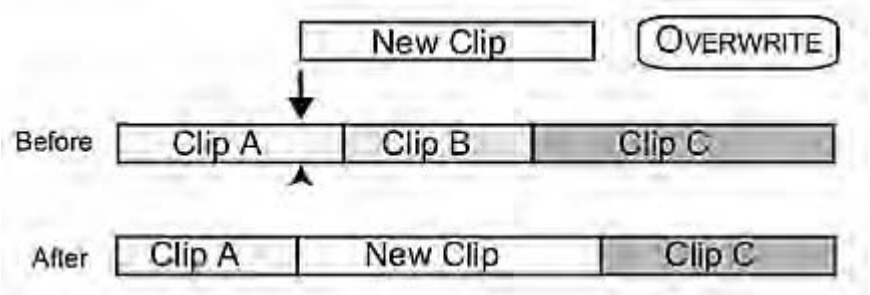
Overwriting a Clip on the Timeline

To overwrite a clip to the timeline using the Overwrite button:

1. Move the timeline positioner to the frame that you want to use as the beginning of the overwrite.
2. From the workspace, select the clip that you want to overwrite.
3. Add In and Out points, if needed.
4. Click the Overwrite button.



Depending where the positioner is placed, existing material is overwritten, and the overall length of the timeline sequence does not change. In the following illustration, New Clip is edited into the sequence at a frame in Clip A. Frames after the insertion point are overwritten by New Clip. All of Clip B and some of Clip C are overwritten.



NOTE You can also perform three-point or four-point overwrite edits with In and Out points on the source and timeline clips. In the case of a four-point edit, a Timewarp may be applied if the number of frames differs between the source and timeline clips. You can enable of disable Auto Timewarp in the Timeline section of the Preferences menu.

To overwrite a clip to the timeline gesturally:

1. Disable Ripple.
2. From the Workspace Media panel or Viewing panel, select the clip that you want to overwrite.
3. Add In and Out points, if needed.
4. Drag the clip to the timeline.

   As you hover over the timeline with your clip, you see a phantom visual guide as to the space on the track or tracks your clip will take when dropped.
5. Release the clip on the timeline.

TIP You can also drag and drop a clip to the timeline from the source tab of the Player.

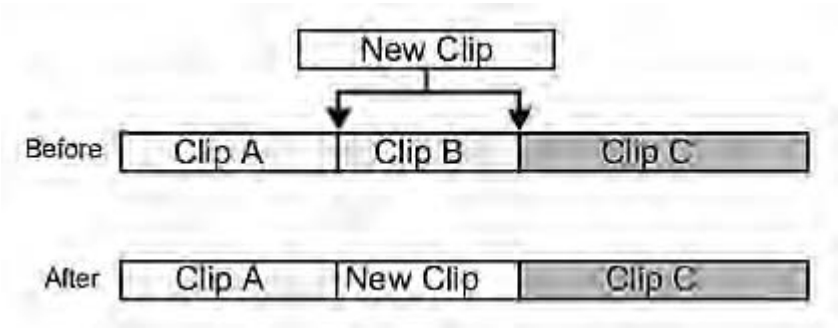
Replacing a Clip on the Timeline

To replace a clip on the timeline using the Replace button:

1. Do one of the following:
   - Move the timeline positioner over the clip that you want to replace.
   - Select a clip or multiple clips on the timeline that you want to replace.
2. From the workspace, select the clip that you want to use as the replacement clip.
3. Click the Replace button.



The overall length of the sequence does not change. The replace clip must be of equal length or longer than each clip it is replacing. If longer, extra frames are loaded as tail frames.



Other Editing Operations

While insert, overwrite, and replace are the editing operations you use most often, there are other editing options available. For each of these operations, you select your source clip in the same manner as for an

insert, overwrite, or replace edit. Then you select the appropriate editing operation from the Secondary Edit box.



Aligned Edit

An Aligned Edit is usually performed on two tracks. You set the positioner on the timeline to an point you want to align to, then move the focus point of the positioner to a different track. Set the positioner of the source track to the point you want aligned. The source clip is added to the track, and both points are aligned.

Append

Append applies an edit that adds your source clip to the end of the edit sequence on the timeline.

Prepend

Prepend applies an edit that adds your source clip to the beginning of the edit sequence on the timeline.

Ripple Replace

Ripple Replace functions as a combination of an insert and replace edit. The source clip replaces the selected clip on the timeline, and the all of the clips to the right ripple to accommodate the length of the new clip.

Replace Media

Replace Media performs a replace edit but retains any effects applied to the replaced clip or clips on the timeline.

Fit To Fill

A Fit To Fill edit functions as a replace edit, but instead of adding tail frames to a longer source clip, a Timewarp is added to the replaced clip so that it fits the same space occupied by the clip it is replacing.

Timeline Editing Settings

The settings you use for editing to and on the timeline can be found to the right and above the timeline. Most of these settings also have a keyboard shortcut associated with them.

Editing Operations



Insert Edit button Click to insert the selected source into the timeline at the positioner. The overall length of the timeline may change.

Overwrite Edit button Click to overwrite material in the timeline (at the positioner) with the selected source. The overall length of the timeline does not change.

Replace Edit button Click to replace the selected timeline clip or clips with the selected source clip. The overall length of the timeline sequence does not change. The replacement clip must be of equal length or longer than each clip it is replacing. If longer, extra frames are added as tail frames.

Secondary Edit box Select an editing operation to apply to the selected source clip.

Select:	To apply:
Aligned Edit	An edit aligning a point on the timeline with a point on the source clip.
Append	An insert edit that adds your source clip to the end of the edit sequence on the timeline.
Prepend	An insert edit that adds your source clip to the beginning of the edit sequence on the timeline.
Ripple Replace	A combination of an insert and replace edit; where the source clip replaces the selected clip on the timeline, and the all of the clips to the right ripple to accommodate the length of the new clip.
Replace Media	A replace edit that retains any effects applied to the replaced clip or clips on the timeline.
Fit To Fill	A replace edit; but instead of adding tail frames to a longer source clip, a timewarp is added to the replacement clip so that it fits the same duration occupied by the clip it is replacing.

Trimming Operations



Editorial Mode box Choose an editorial mode for working with clips on the timeline.

Select:		To:
Select		Select or move a timeline element, without performing any trim operations.
Trim		Set Trim mode to perform general trim (or roll) operations on a clip.
Slip		Set Slip mode, a trimming mode that offsets the frames in a clip without trimming the clips before or after it.
Slide		Set Slide mode, a trimming mode that trims the clips before and after the clip without changing the frames used in it.

Select:		To:
		The state of the Ripple button has an effect on this mode.
Slide Cuts		Set Slide Cuts mode, a trimming mode that trims the clips before and after it, while also changing the frames used in a clip. The state of the Ripple button has an effect on this mode.
Slide Keyframes		Set Slide Keyframes mode, a trimming mode that slides only animation keyframes on a clip.

NOTE Trim, Slide, and Slide Cuts modes are affected by the state of the Ripple button. The icons display as yellow when ripple is enabled, and red when ripple is disabled.

Link button Enable to select and link all elements in an editorial group. This is useful if you want to trim all clips in a group by the same number of frames, for example. You can invert the Link mode on-the-fly during manipulation, with the Invert Group Selection keyboard shortcut.

Ripple button When gesturally adding material into an edit sequence, enable to perform an insert edit, disable to perform an overwrite edit. When moving (or trimming) clips on the timeline, enable to fill the gap left by the removed clip or frames, disable to leave the gap.

Snap button Enable to snap to the closest transition, positioner, or mark to help gesturally align edits on the timeline. Press Shift to invert the snap mode during manipulation.

Keyframe Move Modes box Select how the animation channel is affected when you trim elements with animated effects.

Select:	To:
Reposition Proportionally	Resize the channel as you trim. The animation channel is scaled to fit into the timeline element. This option has no effect when you slip or slide.
Shift With Media	Link the keyframes to their original frame numbers. The animation channel moves to follow the original frames as you trim.
Pin To Segment Start	Unlink the keyframes from their original frame number. The animation channel remains with the timeline element as you trim.

Trimming Clips

Use the trimming tools to fine-tune your rough edits. You can trim video or audio clips (with or without effects), cuts, and transitions on the timeline. You can trim multiple tracks or segments at the same time. Flame Premium keeps track of the source material that goes into trimmed shots, so you can go back and reintroduce material you had previously trimmed out.

You can trim using the Trim View, with keyboard shortcuts, or gesturally on the timeline. You can also trim by any combination of these methods, depending on your editing style. In Trim mode, the Trim View

displays the last (outgoing) and first (incoming) frame above the timeline from the two clips you are trimming, allowing you to visualize your trimming operation. In Slip or Slide mode, the Trim View changes to a four frame display, which, in addition to the incoming and outgoing frames of the clip you are slipping or sliding, also displays the outgoing frame of the preceding clip and the incoming frame of the subsequent clip on the timeline.

The state of the Ripple button has an effect on *some* trimming operations. You can set the ripple state explicitly before trimming, or on-the-fly by pressing `Alt` while you are trimming.

Audio tracks are divided into sub-frames. There are 100 sub-frames of audio for every frame of video. When trimming audio tracks, you can trim on a sub-frame level by holding down the `Shift` key while you trim.

NOTE You cannot trim on a sub-frame level when simultaneously trimming audio and video tracks.

Trimming Quickstart

Use the following procedure to quickly start trimming. This method is certainly not the only way to trim, but should give you a start in familiarizing yourself with trimming in Flame Premium. For more detailed explanations of the various trimming modes, see the related topics below.

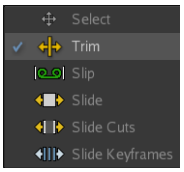
1 Double-click a cut between two clips on the timeline. The positioner moves to this location, and a yellow or red highlight appears on either side of the cut (depending if Ripple is enabled or not).



The Trim View also opens, (provided that Auto-Toggle Trim View is enabled in **Preferences ➤ General ➤ Player**) displaying the outgoing and incoming frame of the two clips.

TIP You can also access Trim View from the View mode box. In this case, the closest cut on the timeline is automatically selected. You can use the Previous or Next Transition buttons to select a different cut.

2 From the Editorial Mode box, select Trim.



3 Drag the Outgoing or Incoming Trim Offset fields to your desired trim offset value (in frames). The corresponding view (incoming, outgoing, or both), and the timeline clips are updated.



Notice that the offset fields have the same colour borders as the highlight on the timeline cut (yellow when Ripple is enabled, or red when Ripple is disabled).

4 Use the loop option of the Play button to loop around your trim with a number of preroll and postroll frames to preview your trim.



Performing a Trim

Performing a Trim or Roll

Trim (or roll) a clip to add frames to, or remove frames from, a clip's head or tail.

To trim (or roll) a clip or transition on the timeline:

1. From the Editorial Mode box, select Trim.
2. Position the cursor near the head or tail of a clip or transition on the timeline.

   The cursor turns to a trim cursor (a yellow arrow and line if ripple is enabled, or a red arrow and line if ripple is disabled). If you want to trim a cut, position the cursor over the cut, and notice that the trim cursor becomes a double arrow with a line.
3. Drag left or right to remove the number of frames that you need.

   You can see the head or tail number on the clip change as you trim.

   TIP Enable Focus On Trim in the Timeline Options menu to snap the timeline positioner to the transition while trimming, allowing you to view the frames you are trimming in the Player, or switch to Trim View to see your incoming and outgoing frames.

To trim (or roll) a clip or transition using Trim View:

1. Double-click a cut between two clips on the timeline, or select Trim View from the View Mode box.

   Trim View appears in the Viewing panel, displaying the last (outgoing) and first (incoming) frame from the two clips you are trimming.
2. Use any of the following techniques to trim your clips to the desired frames:
   - Click and drag directly in the left or right Trim View window. You can see the trim icon as you drag. The icon is displayed in yellow or red, depending on the state of the Ripple button.
   - Click and drag toward the middle of the two Trim View windows. Notice that the trim icon appears as a double-sided arrow, indicating that you are trimming the cut between the two clips.
   - Click any of the timecode or duration fields at the top of Trim View windows to enter specific values. If a timecode or duration field is not enabled, click the corresponding Trim View window or offset field to enable it.
   - Click and drag the outgoing or incoming Trim Offset field to specify the number of frames offset in the clip. You can also click the minus or plus buttons to offset by those amounts.
   - You can also trim on the timeline, and use the Trim View as a viewing aid.

Trimming Keyframes

If you set animation keyframes on timeline effects, you have different options as to their behaviour while trimming. Use the Keyframe Move Modes box to select how the animation channel is affected when you trim elements with animated effects.

Select:	To:
Reposition Proportionally	Resize the channel as you trim. The animation channel is scaled to fit into the timeline element. This option has no effect when you slip or slide.
Shift With Media	Link the keyframes to their original frame numbers. The animation channel moves to follow the original frames as you trim.

Select:	To:
Pin To Segment Start	Unlink the keyframes from their original frame number. The animation channel remains with the timeline element as you trim.

In the Editorial Mode box, you can also select Slide Keyframes mode, which allows you to slide all of the keyframes of a specific timeline effect on a clip. For example, you can slide all of the Axis keyframes on a clip.

Performing a Slip

Performing a Slip

Slip a clip to offset the frames in a clip (by trimming head and tail frames of the clip) without trimming the clips before or after it. The state of the Ripple button has no effect on slip operations.

To slip a clip on the timeline:

1. From the Editorial Mode box, select Slip.
2. Position the cursor in the middle of a clip on the timeline.

   The cursor turns to a slip cursor.
3. Drag left or right.

   You can see the head and tail numbers on the clip change as you slip.

To slip a clip in Trim View:

1. Select the clip you want to slip on the timeline.
2. From the Editorial Mode box, select Slip.
3. Double-click the clip you want to slip, or select Trim View from the View Mode box.

   In Slip mode, the Trim View displays four panels. In addition to the incoming and outgoing frames of the clip you are slipping, you can see the outgoing frame of the preceding clip and the incoming frame of the subsequent clip on the timeline.
4. Use any of the following techniques to slip your clip:
   - Click and drag left or right directly in the Trim View window. You can see the slip icon as you drag.
   - Click any of the timecode fields at the top of Trim View windows to enter specific values.
   - Click the minus or plus buttons to offset by those amounts.
   - You can also slip on the timeline, and use the Trim View as a viewing aid.

   NOTE In Slip mode, the Duration and Offset fields of the Trim View update as you slip, but are non-editable.

Performing a Slide

Performing a Slide

Sliding changes a clip's position in the edit sequence. It simultaneously slides the clip under the cursor and trims the head and tail of the surrounding clips. The state of the Ripple button has an effect on slide operations.

NOTE Instead of using Slide mode, you can select Slide Cuts from the Editorial Mode box, a modified Slide mode that trims the clips before and after it, while also changing the frames used in a clip.

To slide a clip on the timeline:

1. From the Editorial Mode box, select Slide.
2. Position the cursor in the middle of a clip on the timeline.

   The cursor turns to a slide cursor (a white square with yellow arrows if ripple is enabled, or a white square with red arrows if ripple is disabled).
3. Drag left or right.

   The position of the clip changes, and surrounding clips' head and tails are trimmed.

To slide a clip in Trim View:

1. Select the clip you want to slide on the timeline.
2. From the Editorial Mode box, select Slide.
3. Double-click the clip you want to slide, or select Trim View from the View Mode box.

   In Slide mode, the Trim View displays four panels. In addition to the incoming and outgoing frames of the clip you are sliding, you can see the outgoing frame of the preceding clip and the incoming frame of the subsequent clip on the timeline.
4. Use any of the following techniques to slide your clip:
   - Click and drag left or right directly in the Trim View window. You can see the slide icon as you drag.
   - Click any of the timecode fields at the top of Trim View windows to enter specific values.
   - Click the minus or plus buttons to offset by those amounts.
   - You can also slide on the timeline, and use the Trim View as a viewing aid.

   NOTE In Slide mode, the Duration and Offset fields of the Trim View update as you slide, but are non-editable.

Trimming with Keyboard Shortcuts and the Calculator

You can use selection-based keyboard shortcuts to perform your trim, slip, or slide operations (depending on the mode selected in the Editorial Mode box).

1. On the timeline, select the segment that you want to trim.
2. From the Editorial Mode box, select the trim mode that you want to perform (for example, Slip).

   TIP Specific keyboard shortcuts are available to switch to each trim mode (Trim, Slip, Slide, Slide Cuts, and Slide Keyframes). See the Keyboard Shortcut editor for more information.
3. Use the keyboard shortcut appropriate to the action you want to perform (in the descriptions below, Trim refers to the selected trim mode).

Description	Smoke (FCP 7) Shortcut	Smoke Classic Shortcut	Flame Shortcut
Trim 1 Frame Forward	. (period)	N	. (period)
Trim 1 Frame Backward	, (comma)	B	, (comma)

Description	Smoke (FCP 7) Shortcut	Smoke Classic Shortcut	Flame Shortcut
Trim <n> Frames Forward	Shift+. (period)	Shift+N	Shift+. (period)
Trim <n> Frames Backward	Shift+, (comma)	Shift+B	Shift+, (comma)
Trim to In Mark	Ctrl+I	Ctrl+B	Shift+[
Trim to Out Mark	Ctrl+O	Ctrl+N	Shift+]
Trim to Positioner	E	Ctrl+P	Shift+P

Trimming with the Keypad and Calculator

If you know the amount of frames by which you want to trim, a quick way to perform the trim is by using the keypad to enter the amount into the calculator.

1. On the timeline, select the clip or cut you want to trim.
2. On the keyboard keypad, enter the number of frames by which you want to trim (positive or negative).
3. Press Enter.

   NOTE If there is no explicit selection on the timeline, trim operations are not performed using this method. Instead, the positioner is moved by the number of frames that you enter.

Dynamic Trimming

In Trim View, you can use key combinations to dynamically trim while your outgoing and incoming clips are playing.

To trim with the J-K-L keys:

1. In Trim View, select either the incoming, outgoing, or both clips.
2. Use one of the following J-K-L key combinations to trim:
   - Press the `J` or `L` key once to play backward or forward, then press `K` to stop and perform the trim at the desired frame. You can also press `J` or `L` multiple times to play faster before pressing `K`. This method works for Trim mode only.
   - Press and hold `K`, then press `J` to trim one frame backward or `L` to trim one frame forward. This method works for Trim, Slip, and Slide modes.

   The trim viewports and the timeline update as you perform your trim. While pressing `J` or `L`, you can see a phantom positioner moving along the timeline to help you see where you want to trim.

To trim during playback:

1. In Trim View (with Trim, Slip, or Slide mode selected), select a playback option from the Play button (such as Loop), and click Play.
2. While your selection is playing (with the desired number of preroll and postroll frames), you can use the Trim 1 Frame Forward, Trim 1 Frame Backward, Trim <n> Frames Forward, and Trim <n> Frames Backward keyboard shortcuts.

Trim View Settings



In Trim mode, the Trim View displays the last (outgoing) and first (incoming) frame from the two clips you are trimming (as seen in the example, above). In Slip or Slide mode, the Trim View changes to a four frame display, which, in addition to the incoming and outgoing frames of the clip you are slipping or sliding, also displays the outgoing frame of the preceding clip and the incoming frame of the subsequent clip on the timeline.

Use these settings while working in Trim View. Most of the settings are available for all Trim View modes; exceptions are noted below.

Tail Source Timecode field Displays the tail source timecode of the outgoing segment. Non-editable if the outgoing trim offset is not selected.

Outgoing Duration field Displays the duration of the outgoing clip. Non-editable if the outgoing trim offset is not selected. Available in Trim mode only.

Incoming Duration field Displays the duration of the incoming clip. Non-editable if the incoming trim offset is not selected. Available in Trim mode only.

Head Source Timecode field Displays the head source timecode of the incoming segment. Non-editable if the incoming trim offset is not selected.

Duration field Displays the duration of the selected segment. Non-editable. Available in Slip and Slide mode. (Not shown)

Timecode field Displays the current timecode of the focus clip (green signifies a source clip, while red signifies a sequence clip). Editable.

View Mode box Select a view mode for the Viewing panel layout.

Previous Transition button Click to select the previous transition on the selected track.

Next Transition button Click to select the next transition on the selected track.

Outgoing Trim Offset field Displays the number of frames offset in the outgoing clip. Click to select the offset, and drag to change the offset value. A selected offset has a yellow border if Ripple is enabled, and a red border if Ripple is disabled. Available in Trim mode only.

Incoming Trim Offset field Displays the number of frames offset in the incoming clip. Click to select the offset, and drag to change the offset value. A selected offset has a yellow border if Ripple is enabled, and a red border if Ripple is disabled. Available in Trim mode only.

Trim Offset field Displays the number of frames offset as the result of the slip or slide operation performed on the selection. Non-editable. (Not shown)

-5 button Click to trim the selection five frames to the left.

-1 button Click to trim the selection one frame to the left.

+1 button Click to trim the selection one frame to the right.

+5 button Click to trim the selection five frames to the right.

Preroll field Displays the number of preroll frames when playing a clip. Editable.

Play button Click to activate the current play behaviour selected for this button. Click and hold to list a selection of playback types. The Play button will change to display the current play status; Once, Loop, or Back and Forth.

Postroll field Displays the number of postroll frames when playing a clip. Editable.

Player Options box Select an option for working in the player. Available options differ depending on the view selected.

About Timeline Tracks and Versions

Different areas on the timeline form a hierarchy that allows you the flexibility to composite on multiple tracks while maintaining different video streams or versions of your work.

Think of a version as a single stream of video on the timeline. Each version can have multiple tracks. While you can also use sequences on the timeline to accomplish similar results, use versions for the following:

- Compare two streams, such as an offline and online version. You can use the player in a split view to compare versions.
- Create matte containers with an Axis timeline effect, to help you key using a separate fill and matte, for example.
- Create different versions of a track or tracks, to try out different effects without affecting each other.

Tracks on the timeline are stacked vertically within a version to help you composite.

Track Indicators and Tools

On the left side of the timeline, in the Patch Panel area, you can find indicators and other tools for working with tracks and versions.



In the example above, the upper track (version 1 track 2) is highlighted, and therefore selected. The track indicators and tools for this track are (from left to right):

Grab area Grab the handle area of the track to move and reorder a track or group of tracks. As you are dragging, a red message gives you an indication as to the operation being performed.

Track Collapse/Expand arrow Click to collapse or expand all tracks within a version. You can collapse tracks to remove clutter from your timeline.

Patch Identifier (Source track) Indicates which source track is patched to the track (in green).

Track Identifier (Destination or Sequence track) Displays the version and track number (in dark grey).

Track Visibility icon Displays or hides the track.

Track Lock icon Enable track lock to prevent editing operations from being performed on the track. A locked track displays grey diagonal lines.

Sync Lock icon Enable to maintain sequence sync on a track or tracks.

Video Track indicator Indicates if a track is the primary or secondary track. The primary track can also be set by moving the focus point on the positioner.

Adding Tracks and Versions

To add a new video track:

1. On the timeline, select the version you want to add a track to.
2. On the bottom left of the timeline, click Track+, or from the Timeline Gear menu, select **New ➤ Video Track**.

   A new track is added to the version above the last existing track. If you want the track to be added below the last existing track, press **Ctrl** while clicking Track+. You can re-order your tracks, or even move a track to a different version by dragging the track from the handle at the left of the track.

To add a new audio track:

1. On the bottom left of the timeline, click Audio+ for a mono track or `Alt`+Audio+ for a stereo track, or from the Timeline Gear menu, select **New ➤ Mono Audio Track** or **New ➤ Stereo Audio Track**.

   A new audio track is added to the timeline below the last existing audio track.

To add a new version:

1. On the bottom left of the timeline, click Version+ for a mono version or `Alt`+Version+ for a stereo version, or from the Timeline Gear menu, select **New ➤ Mono Version** or **New ➤ Stereo Version**.

   A new version is added to the timeline above the last existing version. If you want the version to be added below the last existing version, press `Ctrl` while clicking Version+.

TIP To add multiple versions or tracks, you can use the keyboard to enter the number of versions or tracks you want to add, before using one of the above methods.

Moving, Resizing, and Deleting Tracks

To move a track:

1. Select the track or tracks that you want to move.
2. Grab the selected track from the left end of the Patch Panel area, and drag up or down to the desired location.

   A red message appears while dragging to help guide while dragging the track.
3. Let go of the track to perform the move.

NOTE You cannot move a video track to the audio track area and vice-versa.

To resize a track:

1. Place the cursor at the bottom of the Patch Panel area for the track that you want to resize. Notice that the cursor becomes a line with a double-headed arrow.



2. Drag down to increase the height of the track (to see more information on the tracks clips, for example), or drag up to decrease the height of the track.

   TIP To increase or decrease the height of all tracks, drag left or right on the vertical scroll bar to the right of the timeline.

To delete a track:

1. Right-click on the Patch Panel area of the track you want to delete.
2. Click Delete Track.

About Timeline Patching

When you record a source clip to a sequence, you must decide what source clip channels you want to use, and to which tracks you want to record them. To connect the source channels to the destination timeline tracks, you use patching. When you select the source clip, green patch identifiers indicate what channels you can record from the source to the timeline.



In the above example, the top track (V1.3) is patched, but no source is assigned. The middle track (V1.2) is not patched and no source is assigned. The bottom track (V1.1) is not patched, but a source is assigned. If you click the source patch or destination identifier on the bottom track, the track becomes patched, identified by the green patch.



With the patch identifiers, patching information is kept with the sequence clip. If you patch a source clip and then add a new track to the timeline, the source clip follows the track to which it was originally patched.

This allows for a natural workflow of setting up the patch for a source clip, adding a new track, then setting up the patching for another source clip.

Patching on the Timeline

To patch a source channel to a sequence track:

1. Select a source in the workspace.

   Source track patch identifiers appear in the Patch Panel area for each channel in the source.
2. Create the patch:
   - If the track is patched but no source is assigned, drag on the grey patch identifier until the source channel that you want to use appears.
   - If the track is not patched and no source is assigned, drag on the grey patch identifier until the source channel that you want to use appears and then click the patch identifier.
   - If the track is not patched but a source is assigned, click the grey source channel number on the patch identifier.

   The patch identifier turns green and its source channel number indicates what source you have recorded to the track.



Navigating and Searching the Timeline

Although you can search for elements by scrolling the timeline and displaying information about each element, the Find and Select in Timeline options make it easier to find specific elements. You can also refine your search when searching specifically on segments. Once your search is complete, or if you have multiple segments selected on the timeline, you can easily navigate through them.

Navigating with the Positioner

The frame directly beneath the positioner is displayed in the Player or is the current location for an edit such as a dissolve or cut. Drag the positioner to the desired frame on your timeline.

In the timecode area below the tracks, a lighter yellow box attached to the positioner indicates the length of the current frame, at the timeline zoom level. Click and drag this yellow box to move the positioner, so as not to accidentally move clips on the timeline.

If you know the amount of frames by which you want to move, a quick way to navigate the timeline is by using the keypad to enter the amount into the calculator.

1. On the timeline, make sure that there is no explicit selection of a clip or cut (in this case, a trim may be performed).
2. On the keyboard keypad, do one of the following:
   - Enter the number of frames by which you want to move the positioner (for example, +10 or -10).

- Enter a number (without a + or -) to navigate to the closest timecode value that finishes with the value entered (for example, enter 10 to move the positioner to the closest timecode value that finishes by :10)

3 Press Enter.

TIP Use the timeline scroll bars to zoom timeline, if needed. Click Home in the Timeline Layout combo box to reset the timeline view.

Searching the Timeline

1 Click the magnifying glass icon at the bottom right side of the timeline.



The Find and Select in Timeline window opens.

2 Enable a button or multiple buttons to set your search criteria. For example, you can enable Dissolve in the Video Transitions area to search for all dissolves.

TIP You can enable Get Information From Current Element to automatically fill in information from the selected element into the search criteria.

3 Choose whether to search segments, containers, gaps, cue marks, or segment marks. You can select more than one or all of these options.

4 Set the strictness of the search by enabling Every Criteria Below or Any Criteria Below.

5 Click Select on Current Track or Select on All Tracks to perform the search.

The Find and Select in Timeline windows closes, and results are highlighted on the timeline.

Navigating Selected Elements on the Timeline

If you have multiple segments selected on the timeline (as the result of a search, or as a manual selection), you can easily navigate through them with keyboard shortcuts or with these procedures.

To navigate on the timeline in a linear manner:

1 In the Player, select and hold the Go To Next button to display more options.

2 Click Selected Element.

3 Use the Go To Previous and Go To Next buttons to scroll through the selected timeline elements.

To navigate to a specific selected timeline segment:

1 In the FX pipeline above the timeline, click the arrow beside the segment thumbnail.



You can see a list of all selected segments.

2 From the list, select the element you want to navigate to.

The timeline positioner moves to the selected segment.

Using Reveal to Locate Clips or Segments

To help you locate timeline clips or segments in the Workspace Media panel, you can use the reveal options. To locate a clip, you can double-click the timeline tab, or right-click the clip on the timeline, and select

Reveal ➤ Clip in Media Panel. You can also locate a specific clip segment by right-clicking it on the timeline, and selecting **Reveal ➤ Segment in Media Panel**. The clip or segment is then highlighted in the Workspace Media panel, and the Thumbnail or Reels view, if displayed.

Using Markers on the Timeline

Use cue marks or segment marks to mark frames of interest on a clip. You can then quickly go to the marked frames. These marks are for reference purposes only; they do not affect the clip and are not used in any editing operations. You can add a mark to all tracks (*cue mark*) or to a single track or version (*segment mark*) on the timeline. Segment marks move on the timeline as you trim, slip, or slide a clip whereas cue marks stay at the same position on the timeline.

You can also add In and Out marks on timeline clips, as you can in the player.

To add marks on the timeline:

1. Go to the frame where you want to insert the mark.
2. From the Timeline Gear menu, in the Marks section, select an option to add an In, Out, Cue, or Segment mark.
3. If you added a cue or segment mark, you can change the name and colour of the mark to help you identify it more easily.

NOTE The Marks section of the Timeline Gear menu offers other options for clearing or deleting marks. The Go To section of the Timeline Gear menu offers navigation aids for existing marks.

About Timeline Containers

Containers are a convenient way to remove clutter from the timeline by grouping or nesting elements together. The elements, which can come from different tracks or versions, are treated as one unit yet remain individually editable. Containers behave like any other element but they appear in a separate timeline tab when you enter their editor. You can also add containers within a container.

Containers can also be used to take a clip and matte that are separate and add them to the timeline as a single element.

NOTE You cannot contain audio across tracks, but you can contain adjacent audio elements.

Creating a Timeline Container

To create a container:

1. Select all the elements on tracks or versions on the timeline that you want to collapse into the same container.

   NOTE The bottom track is always trimmed out so that you can dissolve from an element to a container.

2. From the Timeline Gear menu, select Contain Selection.

   All the selected elements are collapsed into a container and the element changes to a dark blue. You can add effects or perform other timeline operations to the contained element.
3. To edit the contents of the container individually, double-click the container, or click Open in the Container quick menu.

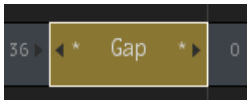
The elements that make up the container appear in their own timeline tab (the tab has a blue line). Yellow marks indicate the duration of the container. From this view, you can add effects, tracks, versions, or any other timeline operation.

To remove a container:

1. Select the container in the sequence where it was created.
2. From the Container quick menu, click Uncontain.
   The container is removed and its contents are restored to the sequence.

About Timeline Gaps

Gaps are spaces on a track that do not contain media. You can use gaps to see through one layer to another or to apply effects to the tracks below the gap. If a gap exists on a track, the next track that contains video media is included in the edit as long as it is also below the focus track.



Since timeline gaps are independent of all media, they are useful for applying Timeline FX or Batch FX (as an Adjustment Segment) that affect all tracks under the gap.

By default, empty gaps are transparent. However, you can make an empty gap opaque if you do not want to see through to the next track. To do so, select the gap on the timeline, then click Opaque in the Gap quick menu.

While editing on the timeline, you can also quickly remove a gap between elements by right-clicking the gap, and choosing Close Gap.

When selecting elements on the timeline with a rectangle selection (that is, when drawing a bounding box around them), you can choose whether to include gaps by enabling or disabling Selection Includes Gaps in the Timeline Options menu.

Playing the Timeline

You can control how the timeline plays back. For example, frame-step through a shot to isolate a frame or jump to the start of an element to play a specific shot. The image window updates as the positioner moves on the timeline. While you can play your timeline in any of the Viewing panel's View modes, if the Player is displayed, you can also use any of the playback controls to move through the timeline. The Player also offers compare modes, where you can set up a split view to play primary and secondary versions or tracks at the same time.

To play the timeline:

1. Press `L` to play forward.
2. Press `J` to play backward.
3. Press `K` to stop playback.

TIP Press `J` or `L` multiple times to play faster. Press `Shift` along with `J` or `L` to play slower.

To scrub the timeline:

1. Drag the positioner, or the scrub area at the bottom of the timeline, left or right.

When scrubbing in the Source tab, you cannot navigate past the first or last frame of the clip.

To jump to a particular location:

1. Do one of the following:
   - Click the scrub area at a particular location.
   - Enter a value in the Current Timecode field.

     TIP To enter a frame number instead of timecode, click the field to bring up the numeric keypad. Click TC to change to FRM, and then enter a frame number.

To frame-step the timeline:

1. Press the left or right arrow key.

   TIP To frame-step a specific number of frames, enter a number in the numeric keypad and then press the left or right arrow key.

Cutting Clips on the Timeline

When you splice clips together, the location at which one clip changes to another is called a cut. A cut is useful when you want to extract part of an element or end a shot at a given frame.

You can make a cut at any point on a segment. A cut creates a transition with a head equivalent to the duration of the segment before the cut and a tail equivalent to the duration after the cut. For example, if you make a cut after the 5th frame in a 10-frame clip, the cut has a tail of 5 frames and a head of 5 frames.

In a match frame cut, the outgoing and incoming shots are from the same source and the outgoing and incoming frames are consecutive. Match frame cuts are indicated by an "=" on the cut point.

To add a cut on the timeline:

1. Move the positioner over the frame where you want to add a cut. Make sure:
   - The element is explicitly selected (yellow) or implicitly selected (surrounded by a white bounding box). If the element is neither yellow nor surrounded by a white bounding box, another element, cut, or transition is selected on the timeline and the cut will not occur at the positioner location.
   - The focus point is over the appropriate track.
2. Select Cut from the Timeline Gear menu.

   A match frame cut is added at the specified location.

   TIP You can also add cuts while the clip is playing by pressing the cut keyboard shortcut.

To remove a match frame cut:

1. Select the cut on the timeline.
2. Right-click and choose Remove Match Cut.

   The cut is removed and the two elements are joined together.

Swapping Timeline Elements

You can quickly swap elements on the timeline. If segments contain timeline effects, they are swapped as well.

You can swap the following elements:

- Video segments or containers
- Audio segments or containers
- Tracks
- Versions
- Contiguous sequences of elements
- Video transitions (unless they start or end a contiguous sequence of elements)
- Audio transitions
- Gap effects

Elements must be compatible to be swapped. Compatible elements are:

- Video segments, video containers, gap effects
- Audio segments and audio containers
- Video transitions
- Audio transitions
- Tracks and versions

The following elements cannot be swapped:

- Gaps (unless they are between elements in a contiguous sequence)
- Cuts
- Cue marks
- Individual timeline effects

To swap timeline elements:

1. Select two elements belonging to the same family.

   A contiguous sequence of elements is treated as a single element as long as the transition between the elements is selected. A gap can be part of a contiguous element as long as it is not at the beginning or end of the sequence.

2. Press `Shift+S`.

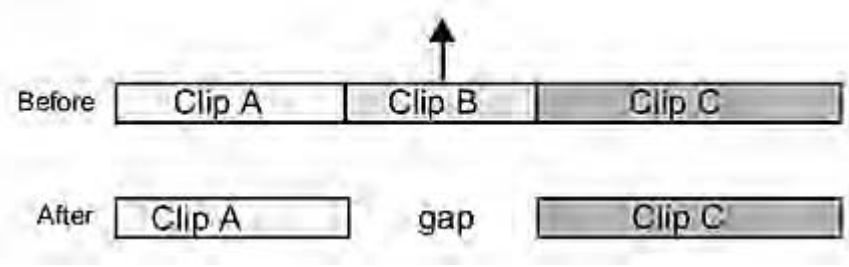
   The elements are swapped according to the ripple setting. If the segments contain any timeline effects, they are swapped as well.

   If Ripple is disabled, the segments are timewarped using a Constant Fit-to-Fill timewarp to fill each other's location. If there is already a timewarp on the segment, the timewarp is replaced by the Constant Fit-to-Fill timewarp. Gap effects, Matte containers, and contiguous sequences ripple regardless of Ripple status because they cannot be timewarped.

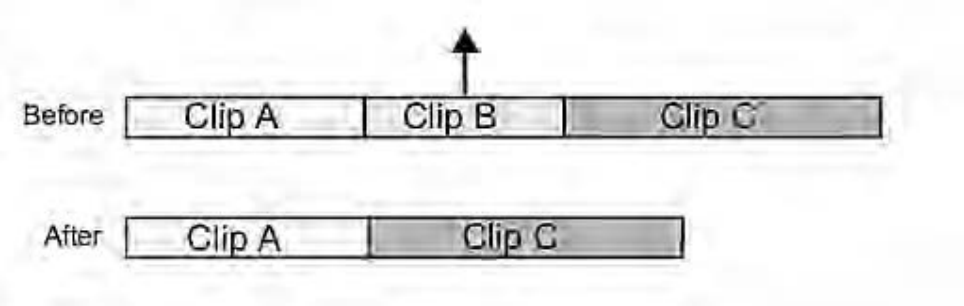
   Transitions keep their alignment but their duration changes based on the head and tail frames of the destination segments.

Removing Elements From the Timeline

When you remove an element from an edit sequence gesturally, the effect depends on whether Ripple is on or off. When Ripple is off, you *lift* an element, leaving a gap in place of the lifted element.



When Ripple is on, you *extract* an element, and the edit sequence collapses to fill the gap left by the extracted element. This is also called a *ripple delete*.



You can lift or extract elements from the timeline independently of the Ripple mode using the contextual menu or keyboard shortcuts.

To lift or extract an element from using the contextual menu:

1. Select the element you want to lift or extract or mark in and out points around the material.
2. Right-click the element and select one of the following:
   - Select Lift to lift the element from the timeline, leaving a gap in place of the lifted element.
   - Select Ripple Delete to extract the element from the timeline and ripple the remainder of the timeline to fill the gap.

To gesturally remove an element from an edit sequence:

1. Do one of the following:
   - Turn off Ripple to lift.
   - Turn on Ripple to extract.
2. Select the element that you want to lift or extract and drag it out of the timeline.

   NOTE You can also lift or extract gaps on the timeline.

Matching Clips with Their Sources

Matched sources are the original, untrimmed clips with their original names and timecode information. There are numerous ways to match clips with their sources:

- **Match Frame:** Displays the source and matches the frame and the In / Out markers of the selected clip.
- **Match Source:** Displays the source of the selected clip while retaining its original markers, if any.
- **Match Source and Keep Handles:** As above while retaining the handles.
- **Match Content:** Displays the sources of clips contained within transitions, containers, matte containers and Batch FX clips.
- **Match Content and Keep Handles:** As above while retaining the handles.

  NOTE The Match Content options can also be performed on timeline clips.

- **Match All Sources:** Displays all the sources within a selected sequence.
- **Match All Sources and Keep Handles:** As above while retaining the handles.

If the source is already on the Desktop, it is centred on its reel. If the source is not on the Desktop, the cursor becomes a white arrow, prompting you to select a destination reel on which to display the source(s).

To match a frame:

1. Select a clip on the Desktop and place the cursor at the frame you want to match.
2. From the contextual menu, select **Match ➤ Frame**.
3. Select a destination if the source is not already on the Desktop.
   The source is displayed centred on the destination reel at the appropriate frame.

To match a source:

1. Select a clip on the Desktop .
2. From the contextual menu, select **Match ➤ Source** .
3. Select a destination if the source is not already on the Desktop.
   The source is displayed centred on the destination reel at the appropriate frame.

To match content:

1. Select a transition, container, matte container, or Batch FX clip on the Desktop , or a timeline clip.
2. From the contextual menu, select **Match ➤ Content**.
3. Select a destination if the sources are not already on the Desktop.
   The sources are displayed centred on the destination reel.

To match all sources within a sequence:

1. Select a sequence on the Desktop.
2. From the contextual menu, select **Match ➤ All Sources**.
3. Select a destination if the sources are not already on the Desktop.
   The sources are displayed centred on the destination reel.

Grouping and Syncing Elements on the Timeline

You can create editorial groups to preserve the relationship between timeline elements in a vertical composition. For example, if you have a video clip with two tracks of audio overlapping on the timeline, you can create a group for them. When you edit one segment that is part of a group, all segments in the group are edited in the same way. If your group becomes out of sync, a visual cue is displayed on the timeline, and tools are available to help you resync the elements.

To create an editorial group:

1. On the timeline, select the overlapping video or audio elements to include in the group.
2. From the Timeline Gear menu, select **Group ➤ Group**.

   NOTE You can also right-click your selection, and choose Group from the contextual menu. If Group is unavailable in the contextual menu or the Timeline Gear menu, the selection is not able to be grouped.

   Once a group is created, file names on the elements are underlined to indicate group status.
3. Enable Link to allow elements in the group to be edited together (you can use the keyboard shortcut for Invert Group Selection to temporarily change the status of the Link button during manipulation).

   TIP If you cut a segment that is part of a linked group, all elements in the group are cut at the positioner location, and two separate groups are created from the cut segments.

Resetting Sync

If an editing operation removes the sync for a track in a group, a red plus (+) or minus (-) symbol appears in the element with the amount of frames that are out of sync to the left (-) or right (+).



You can attempt to manually fix the sync issue by editing the out-of-sync element. In this case, make sure that Link is disabled. Once you have resynced, you can enable Link again.

You can either resync the elements or reset the offsets, creating a new sync relationship based on the current position of the segments.

To regain sync:

1. Select the element that is out of sync.
2. Right-click the element, and select Resync.

   The element is resynced, if possible, and the out-of-sync symbol disappears.

To reset sync offsets:

1. Select any element in the edit sync group.
2. Right-click the element, and select Reset.

   The segments remain in their current positions. The sync offsets are removed creating a new edit sync group.

Deleting and Disabling Editorial Groups

When you delete an editorial group, the sync is removed from all elements that were part of the group. You can also temporarily disable an editorial group.

To delete an editorial group:

1. Select one of the elements that make up the group.
2. From the Timeline Gear menu, select **Group ➤ Ungroup**.

   NOTE You can also right-click your selection, and choose Ungroup from the contextual menu.

To remove sync temporarily:

1. Select one of the elements that make up the group.
2. Disable Link.
3. Edit an element of the group.

   No other elements in the edit sync group are affected. A red plus (+) or minus (-) symbol appears on every element that is out of sync.

   TIP You can change the Link mode temporarily on-the-fly by using the keyboard shortcut for Invert Group Selection.

Syncing Tracks and Versions

You can use the track and version-based sync locks to decide the behaviour of tracks or versions when performing editing operations, such as trimming or an insert edit. In this case, you do not have to select individual elements on tracks.

The contents of a track set to Sync can ripple even when nothing is selected for the edit on it. This is to keep segments in sync throughout many tracks and versions. For example, performing an insert edit on a track can have the effect of a gap of the same length as the inserted material being created on another synced track.

To turn on or off syncing for tracks or versions, click the sync lock icon in the Patch Panel area. In the following example, sync is enabled for tracks V1.1 and V1.3, and disabled for track V1.2.



You can also right-click a sync lock icon and select to sync or remove sync from all tracks or versions.

Affected Editorial Operations

The state of the track or version sync lock is taken into account when performing the following operations:

- Insert
- Extract
- Delete*

- Trim*
- Move*
- Slide*

*Ripple must be enabled.

These editing operations ignore sync lock:

- Aligned Edit
- Overwrite
- Replace
- Append
- Prepend
- Lift
- Slip

Timeline Reference

Use these settings to help you navigate and perform tasks on the timeline. You can also right-click anywhere on the timeline to display a contextual menu of relevant operations for the type of element selected.

Navigation

Positioner The "playhead" for playing the clip, displayed as a vertical yellow bar. The frame directly beneath the positioner is displayed in the Player or is the current location for an edit such as a dissolve or cut.

In the timecode area below the tracks, a lighter yellow box attached to the positioner indicates the length of the current frame, at the timeline zoom level. Click and drag this yellow box to move the positioner, so as not to accidentally move clips on the timeline.

Focus point A horizontal yellow line on the positioner indicating the current track.

Back button Click to move back one page in the timeline window.

Horizontal scroll bar To pan, drag left or right. To zoom the timeline, drag up or down.

Forward button Click to move forward one page in the timeline window.

Current Timecode field Displays the timecode of the current position. Editable.

- button Click to expand track height.

+ button Click to decrease track height.

Vertical scroll bar To pan, drag up or down. To zoom the timeline, drag left or right.

Timeline Search button Click to open the Find and Select in Timeline window to search the timeline using many different criteria.

Timeline Layout combo box Select an option to display the timeline.

Select:	To:
Home	Reset the view of the timeline.

Select:	To:
Reset Height	Reset the height of the tracks in the timeline.
Fit to Width	Reset the horizontal scale of the timeline.
Fit Selection	Centre the timeline on the selected element.

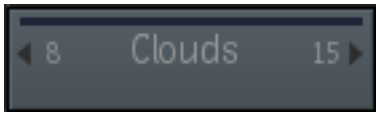
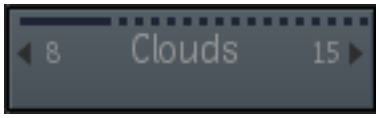
Rendering

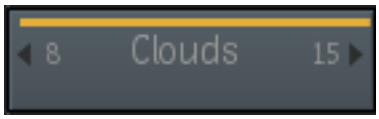
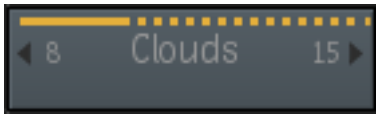
Rendering combo box Select an option to render timeline effects. You can edit effects after rendering.

Select:	To:
Render	Render the selected elements. This is the default option.
Force Render	Force a render on a timeline effect that needs to be refreshed. Use this to force a render on a locked element, or when Flame Premium fails to recognize that a processed clip is no longer valid. For example, if you change a track in a complex vertical edit, you may need to force a render to update all the tracks in the stack.
Render Proxy	Render proxies only, not the corresponding high-resolution images. An amber outline appears on the timeline segment to indicate the state of the rendering.
Force Proxy	Force the rendering of a proxy on a timeline effect that needs to be refreshed.

NOTE If Burn is present, additional options are available.

Depending on how a clip is rendered, you can see a visual representation on the render bar at the top of the clip on the timeline:

Render Bar:	State:
	Unrendered.
	Rendered.
	Partially rendered.

Render Bar:	State:
	Proxy rendered.
	Proxy partially rendered.
	Sent to Burn.

Viewing

Timeline View Mode box Select a viewing mode for working in the timeline.

Select:	To Display:
Timeline	Video and audio tracks in a timeline format with a positioner at the location of the current frame.
Collapsed	Only the current frame of each edit sequence. The edit sequence appears in the form of a stack. Shuttle through the sequence by dragging the cursor across the bottom of the frame.
Head-Tail	The first and last frames of each clip. Shuttle through the sequence by dragging the cursor across the bottom of any frame.
Storyboard	Only the first frame of each clip in the edit sequence. Shuttle through the sequence by dragging the cursor across the bottom of any frame.
Frames	Every frame in the edit sequence is displayed. Although you cannot edit in this mode, it is useful for viewing your entire edit.
History	A schematic or list representation of the majority of operations performed on a clip or timeline element.
Clip List	A list of objects and information about these objects that make up the timeline.

TIP In Collapsed or Storyboard view, you can drag sequences to reorder them. If you switch back to Timeline view, the new order of sequences is maintained.

Timeline Tasks

Timeline Gear menu Select a task to perform on the timeline. Some of the options have sub-menus with more options.

Select:	To:
New	Add new tracks or versions to the timeline.
Edit	Perform various element or timeline editing operations.
Rename	Rename the selected timeline element.
Rename Track	Rename the selected track.
Media	Break the link between an element's metadata and media or high-resolution media.
Select All	Select all elements in the timeline.
Deselect All	Deselect all elements in the timeline.
Container	Work with containers on the timeline.
Group	Create an edit sync group for selected timeline elements.
Go To	Navigate to various areas of the sequence.
Marks	Work with In and Out, Cue, or Segment marks.
Stereo	Work with stereo tracks on the timeline.
Cut	Add a cut to the selected element at the positioner location.
Cut All Tracks	Add a cut to all tracks at the positioner location.
Merge Tracks	Merge timeline tracks into a single track.
Swap Effects	Copy timeline effects between clips.
Lift	Remove the selected element from the edit sequence, and leave a gap in its place.
Ripple Delete	Remove the selected element from the edit sequence, and close the gap.
Close Gap	Remove a gap and collapse the elements to fill the gap.
Commit	Manage your timeline to simplify or save space.

Select:	To:
Render	Manage render options for the timeline.

Other Options

Timeline Options menu Select an option for working on the timeline.

There are a number of audio options in this menu:

Select:	To:
Scrub Audio	Enable scrubbing audio while dragging the positioner.
Show Gain Animation	Display the animatable Segment Gain level.
Show Waveforms With Effects	Display waveforms after an effect is applied to an audio segment.
Show Waveforms Without Effects	Display waveforms before an effect is applied to an audio segment.
Hide Waveforms	Not display waveforms.
Increase Waveform Range	Zoom in on the waveform.
Decrease Waveform Range	Zoom out on the waveform.

Use the Rectangle Selection options to select how timeline elements are included in a selection when drawing a bounding box around them.

Select:	To select:
Inclusive	All elements that are partially or entirely included in the bounding box.
Partial	Only the portions of the elements that fall within the bounding box. Audio is selected on a sub-frame basis if Sub-frame Positioner is selected in the Timeline section of the Preferences menu.
Partial A/V	Only the portions of the elements that fall within the bounding box. Audio is selected on a frame basis even if Sub-frame Positioner is selected in the Timeline section of the Preferences menu.
Bounded	Only elements that are fully in the bounding box.

Enable Selection Includes Gaps to include empty gaps when performing a timeline selection.

Enable one or both snap options to include in the timeline snap criteria:

- Snap To Positioner on Gestural Insert
- Snap Includes Marks

Enable Focus on Trim if you want the timeline positioner to snap to the transition while trimming, allowing you to view frames in the player.

Timeline Media Management Options

The Commit area of the Timeline Gear menu offers a number of options to help you clean up your timeline or save space when archiving a project.

Commit Options

When you commit an edit sequence, it becomes one continuous clip. The soft properties of the sequence such as transitions, head and tail frames, and timewarp data are removed. The visual effects of the transitions and timewarps remain but you cannot edit them.

You can commit any selection of elements or tracks, or the entire timeline. Commit a portion of an edited sequence to simplify a timeline that is too complex or to recoup disk space.

You cannot recapture or reimport committed clips.

The following rules apply to hard commits:

- Hard commits are permanent; however, they can be undone using the Undo button (as long as undos remain in the undo buffer). If you want to consolidate elements but have the ability to make changes at a later time, use containers.
- Hard commits cannot be restored—the commit is permanent.
- Hard commits force a render on the selected elements.

You can also choose to only commit Timeline FX or Batch FX segments on the timeline.

Consolidate Handles

If you have video or audio elements on the timeline that have excessive handles, use the Consolidate option to remove head and tail frames. Consolidate your clip before archiving a project to avoid archiving a large number of unused frames. You can keep a specified number of head and tail frames when you consolidate (the calculator appears when you select Consolidate for you to enter the number of handles you want to keep). You can consolidate a single element or a selection of elements.

History Options

You can commit a clip with a history, or a clip segment with a history just as you would any other clip. When commiting a clip, you delete the clip history without removing the clip or element's soft properties. Delete History deletes history frames for all selected segments.

Editing on the Desktop 11

About Editing on the Desktop

You can apply many types of edits to clips on the Desktop - splitting clips in two, splicing clips together, and adding dissolves.

Editing functions such as splicing clips create soft edits, which you can modify any time until you commit them. Soft edits created on the Desktop can be brought into the timeline for further manipulation.

All Desktop editing operations are applied to the audio and video by default. To edit the video only, select Edit Video Only from the Option box.

Some Desktop editing operations can be accomplished in two ways - by using menu commands and by using drag and drop gestures with the mouse or stylus. For these cases, this chapter describes both methods.

You can disable broadcast monitor support option in the User Interface tab of the Preferences. Disabling this option enhances interactivity when manipulating clips and performing gestural edits on the Desktop Reels.

NOTE When this option is disabled, Timeline FX are not automatically rendered and display Unrendered Frame on the Reels. However, the frame on which the Timeline positioner is parked, is rendered.

Editing Multiple-Resolution Clips

Editing operations that are performed on more than one clip, such as splicing two clips together, can only be done on clips of the same resolution, aspect ratio, and bit depth. For most editing operations involving two clips, if you attempt to use clips of differing resolutions, a pop-up menu appears stating that the source and destination clips are mismatched. If you click confirm, the source clip will be resized to match the destination clip.

Gestural Editing on the Desktop

Gestural editing is a technique in which you use your mouse or stylus to move clips among reels on the Desktop, and to perform cut, copy, insert, splice, and replace edits.

Gestural Editing Cursors

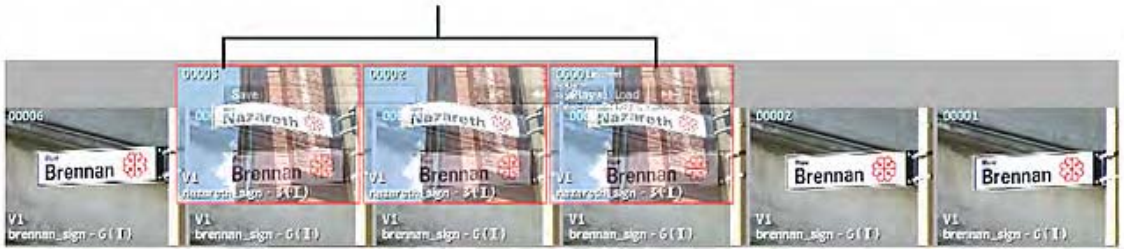
When you drag a source clip over a target clip, the cursor changes as the source clip moves over different areas of the target clip. The type of cursor indicates the type of edit that occurs when you drop the source clip on that spot.

All gestural splices, insert edits, and replace edits ripple the edit sequence; that is, they change the duration of the target clip unless you replace a clip with another clip of the same duration.

Cursor:	Appears when:	Use to:
 Insert	The source clip is positioned between frames on the target clip.	Perform an insert edit.
 Replace	The source clip is grabbed at any frame other than its first or last frame, and is positioned over a frame on the target clip.	Perform a replace edit. You cannot gesturally replace frames with a two-frame clip. However, you can perform this type of edit with a one-frame clip.
 Replace ahead	The source clip is grabbed at its first frame and is positioned directly over a frame on the target clip.	Perform a replace edit starting at the frame under the cursor and moving in the direction of the arrow.
 Replace back	The source clip is grabbed at its last frame and is positioned directly over a frame on the target clip.	Perform a replace edit starting at the frame under the cursor and moving in the direction of the arrow.

To perform a gestural insert or replace edit:

1 Drag the source clip over the frame in the target clip where you want to make the edit until you see the appropriate edit cursor.



2 Drop the source clip.

The clip is updated with the edit. The source clip replaces the frames of the target clip



Ganging Clips on the Desktop

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

To gang two clips on the Desktop:

1. Make sure the clips are on different reels.
2. Align each clip to be ganged to the timecode you want to lock.
3. Select the clips while holding the `Ctrl` key, to make a multi selection.
4. From the contextual menu, select Gang.
   The selected clips are ganged and the clip information turns green.

To select all clips in a gang:

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

To remove a clip from a gang:

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

Cutting Clips

You can split a clip into two or more clips using the Cut command. When you cut a clip, you can retain the cut frames as head and tail frames (handles) on each resulting clip, creating a soft clip, or you can discard the frames.

The Cut options are described as follows.

Select:	To cut the clip:
After Selected Frame	After the frame that you select when in Cut mode.
At Splices	At every soft splice that occurs on the clip.
Every Nth Frame	At the intervals that you specify in the numeric field that appears when you select this option.

To cut a clip after the selected frame:

1. Select the frame after which you want to apply a cut.
2. From the contextual menu, select **Cut ➤ After Selected Frame**.
   The clip is divided into two cuts after the selected frame. The frames in the second clip are renumbered starting at frame number 00001.

To cut a clip at its splices:

1. Select the clip on which you want to perform the cut operation.
2. From the contextual menu, select **Cut ➤ At Each Splice**.
   The clip is cut into segments at every splice point. If the clip contained no splices, no cuts are made.

To cut a clip at every Nth frame:

1. Select the clip on which you want to perform the cut operation.
2. From the contextual menu, select **Cut ➤ At (n) Frame**.
3. Enter the number of frames to be contained in each cut in the Frame Number field.
   The clip is cut into segments of the selected number of frames.

Cutting Dissolves and Timewarps

Performing a cut on an uncommitted dissolve or timewarp divides the soft edit into two soft cuts.

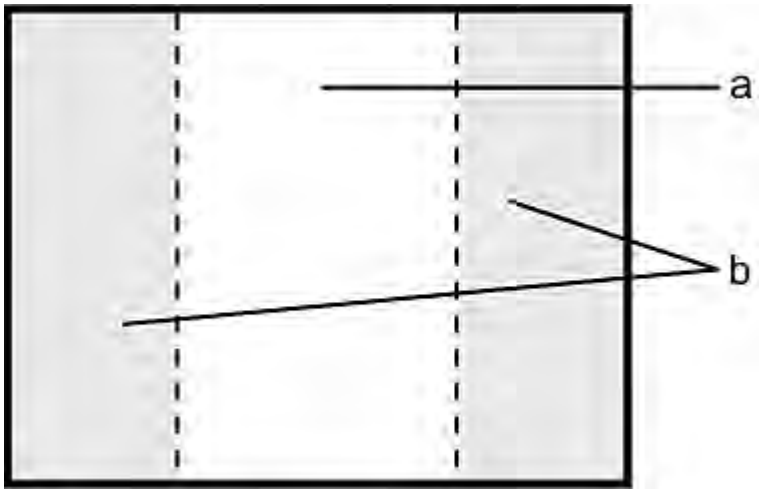
If the cut is performed on a frame within a dissolve, the dissolve is committed before the cut is made. The two resulting cuts are soft edits linked to the original source clips.

If the cut is performed on a frame within a timewarp, two clips with unrendered timewarps are created.

NOTE You should commit a soft cutout prior to archiving to avoid saving the soft cutout's source clip. See Hard Committing from the Desktop (page 308).

Cutting Clips Gesturally

Each frame in a clip contains two types of “hot spots”. The Move Clip hot spot is the part of the clip that is used to move clips around on the Desktop. The Cut Clip hot spots are the parts of the clips that are used to cut the clip.



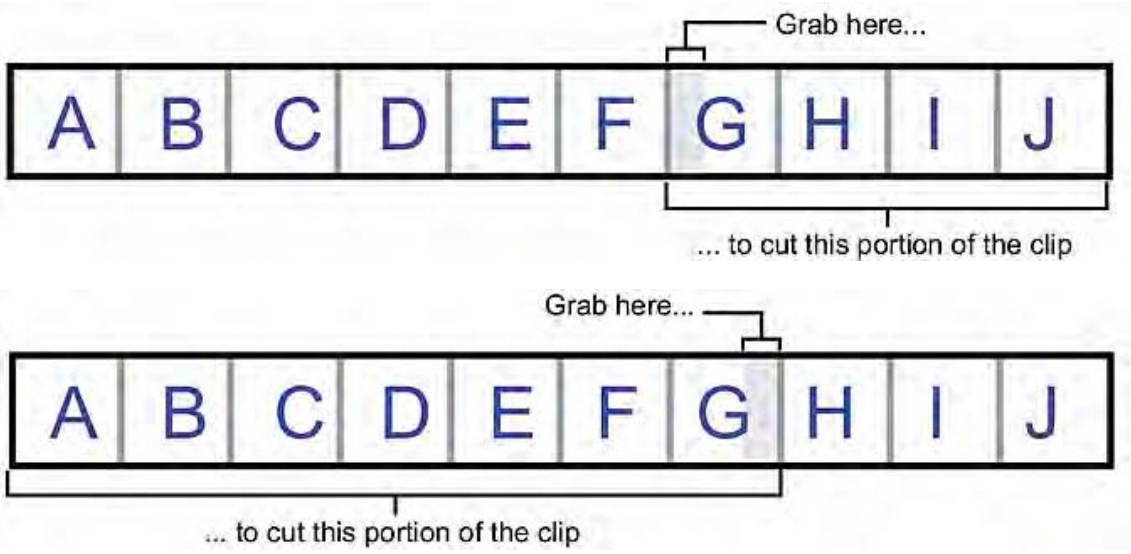
(a) Move Clip hot spot (b) Cut Clip hot spots

To cut a clip gesturally:

1. Position the cursor over the frame where you want to make the cut. Place the cursor in the Cut Clip hot spot on the frame edge closest to the cutting point.

2 Grab the frame and drag up or down to cut it from the clip.

3 Drop it in another location on a reel.



Splicing Clips

You can join together either two or more clips on a reel. This creates a single clip with a soft edit (a splice) at the location where the clips were joined.

To splice two or more clips together:

1 Select the clips you want to splice together, by Ctrl+clicking them.

2 From the contextual menu, select Splice Selected.

The two clips are spliced together to form a single clip. A yellow transition handle indicates where the two clips were spliced together. To remove the yellow handle, hard commit the clip. See Hard Committing from the Desktop (page 308).

To splice together all clips on a reel:

1 Place the cursor in an empty area to the right or left of the reel.

2 From the contextual menu, select Splice Reel.

All clips on the reel are spliced together into one clip.

Yellow transition handles indicate where the clips were spliced together. To remove the yellow handle, hard commit the clip. See Hard Committing from the Desktop (page 308).

Splicing Clips Gesturally

To splice two clips together gesturally:

1 Select the source clip you want to splice into a target clip, using the Move Clip hot spot.

2 Move the selected clip so that it overlaps either the first or last frame of the target clip. The Insert cursor is displayed.

3 Release the clip.

The two clips are spliced together to form a single clip. A yellow transition handle indicates where the two clips were spliced together. To remove the yellow handle, hard commit the clip. See Hard Committing from the Desktop (page 308).

Inserting Frames and Clips

You can insert a source clip or a portion of a source clip after a frame in the target clip.

Inserting Frames Gesturally

You can insert a portion of a source clip after a selected frame in the target clip.

To insert frames:

1. Select the source clip from which you want to insert frames into a target clip.
2. In Collapsed mode, scrub the source clip to the first frame you want to insert and drop a marker, using the Open Bracket key ([ ).
3. Scrub the source clip to the last frame you want to insert and drop a marker, using the Close Bracket key ( ]).
4. While holding Ctrl+Alt+Shift, drag up from the source clip to extract the selected frames.
5. Move the selected frames so that they overlap the target clip in between the two frames within which you want to insert the source frames. The Insert cursor is displayed.
6. Release the frames.

The source frames are inserted in between the selected frames in the target clip. The remaining frames of the target clip are placed at the end of the inserted clip.

Inserting Clips Gesturally

You can insert an entire clip between two frames in the target clip.

To insert a clip:

1. Select the source clip you want to insert into a target clip, using the Move Clip hot spot.
2. Move the selected clip so that it overlaps the target clip in between the two frames within which you want to insert the source clip. The Insert cursor is displayed.
3. Release the clip.

The source clip is inserted in between the selected frames in the target clip. The remaining frames of the target clip are placed at the end of the inserted clip.

Replacing Frames and Clips

You can replace frames in the target clip with a source clip or with frames from a source clip.

Replacing Frames Gesturally

You can replace frames in a clip with an entire source clip. There are two ways to replace clips gesturally:

- **Replace** will replace the frame on which you drop the source with the current frame of the source and the source frames before and after the current source frame will replace the corresponding frames in the target clip.
- **Replace Ahead** will replace the frames after the selected frame in the target clip (including the frame on which the source clip was dropped).
- **Replace Back** will replace all the frames before the selected frame in the target clip (including the frame on which the source clip was dropped).

To perform a Replace frames operation:

1. Make sure the source clip used is not at the first or last frame.
2. Move the selected clip over the first frame that you want to replace in the target clip. The Replace cursor is displayed.
3. Release the clip.

The current frame in the source clip is matched to and replaces the frame on which you dropped the source clip and the source frames before and after the current source frame replace the corresponding frames in the target clip.

To perform a Replace Ahead operation:

1. Make sure the source clip used is at frame 1.
2. Move the selected clip over the first frame that you want to replace in the target clip.

   The Replace Ahead cursor is displayed.
3. Release the clip.

   The current frame in the source clip is matched to the frame on which you dropped the source clip and the frames after the selected frame in the target clip are replaced with the frames of the source clip, including the frame on which the source clip was dropped.

To perform a Replace Back operation:

1. Make sure the source clip used is at the last frame.
2. Move the selected clip over the first frame that you want to replace in the target clip.

   The Replace Back cursor is displayed.
3. Release the clip.

   The current frame in the source clip is matched to the frame on which you dropped the source clip and the frames before the selected frame in the target clip are replaced with the frames of the source clip, including the frame on which the source clip was dropped.

Replacing Clips Gesturally

You can replace frames in a clip with an entire source clip. There are two ways to replace clips gesturally:

- **Replace Ahead** will replace the frames after the selected frame in the target clip (including the frame on which the source clip was dropped).
- **Replace Back** will replace all the frames before the selected frame in the target clip (including the frame on which the source clip was dropped).

To replace frames with a clip using the Replace Ahead operation:

1. Make sure the source clip used is at frame 1.
2. Move the selected clip over the first frame that you want to replace in the target clip.

   The Replace Ahead cursor is displayed.
3. Release the clip.

   The source clip replaces the frames after the selected frame in the target clip, including the frame on which the source clip was dropped.

To replace frames with a clip using the Replace Back operation:

1. Make sure the source clip used is at the last frame.
2. Move the selected clip over the first frame that you want to replace in the target clip.

   The Replace Back cursor is displayed.
3. Release the clip.

   The source clip replaces the frames before the selected frame in the target clip, including the frame on which the source clip was dropped.

Matching Clips with Their Sources

Matched sources are the original, untrimmed clips with their original names and timecode information. There are numerous ways to match clips with their sources:

- **Match Frame:** Displays the source and matches the frame and the In / Out markers of the selected clip.
- **Match Source:** Displays the source of the selected clip while retaining its original markers, if any.
- **Match Source and Keep Handles:** As above while retaining the handles.
- **Match Content:** Displays the sources of clips contained within transitions, containers, matte containers and Batch FX clips.
- **Match Content and Keep Handles:** As above while retaining the handles.

  NOTE The Match Content options can also be performed on timeline clips.
- **Match All Sources:** Displays all the sources within a selected sequence.
- **Match All Sources and Keep Handles:** As above while retaining the handles.

If the source is already on the Desktop, it is centred on its reel. If the source is not on the Desktop, the cursor becomes a white arrow, prompting you to select a destination reel on which to display the source(s).

To match a frame:

1. Select a clip on the Desktop and place the cursor at the frame you want to match.
2. From the contextual menu, select **Match ➤ Frame**.
3. Select a destination if the source is not already on the Desktop.

   The source is displayed centred on the destination reel at the appropriate frame.

To match a source:

1. Select a clip on the Desktop .
2. From the contextual menu, select **Match ➤ Source** .
3. Select a destination if the source is not already on the Desktop.

The source is displayed centred on the destination reel at the appropriate frame.

To match content:

1. Select a transition, container, matte container, or Batch FX clip on the Desktop , or a timeline clip.
2. From the contextual menu, select **Match ➤ Content**.
3. Select a destination if the sources are not already on the Desktop.
   The sources are displayed centred on the destination reel.

To match all sources within a sequence:

1. Select a sequence on the Desktop.
2. From the contextual menu, select **Match ➤ All Sources**.
3. Select a destination if the sources are not already on the Desktop.
   The sources are displayed centred on the destination reel.

Swapping Segments

You can swap a segment within a sequence with a source clip, directly from the Desktop, while preserving any Timeline FX applied to the original segment. When you swap segments, you select a frame in the target clip and a frame in the source clip. When the swap operation is performed, the selected frame from the source clip takes the place of the selected frame in the target segment and the remaining frames are aligned relative to this position.

To swap segments:

1. In the target clip, select a frame in the segment to be swapped.

NOTE If you are in collapsed view, the soft positioner at the bottom of the clip determines the selected frame. If you are in storyboard view, the first frame of the segment is selected by default.

2. From the contextual menu, select Swap Segment.

   NOTE If you are swapping a shot in a dissolve, from the contextual menu, select **Swap Segment ➤ Incoming** or **Swap Segment ➤ Outgoing** to determine whether you are swapping the outgoing or incoming segment.

3. Click a frame in the source clip.

The selected segment in the target clip is replaced by the source clip and the selected frame from the source clip takes the place of the selected frame in the target segment and the remaining frames are aligned relative to this position. The selected frames are aligned in the swap operation.

NOTE If there are not enough frames before or after the align frame in the source clip to fit before or after the align frame in the selected shot an error message appears in the message bar.



Two clips on the Desktop prior to the Swap Segment operation. In this example, the operation will swap the green segment for the blue clip while aligning frame 8 from the green clip to frame 11 from the blue clip.



After the Swap Segment operation, the green segment has been swapped with the blue clip, while aligning frame 8 from the green clip and frame 11 from the blue clip.

Hard Committing from the Desktop

You can hard commit sequences and segments from the Desktop. When you commit a sequence or segment, the soft properties of the sequence such as cuts, transitions, head and tail frames, and timewarp data are removed. The visual effects of the transitions and timewarps remain but you cannot edit them.

To hard commit a segment:

1. Make sure the clip is at a frame within the segment to be committed.
2. From the contextual menu, select **Hard Commit ➤ Segment**.
   The segment is committed.

To hard commit a sequence:

1. Select a sequence on a Desktop reel.
2. From the contextual menu, select **Hard Commit ➤ Sequence**.
   The sequence is committed.

NOTE You can hard commit a sequence via Burn, by selecting **Hard Commit ➤ Burn**, from the contextual menu.

Rendering from the Desktop

You can render segments or sequences directly from the Desktop.

To render a segment:

1. Make sure the clip is at a frame within the segment to be rendered.
2. From the contextual menu, select **Render ➤ Segment**.
   The segment is rendered.

To render a sequence:

1. Select a sequence on a Desktop reel.
2. From the contextual menu, select **Render ➤ Sequence**.
   The sequence is rendered.

NOTE You can render a sequence via Burn, by selecting **Render ➤ Burn**, from the contextual menu.

Audio 12

You can perform many of the same editing operations on audio that you use to edit video, such as cutting, trimming, adding fades, and creating stereo tracks. A comprehensive set of audio effects tools is available in the timeline and Audio Desk for mixing and mastering your audio.

When working with audio you can:

- In the MediaHub, import and export audio files using a variety of formats.
- In the timeline, synchronize audio with video, and apply audio fades. You can edit these on the fly while the clip is playing. You can also assign input strips to output strips. You can also assign audio tracks to Audio Desk input strips.
- In the Audio Desk, adjust Gain, Pan, Mute, Phase Shift, and Solo settings for up to 32 individual input strips. You can apply these on the fly while the clip is playing. You can also assign input strips to output strips.
- In the EQ Desk, apply and adjust High Shelf, Mid Notch, Mid Presence, and Low Shelf filters to input strips.
- In the Auxiliary Effects Desk, adjust global settings for the Modulation, Delay, and Reverb Auxiliary Effects.
- In the Audio Desk, adjust the output Gain and Limiting for output strips.

NOTE You can only see the Audio Desk in the single player.

Accessing the Audio Desk

The Audio Desk components are split into two panels; the patch panel will display to the left of the viewer, while the EQ Desk and the Auxiliary Effects Desk will display to the right of the viewer.

NOTE The Audio Desk is accessible from the Player.

To access the Audio Desk:

1. From the Player view, select the Options drop down.
2. Click Show Audio Desk.

   The Audio Desk panels are split to either side of the viewer.
3. Click the Audio Desk Panel button to switch between the Auxiliary Effects Desk and the Eq Desk panels.

Importing Audio Files

To import an audio file:

1. Click the MediaHub tab.
2. Enable Browse for: Files.
3. Browse to the location of the audio file.
   The audio file thumbnail appears in the browser display.
4. Drag and drop the audio file from the browser into a media panel folder.

Adding an Audio Effect on a Segment

To add an audio effect on a segment:

1. Select the audio segment.
2. Right-click on the segment and select Add Effects.
3. Enable one of the audio effects.
4. Adjust the parameters for the audio effect on the Effects Ribbon.

Adjusting the Audio Levels on Part of an Audio Segment

To adjust the audio level on part of an audio segment:

1. Select an audio segment.
2. Set the positioner to the relevant part of the segment.
3. Right-click on the segment and select Add Effect.
   The Effects Ribbon for audio effects is displayed.
4. Select Gain Audio effects.
   The toolbar displays the Gain features.
5. Enable Edit and Auto Key.
6. Right-click on the Gain slider and set Key frame.
7. Move the positioner down the sequence.
8. Adjust the Gain slider.
   Another key frame is automatically created at the gain level selected.
   The audio level is adjusted to the new gain.

You can click and drag the keyframes in the segment to adjust the audio level. Holding Shift + dragging constrains the adjustment vertically or horizontally. Setting the gain slider to zero brings the audio level back to the segment default.

Cross-fading Two Audio Segments Together

To cross-fade two audio segments together:

1. Move the vertical focus point inline with the relevant audio track.

2 Navigate to the cut point of the two audio segments with the positioner.
3 Click the Transition button.
4 In the Audio Transitions tab, enable Fade.
An audio transition is applied to the cut point of the two audio segments.
5 Adjust the length of the fade.
The audio cross-fades between the two audio segments.

Fading Audio Into or Out of a Clip

To fade audio into or out of a clip:

1 Move the vertical focus point inline with the relevant audio track.
2 Navigate to the beginning (or end) of the segment.
3 Click the Transition button.
4 In the Audio Transitions tab, enable Fade.
An audio transition is applied to the audio segment.
5 Adjust the length of the fade.
The audio is now fading into (or out of) the clip.

Scaling the Size of the Waveform

To scale the size of the waveform:

1 Do one of the following:
- Drag the audio track to increase or decrease the track size, or
- From the Audio pull down menu in the Timeline Menu Bar, choose Increase Wave Form (or to decrease it, Decrease Wave Form).

Displaying Audio Waveforms

To turn audio waveforms on:

1 From the Timeline Menu Bar at the bottom of the screen, select Audio > Show Waveforms With Effects.
The Waveform display for all audio segments is turned on.

To turn audio waveforms off:

1 From the Timeline Menu Bar, select Audio > Hide Waveforms.
The Waveform display for all audio segments is turned off.

To show audio waveforms without effects:

1 From the Timeline Menu Bar, select Audio > Show Waveforms Without Effects.
All Waveforms from the source are displayed and any audio timeline effects modifications are not reflected in the Waveforms.

Muting Audio Tracks

To mute an audio track in the timeline:

1. Locate the track you want to mute.
2. Click the speaker icon in the track identifier strip.
   The speaker icon displays as a crossed-out icon to indicate the track is muted.

To mute an input strip:

1. In the Audio Desk, enable the Mute button for the input strip that you want to mute.
   The strip is muted. In the case of a stereo audio track, both strips are muted.

Mapping Audio Tracks to Audio Desk Input Strips

In the patch panel each audio track has a small square box with a number. That number is the audio input that is matched to the numbers at the bottom of each audio strip at the bottom of the audio desk.

To map audio tracks:

1. Click on the audio input channel box in the patch panel.
2. Drag right or left and choose the audio input strip into the audio desk you would like this particular audio track to be patched too.
   An audio track input is selected.

If the input channel box is empty, the audio track is not patched to an audio desk input. You will not hear audio segments for that track.

Normally all these mappings are automated. It is important to understand these outputs are either the audio channel outputs for your hardware or discrete channel outputs for files exported to the application.

Converting Audio from Stereo to Mono or Mono to Stereo

To convert a stereo audio track to two mono audio tracks:

1. Select the stereo track by clicking on the audio channel strip.
2. From the Timeline Gear menu, select **Stereo ➤ Split Stereo Track**.

To convert mono audio to stereo audio:

1. Select two mono tracks by control-clicking on the audio channel strips.
2. From the Timeline Gear menu, select **Stereo ➤ Merge Into Stereo Track**.

Timeline Transitions

13

Timeline transitions are effects you apply directly to the transitions between clips. As such, they accelerate the process of creating and modifying effects because you don't have to enter the editor to apply them.

In order to access the Timeline transitions, you must move the positioner directly over a transition in the timeline or explicitly select it. There are separate video and audio transitions, depending on what type of clip you want to apply effects to.

The Timeline video transitions are:

- Dissolve
- Wipe
- Axis.



The Timeline audio transitions are:

- Fade



Adding Timeline Transitions

To add Timeline transitions to the timeline:

1. Select a transition in the timeline to enable the Transitions button. A proxy of the last frame of the left-hand segment, and the first frame of the right-hand segment, with a cut between the two segments, appears next to the Transition button.
2. Click the Transition button to display the Effects menu, and select the effect you want to apply. You can also right-click one of the proxies or the cut between the proxies and select one of the effects. Once you have selected an effect, a proxy of the effect replaces the cut between the left and right segment proxies.
3. You can adjust the effect's settings using the quick menu, or click the Enter Editor button to enter the full editor of the effect.

   NOTE Only one Timeline transition effect can be applied at a time. Each time you apply a new effect, only the proxy of the last effect you applied will appear between the segment proxies, and you will only be able to modify the settings of the latest effect applied.

Copying, Moving and Removing Timeline Transitions

To copy, move or remove Timeline Transitions, follow the steps below.

To copy Timeline Transitions:

1. Select a Timeline Transition.
2. Right-click the Timeline Transition and choose Copy or press Ctrl+C.
3. Select another transition in the timeline and either Right-click and select Paste or press Ctrl+P.

   NOTE Copying a Timeline Transition on to another Timeline Transition overwrites the one that was originally there.

To move Timeline Transitions:

1. Select a Timeline Transition.
2. Drag the Timeline Transition to another transition and release the cursor.

To Remove Timeline Transitions from the timeline:

1. Select a Timeline Transition.
2. Either Right-click on it and select Delete or press Alt+Click. You can also drag it to the bottom of the screen and release the cursor.

Adding Dissolve Timeline Transition

Use the Dissolve Timeline Transition to blend transitions between timeline elements in which the first clip fades out at the same time as the second clip fades in.

NOTE To create a transition, you must have enough head and tail frames in the outgoing and incoming shots.

To apply the Dissolve Timeline Transition:

1. Move the positioner between the two video clips where you want to add the transition.
2. Click the Transitions button to display the Effects menu and select the Dissolve button or right-click the positioner and select Add Dissolve.
3. The settings for the Disslove Transition appear on the Effects menu.
4. Adjust the settings for the transition. You can also click the Enter Editor button to access the full editor for the transition.

You can also create a fade to black (fade-out) by adding a dissolve at the end of the last segment or at the end of a segment followed by a gap.

Additionally, instead of fading to or from black, you can specify a colour for the dissolve. You can use the custom colour for fade-ins, fade-outs, and crossfades.

Select the dissolve you want to modify.

Depending on the type of dissolve you selected, one of the following custom colour buttons appears in the Effects menu:

1. If you selected a fade-in, a From Colour button appears.
2. If you selected a fade-out, a To Colour button appears.
3. If you selected a crossfade, a To/From Colour button appears.
4. Click the custom colour button.
5. Select a colour from the colour picker. The custom colour dissolve is created.

   NOTE If you selected a dissolve that is between two elements (not a fade-in or fade-out), the dissolve is split into a fade-out for the outgoing shot and a fade-in for the incoming shot.

Adding Wipe Timeline Transition

Wipe Timeline Transitions are non-blending transitions. Use them to enable masks that hide or reveal the outgoing or incoming shots.

NOTE To create a transition, you must have enough head and tail frames in the outgoing and incoming shots.

To apply the Wipe Timeline Transition:

1. Move the positioner between the two video clips where you want to add the transition.
2. Click the Transitions button to display the Effects menu and select the Wipe button or right-click the positioner and select Add Wipe.
3. The settings for the Wipe Transition appear on the Effects menu.
4. Adjust the settings for the transition. You can also click the Enter Editor button or click on the Wipe button in the Effects pipeline to access the full editor for the transition.

   NOTE You can enter the Stabilizer from the Wipe Editor by selecting a vertex and then clicking the Stabilizer button.

Accessing the Wipe Schematic

Axis and Geom nodes are added to the schematic when you create a garbage mask. Axis nodes are used to control a mask's position, rotation, and scaling, for example. Geom nodes contain information about how the mask will affect the image (softness, opacity, alpha, and axis offset).

To access the Wipe schematic:

1. Add a Wipe Timeline transition to the timeline.
2. Double-click the Wipe button in the Effects pipeline or click the Enter Editor button.
3. Click the World View box and select Schematic.

   NOTE Once you are in the Schematic view, you can click the Node box to add either geometry or an Axis node to the schematic.

Adjusting Preset SMPTE and Custom Wipes

Templates are included with the preset SMPTE wipes that come with your application. If you modify the start position or increase the duration for a SMPTE wipe, the interpolation for the transition may not be as even as it should be. For example, if you load a preset SMPTE pattern and change its default start position by a large number of frames, the transition may appear to jump at the last frame. Or, if you increase the duration of the transition by a large number of frames, the interpolation may also appear to jump between some frames. Use Adjust to fix this problem.

With custom wipes, you may need to create a template if:

- You intend to change the default start position for the wipe by using the X and Y Position fields in the Wipe Effects menu.
- You intend to increase the duration of the transition by a large amount.
- You intend to use the custom wipe often.
- The custom wipe has no tracking data.

When you create a template, you must specify the default start position and then define an End Bias for all nine possible end positions. Once you create the template, you must enable the blue light on the Template button to make the Adjust option available.

To adjust a preset SMPTE wipe:

1. Click the Wipe icon on the timeline or click the Wipe Timeline Transition button in the Effects pipeline to display its settings on the Effects menu.
2. Click Pattern, and then select a wipe pattern in the file browser.
3. Move the positioner to the first frame of the transition.
4. Select Position from the Global Axis box, and then change the start position of the transition of the wipe using the X and Y Position fields.

   If you scrub the transition, you should see the wipe make a large jump at the last frame.
5. If the Adjust button is not greyed out, enable it.

   NOTE If the Adjust button is greyed out, a template does not exist for the wipe and you must select a different pattern in order to use the Adjust feature.

   The jump in the wipe should be gone and the interpolation for the wipe should be fairly even.

To start, end and adjust custom wipes, follow the instructions below.

To set the start position for a custom wipe:

1. Create your wipe. See Creating Customized Wipes with Garbage Masks (page 930).
2. Select the transition on the timeline and click Pattern. The File browser appears.
3. Select Custom in the SMPTE wipes box to view saved custom transitions.
4. Select the custom transition you want to use. Your custom wipe is applied to the transition icon on the timeline.
5. Click the Enter Editor button or double-click the Wipe Timleine Transition button in the Effect pipeline to enter the Wipe Editor.
6. Click Template to display the Template menu.
7. Enable the Template Mode button to view the template in the image window.
8. Click Start Position.
9. Set the default start position by enabling a Start Position button, for example, Top Left.

To set the end bias for a transition:

1. Click End Bias.

   The positioner goes to the last frame of the wipe.
2. Set the value in the currently enabled End Bias box to 0. (The currently enabled End Bias box should be the same as the default start position.)

   On the last frame, the point closest to the centre of the axis should be just outside the image. This ensures that the mask fully covers the incoming image. If it does not, click the Mask button and proportionately increase the scale of the mask.
3. Set an End Bias for every End Bias position:
   - Click each End Bias box and drag the value until the mask is just large enough to encompass the entire image.

To set the end bias for a transition:

1. Click End Bias.

   The positioner goes to the last frame of the wipe.
2. Set the value in the currently enabled End Bias box to 0. (The currently enabled End Bias box should be the same as the default start position.)

   On the last frame, the point closest to the centre of the axis should be just outside the image. This ensures that the mask fully covers the incoming image. If it does not, click the Mask button and proportionately increase the scale of the mask.
3. Set an End Bias for every End Bias position:
   - Click each End Bias box and drag the value until the mask is just large enough to encompass the entire image.

To adjust the custom wipe:

1. Enable Template.
2. Exit the Wipe Editor.
3. Change the X and Y position for the wipe in the Wipe quick menu, or increase the duration of the transition.

   Notice how the interpolation causes the last frame of the wipe to jump.

4 Enable Adjust in the Wipe quick menu. The interpolation of the transition should be much smoother.

Creating Customized Wipes with Masks

Use masks on the incoming clip to create a customized transition that wipes into the outgoing clip.

Before you create a mask, set how the points of the mask are drawn.

To set the drawing options:

1 From the Wipe Editor, click Setup.

2 Specify the setup options.

Enable:	To:
Auto Tangents	Create tangents for every new point you set.
Show Border	View the border defined in the Offset field of the Shape menu. The colour pot next to the Show Border button defines the colour of the border.
Invert	Reverse your wipe. For instance, if you originally create an expanding box wipe, enabling Invert creates a shrinking box.
Spline Keyframing	Allow animation of points on the mask.

3 To change the colour of the mask's wireframe, colour points, or offset border, click a colour pot and pick a new colour.

An Axis node contains all the rotation, scaling, and position data for the mask. You can change all this information in the Mask menu. The Axis controls are identical to those in the Action Axis menu.

You can animate the following mask properties:

- The characteristics of the camera in the scene
- The position, rotation, scaling, and shearing
- The mask shape
- The offset, border offset, border alpha, colour, and opacity

To display the Animation menu, click the Animation button.

Creating a Mask

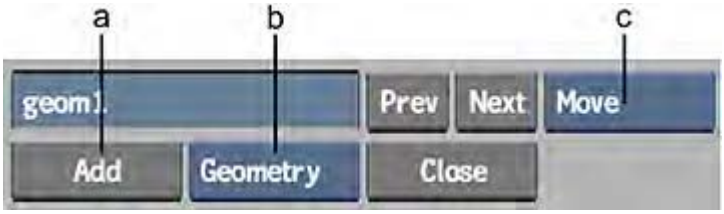
To create a mask, you must enter the full editor for the Wipe effect.

NOTE When you create a mask, an Axis node and a Geom node are added to the Wipe schematic.

To create a wipe mask:

1 From the Wipe Editor, click Mask.

2 Select Geometry from the Node box.



(a) Add button (b) Node box (c) Edit Mode box

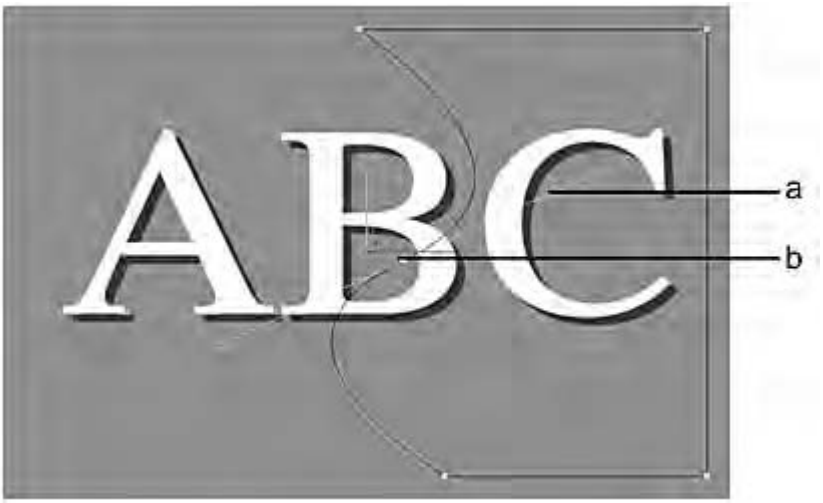
3 Click Add, or select Create from the Edit Mode box.

TIP You may want to display the outgoing or incoming clip while drawing the mask since the composite is created in Result mode.

4 Click the image to create the first point.

NOTE If you are drawing a mask freehand, press `Shift` and then press down on the cursor to draw the mask. When you release the cursor, the mask closes automatically.

5 Click again to draw the second point.



(a) Tangent (b) Vertex

NOTE If you are not in Auto Tangents mode, you can still create a tangent while setting a point by clicking and dragging the cursor.

6 To close the mask, click Close or click the first point.

Modifying a Mask's Shape

You can manipulate points on a mask while displaying the incoming or outgoing clip.

Select points using the Select option in the Edit Mode box. You can select individual points, individual tangents, or a group of points:

1 To select a point or tangent, click it.
2 To select additional points, press `Shift` and click the point. You can also press `Shift+Ctrl` and drag across a series of points.
3 To select multiple points, press `Ctrl` and drag the selection box over a series of points.

You can break or modify a tangent using the Break option in the Edit Mode box:

1. Click a point to remove any tangents connected to that point.
2. Click a tangent and hold down the cursor to edit it separately from its opposite tangent. You can also select the Move option to edit the tangent without editing its continuity. Click the tangent again and hold down the cursor to edit the continuity of the tangent. Click the tangent again and hold down the cursor to edit the continuity of both tangents.
3. Click a point to remove any tangents connected to that point.

You can use the Auto option in the Edit Mode box to calculate the tangent for a point according to the curve the point is on. Select the Auto option and click a tangent.

Controlling a Mask's Effects

You can control a mask's effect in the Mask menu.

NOTE The Geom node contains all the information about how a mask affects the final transition.

To control a mask's effect:

1. Select Matte or Result.
2. Select the mask in the image window.
3. Click Mask.
4. Set the mask opacity in the Opacity field.

   The Opacity field defines the effect the mask has on the matte. A value of 100% means the inside of the mask is completely opaque. A value of 50% means the inside of the mask is 50% transparent. A value of 0% has no effect on the image.



(a) Colour and Opacity fields (b) Alpha and Offset fields (c) Axis Offset fields



Mask with 0% opacity



Mask with 100% opacity

5 Set the mask colour in the Colour field.

The Colour field defines the blend between the outgoing and incoming image inside the mask. A value of 50% is a 50/50 blend between the outgoing and the incoming clip. A value of 100% displays only the outgoing clip. A value of 0% displays only the incoming clip.



Mask with 25% colour and 100% opacity



Mask with 75% colour and 100% opacity

6 Set the mask softness (see below).

7 Set the axis offsets in the Axis Offset fields.

The Axis Offset fields defines how much a mask is offset from its axis.

8 Enable Outside to apply the effect to the part of the image outside the mask shape.

You can adjust the softness gradient of a mask to smoothen the edges of your wipe. You can create a uniform gradient around the edge of the mask or use an advanced gradient to control the shape of the gradient at different parts of the mask.

To create a uniform gradient, you define how far you want the gradient to be offset from the edge of the mask and then set its transparency. To vary the shape of the gradient, you move vertices on inner and outer softness splines. The gradient will be based on how far each vertex point is offset from the mask.

For either type of gradient, you can smoothen the gradient towards the inside edge, the outside edge, as well as the area where the inside and outside adjustments have an effect.

To smoothen the gradient of a mask:

1. From the Wipe Editor, click Mask.
2. To create a uniform gradient around your mask, do the following:
   - If necessary, toggle the Edge Softness button to Softness.
   - Set the border of the softness gradient using the Offset field. Your gradient will be affected by how far the softness border is offset from the edge of the mask.
   - Set the transparency of the gradient using the Alpha field.
3. To change the shape of a gradient at different parts of the mask, do the following:
   - If necessary, toggle the Edge Softness button to Advanced Gradient.
   - Adjust the distance of the inner and outer splines from the mask by adding and moving vertex points on the splines. If you do not see the inner and outer splines, enable Splines.
4. Use the Inner Edge field to smoothen the softness gradient towards the inside.
5. Use the Outer Edge field to smoothen the softness gradient towards the outside. Adjusting this value is especially noticeable if you are smoothening a transition from a black inside edge to a white outside edge.
6. Use the Distance field to modify the area over which the Inner and Outer Edge adjustments have an effect.

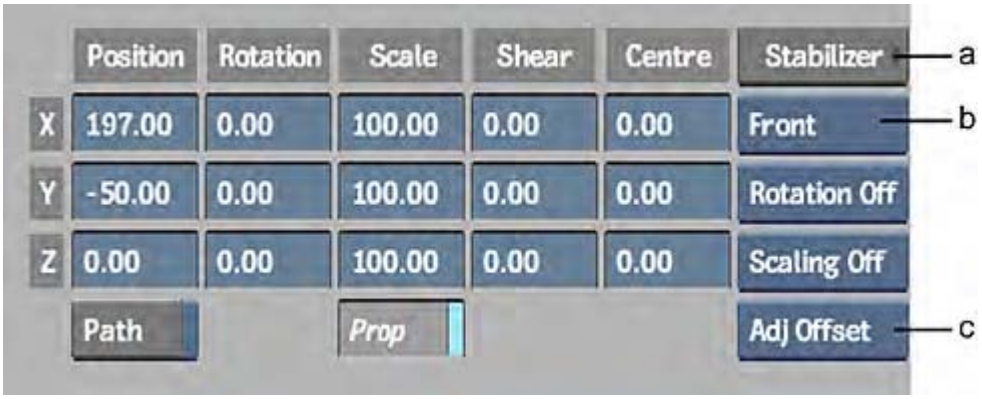
Tracking with Masks

You can track a point on your clip and apply that data to the axis of a piece of geometry or a hierarchy of objects. You can also track individual points on the mask.

NOTE The points in the mask are not affected by the tracking data. The tracking data is applied to the entire mask.

To track the entire mask:

1. After creating a wipe, access the Wipe Editor and click Mask.
2. Select the wipe's mask axis.
3. Select Front or Back from the Layer box to specify whether you want to track on the outgoing clip (Front) or the incoming clip (Back).



(a) S (Stabilizer) button (b) Layer box (c) Adjust option box

4 Select an option from the Adjust option box.

Select Adjust Offset if the selected mask is parented to one axis. Select Adjust Axis if the selected mask is parented to a hierarchy of objects.

5 Go to the first frame of the transition.

6 Click Stabilizer.

7 Set up the tracking point as needed.

8 Click Analyse.

Fine-tune your analysis if necessary.

9 Click Return.

The Wipe Editor reappears.

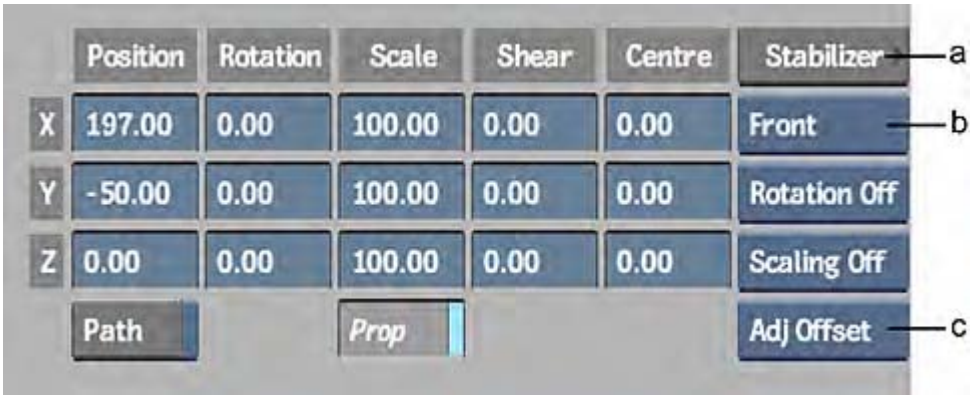
You can also track individual points on a mask so that portions of the mask follow a point in the clip. Each point you select is assigned a tracker box in the Stabilizer. The points are repositioned according to the reference point you set in the Stabilizer.

To track individual points on a mask:

1 After creating a wipe, access the Wipe Editor.

2 Select a vertex or a group of vertices on a wipe's mask.

3 Click Mask.



(a) S (Stabilizer) button (b) Layer box (c) Adjust option box

4 Select Front or Back from the Layer box to specify whether you want to track on the outgoing clip (Front) or the incoming clip (Back).

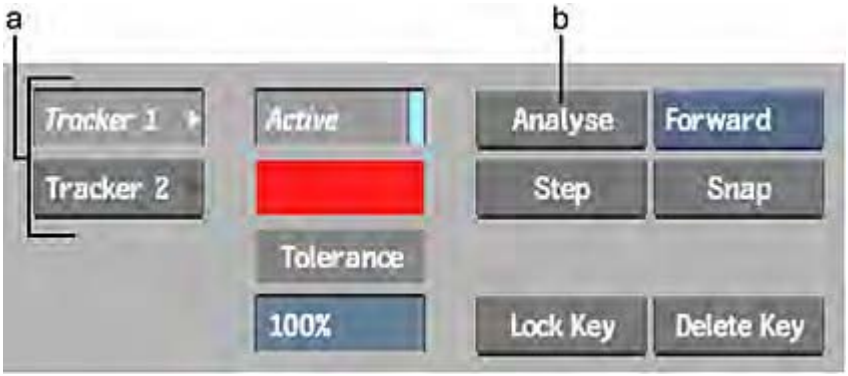
5 Select an option from the Adjust option box.

Select Adjust Tangents if you want the tangents for the selected points to be adjusted while the points are being tracked.

6 Go to the first frame of the transition.

7 Click Stabilizer.

The Stabilizer appears. A Tracker button appears for each selected vertex.



(a) Tracker buttons (b) Analyse button

8 Click Tracker 1, and then set up the first tracking point.

9 Click Tracker 2 to set up the second tracking point, and then continue setting up tracking points for all remaining vertices.

10 Click Analyse.

Fine-tune your analysis if necessary. For example, disable problem trackers and analyse again.

11 Click Return.

The Wipe Editor reappears.

12 Fine-tune your mask if necessary.

Adding Axis Timeline Transition

Use the Axis Timeline Transition to modify the scaling, rotation and position for the axis of one image to create the transition effect.

You can quickly swap the incoming and outgoing sources to reverse the animation. Swapping sources accommodates applications such as Apple's Final Cut Pro and Avid products, where the effect is applied to the outgoing source rather than the incoming source.

NOTE The Axis Timeline Transition menu is the same as the Axis Timeline FX menu.

To apply the Axis Timeline Transition:

1 Move the positioner between the two video clips where you want to add the transition.

2 Click the Transitions button to display the Effects menu and select the Axis button or right-click the positioner and select Add Axis Transition.

3 The settings for the Axis Transition appear on the Effects menu.

4 Adjust the settings for the transition. You can also click the Enter Editor button to access the full editor for the transition.

To swap the sources of an Axis Timeline Transition:

1 Select a transition point in the timeline.

2 Right-click on the transition point or click the Transition button to bring up the Effects menu.

3 Select an Axis transition.

4 Create the transition. For example;
- At the first frame of the transition, set the Scale X and Y values to 0.
- At the last frame of the transition, set the Scale X and Y values to 100

5 Play the transition.

6 In the Effects menu, enable the Swap button.

NOTE In the full Axis editor, the Swap button is called the Swap Source button.

7 Play the transition again. This time the incoming and outgoing sources are switched and the animation is reversed.

Adding Fade Timeline Transition

Use the Fade Timeline Transition to specify the duration of a fade, the interpolation of the fade-in and fade-out, and the rate of the fade. The Fade Timeline Transition is the only transition on the Audio Effects menu and in the pipeline.

To apply a Fade Timeline Transition:

1 Move the positioner between the two audio clips where you want to add the transition.

2 Click the Transitions button to display the Effects menu and select the Fade button or right-click the positioner and select Add Fade.

3 The settings for the Fade Transition appear on the Effects menu.

4 Adjust the settings for the transition. You can also click the Enter Editor button to access the full editor for the transition.

Timeline FX

14

Timeline FX are effects you add directly to clips on the timeline. As such, they accelerate the process of creating and modifying effects because you don't have to enter the editor to apply them.

There are separate video and audio effects, depending on what type of clip you want to apply effects to.

Timeline FX follow a specific processing order. Each effect, regardless of when it is applied to a clip, will be processed in the same order in the pipeline.

The processing order of the Timeline Video effects is as follows:

- Time Warp
- Resize
- Text
- Colour Correct
- Spark
- Blend
- Wipe
- Axis



The processing order of the Timeline Audio effects is as follows:

- Time Warp
- Gate
- Compression
- Equalizer
- Gain
- Reverb Send
- Delay Send
- Mod Send



Modifying Clip Format and Import Options on the Timeline

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

For example, after importing R3D footage and adding it to the timeline, you can edit the colour curves of a clip in the Effects pipeline, and also copy the option from the Effects pipeline directly to other R3D clips that are used in the same timeline.

To access the Format and Pre-processing Options Editor:

1. Click the Timeline tab.
2. Select the clip in the timeline to display its Effects pipeline.
3. Do one of the following:

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

   - Select the Format Options button in the Effects pipeline, and click the Editor button in the quick menu.

   The Basic menu opens. Format options will be saved to this menu.
   - Select the Pre-Processing button in the Effects pipeline, and click the Editor button in the quick menu.

   The Resize menu opens. Import options will be saved to this menu and the RGB LUT menu.

To copy format and import options to another segment on the timeline:

1. Select the clip you wish to copy in the timeline.
2. In the Effects pipeline, do one of the following:
   - To copy basic clip format settings, Click and drag the Format Options button to another segment on the timeline.
   - To copy resize and RGB LUT settings, click and drag the Pre-processing button to another segment on the timeline.

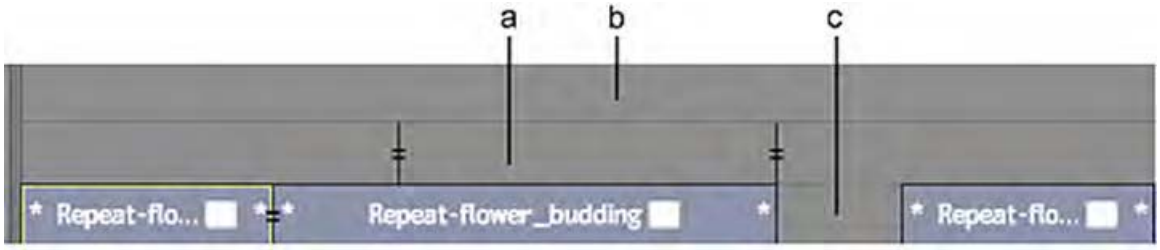
Gaps

Gaps are empty spaces between elements in an edit sequence. When you apply Timeline FX to a gap, you create a gap effect,.

While Timeline FX can only be applied to a clip, the effects applied to gaps are independent of all media. They effect the media under them.

Gaps usually appear in the following places:

- The empty space between media elements on a video track.
- An empty video track.
- The space between cut points on an empty video track. Cut points are useful for containing a gap effect applied to an empty video track. The areas to either side of the cut points are also gaps.



(a) Space between cut points (b) Empty Track(c) Empty space between elements

Gap effects are not restricted by in points, out points, cuts and transitions, so they are easily trimmed, moved and duplicated. You can trim a gap effect over the entire duration of the video track regardless of the underlying cut points.

You can also freely edit cuts and transitions between elements on video tracks under the gap effect. You can cut an element, add a transition at the cut point, and then replace the incoming element without affecting the gap effect. In addition, gap effects can be copied and placed over different media in the timeline.

Adding Timeline FX to Gaps

To add Timeline FX to a gap:

1. Move the positioner and its focus point over the gap to select the gap, or explicitly select the segment.
2. Click the FX button and select the Timeline FX you wish to apply to the gap, or select a Timeline FX from the Effects Menu and drag it to the gap.
3. Edit the Timeline FX using the quick menu or click the Enter Editor button to access the full editor.

Modifying Multiple Timeline FX

You can modify Timeline FX from different clips at the same time by mulit-selecting clip segments. Selecting multiple segments allows you to apply changes to effects in one segment, and have those changes applied to the same effects in the other segments.

To modify multiple Timeline FX:

1. Select multiple segments in the timeline.
2. Add or edit an effect in one segment using the Effects Menu. Those modifications will be applied to all segments which you have selected.

Copying Timeline FX

You can copy Timeline FX from the timeline to the workspace, from the workspace back to the timeline, or from one segment on the timeline to another. Additionally, you can copy multiple Timeline FX to the workspace or another segment simultaneously, as long as the Timeline FX you want to copy are from the same segment.

To copy Timeline FX from the timeline to the workspace:

1. Select the Timeline FX you wish to copy either by clicking the specific FX button in the Effects pipeline or by clicking the FX proxy on the track.

   NOTE If you want to select the FX proxy on the track, you may have to resize the video or audio track so that the effect is visible on the timeline.

2. Drag the Timeline FX to the workspace. If you want to copy multiple Timeline FX, Ctrl-click the ones you wish to copy and drag them to the workspace.

The Timeline FX proxy appears in the workspace. It displays the front view of the effect, the effect you copied, and the name of the effect. If you copied multiple Timeline FX, they will appear as a single black proxy displaying the names of the Timeline FX.

To copy Timeline FX from the workspace to the timeline:

1. Select the Timeline FX proxy you wish to copy or click the specific Timeline FX button in the Effects pipeline and drag it to a segment on the timeline.

To copy Timeline FX from one segment in the timeline to another:

1. Click the Timeline FX proxy you wish to copy, or click the specific Timeline FX in the Effects pipeline, and drag it to the other segment. If you want to copy multiple effects, `Ctrl`-click the ones you wish to copy and drag them to the other segment.

   NOTE If you want to select the FX proxy on the track, you may have to resize the video or audio track so that the effect is visible on the timeline.

Sliding Timeline FX KeyFrames

To slide Timeline FX within a clip:

1. Select the Timeline FX that you want to slide by clicking it in the Effects Menu or clicking the effect icon in the clip.
2. If the clip does not already contain animation keyframes, set keyframes on the clip to animate the FX.
3. Select how the animation channel is affected when you slide the animated effects by choosing an option from the Keyframe Move Modes box.

Select:	To:
Reposition Proportionally	Resize the channel as you trim. The animation channel is scaled to fit into the timeline element. This option has no effect when you slip or slide.
Shift With Media	Link the keyframes to their original frame numbers. The animation channel moves to follow the original frames as you trim.

Select:	To:
Pin To Segment Start	Unlink the keyframes from their original frame number. The animation channel remains with the timeline element as you trim.

4 From the Editorial Mode box, select Slide Keyframes, which allows you to slide all of the keyframes of a specific Timeline FX on a clip.

NOTE You can only slide one Timeline FX at a time.

Muting and Deleting Timeline FX

If you want to temporarily remove Timeline FX from a clip, you can mute it.

To mute Timeline FX:

1 From a segment on the timeline, select the Timeline FX you wish to mute.
2 Then from the Effects menu, Click the blue LED on the Timeline FX button. The LED on the button will turn dark to indicate that the Timeline FX has been turned off. The blue LED on the Timeline FX in the segment on the timeline will turn off as well, indicating that the effect has been muted. To unmute an effect, click the LED on the Timeline FX button in the Effects menu. The LED will turn blue, to indicate that the Timeline FX has been unmuted.

To delete Timeline FX:

1 Then from the Effects menu, Click the blue LED on the Timeline FX button. The LED on the button will turn dark to indicate that the Timeline FX has been turned off. The blue LED on the Timeline FX in the segment on the timeline will turn off as well, indicating that the effect has been muted. To unmute an effect, click the LED on the Timeline FX button in the Effects menu. The LED will turn blue, to indicate that the Timeline FX has been unmuted.
2 Select the Timeline FX from a segment in the timeline that you want to delete.
3 Either click the Timeline FX and drag it to the bottom of the screen, or right-click the Timeline FX, and from the context menu select Delete. You can also click the Delete button on the right side of the Effects menu.

Adding Timewarp Timeline FX

Use the Timewarp Timline FX to speed up or slowdown clips on the timeline. The Timewarp Timeline FX is the first effect on both the Video and Audio Effects menu and in the pipeline.

There are two Timewarp Timeline FX, one for video and one for audio; depending on what type of clip you select, you will have access to one or the other when you click the FX button.

To apply Timewarp Timeline FX:

1 Select the video or audio clip you wish to apply the effect to.
2 Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3 Click the Timewarp button. A timewarp icon appears on the clip.
4 Adjust the settings for the effect. You can also click the Enter Editor button to access the full editor for the effect.

Adding Resize Timeline FX

Use the Resize Timeline FX to set how the segment adjusts itself to the timeline resolution and change the bit depth directly on the timeline. The Resize Timeline FX is the second effect on the Video Effects menu and in the pipeline.

A Resize Timeline FX is automatically applied to clips added to the pipeline that do not have the same resolution or scan mode as the current clip on the timeline. When you mix different clips of different resolutions on the timeline, the Resize Timeline FX is applied using the following attributes:

- Resolution
- Bit Depth
- Scan mode

To apply Resize Timeline FX:

1. Select the video clip you want to apply the effect to.
2. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3. Click the Resize button. A resize icon appears on the clip.
4. Adjust the settings for the effect. You can also click the Enter Editor button to access the full editor for the effect.

Adding Text Timeline FX

Use the Text Timeline FX to add movement and colour to text over time. The Text Timeline FX is the third effect on the Video Effects menu and in the pipeline.

Most Text Timeline FX can be previewed in real time. You can add several stacked Text Timeline FX to a timeline without significantly affecting performance.

Text Timeline FX differ from the full Text editor in several ways. With Text Timeline FX you can:

- Create a text layer with or without a matte
- Create a matte key
- Add text to a gap effect

You can also edit attributes such as colour and font directly in the XML file.

NOTE To use logos, you will have to work with the full Text editor.

To apply Text Timeline FX:

1. Select the video clip you want to apply the effect to.
2. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3. Click the Text button. A text icon appears on the clip.
4. Adjust the settings for the effect. You can also click the Enter Editor button to access the full editor for the effect.

Creating a Text Matte Key

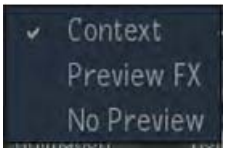
A matte key allows you to create a text effect in which the text's fill is a background clip. When used with an Axis Timeline FX (or a Sparks effect that takes a matte), it can be used to layer text made of one background on top of another background.

NOTE All static (non-animated) fill colours are set to a transparency of 0%.

To create a text matte key:

1. Add two layers to the timeline.
2. Click the FX button or right-click on the clip and select Add Effect. The Effects Menu appears.
3. Click the Text button.
4. To enter the Text menu, click the Text in the Effects Pipeline or click the Enter Editor button.
5. In the Text editor, enable the Matte Key button.

   The Alpha Processing Mode box changes to RGBA and the Fill field resets to 0%.
6. Select Context from the Preview Options box to preview the effect.



   The text fill is composited over the background layer. This text layer is then applied to the timeline background layer.

Adding Colour Correct Timeline FX

Use the Colour Correct Timeline FX to add colour corrections directly on the timeline. You can also colour correct segments on the timeline using the Colour Warper. The Colour Correct Timeline FX is the fourth effect on the Video Effects menu and in the pipeline.

To apply a Colour Correct Timeline FX:

1. Select the video clip you want to apply the effect to.
2. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3. Click the Colour Correct button. A colour correct icon appears on the clip.
4. Adjust the settings for the effect. You can also click the Enter Editor button to access the full editor for the effect.

   NOTE Click the Colour Correct button to toggle between Colour Correct and Colour Warping Settings.

Colour Matching on the Timeline

To colour match using the Triptych player:

1. Click the Layout button and select the Triptych player, or swipe to the side of the Viewing Panel to prompt the Layout Selection Overlay.

2 Move the positioners over the elements you want to use for the colour match. See Comparing Three Shots in the Triptych Player (page 251).

3 Click the FX button or right-click on the clip and select Add Effect. The Effects Menu appears.

4 Click the Colour Correct button.

5 To enter the Colour Corrector menu, click the Colour Corrector in the Effects Pipeline or click the Enter Editor button.

6 In the Colour Corrector editor, click the Curves tab to display the Curves menu.

7 Click the Source colour pot.

8 Plot the colour you want to modify by selecting it with the eyedrop cursor.

9 Plot the match colour in Dest 1 or Dest 2.



10 Specify the match parameters by enabling the Channel Match button and selecting a channel from the Channel Selection box.

11 Click either Match 1 or Match 2.

The Match curves update. You can fine-tune the result by adjusting the curves handles.

Adding Sparks Timeline FX

Use the Sparks Timeline FX to apply video Sparks to video elements in the timeline. The Sparks Timeline FX is the fifth effect on the Video Effects menu and in the pipeline.

Sparks Timeline FX are applied below Axis and Wipe effects. If you want to apply a Sparks Timeline FX to an element that contains an Axis or Wipe effect, you need to contain it first if the result of the Axis or Wipe effect is required by the Sparks. You can also apply a Sparks Timeline FX to a gap above the element containing an Axis or a Wipe effect.

To apply Sparks Timeline FX:

1 Select the video clip you want to apply the effect to.

2 Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.

3 Click the Sparks button. A sparks icon appears on the clip.

4 If the Sparks you are using requires a matte, click the Matte button in the Effects menu to load the matte.

5 Click the Sparks button to load your Sparks.

6 Depending on what Sparks you load, there may be additional setting on the Effects menu. Additionally, and also depending on what Sparks you load, you my be able to enter the full editor by clicking the Enter Editor button.

Adding Blend Timeline FX

Use the Blend Timeline FX to adjust the transparency between layers on the timeline. The Blend Timeline FX is the sixth effect on the Video Effects menu and in the pipeline.

To apply Blend Timeline FX:

1. Select the video clip you want to apply the effect to.
2. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3. Click the Blend button. A blend icon appears on the clip.
4. Adjust the settings for the effect. You can also click the Enter Editor button to access the full editor for the effect.

Adding Wipe Timeline FX

Use the Wipe Timeline FX to create vertical wipes between layers on the timeline. The Wipe Timeline FX is the seventh effect on the Video Effects menu and in the pipeline.

To apply Wipe Timeline FX:

1. Select the video clip you want to apply the effect to.
2. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3. Click the Wipe button. A wipe icon appears on the clip.
4. Adjust the settings for the effect. You can also click the Enter Editor button to access the full editor for the effect.

   NOTE For information on creating wipes with masks, see: Creating Customized Wipes with Masks (page 320).

Adding Axis Timeline FX

Use Axis Timeline FX to create a single layer composite directly on the timeline. The Axis Timeline FX is the eighth effect on the Video Effects menu and in the pipeline.

NOTE Adding an Axis Timeline FX will automatically mute a Resize Timeline FX, however the Resize Timeline FX can be unmuted.

To apply Axis Timeline FX:

1. Select the video clip you want to apply the effect to.
2. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3. Click the Axis button. An axis icon appears on the clip.
4. Adjust the settings of the effect. You can also click the Enter Editor button to access the full editor for the effect.

Adding Matte Containers

A matte container is a container with the RGB portion of an image on one track and its matte on another. If the matte for a clip exists separately from it, you can add the clip and its matte to the timeline as a single element by creating a matte container for them.

To create a matte container:

1. Select the clip you want to add the matte container to.
2. Click the FX button and add an Axis Timeline FX or right-click on the clip, select Add Effect and choose the Axis Timeline FX.
3. From the Axis Timeline FX settings, click the Matte button.
4. Select a Matte from the Workspace. The Matte Container button appears in the Effects Menu.
5. To open the Matte Container, click once on the Matte Container button and then click open, or double-click on the Matte Container button.
6. Once the Matte container is open, you can adjust the settings for the matte or add other effects to it.
7. If you want to mute the matte, right-click the Matte Container button and select Mute.
8. If you want to remove the matte, click once on the Matte Container button and then click on the Uncontain button.

Accessing the Axis Keyer

You can create chroma keys directly on the timeline by accessing the Axis Keyer using the Axis Timeline FX. When you enter the Axis Keyer, the clip loaded as the Front is the clip with the Axis soft effect, and the clip loaded as the Back is the next available track on the timeline.

NOTE The preferred keying workflow is to access the Modular Keyer via the Timeline FX. See Setting Up the Nodes and Media to Pull a Key (page 792).

To access the Axis Keyer:

1. Build a multilayer timeline with a front track and a background.
2. Select the track you want to key, and then right-click and choose Add Effect, or click on the FX button. The Effects menu appears.
3. Click the Axis button. An Axis icon appears on the track and an Axis button appears in the Effects pipeline.
4. To enter the Axis editor, click the Axis button in the Effects pipeline, or click the Enter Editor button.
5. In the Object menu, click the Keyer. The Axis Keyer appears.

   The Front clip is provided by the clip with the Axis soft effect.
6. Create the key.

   The back is provided by the next available clip in the multilayer timeline.

Adding Gate Timeline FX

Use the Gate Timeline FX to lower or remove the perceptible level of noise in an audio signal from an audio segment. It does not, however, remove noise from the audio signal. The Gate Timeline FX is the second effect on the Audio Effects menu and in the pipeline.

A noise gate is commonly used when the level of the desired signal generally stays above the level of the noise. In this case, the threshold is set above the level of the noise. When the level of the signal remains above the threshold level, the gate is open and both the signal and the noise are allowed to pass through. When the level of the signal falls below the threshold level, the gate is closed and no signal is allowed to pass. In effect the perceptible noise is attenuated.

To apply Gate Timeline FX:

1. Select the audio clip you want to apply the effect to.
2. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3. Click the Gate button. A gate icon appears on the clip.
4. Adjust the settings for the effect. You can also click the Enter Editor button to access the full editor for the effect.

Adding Compression Timeline FX

Use the Compression Timeline FX to reduce the dynamic range of an audio signal from an audio segment. The Compression Timeline FX is the third effect on the Audio Effects menu and in the pipeline.

When you compress an audio signal, there is less of a difference between the lowest measured gain of the signal and the highest measured gain of the signal. Compression is useful if your audio signal has many different loudness levels that you want to play back at a similar perceived loudness level. For example, when recording voice-overs, you may want to maintain a constant perceived level of loudness in the actor's voice.

You can also compress an audio signal to attenuate portions of the signal that are too loud. For example, if your recording is at a consistent loudness level, but for some reason there is an unwanted peak in loudness level, you can compress the loud portion to give it the same perceived loudness as the rest of the recording.

Because compressing an audio signal could have a drastic effect on the overall dynamic range of the signal, it is not effective in every situation. For example, for a complex recording that has been mixed down from many other audio sources, such as a vocal track with music and sound effects, the audio signal might have complex differences in loudness levels. If you compress a master audio signal such as this, you will lose many of the original qualities of the audio signal. In such cases, it is advisable that you compress the source audio track before mixing down.

If you are editing a pair of mono tracks that you wish to process as stereo tracks, you should merge both tracks into one stereo audio track. This ensures that the same compression is used on both left and right channels.

To apply Compression Timeline FX:

1. Select the audio clip you want to apply the effect to.
2. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3. Click the Compression button. A compression icon appears on the clip.
4. Adjust the settings fo the effect. You can also click the Enter Editor button to access the full editor for the effect.

Adding EQ Timeline FX

Use the EQ Timeline FX to perform precise manipulations of the audio frequency content using the EQ Editor, which is based on the graphical display of EQ settings. The EQ Timeline FX is the third effect on the Audio Effects menu and in the pipeline.

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

NOTE These filters can have a dramatic effect on the audio so they should be used sparingly.

To apply EQ Timeline FX:

1. Select the audio clip you want to apply the effect to.
2. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3. Click the Gate button. A gate icon appears on the clip.
4. Adjust the settings for the effect. You can also click the Enter Editor button to access the full editor for the effect.

Adding Gain Timeline FX

Use the Gain Timeline FX to adjust the gain for individual segments in an audio track. The Gain Timeline FX is the fourth effect on the Audio Effects menu and in the pipeline.

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.

1. To apply Gain Timeline FX:
2. Select the audio clip you want to apply the effect to.
3. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
4. Click the Gain button. A gain icon appears on the clip.
5. Adjust the settings for the effect. You can also click the Enter Editor button to access the full editor for the effect.

TIP If Show Gain Animation is enabled in the Timeline options menu, you can interact with a Gain Animation curve directly on a timeline audio clip. In this case, a Gain Timeline FX is automatically applied when you modify the Gain curve.

Adding Reverb Send Timeline FX

Use the Reverb Send Timeline FX to patch audio segments to the Reverb Auxiliary Effects. The Reverb Send Timeline FX is the sixth effect on the Audio Effects menu and in the pipeline.

All parameters for the Reverb Auxiliary Effect are set through the Auxiliary Effects panel. You can route any number of audio segments through the Reverb Auxiliary Effect but the Reverb effect is output in stereo to two separate output channels, combining the effect on all input segments. You can adjust the amount of reverberation that can be applied to an individual segment using the Reverb Amount slider.

To apply Reverb Send Timeline FX:

1. Select the audio clip you want to apply the effect to.
2. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3. Click the Reverb Send button. A reverb send icon appears on the clip.
4. Adjust the settings for the effect. You can also click the Enter Editor button to access the full editor for the effect.

Adding Delay Send Timeline FX

Use Delay Send Timeline FX to patch audio segments to the Delay Auxiliary Effect. The Delay Send Timeline FX is the sixth effect on the Audio Effects menu and in the pipeline.

All parameters for the Delay Auxiliary Effect are set through the Auxiliary Effects panel. You can route any number of audio segments through the Delay Auxiliary Effect but the Delay effect is output in stereo to two separate output channels, combining the effect on all input segments. You can adjust the amount of reverberation that can be applied to an individual segment using the Delay Amount slider. You can also send the segment levels to the Left and Right Delay Channels independently using the Pan Delay slider.

To apply the Delay Send Timeline FX:

1. Select the audio clip you want to apply the effect to.
2. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3. Click the Delay Send button. A delay send icon appears on the clip.
4. Adjust the settings for the effect. You can also click the Enter Editor button to access the full editor for the effect.

Adding Modulation Send Timeline FX

Use the Modulation Send Timeline FX to patch audio segments to the Modulation Auxiliary Effect. The Modulation Send Timeline FX is the seventh effect in the Audio Effects menu and in the pipeline.

All parameters for the Modulation Auxiliary Effect are set through the Auxiliary Effects panel. You can route any number of audio segments through the Modulation Auxiliary Effect. The Modulation effect applies globally to the entire sound track and is output in stereo to a pair of output channels, combining the effect on all inputted audio segments. You can adjust the amount of modulation that can be applied to an individual segment by using the Mod Amount slider.

To apply Modulation Send Timeline FX:

1. Select the audio clip you want to apply the effect to.
2. Either click the FX button or right-click on the clip and select Add Effect. The Effects menu appears.
3. Click the Modulation Send button. A modulation send icon appears on the clip.
4. Adjust the settings for the effect. You can also click the Enter Editor button to access the full editor for the effect.

Creating New Sources Using Tools from the Tools Tab

15

The Tools tab provides access to numerous effects and tools allowing you to create modified clips that you can then use as sources on the timeline or in Batch or Batch FX. The tools use input clips (or in some cases are generated from scratch) that are then modified through parameters to produce a result directly in the workspace.

Some tools allow you to set parameters directly in the Tools quick menu, and instantly see a rendered result in the workspace. Other tools have full editors that you enter, allowing you to tweak numerous settings before rendering your result back to the workspace. In either case, new modified clips are created that you can use in your project.

Saving and Loading Tool Setups

For each tool that has a full editor, you are able to save and load setups. A setup is a file that contains a record of all changes you make to a clip in a particular tool. This record includes references to clips used—not the clips themselves. Setups let you save your work separately from clips, so you can load and work on the setup at anytime or apply the setup to other clips. Setups can be shared between instances of the same effect, whether accessed from the Tools tab, as a Timeline FX, or from the Batch or Batch FX node bin.

In the tool editor, use the Save and Load buttons to enter the file browser (pointing to the default directory location for each tool type). From the Load browser, you can also delete previously saved setups. For most tools, you can load or save setups or preferences/defaults. Preferences are settings that let you customize the display or functioning of some tool elements, keyboard shortcuts, pen and tablet, and audio (defaults are the default preferences). Some tools, such as Paint, have more saving and loading options.

Accessing Tools

To access specific tools

1. From the Tools tab, find and click the button for the tool you want to use.
2. If needed, use the Input Mode box to select the number of source clips you want to modify (for example, front only or combinations of front, back, and matte).
3. Set any tool parameters, as needed.
4. In the workspace, select the source clip(s) needed.

   The colour and text of the cursor indicates which clip you should select next. After selecting the source clips, the cursor changes to white and the text Render Here appears.
5. Select the area to place your modified clip.

If the result requires no other settings (other than tool parameters), then the rendered result appears immediately. If more settings are required, a full editor appears.

TIP Some tools, such as Paint, allow you to enter the full editor without inputs. In this case, you can double-click the tool name, and directly apply the Render Here location.

To access a tool with the same clips as previously used:

1. From the Tools tab, find and click the button for the tool you want to use.
2. Click the "S" button to the right of the tool name, or click Use Current Setup.
3. Select the area to place your modified clip.

   The result is rendered or the editor opens with the clips from the previous session. The most recent settings are also restored.

Procedural Compositing with Batch and Batch FX

16

About Batch and Batch FX

Batch provides a flow graph environment where you perform procedural compositing with integrated access to almost all Flame effects and image-processing commands. Simply click the Batch tab to enter the Batch view. You can switch tabs, to Timeline or Tools for example, without losing your current Batch work.

A Batch FX is a setup applied directly to one or more segments on the timeline. Creating a Batch FX allows you to take a selection of timeline segments into a Batch flow graph environment for procedural compositing. You can edit and reorder any Batch FX node without affecting anything else in your pipeline.

You use the Batch processing environment to assemble a process tree of clips and nodes, where the result of each operation serves as the source for the next one. Because a Batch pipeline is not fixed, you have the flexibility of reordering and editing nodes.

Because the Media panel is integrated into Batch, it is easy to switch between the desktop, timeline, and Batch—and still keep track of all of your sources. You can add clips to Batch easily using the Media panel, or use the Read File node within Batch to bring in clips from anywhere on your network. You can bring in clips containing stereo tracks as well as clips of any resolution and bit depth. Some nodes require that the clips be of the same resolution and bit depth. You can change these clip settings directly in Batch.

- The default viewport layout for Batch and Batch FX is a 1-Up view displaying the schematic (displaying the process tree). Use the Layout settings under the left viewport to change the layout, or return to this default view. For Batch FX, you can also change the default layout to 2-Up in the Timeline FX / BFX section of the Preferences menu (displaying the schematic on the left, and the result view on the right).
- You can change the actual view of the selected viewport using the View box under the right viewport. If a selected node has its own views, such as Action, you can select node-specific views also.
- The bottom portion of the Batch or Batch FX view displays the nodes that you can add to your process tree. This bottom area can also display the menu for the selected node in the schematic view. To switch between the node view and selected menu view, use the FX Nodes button (located under the left viewport).

Accessing Batch

To access Batch:

1. Click the Batch tab.
2. Add clips and nodes to build your pipeline.

You can switch tabs, to Timeline or Tools for example, without losing your current Batch work.

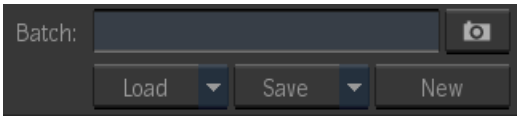
Any media used in Batch is displayed in the Workspace Media panel under Batch Sources. You can drag media from any folder in the Media panel to the Batch Sources folder. This places a copy of the media in the Batch Sources folder and displays it in Batch. Dragging media directly from the Media panel to the Batch schematic also creates a copy of the media in the Batch Sources folder.



TIP Drag clips to the Batch header in the Workspace Media panel (above the Batch Sources folder) to add clips to Batch, and automatically switch to the Batch tab.

To clear a Batch setup:

1. Do one of the following:
   - From anywhere in Flame Premium where you can access the Workspace Media panel, right-click in the Batch area, and select Clear Batch Setup.
   - From within Batch, click the New Batch button.



   All nodes and sources are cleared, and any unsaved changes to the current Batch setup are lost.

Creating Batch FX

To create a Batch FX on a timeline segment:

1. Select one or more video segments on the sequence to which you want to apply a Batch FX.
2. Click the FX button.
3. Click Create Batch FX to quickly enter the Batch FX view. Use this method if you want to enter Batch FX with only one timeline segment without existing Timeline FX, or with Timeline FX that you don't want carried over into Batch FX.

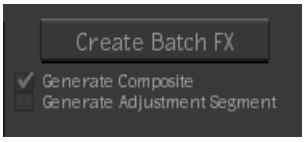


NOTE Once you have returned to the timeline sequence from Batch FX, you can re-enter the same Batch FX by clicking Enter Editor in the FX pipeline, or by double-clicking the BFX icon on the segment, or in the pipeline.

Generating a Composite Batch FX

Use the Generate Composite option when creating a Batch FX if you want to convert any existing Timeline FX to a Batch FX pipeline. When you enter Batch FX, you'll see that your pre-existing Timeline FX are now displayed as Batch FX nodes, allowing you to edit these effects directly in the Batch FX view, or add or delete as many other effects as needed.

1. Select one or more video segments on the sequence to which you want to apply a Batch FX.
2. Click the FX button.
3. Select Generate Composite.



NOTE If you select multiple video elements on the sequence, this option is automatically selected.

4. Click Create Batch FX to enter the Batch FX view.

When you return to the timeline, only the BFX icon appears on your clip, as the previous Timeline FX are now included within the BFX.

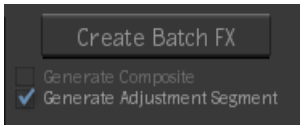
In the case of entering Batch FX with multiple video elements, you can also see that upon exiting Batch FX, only one element exists, comprising the composite of the previous elements. To restore your previous vertical edit on the timeline, you can click Remove BFX and Recover Stack. The previous configuration of the video elements is restored, but any pre-existing Timeline FX are lost.

Generating an Adjustment Segment Batch FX

You can create an edit on the timeline in a gap above multiple clips, without losing any of the effects already on those clips. After creating an adjustment segment, you can add new clips or edit existing clips on the timeline under the adjustment segment gap, and the Batch FX is maintained above. You can perform edits on an adjustment segment as it is treated like any other timeline segment, such as trimming or even adding other Timeline FX.

There are two ways to add an adjustment segment to the timeline:

- Add a Batch FX to a gap in the timeline. In this case, an adjustment segment added by default.



- Manually click Generate Adjustment Segment when adding a Batch FX anywhere on the timeline. In this case, a gap is automatically created on the track above selected segment.

The adjustment segment uses the top media of the tracks below the timeline gap as the input. Inside Batch FX, that input is called the Back Clip. For example, if you have three tracks composited together, and you add an adjustment segment on top of them, the image available through the Back Clip is only the output of the third track.

The Back Clip node is available in the Batch FX I/O bin. The Back Clip node does not contain any media; instead, it offers a link to the Timeline, giving access to the topmost media located below the BFX segment you are currently working on. Since the Back Clip is only a link to media, you cannot access common clip settings within Batch FX. The only settings that are available if you double-click the Back Clip node are to set what media is available for the head and tail of the adjustment segment.



About the Batch and Batch FX Process Tree

Use Batch or Batch FX to assemble a series of tasks using nodes. Each node represents a specific Flame Premium function. You connect clips with nodes and use the result of one node as the source for the next node. As you assemble this arrangement in the Batch or Batch FX schematic, you are building a process tree from left to right, that processes as many output clips as you want.

A process tree begins with a clip, contains at least one node, and ends with a Render, Write File, or BFX Output node.

Once you are working in Batch or Batch FX, it is easy to add more clips and nodes to achieve your desired result. Large schematic pipelines are common, so you can group nodes or add notes to nodes to help you manage clutter.

About Nodes

Nodes have one or more colour-coded input and output points (also called tabs) used for connecting to clips and to other nodes. All nodes have an output tab. For example, a Colour Correct node can accept a Front, Matte, and Back connection, whereas an Auto Matte node accepts a Front input.

As you add nodes, you connect them to the process tree by linking the result from one node and using it as a source (front, matte, or back) for the next node in the process tree. You can also connect by linking backward from the source of one node to the output of another (to reuse a node's output).

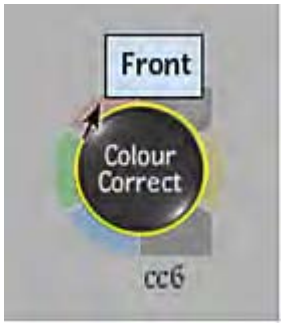
To connect nodes, you use the coloured tabs on the node's left side; these tabs are called *source tabs*. The colours of the source tabs represent different input types. The yellow tab on the node's right side is called the Result tab (for some nodes, a blue tab is also present on the right side, this is the OutMatte tab). You use the Result tab of a node to connect its result to the input tab of another node.

This is a typical node tab configuration. Some nodes have specific tabs related to their function (such as the forward vector tab on the Pixel Spread node).



(a) Matte tab (b) Back tab (c) Front tab (d) Warning tab (e) Result tab

TIP If Auto Display of tooltips is enabled in the Preferences menu (**Preferences ➤ User Interface ➤ Tooltips**), hover over a tab to see the name of the tab.



Colour	Tab	Description
Red	Front	Connects a front clip to a node.
Green	Back	Connects a back clip to a node.
Blue	Matte	Connects a matte clip to a node.
Light Blue	Misc	Miscellaneous tab specific to certain nodes, such as a Z-Depth or Forward Vector input.
Turquoise	Audio	Connects a clip with audio to a Render node.
Yellow	Result	Connects the result of a node to other nodes.
Blue	Output Matte	Connects the output matte of a node to other nodes.
Red circle	Warning	Warns that clip input to this node is unconnected or is missing media, or clips parented to this node do not share the same resolution or a compatible bit-depth.

The available source tabs depend on the node. If the node accepts a front, back, and matte clip, all coloured source tabs are available. If the node only accepts a front clip, the red source tab is available and the other source tabs are grey. When a source tab or Result tab is not connected to a clip or to another node, the coloured tabs are dimmed.

TIP Right-click any node in the schematic to reveal a contextual menu of operations available for that type of node.

Adding Clips to the Process Tree

The quickest way to add clips to your Batch process tree is through the Workspace Media panel. Even though Batch FX process trees are pre-populated with clips from the timeline, you can also add more clips from the Workspace Media panel.

Another way to add clips is through the Read File node (page 351).

To add clips from the Workspace Media panel:

1. Select one or more clips from the Workspace Media panel.
2. Drag the clips into the Batch or Batch FX schematic.

   The clips are now available to use in the process tree. Notice that the clips are also displayed in the Media panel's Batch or BFX Sources folder.

   TIP You can also drag clips directly into the Batch or BFX Sources folder (you don't even have to be in Batch or Batch FX to do this). Any clips in the Sources folder can be seen in the schematic (with the exception of Read File clip nodes). Drag clips to the Batch header in the Workspace Media panel (above the Batch Sources folder) to add clips to Batch, and automatically switch to the Batch tab.

Once a clip is present in the Batch or Batch FX schematic, you can connect its output tab to other nodes in the same manner as connecting nodes to nodes. The same clip can be connected to multiple nodes in the process tree.

To replace a clip:

1. Drag a clip from the Workspace Media panel into the Batch or Batch FX schematic.
2. Release the clip on top of an existing schematic clip, once you see a replace icon.

   The clip is replaced.

   TIP If the clip proxy is black after replacing, right-click the clip and select Reset to reset the timing offset of the clip.

Clip Information Reference

You can display detailed information about clips in the schematic such as resolution, frame rate, and size. As well, visual cues such as symbols and letters identify clips as having had a specific type of operation applied.

The colour of the clip information and letters and symbols also provides specific information.

Clip Information

You can choose what clip information is displayed under clips in the schematic by selecting an option from the Clip Info box in the Batch or Batch FX Schematic Preferences:

Select:	To display:
Resolution	The clip's name, number of frames, dimension, frame depth, and aspect ratio.
Size	The clip's name, number of frames, and dimension (in pixels).
Framerate	The clip's name, number of frames, and number of frames per second (fps).
Size+Rate	The clip's name, number of frames, dimension (in pixels), and number of frames per second (fps).
Resolution+Rate	The clip's name, number of frames, dimension, frame depth, aspect ratio, and number of frames per second (fps). This option provides the most detailed level of clip information.

Select:	To display:
No Info	No information except the clip name and number of frames.

Clip Name Colouring

The colour of the clip name in the schematic helps you locate where the clip originated.

Colour:	Origin:
White	Media Panel
Blue	Batch Snapshots
Pink	Read File node
Red	No media found (clip is also displayed as a black proxy)
Green	BFX Back Clip

Clip Status Symbols

The following visual cues can help you track the type of clip and clip status.

(A) symbol and Audio Context outline Indicates that an audio context is set for this clip or node. In addition to the A, the clip has a dotted purple outline. The letter A is followed by the number of the context, for example, A2 is the second Audio context.

(C) symbol and Context outline Indicates that a context is set for this clip or node. In addition to the C, the clip has a dotted green outline. The letter C is followed by the number of the context, for example, C1 is the first video context.

LUT symbol Indicates that a LUT was applied to the clip. The first and second number represent the source and destination bit depths, for example [8 -> 16f].

[R] symbol Indicates that a Resize was applied to the clip.

(BFX) symbol Indicates that a Batch FX was applied to a timeline made up of one segment, or the clip was converted as a Create BFX. The clip can be expanded.

(bfx) symbol Indicates that a Batch FX was applied to a timeline made up of multiple segments. You cannot expand setups of clips containing more than one segment.

Importing With the Read File Node

To import images into Batch or Batch FX using the Read File node:

1. In the I/O node bin, select the Read File node and drag it to the schematic.
   The Read Files browser appears.
2. Optional: Set General and Format Specific Options for the files to import.
3. Optional: For each file, set In and Out points using the Preview panel.
4. Select the clips to import.

5 Click Load.

You are returned to Batch or Batch FX and the file appears in the schematic. If you import more than one file, a Read File clip is created for each file. The clip name appears in pink to help you differentiate from other schematic clips.

NOTE Whenever you open a Batch or Batch FX setup which uses Read File clips that have been updated, Flame Premium asks for your permission to update the setup with updated versions of these clips (when updates exist). Updating the setup replaces the current Read File clips with the latest found versions. This ensures that you are using the correct versions, and in any case, Flame Premium maintains the appropriate schematic connections.

To Expand a Read File Clip

1 Select the Read File clip.

2 Press `Shift+C`.

To Replace the Current Read File Clip With Another One

1 Double-click the proxy of the Read File clip.

The Read Files browser appears.

2 Optional: Set General and Format Specific Options for the files to import.

3 Optional: For each file, set In and Out points using the Preview panel.

4 Select the clip to import.

5 Click Load.

You are returned to Batch or Batch FX and the Read File clip is updated with the selected file. The Schematic is also updated, using a channel matching strategy that preserves the links between the clip's channels and the linked nodes.

To Load a Clip as a Version

1 Double-click the proxy of the Read File clip.

The Read Files browser appears.

2 Optional: Set General and Format Specific Options for the files to import.

3 Optional: For each file, set In and Out points using the Preview panel.

4 Select the clip to import.

5 From the Load drop down menu, select Load as Version.

You are returned to Batch or Batch FX and the Read File clip is updated with the selected file. The previous clip is now available as a version from the Clip Versions box. You can have as many versions as you need.

To Switch Between Versions of a Read File Clip

NOTE Only possible with Read File clips loading clips with versions, or after loading a clip as a version.

---

1 Double-click the Read File clip.

2 Select a Clip Version using the Clip Version box.

The Read File clip now uses the selected version and the schematic is updated accordingly, using a channel matching strategy that preserves the links between the clip's channels and the linked nodes.

To Check for New Versions of Clips

NOTE The Check for Updated Versions button is only available if at least one Read File clip in the schematic has been updated.

1. Open the Batch or Batch FX Preferences menu.
2. Under Read File Nodes in the Preferences section, click Check for Updated Versions.

   Every Read File clip in the schematic is checked for updated versions of their clip.

To Edit a Batch Setup Included with a Read File Clip

NOTE Only available to Read File clips with clips that were created using the Include Setup option.

1. Double-click the Read File clip.
2. Click Append to current Batch.

   The Batch setup that created the open clip for the displayed Clip Version is added to the schematic as a Group Node.

Modifying the appended setup does not modify the open clip imported with the Read File clip. But since it includes the original Write File node, you can export a new open clip with the exact same export settings. This means that you can create a new version of the open clip, and then use the Check For Updated Versions button to update the current open clip. A high level workflow could be as follows.

1. A Flare user processes a Batch setup to export a clip with the following options enabled.
   - Create Clip Data
   - Include Setup
   - Version (set to version 1)

   It creates open clip with a Batch setup, and versioning enabled.
2. A Flame user imports the open clip and starts using it in his own Batch setup.

   Looking at the Read File clip, he can see that there is only one version of the clip (only one version in Clip Versions box) and that the Batch setup that created this open clip is included (Source Setup field has the name of the setup).
3. In the meantime, the Flare user touches up his Batch setup. Before processing it, he also updates the Write File node's Exports Settings (from JPEG to DPX), but makes sure to not change any of the Clip Data, Path, and Naming fields (the Flame updates the version automatically). Without further ado, he processes it.

   The Write File clip updates the open clip file with new information: a new version which points to updated media files.
4. The next time the Flame user opens his Batch setup, he is prompted to check for updated open clips. He does check, and the clip he imported in a previous session is updated and set to the current version.

   Or, during his Batch session, the Flame user can use the Check for Updated Versions.

About the Read File Node

Why Use a Read File Node?

While you can import many types of media files using a Read File node, its primary use is to facilitate integration of Flame Premium in render pass workflows.

Using the Read File Node With Render Passes

The Read File node is more than just another way to import into the Batch or Batch FX schematic. It actually allows you to integrate Flame Premium inside a render pass pipeline very efficiently.

While you can use the Read File node to import almost any file format, three are of interest for multi-channel render pass workflows:

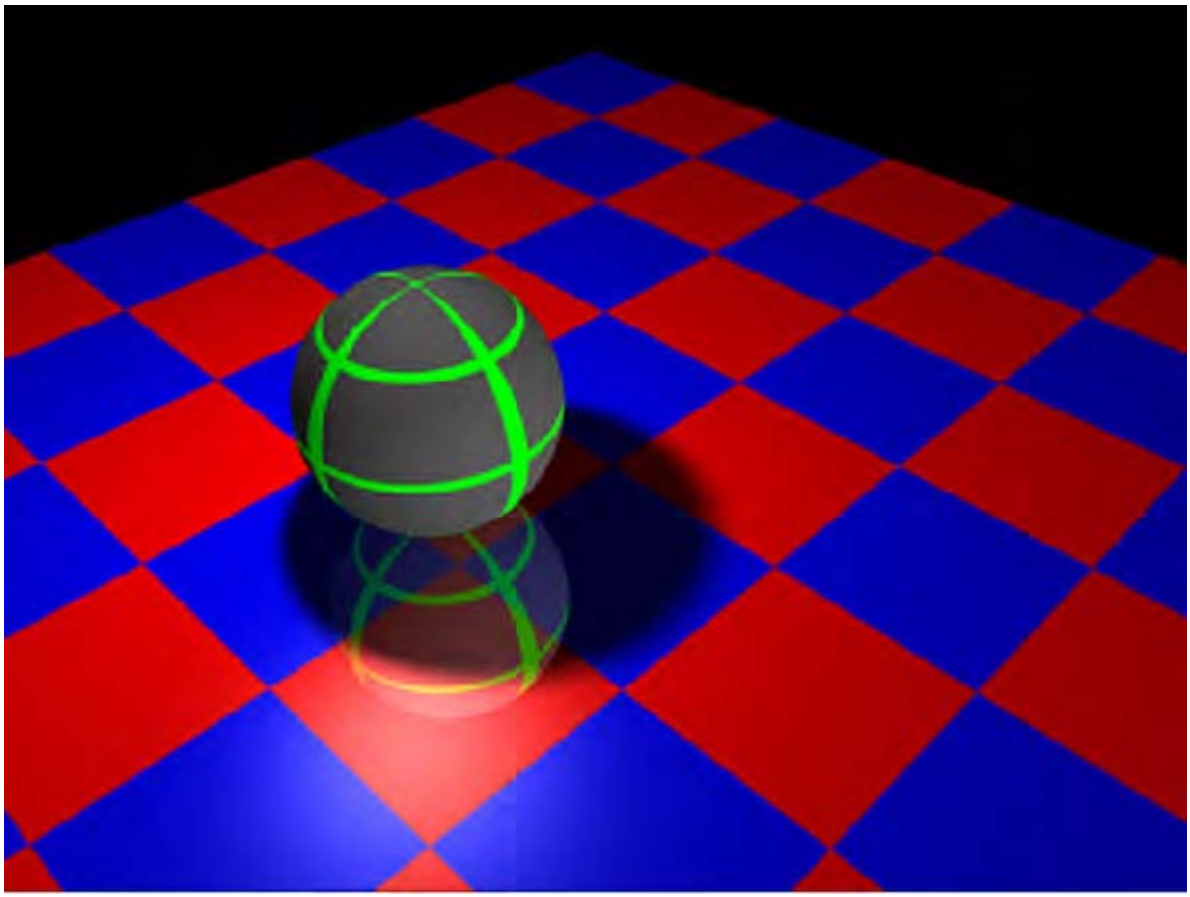
- open .clip
- multi-channel OpenEXR
- Maya .precomp

And while they differ in their basic structures, they all provide access to a powerful CG concept: Read File Nodes and Autodesk Maya. The render pass allows you to composite within Flame Premium the many components, or passes, making up the scene. By working directly with the render passes, you end up performing tasks quicker, such as colour correcting, that are much more time-consuming in the 3D application. The fact is that Flame Premium only needs to compute the value of pixels in a scene, while the 3D application needs to recompute the lighting of whole geometries: Flame Premium is just better for these tasks.

How to Use Flame Premium With Render Passes?

Working with render passes is not difficult, especially if you use the Read File node and one of the following file types: Open .clip, multi-channel OpenEXR, or Maya .precomp.

Consider the following example of an image generated in Maya.

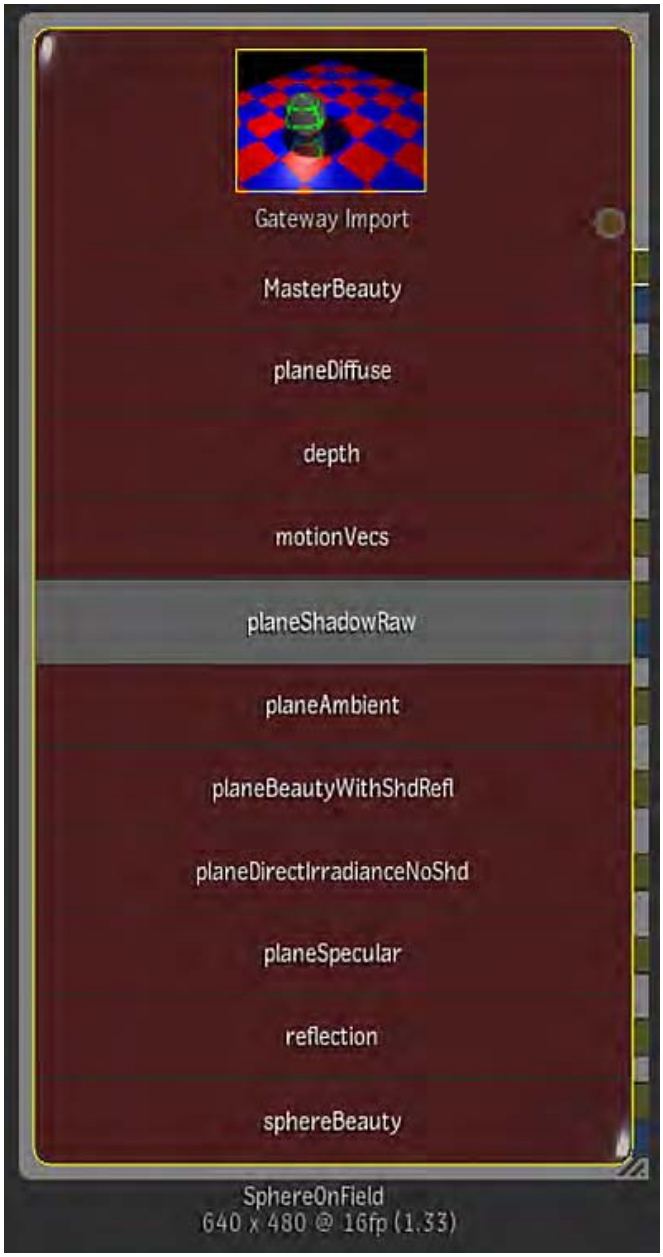


This image was exported from Maya using the following render passes.

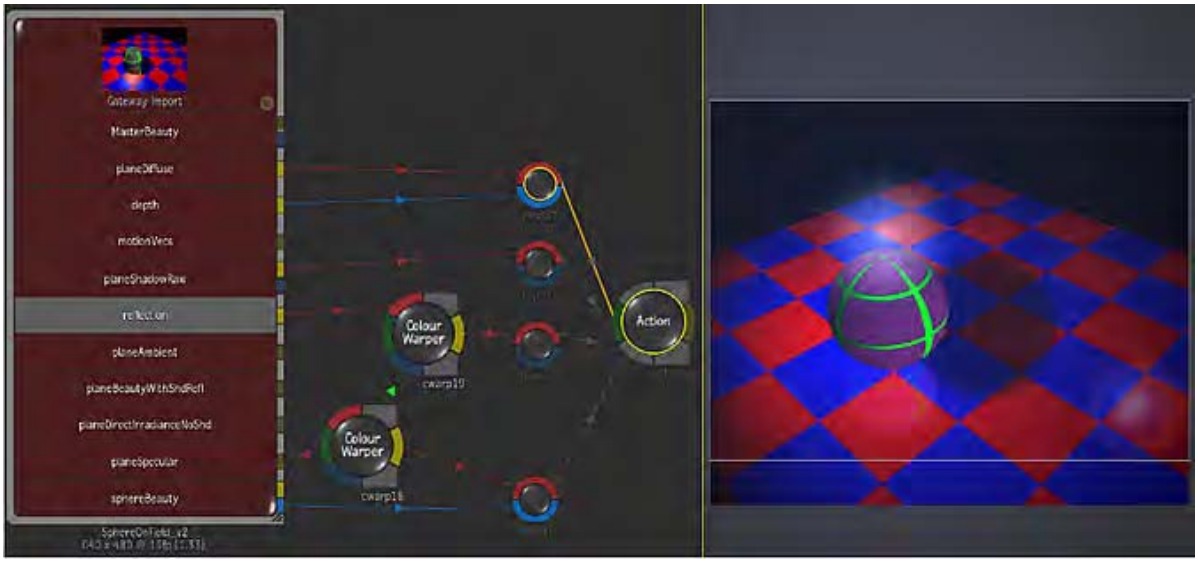
- Beauty - scene

- Camera Depth
- Motion Vectors
- Ambient
- Beauty with Shadow and Reflections
- Diffuse
- Direct Irradiance Without Shadows
- Raw Shadow
- Specular
- Reflection
- Beauty - sphere-only

Importing this file using the Read File node, you have access to all these passes.



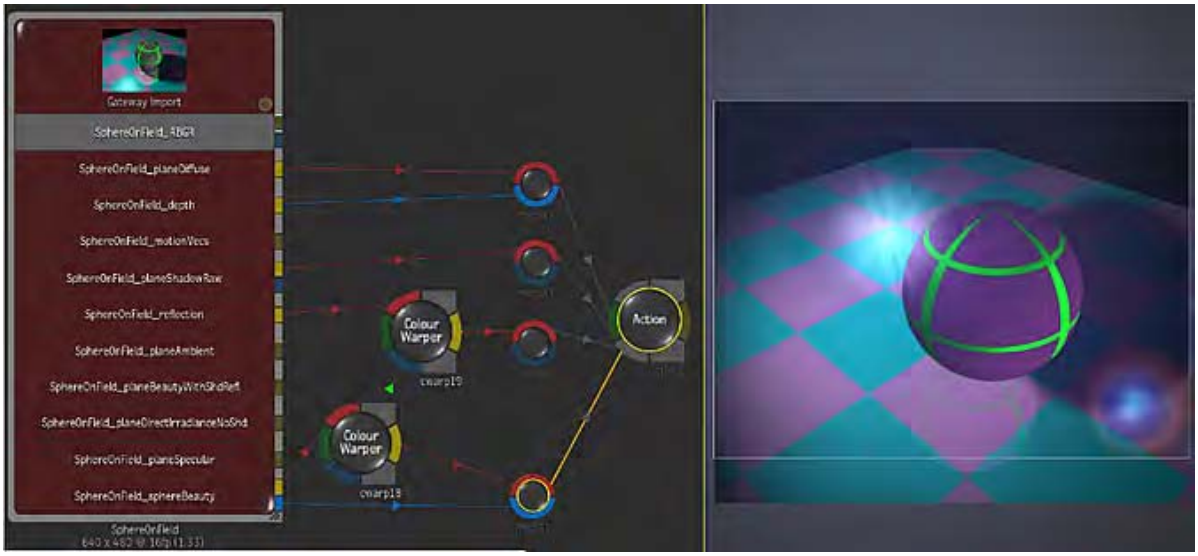
Once imported, you can create a simple Batch or Batch FX schematic and composition.



In this Batch or Batch FX setup, you can do the following, for example:

- Warp the sphere's colour.
- Warp the sphere reflexion's colour by the same amount.
- Remove the original shadow and replace it with a new one.
- Add a spot on the sphere.
- Change the overall lighting look.
- Add a lens flare.

If the CG artist updates their work, you can just replace the current clip with the updated version, performing a simple replace operation (page 352); all the compositing information is maintained.



In fact, the Read File node supports matching channels based not only on their UID and name, but also on their channel type. This allows you to replace sources with multiple render passes in a clip that are similar, but not identical, and maintain schematic connections to the Read File node within a composite.

About .precomp Files

A .precomp file is a file generated by Maya to facilitate data exchange with its Composite component. Flame Premium can read that file and extract render pass information. An actual .precomp export from Maya consists of two distinct elements:

- One .precomp file which describes the components of the scene. It is a Python plain-text file with compositing information and path to the render passes.
- Media files referred to by the .precomp file. In the simplest of cases, there is one OpenEXR file for every rendered frame. Each OpenEXR contains all the render passes for that frame.

An interesting aspect of .precomp files is their use of render layers as versions: this allows the Maya artist to create different versions of the same scene using render layers. You can then use the Read File node to import the .precomp file, and then use the Versions box to switch between looks.

Read File Node and Autodesk Maya

The Read File node automatically recognizes pass types from mental ray in Maya, allowing for more robust channel matching during clip updates or replacement.

Below is a list of supported mental ray render passes, with a short description.

Render Pass	Description
2D Motion Vector	Relative motion (in raster coordinates) of objects in your scene; in other words, how far each pixel is moving between two frames. Vector is expressed in normalized pixels.
3D Motion Vector	3D motion vector in world space. In mental ray for Maya, the 3D motion vector is expressed in internal space.
Ambient	Ambient contribution of the surface. In Maya, this is the material color multiplied by the ambient light color.
Ambient Irradiance	Amount of ambient light received by the surface.
Ambient Occlusion	Ambient occlusion contribution from both self ambient occlusion as well as primary ambient occlusion, which is derived from surrounding objects.
Ambient Material Color	Reflectivity of the material with respect to ambient light.
Beauty	Final colour computed by mental ray for Maya.
Beauty Without Reflections or Refractions	Beauty pass without reflections or refractions. Used to tune the tint and intensity of the reflection/refraction separately from the rest of the passes.
Camera Depth	Extracts the distance between the camera and the intersection point.
Coverage	mental ray Coverage frame buffer. This frame buffer offers only silhouette coverage. Self-coverage is currently not supported.
Diffuse	Diffuse shading of material.

Render Pass	Description
Diffuse Without Shadows	Diffuse pass without shadowing information.
Diffuse Material Colour	Provides constant diffuse colour or textured diffuse colour, excluding light contribution.
Direct Irradiance	Direct light arriving at each sample location.
Direct Irradiance Without Shadows	Direct irradiance without shadowing information.
Glow Source	outGlow output of surface shaders; affected by pass contribution maps.
Incandescence	Additive colour.
Incidence	Measures the difference between the direction of the light ray and the surface normal. If the surface normal is facing the light, this value is 1. If the normal is facing away from the light, the value is 0.
Indirect	Indirect lighting from final gather, global illumination, and caustics.
Light Volume	Extracts all light-centric volume effects, for example, a light cone volume effect.
Material Incidence	Measures the difference between the direction of the camera ray and the surface normal. If the surface normal is pointing to the camera, this value is 0. If the normal is facing away from the camera, the value is 1. Any angle greater than 90 degrees is also translated to 1. If bump mapping is applied to the shading network, it will appear in this pass.
Material Normal	Interpolated surface normal. If bump mapping is applied to the shading network, it will appear in this pass.
Matte	The object's matte, excluding transparency/opacity. This pass serves as the render layer compositing mask. Should be solid white in areas where objects are intersected. Independent of transparency/translucency.
Normalized 2D Motion Vector	Relative motion (in raster coordinates) of objects in your scene; in other words, how far each pixel is moving between two frames. Pixel displacement is normalized to (0—1). Static objects are expressed with 0.5,0.5 values.
Object Incidence	Similar to the Material Incidence pass but without support for bump mapping.
Object Normal	Similar to the Material Normal pass but without support for bump mapping.

Render Pass	Description
Object Volume	Extracts all object-centric volume effects, for example, smoke that is contained in a glass object. Also includes volume particles, volume fur, and fluids.
Opacity	The object's opacity, which is derived from transparency/refraction. In compositing, the object's opacity can be controlled independently from the render layer matte.
Raw Shadow	Similar to the Shadow pass but calculated only with respect to the irradiance in the scene.
Reflection	Reflection results. Includes self-reflection, primary reflections, secondary reflections and environment reflections.
Reflected Material Colour	The reflected colour parameter of the material. Pure constant reflection colour or textured reflection. Used as a reflection matte to determine where reflection would be revealed (coloured and non-coloured).
Refraction	Refraction results. Includes self-refraction, primary refraction, and environment refraction.
Refraction Material Colour	The transparency colour parameter of the material. Pure constant refraction colour or textured reflection. Used as a refraction/transparency matte to determine where refraction is revealed (coloured and non-coloured).
Scatter	Scattering effects that result from the material's scattering attributes.
Scene Volume	Extracts all scene-centric volume effects such as fog, layered fog, haze, and so forth.
Shadow	Pure shadow contribution for both self-shadowing as well as direct shadows. The shadow pass can be luminance or coloured shadows. Takes into account material contributions.
Specular	Specular shading. The specular component is modulated differently depending on the type of material associated with the object. For example, Phong, PhongE, Blinn, and Anisotropic materials produce specular contributions differently. On a Phong material, the specular pass can be modulated using cosine power and specular colour.
Specular Without Shadows	Similar to Specular but without shadow occlusions.
Translucence	Back shading contribution revealed on the front surface.
Translucence Without Shadows	Similar to Translucence but without shadows.

Read File Clip Settings

To access Read File clip settings, double-click a Read File node in the schematic.

Basic Settings

NOTE The Import Settings are dynamically generated according to the type of clip imported. You can get a quick description of each setting by viewing its tooltip.

Basic button Opens the Basic menu where you set the clip version, append the available setup, modify the import settings, and set the missing media options.

Versions box Select the active version of the clip. If the available channels differ between versions, the application updates the links to other nodes.

Creation Date field Displays the creation date of the version selected in the Clip Version box. Non-editable.

Source Setup Name field Displays the name of the Batch setup saved with the open clip. Only available to open clip output using a Write file node with Include Setup enabled.

Append to current Batch button Click to add to the current Batch schematic a group made of the Batch setup stored with the open clip. Only available to open clip output using a Write file node with Include Setup enabled.

Head Media box Select to substitute missing media at the beginning of a clip with black frames, the first frame of media, or leave as is.

Tail Media box Select to substitute missing media at the end of a clip with black frames, the first frame of media, or leave as is.

In field Set, as an offset of frames from frame 0, the In mark of a clip. Only frames located after the mark are available and rendered.

Out field Set, in frames from frame 0, the Out mark of a clip. Only frames located before the mark are available and rendered.

Frame field Displays the absolute frame number of the current frame, from the file with the name displayed in the File Name field. Non-editable.

File Name field Displays the name of the file from which the current frame is being read. Non-editable.

Clip Information tabs

Available in the Basic and Extended menus of the Read File clip.

The Clip Information tabs display non-editable information about imported clip.

Clip Info tab The Clip Info tab displays the name of the original media, file format, resolution and FCM, as well as the number of tracks in the clip.

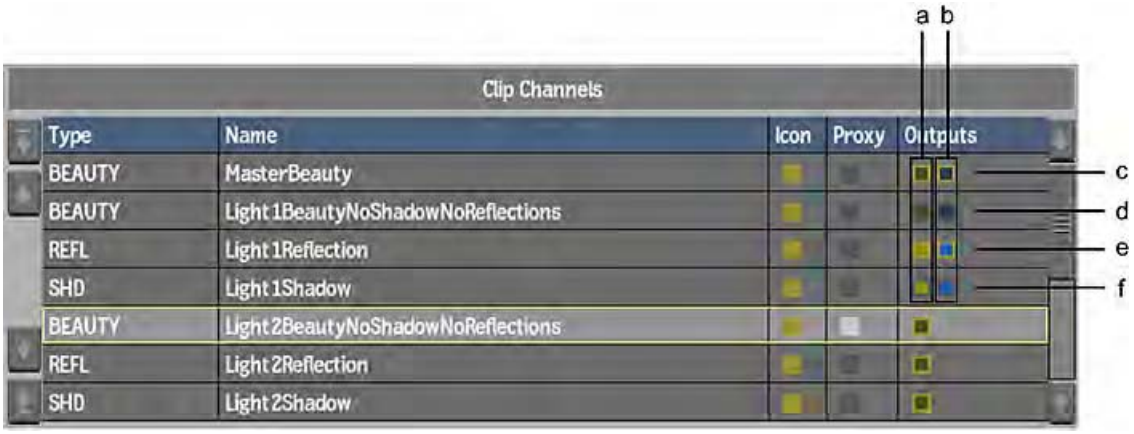
Metadata tab The Metadata tab displays metadata information about the imported media, including but not limited to: path to the sources, camera settings (RED and ARRI), file creator.

Versions tab The Versions tab lists all the versions available for the selected clip. Available to open clips and Maya .precomp imports.

Extended Settings

Extended button Opens the Extended menu where you set the channel options, from the channel displayed in the node proxy to the available outputs.

NOTE With a Read File clip selected in the schematic, press `Shift+C` to expand the node and display its channels in the schematic.



(a)Media output (b)Alpha output (c) Displayed and disconnected outputs (d)Hidden and disconnected outputs (e) Displayed and connected outputs (f) Hidden but connected outputs

Type field Displays the type of channel. Non-editable.

Name field Displays the name of the channel. Non-editable.

Icon field Enable to display the channel in the schematic; disable to hide the channel. The Read File clip must be expanded (press `Shift+C`) to view the channels in the schematic. Hiding a channel does not break its link to a node.

Proxy field Enable to use a particular channel as the Read File clip proxy in the schematic.

Outputs fields Click to display or hide the outputs available for each channel from the Read File clip in the schematic. Hiding an output does not break its link to a node. There is one output each for RGB and its alpha (if any).

Resize Settings

The Resize settings are the same as when you are using the Resize node in Batch or Batch FX. Click the Active button to activate the Resize settings.

RGB LUT Settings

The RGB LUT settings are the same as when you are using the LUT Editor node in Batch or Batch FX. Click the Active button to activate the RGB LUT settings.

Clip Settings

In Batch, you can also double-click a clip in the schematic to switch to the Timeline with the selected clip available as a source clip. You can apply any timeline editing operation on your clip, and when you return to Batch, your clip is updated.

The following settings are available when selecting a clip in the schematic. You can also right-click a clip node in the schematic to reveal a contextual menu of operations, such as Render options.

Basic Settings

Basic button Opens the Basic menu where you set timeline and timewarp options, control clip locking and slipping, and define how missing media is displayed and rendered.

Head Media box Select to substitute missing media at the beginning of a clip with black frames, the first frame of media, or leave as is.

Gap Media box Select to substitute missing media in gaps with black frames or leave as is.

Tail Media box Select to substitute missing media at the end of a clip with black frames, the last frame of media, or leave as is.

Offset field Displays the number of frames by which selected clips and/or segments are offset. This same setting can be found in the Timing View as well as the Clip Settings. Editable.

Lock Frame button Enable to display the current frame for the duration of the clip.

Explode All Timeline SFX button Click to explode all Timeline FX in the clip into a process tree.

NOTE If a clip proxy is black after exploding, right-click the clip and select Reset to reset the timing offset of the clip.

Explode History button Click to access pre-processing settings for the clip.

Explode One button Click to explode one level of the clip's BFX setup.

Explode All button Click to explode all levels of the clip's BFX setup.

Alpha Active button Enable to generate an alpha or matte from a clip.

Alpha Colour box Select whether to output a white or black alpha of your clip.

Resize Settings

Most of the Resize menu is the same as when using the Resize node in Batch or Batch FX. The following settings are specific to the Resize tab for a clip:

Resize button Opens the Resize menu where you change the size of a clip, as well as its aspect ratio.

Active button Enable to activate the Resize settings.

Load button Loads a Resize setup from the library.

Save button Saves the current resize setup.

RGB LUT Settings

Most of the RGB LUT menu is the same as when using the LUT Editor node in Batch or Batch FX. The following settings are specific to the RGB LUT tab for a clip:

RGB LUT button Opens the LUT Editor for the selected clip.

Active button Enable to activate the LUT settings.

Back Clip Settings

These settings are available for a Batch FX Back Clip created for a timeline adjustment segment.

Head Media box Select what media is available before the In Point of the adjustment segment. Select Timeline Level to have the Back Clip read the head media from the segments preceding (and below) the adjustment segment.

Tail Media box Select what media is available after the Out Point of the adjustment segment. Select Timeline Level to have the Back Clip read the tail media from the segments following (and below) the adjustment segment.

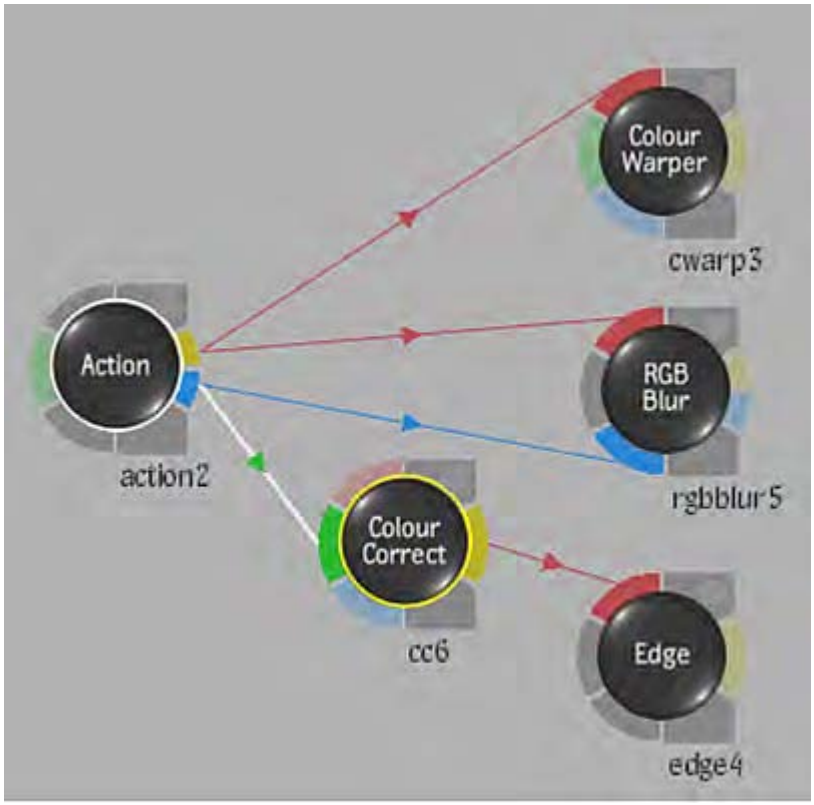
Adding and Connecting Nodes to the Process Tree

To add a node from the node bin to the schematic, double-click a node, or drag a node directly to the schematic. There are several ways of connecting clips or nodes together. You can connect them manually or automatically.

To insert a node between connected nodes:

1. Do one of the following:
   - If Auto Insert is enabled in the Prefs menu Schematic settings, drag a node and navigate to the link between two connected nodes. The link is highlighted in orange.
   - If Auto Insert is disabled in the Prefs menu Schematic settings (this is the default), press and hold `Shift`, then drag a node and navigate to the link between two connected nodes. The link is highlighted in orange.
2. Release the node.

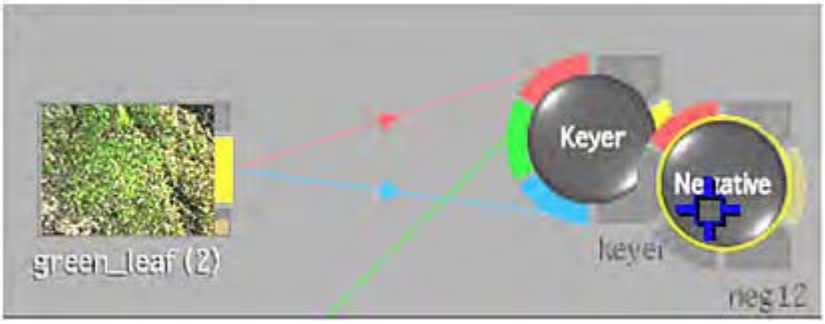
The node is inserted, while retaining the input and output connections.



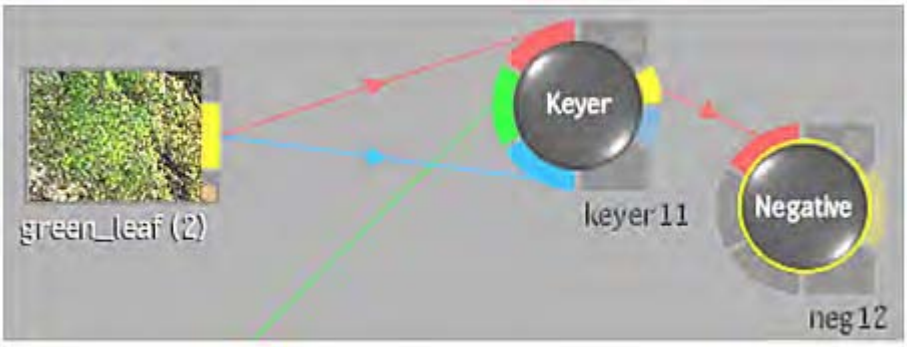
To connect nodes automatically with autolink:

1. Use the Autolink feature by pressing `Shift` and dragging a node to another node so their tabs touch.

   Action: Press and hold Shift while dragging the Negative node to the Keyer node



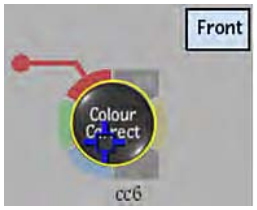
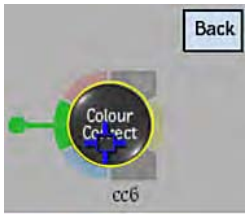
Result: The Keyer output is the front for the Negative node



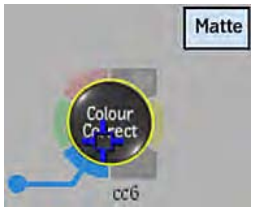
To connect nodes using the advanced autolink:

1 To distinguish between the various source tabs, use the Advanced Autolink feature by pressing `Alt` repeatedly while still holding `Shift` to extend a similarly coloured arm from each source tab (starting with the topmost source tab, and cycling counter-clockwise with each press of `Alt`). The name of the tab is also displayed above the node. You can then touch the extended arm to the tab you want to connect to.

Approaching a node with multiple outputs, such as a multi-channel clip imported with a Read File node, automatically expands that node.

Hold `Shift` and press `Alt` to extend the Front tab.	
While still holding `Shift`, press `Alt` again to extend the Back tab.	

While still holding `Shift`, press `Alt` a third time to extend the Matte tab.



You can also retain connections using this method by releasing the node over an existing connection (when it turns orange).

To create a node connection using tap-tap linking:

1. Click the tab of the first node or clip you want to connect.
2. Click the tab of the node you want to connect the first node to.

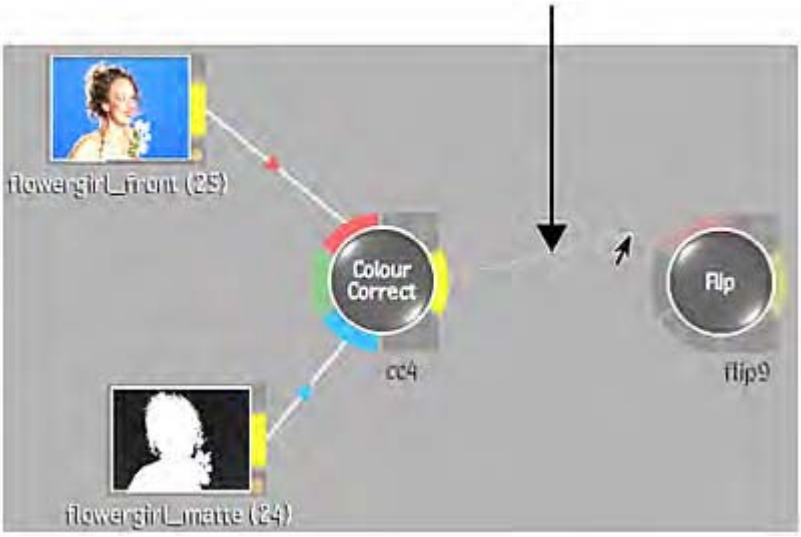
   The nodes are connected.

   NOTE If the first node you click is an Output node, you can add as many subsequent input tab connections as you like. To finish selecting, click anywhere in the schematic work area.

To connect nodes manually:

1. Click the Result tab of a node whose result you want to use and drag the cursor to one of the source tabs of the next node in the process tree.

   An arrowed line is drawn from the node to the source tab. For example, click the Result tab of the Colour Correct node and drag the cursor to the front tab of the Flip node to flip the result of the colour-corrected clip.



To extract a linked node while maintaining connections:

1. Press `Alt+Win` and drag a linked node away from the link.

   The node is disconnected and connections between existing nodes are reformed.

To disconnect nodes or clips:

1. Drag the cursor across the connecting line between a clip and a node or between two nodes.

   The arrowed line is cut and the source tab is dimmed.
2. You can cut multiple connections in a single stroke by clicking and dragging over several connecting lines in the schematic.

To delete a node:

1. Do one of the following:
   - Right-click a node and choose Delete.
   - Drag a node towards the bottom of the screen, and release once you see the garbage can icon.

     TIP You can combine the extract `Alt+Win` and dragging a node to delete it and maintain connections.

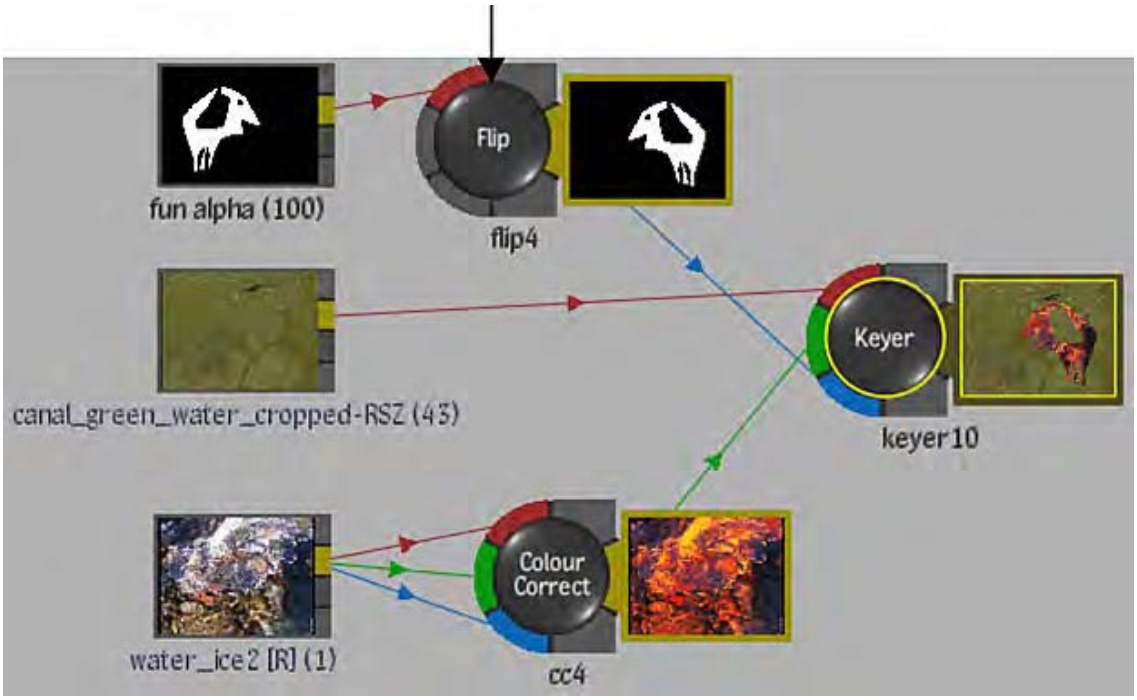
Bypassing Node Inputs

You can improve your workflow in Batch or Batch FX by deactivating certain nodes and rendering only the nodes that you want to process right away.

For any node, you can pass the Front, Back, Matte, or Key-in clip—depending on the type of node you select and its source tab inputs—as the input to the next node in a branch. If you bypass a node with multiple outputs (such as Action and Modular Keyer), the matte output will be the same as the result output (depending on the current selection in the Bypass box).

To bypass a node input:

1. In the schematic, select the node whose input you want to bypass.



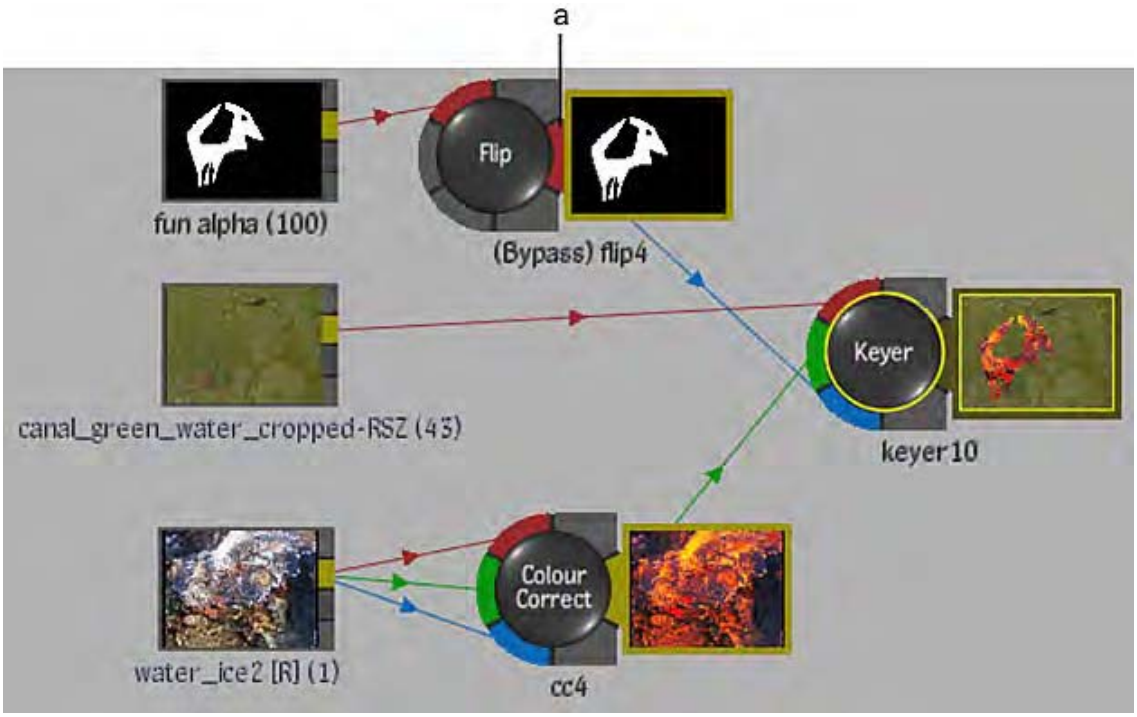
2. Enable Bypass.

The Bypass box becomes active.

3 Select the clip that you want to pass as the input to the next node from the Bypass box.

NOTE The Flip node, which was selected in step 1 of this example, has input tabs for a front source. Therefore Front is the only option in the Bypass box.

The output tab of the bypass node changes colour. In the following example, the tab turns red indicating that the Front input is passed to the next node in the tree. As well "(Bypass)" is added to the node name.



(a) Output tab of bypassed Front input

To unbypass nodes:

1 Do any of the following:

- Select the node with the bypassed input and disable the Bypass button.
- If the bypassed node is part of a multi-selection in the schematic, click Disable Bypass On Selection.

NOTE Click Restore Bypass On Selection to redo the bypass.

Adding Notes to the Schematic

You can add notes to the Batch or Batch FX schematic, or to a specific clip, node, or group. Notes are useful when collaborating on an effect or project with other users. Notes are saved with the setup so they are visible to all. You can copy and paste content between notes.

To create a freeform schematic note:

1 Drag the note icon from the All Nodes bin to the schematic.



2 Double-click the note icon.

The Note text editor appears.

3 Click the editor window to activate it, and type your note text.

4 Click the upper-left corner of the text editor to close the editor.

To create a note on a clip, node, or group:

1 In the schematic, select the clip, node, or group to which you want to add the note.

2 Press `Shift+V`.

The Note text editor appears.

3 Type text into the Note field.

4 Click the upper-left corner of the text editor to close the editor. The note icon appears to the right of the clip, node, or group.

This note is attached to the clip, node, or group.

Viewing Schematic Notes

You can modify how notes are displayed in the schematic.

To display an existing note:

1 Use one of the following commands with the cursor over the note (for freeform notes), or over a clip, node, or group that has a note.

Press:	To:
V	Temporarily display a note.
Ctrl+Shift+V	Expand a note for display.
Shift+V	Expand a note for editing.

Selecting Nodes in the Schematic

Use the following keyboard shortcuts to help you select and work with nodes in the schematic.

To:	Do this:
Select a node.	Click node.
Select multiple nodes.	Press Control, then click and drag a rectangular selection around the nodes.

To:	Do this:
Deselect everything.	Click on an empty space in the schematic, or press Control and click-drag on an empty space in the schematic.
Add or remove nodes from a selection.	Press Control and click the nodes you want to add or remove.
Select all the nodes before the current node (ascendants).	Press Alt and click the node.
Select all the nodes after the current node (descendants).	Press Alt+Spacebar and click the node.
Remove a branch from a selection.	Press Ctrl+Alt and click the branch.
Add or remove the branch after a node (descendants) to the selection.	Press Ctrl+Alt+spacebar and click the node.
Select the entire tree.	Press Alt+Win+Spacebar and click any node in the tree.
Add or remove a tree to or from the selection.	Press Ctrl+Alt+Win+Spacebar and click a node in the tree.
Select the entire setup.	Press Control+A.
Move only one node in a selection.	Press Shift and move the node in the selection.

Adding Media to an Action Node

To add indirect media to an Action node:

1. In the Batch or Batch FX schematic, double-click the Action node and then click Media in the Action menu.
2. From the Media List box, select New Input.

   A Media node is added to the Action node.

   Notice that the Media list is empty. Although you have added a Media node to the Action node, you have not yet added any media.
3. Connect front and matte clips (image clips, or the output of another node in the Batch or Batch FX process tree) to the red and blue input tabs of the Media node, respectively.

   The Media list is updated with the indirect media. Brackets (“[ ]”) around the clip names indicate the media is indirect.

   When you select a Media node in the schematic, the corresponding media in the Media list is highlighted. Conversely, selecting media in the Media list highlights the corresponding Media node and the link to the Action node.

To add direct media to Action using Clip Select:

1. In the Batch or Batch FX schematic, double-click the Action node and then click Media in the Action menu.
2. From the Media List box, select New Media.

3 From the workspace, select a front and matte clip to load as media.

NOTE You can select any number of front and matte clips by holding the Control key while selecting clips. Each front/matte selection is added to its own line in the Media list, and image nodes are automatically added to the schematic if Auto Image is selected in the Node Prefs menu.

To add direct media to Action gesturally:

1 From the Workspace Media panel, drag a clip to the Action schematic or Media list.

An image node and axis are added to the Action schematic (if Auto Image is selected in the Node Prefs menu), and the Media list is updated with the front media.

2 To replace the media in an image node, drag a new clip from the workspace Media panel onto an existing image node in the Action schematic. Release the clip when you see the red replace arrow.

The image is replaced in the schematic, and the Media list is updated with a new entry for the media. The replaced media remains as a Media list entry in case you are using it elsewhere in your Action scene.

To convert direct media to indirect media:

1 Select the media that you want to convert from the Media list.

2 Click Extract.

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

About Action in a Batch or Batch FX Workflow

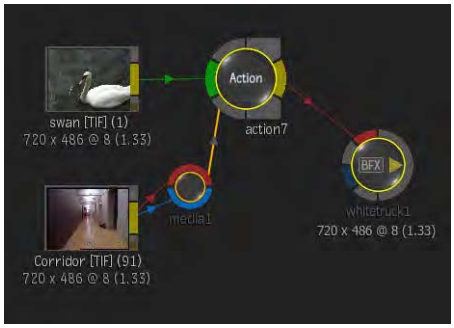
Action is a multilayer compositing tool for creating complex visual effects, with its own distinct schematic and node bin. The Action node supports direct media and indirect media.

Indirect media is connected directly to an Action Media node and appears in the Batch or Batch FX schematic. With indirect media, you can connect any source (such as images, or the output of another node in the Batch or Batch FX process tree) to an Action node. Media nodes are permanently parented to the Action node (you cannot sever the process lines). You can add multiple Media nodes to an Action node.

Direct media appears directly inside an Action node, and in the Batch or Batch FX Sources folder in the Workspace Media panel, as well as in the Batch or Batch FX Timing View. Although direct media does not appear in the Batch FX schematic, media and all related settings are saved with the Batch or Batch FX setup.

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

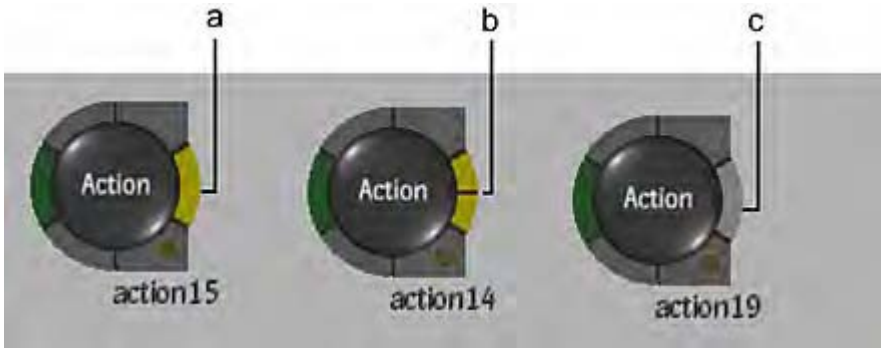
The following example displays indirect media in Action. The *swan* clip is attached to the Action node as the back clip, whereas the *Corridor* clip is attached to an Action media node as the front and matte media.



The Action node supports multiple outputs. For example, you can set up your scene to output your entire composition or just the mattes and then select the type of output you want to process. You can also output stereo results of your scene.

About Action Node Output Tabs

The Action node's output tab changes depending on the number and type of outputs it has, as shown in the following example.



(a) One output (b) Two outputs (c) Multiple outputs (more than two)

NOTE The colours of a dual-output tab are based on the type of output (for example, yellow for composite, blue for matte).

The number of outputs is determined by the outputs set up in the Output list. See About Rendering Outputs from Action (page 435).

Setting Stereo Startup Mode for Action

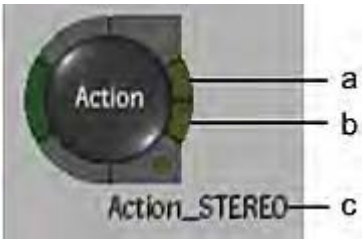
When dragging an Action node to the Batch or Batch FX schematic, you can automatically set the node to stereo startup mode. If you change any of the stereo startup settings, you can revert back to the default stereo startup settings. You can also change the settings of an existing Action mono node to the stereo startup settings.

For information on the settings that are affected by the stereo startup mode, see Rendering Tab (page 425).

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

1 Enable the Stereo Mode button below the node bin.

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



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

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

1. Select the Action node.
2. In the Node Prefs menu for the Action node, click Reset To Stereo Mode in the Rendering section.



Once confirmed, the current setup is replaced by the stereo setup and all media is deleted.

Viewing Nodes in Context

View a node in context with another node to compare intermediate results throughout the process tree. By working with context points, you can modify nodes in the process tree and immediately view the impact those changes have on the nodes further along in the process tree. You can set two context points in a process tree.

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

To view nodes in context:

1. In the schematic, select the node you want as the context (for example, an Action node).
2. Right-click the node, and select Set as Context.

   A dotted green line appears around the node, and (C1) appears next to the node name in the schematic.



3. In the schematic, select the node you want to edit. For example, select a Colour Correct node and modify the setup.
4. View your colour correction changes in the context of Action by selecting Context 1 from the View box.

   In this way, you are colour correcting “in context” of the intermediate result. Furthermore, as you move to different frames in the tool, Context 1 also displays the result at the same frame. For instance, if you are at frame 15 in the Colour Corrector, Context 1 (Action) is also at frame 15. (Exceptions to this occur when using time-based nodes such as Pulldown, Interlace, and DeInterlace.)

5 If you want to set a second context, select another node in the schematic—such as a Keyer RGB node—right-click, and select Set as Context.

(C2) appears next to the Keyer RGB node name.

6 Repeat steps 3 and 4.

NOTE For group nodes, stereo nodes, or nodes with multiple output tabs, you can set the context on specific output tab instead of the complete node.

About Previewing Results in Batch or Batch FX

While creating your process tree, you can preview results at any time. If you do not like the result of one operation, you can modify or delete it without affecting the other operations in the sequence. You can preview your results in Batch or Batch FX by:

- Using proxies in the schematic.

  Double-click the result tab on the node to display or hide the proxy. If Auto Update is enabled in the Schematic section of the Preferences menu, the proxy updates automatically as changes are made.

- Selecting one of the options in the View box to view results at the current pipeline level in the image window.

  Select the node whose results you want to preview, then select an output or result view from the View box (use a 2-Up view to view the schematic and result at the same time). In Batch FX, you can select BFX Timeline Result to see your composite in the context of the timeline.

- Setting up to two context nodes in Schematic view and viewing the results in context in the image window.

Working with Audio Contexts

While working on one clip or branch of the process tree, you can listen to the audio of another clip. Audio contexts work in much the same way as video contexts. You can set up to two audio contexts in Batch or Batch FX.

To set an audio context:

1 In the schematic, select a clip that contains audio.

2 Right-click the clip, and select Set as Audio Context.

Clips with an audio context are labelled A1 or A2 and are outlined by a pink dotted line.

3 To hear an audio context, select Context1 or Context2 from the Audio Source box in the Batch or Batch FX Preferences menu.

NOTE Playback of audio is only available when playing a clip. However, you can scrub audio anytime in Batch or Batch FX. You can scrub your audio context by using `Ctrl` (quick scrub) or `Shift+Ctrl` (real-time scrub) on any node.

About Batch Rendering

When you are ready to generate final clips or intermediate results, define the settings for each Render and Write File node that you want to process.

Use the Render node to output clips to the Desktop or Media Library. Use the Write File node to export open clip files or image sequences in any supported format. With the Render node, metadata is kept; with the Write File node, metadata is not kept, unless you enable Create Clip Data.

Render and Write File nodes support RGB, RGB-A, and Stereo processing modes.

When you work with multiple resolutions, the Render or Write File node's resolution is determined by the clip or node feeding it. However, you can resize your output to any resolution.

You can also set up the Render List to manage multiple Render and Write File processes.

Naming Output Nodes

To help you organize outputs, you can automatically name an Output node with the name of any output socket in the process tree. With the Output node selected in the schematic, press N and click a node output—the name of the Output node is updated. If you want to change the Output node name, or if you change the name of the original output socket, simple redo the naming process.

Exporting With Open Clip Data

Open clip files can be created using a Write File node with Create Clip Data enabled. This gives you the ability to save a copy of the active setup with the exported material, allowing you to package exports with the setups that created them, or to version your exports.

NOTE An open clip file is an XML file which describes channels, media, and versions. You can also create your own open clip files to be used inside Batch, with the Read File node; you might want to do this within a context of render passes interop between Flame Premium and a 3D application.

Advanced: Executing Batch from the Command Prompt

You can execute Batch directly from a command shell so that you can render complex process trees while the machine is not in use, such as overnight. The following command starts Flame Premium for a specified user, and renders a specified Batch setup:

flame -b batchsetup -U user - J project

NOTE The specified user and project name must be valid.

About Open Clip Files

Open .clip files are files created using the Write File node in Batch or by a third party application.

How to Create an Open Clip?

Three ways to create an open clip:

- Using the Batch Write File node with Create Clip Data is enabled.
- Using a third party application or utility.
- Creating one by hand

What Is an Open .Clip File?

In its simplest form, an open .clip file consists of two parts:

- A .clip file
- Media files

The media is any supported media files, from DPX sequences to RED clips, including multi-channel OpenEXR renders. The other component, the .clip file, is written in a human-readable format, the XML format; you can open a .clip in any word processor (kedit, notepad, TextEdit) and decipher its content.

In essence, the .clip files contains all the metadata and references to media that are required to define a source and its versions. The .clip file does nothing by itself, but is essential to recreate the sources.

One of the strengths of the open .clip is that anyone can create .clip files. With one, you can organize media outside of your applications for later use.

Using an open .clip file, you can define many aspects of a clip, including:

- Source Information
- Versioning
- Multiple channels of media
- Paths to media

Why Use an Open Clip?

For open clip history and data exchange.

Open clip history Open clip history provides you with the ability to save with a processed clip the Batch setup used to create that clip. Later on, to open that clip in Batch and edit previous nodes and their settings.

Data exchange Data exchange requirements with a facility's shot management system. In such a case, the .clip is a file created by a third party system, allowing something else to define a source using instructions contained within the file. In this case, the clip can contain a list of render passes for you to composite in Flame Premium. Using of the Read File node, you can easily navigate a source's versions.

Open Clip Example

```
<?xml version="1.0" encoding="UTF-8"?>
<clip type="clip" version="3">
    <name type="string">ClipExport</name>
    <userData type="dict">
        <appName type="string">flame</appName>
        <appVersion type="string">2012</appVersion>
    </userData>
    <tracks type="tracks">
        <track type="track" uid="7119bf06-1ddb-4bba-823a-cd1f2ff27aa0">
            <trackType>video</trackType>
            <dropMode type="string">NDF</dropMode>
            <duration type="time" label="00:00:04+04">
                <rate type="rate">
                    <numerator>24000</numerator>
                    <denominator>1001</denominator>
                </rate>
                <nbTicks>100</nbTicks>
                <dropMode>NDF</dropMode>
            </duration>
            <name type="string">ClipExport</name>
            <feeds currentVersion="v1">
                <feed type="feed" vuid="v1"
uid="d1525f59-bbc7-4da0-9c80-6b251eb5e77a">
                    <storageFormat type="format">
                        <type>video</type>
```

```
                        <channelsDepth type="uint">8</channelsDepth>
                       <channelsEncoding type="string">Integer</channelsEncoding>

                        <channelsEndianess type="string">Little
Endian</channelsEndianess>
                        <fieldDominance type="int">2</fieldDominance>
                        <height type="uint">1080</height>
                        <nbChannels type="uint">4</nbChannels>
                        <pixelLayout type="string">RGBA</pixelLayout>
                        <pixelRatio type="float">1</pixelRatio>
                        <rowOrdering type="string">down</rowOrdering>
                        <width type="uint">1920</width>
                    </storageFormat>
                    <userData type="dict">
                        <recordTimecode type="time" label="00:00:00+00">
                            <rate type="rate">
                                <numerator>24000</numerator>
                                <denominator>1001</denominator>
                            </rate>
                            <nbTicks>0</nbTicks>
                            <dropMode>NDF</dropMode>
                        </recordTimecode>
                    </userData>
                    <sampleRate type="rate" version="3">
                        <numerator>24000</numerator>
                        <denominator>1001</denominator>
                    </sampleRate>
                    <spans type="spans" version="3">
                        <span type="span" version="3">
                            <duration>100</duration>

<path>/ClipExport/01/ClipExport_2011_02_03.[000001-000100].dpx</path>
                        </span>
                    </spans>
                </feed>
            </feeds>
        </track>
    </tracks>
    <versions type="versions" currentVersion="v1">
        <version type="version" uid="v1">
            <name>v1</name>
            <creationDate>2011/02/03 10:53:44</creationDate>
        </version>
    </versions>
</clip>
```

Render Node Settings

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

To access Render Node settings, double-click a Render node in the schematic. If you have multiple Render nodes and Write File nodes in your Batch pipeline, you can manage them in the Render List.

Basic Settings

Basic button Opens the Basic menu where you set Render Node formatting and destination settings.

Export Format box Select the type of clip rendered by the node. RGB-A sets one red RGB input tab and one blue matte tab. Stereo sets two RGB input tabs (one for each eye) as well as two blue matte tabs (one for each eye). Stereo requires a <polarity> token in the Naming field; if there is no <polarity> token, the application automatically adds it to the name of the exported files.

Framerate box Select the framerate of the output clip.

Output Depth box Select the bit-depth of the output clip.

Storage box Select a 12-bit storage option. Select Packed to save image data so that it requires less space on the framestore. Select Unpacked to save an image that can be retrieved without an intermediate step. Active when the Output Depth option is set to 12-bit.

Offset Timecode button Enable to apply any timecode offsets made in the Render List. For example, if you specify a timecode of 00:00:00:00 and render from frame 10, the first frame of the render is TC 00:00:00:09.

Source Timecode field Displays the source timecode for the output. Editable.

Record Timecode field Displays the record timecode for the output. Editable.

Renders Destination box Select whether the render is written to the Desktop or Media Library.

Range From field Displays the first impacted frame. Editable.

Range To field Displays the last impacted frame. Editable.

Render Range button Click to output based on the ranges in the To and From fields.

Burn Range button Click to burn based on the ranges in the To and From fields.

Resize Settings

The Resize settings are the same as when you are using the Resize node in Batch. Click the Active button to activate the Resize settings.

RGB LUT Settings

The RGB LUT settings are the same as when you are using the LUT Editor node in Batch. Click the Active button to activate the RGB LUT settings.

Write File Node Settings

Use these settings to define how individual Write File nodes are processed and to specify a destination for each processed result.

To access Write File Node settings, double-click a Write File node in the schematic. If you have multiple Render nodes and Write File nodes in your Batch pipeline, you can manage them in the Render List.

Basic Settings

Basic button Opens the Basic menu where you set Write File formatting, rendering, and naming settings.

Export Format box Select the type of clip rendered by the node. RGB-A sets one red RGB input tab and one blue matte tab. Stereo sets two RGB input tabs (one for each eye) as well as two blue matte tabs (one for each eye). Stereo requires a <polarity> token in the Naming field; if there is no <polarity> token, the application automatically adds it to the name of the exported files.

Framerate box Select the framerate of the output clip.

File Format box Select the file format of the images to export.

File Extension field Select the extension for the file type selected in the File Format box. Editable.

Output Depth box Select the bit-depth of the output clip.

Source Timecode field Displays the source timecode for the output. Editable.

Record Timecode field Displays the record timecode for the output. Editable.

Frame Pad field Displays the padding of the frame ID appended to the file name. "4" indicates that up to four zeros are prefixed to the file name for each frame; frame 1 is written as 0001, frame 22 as 0022, frame 555 as 0555.

Offset field Displays any timecode offsets that you made in the Render List. For example, if you specify a timecode of 00:00:00:00 and render from frame 10, the first frame of the render has TC 00:00:00:09.

Transfer Characteristics box Select an option to identify the attributes associated with a particular film or video format, such as resolution, frame rate, or colour space. Active when File Format is set to DPX.

Compress button Enable to apply compression to the exported file. This results in smaller file size but lower image quality. Available for Tiff, Sgi, and OpenEXR format types.

JPEG Quality field Displays the degree of quality versus compression for the Jpeg format type. 0 gives the lowest quality (and highest compression). 100 gives the best quality (but applies no compression). Editable.

Create Clip Data button Enable to create an open clip file, which includes extended clip data with the exported files.

Include Setup button Enable to include the Batch setup information with the rendered media. This allows someone accessing the open clip file to view and modify the original Batch setup, similar to how you can edit Clip History.

Version button Enable to version the rendered exports, using an open clip. The resulting open clip has versions accessible through the Clip Versions box in the Read File node. If enabled, it requires a <version> token in the Naming field; if there is no <version> token, the application adds the value of the Version field to the name of the exported files, prefixed by "v_".

Version field Displays the version identifier, which automatically increases at each subsequently rendered export. Or set the version value manually. Ctrl+click to display the next available value. Using a previously used version value overwrites that version with the new export. Editable.

Version Pad field Displays the number of zeroes to left-pad the version number with. Editable.

Range From field Displays the first impacted frame. Editable.

Range To field Displays the last impacted frame. Editable.

Render Range button Click to output based on the ranges in the To and From fields.

Burn Range button Click to burn based on the ranges in the To and From fields.

Name field Displays how to name the exported media files. Build a dynamic naming scheme using the Add Token menu, as well as use any of the characters allowed in a Linux file name. Create folders using the slash (/) character. The application always appends the file ID and the extension of the file format selected for the export. Editable.

Destination Path field Displays where the exported files are created. The path in bold characters is created by editing the Path field. The text in grey is the content of the Name field. The open clip file (Create Clip Data enabled) is always created in the path in bold. Polarity and Version are automatically added unless specified in the Name field. Editable.

Add Token menu Inserts in the Name field one of four tokens. <name> for the name of the Write File node. <date> token for the current date, formatted as YYYY_MM_DD. <version> for the value of the Version field; ignored if either Version or Create Clip Data are disabled. <polarity> to differentiate left eye and right eye files in Stereo exports; ignored if Export Format is set to RGB-A.

Load Name button Click to load a previously saved naming pattern to the Name field.

Save Name button Click to save the content of the Name field, to be loaded at a later time.

Resize Settings

The Resize settings are the same as when you are using the Resize node in Batch. Click the Active button to activate the Resize settings.

RGB LUT Settings

The RGB LUT settings are the same as when you are using the LUT Editor node in Batch. Click the Active button to activate the RGB LUT settings.

Matte LUT Settings

The Matte LUT settings are the same as when you are using the LUT Editor node in Batch. Click the Active button to activate the Matte LUT settings.

Render List Settings

Use the Render List to manage output and export processes. You can set up several output or export processes, specify the frames at which to start and end a process, as well as set job priorities.

You can execute jobs immediately or at a later time using a Batch script. You can attach scripts that convert processed images to a movie file (*.mpg*) or that send messages about the status of Batch processes. The jobs that appear in the Render List are saved in the Batch setup file.

The Render List also displays some settings you defined for a specific Render or Write File node. For example, it lists the destination you specified for a processed output as well as the type of output.

To access the Render List, click the Render List button. You can switch between the Render Node or Write File node menu and the Render List as needed.

List Filtering Settings

Nodes box Select whether to activate or deactivate all Render and Write File nodes in the Render List.

List box Select whether to render only visible Render and Write File nodes in the current Render List view, or include Render and Write File nodes nested in groups.

Render List Columns

Process Indicator column Activate individual Render and Write File nodes for processing. Click an indicator to activate or disable one job at a time.

The arrow in the P column is yellow when a job is active, grey when disabled, and red when there is a processing error.

Node Name column Displays the name for the processed clip. Click a Node Name field to rename the clip.

Start and End frames columns Specify the range of frames you want rendered in a processed result. Corresponds to the Range From and To fields in the Render or Write File node settings.

For example, one node could be set to process the first 30 frames of a 60-frame clip, and another node could process the remaining 30 frames.

NOTE The Start and End fields override the total number of frames in the timebar. For example, if the timebar shows a 60-frame clip and a node is set to start at frame 20 and end at frame 100, then 80 frames are processed.

Destination column Displays the destination for the output or exported result. Use the Output Destination box in the Render or Write File node settings to change the destination.

Priority column Dsiplays the priority of the jobs to be processed. If two processing nodes are connected to the same process tree, they have the same priority.

If there is a processing error, for example, if the Render or Write File node is not connected to a process tree, the job is deactivated and the priority is automatically set to 0.

Script column Attach a script file whose content is executed when the Render or Write File node is finished rendering. Scripts must have the file extension *.bscript*.

Click this field to load an existing script from the file browser. See the Script Reference section below for more information.

Source column Generate your output with a source timecode. Corresponds to the Source Timecode field in the Render or Write File node settings. To edit the timecode, double-click the field and enter a new value.

Record column Generate your output with a record timecode. Corresponds to the Record Timecode field in the Render or Write File node settings. To edit the timecode, double-click the field and enter a new value.

FCM column Displays the framecode mode. Change this value using the Framerate box in the Render or Write File node settings.

Output Type column Displays the type of clip to be processed: RGB, RGB-A, or Stereo. Corresponds to the Export Format box in the Render or Write File node settings.

Script Reference

Batch scripts are ASCII files that contain instructions related to a Batch job. They must have the file extension *.bscript* for Batch to recognize them. A Batch script is executed after the Render or Write File node to which the script is associated has finished rendering.

NOTE Use the *imgview* and *movie* scripts with the Read File node, and use the *listVariables* script with the Render or Write File nodes.

You should consult with your system administrator when creating scripts. Scripts are created using either C shell, Korn shell, or Perl. Also, Batch scripts must have the proper file access, directory access, and execution permissions for the files, directories, and applications in the script.

When the script is executed, Flame Premium continues with its own processes and does not wait for the script to terminate. Flame Premium imposes no restrictions with executing scripts; however, make sure that no one else is using the system when a script is executed and that the script is approved by your system administrator.

Script variables that you can set are defined in the following table.

Variable	Description
BATCH_NODE_NAME	Corresponds to the name of the Render or Write File node. This is the same name that you provide in the Render List Node Name field.
BATCH_FIRST_FRAME	Corresponds to the first frame displayed in the Render List for the selected node name.

Variable	Description
BATCH_LAST_FRAME	Corresponds to the last frame displayed in the Render List for the selected node name.
BATCH_ASPECT_RATIO	Corresponds to the framestore aspect ratio.
BATCH_FRAME_RATE	Applies only to scripts associated with Write File nodes. This is the framerate of the final clip result.
BATCH_FRAME_WIDTH	Applies only to scripts associated with Write File nodes. Corresponds to the width of the final clip result.
BATCH_FRAME_HEIGHT	Applies only to scripts associated with Write File nodes. Corresponds to the height of the final clip result.
BATCH_OUTPUT_DST	Applies only to scripts associated with Render nodes. Indicates the destination of the final clip result as either the Desktop or Media Library.
BATCH_BIT_DEPTH	Applies only to scripts associated with Write File nodes. Specify the bit depth of the final clip result.
BATCH_CROP_WIDTH	Applies only to scripts associated with Write File nodes. Specify the crop width of the final clip result.
BATCH_CROP_HEIGHT	Applies only to scripts associated with Write File nodes. Specify the crop height of the final clip result.
BATCH_FILE_FORMAT	Applies only to scripts associated with Write File nodes. Indicates the numbering format of a sequence of exported images in the final clip result. This numbering sequence ensures that the order of frames in the clip is intact when you import the clip.
BATCH_FILE_EXT	Applies only to scripts associated with Write File nodes. Specifies the format to which the final clip result is being exported: Alias *(.als)*, Cineon, Dpx (Spirit), Jpeg *(.jpg)*, Pict *(.pict)*, Pixar *(.picio)*, Sgi *(.sgi)*, SoftImage *(.pic)*, Targa *(.tga)*, Tdi/Maya *(.iff)*, Tiff *(.tif)*, or Wavefront *(.rla)*.
BATCH_FILE_DIR	Applies only to scripts associated with Write File nodes. Indicates the file location of the final clip result.

About BFX Output Nodes

A Batch FX pipeline always terminates with a BFX Output node. This node is created automatically when you enter Batch FX from the timeline, and can't be deleted.

You can generate both RGB and alpha results from a BFX Output node to the timeline.

Since an effect you create in Batch FX is a setup applied directly to one or more timeline segments, you can choose to render your BFX directly in Batch FX, or wait until you return to the timeline.

Naming Output Nodes

To help you organize outputs, you can automatically name an Output node with the name of any output socket in the process tree. With the Output node selected in the schematic, press N and click a node output—the name of the Output node is updated. If you want to change the Output node name, or if you change the name of the original output socket, simple redo the naming process.

Outputting an Alpha to the Timeline

You can output alpha results in addition to the RGB result from a BFX Output node. You connect the RGB result to the front input of the BFX Output node, and the alpha result to the matte input of the BFX Output node (The output node now displays BFXa). Both results are fed directly to the timeline.

An Axis Timeline FX is automatically added on output, provided that the Add Axis On Matte Output button in the Batch FX section of the Preferences menu is enabled (this is the default).

Output Node Settings

Double-click the BFX Output node in the schematic to see its settings.

Basic Settings

The settings in the Basic menu are non-editable, and are available so you can see the output resolution and timecode of the timeline segment you entered Batch FX with.

Width field Displays the width resolution of the output clip. Non-editable.

Height field Displays the height resolution of the output clip. Non-editable.

Frame Depth field Displays the output frame depth. Non-editable.

Aspect Ratio field Displays the output aspect ratio. Non-editable.

In Timecode field Displays the in point timecode of the output clip. Non-editable.

Out Timecode field Displays the out point timecode of the output clip. Non-editable.

Duration Timecode field Displays the timecode duration of the output clip. Non-editable.

Resize Settings

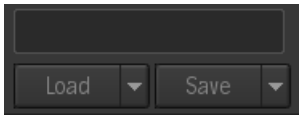
The Resize settings for the BFX Output node are only available if you attach a clip to the pipeline that is a different resolution than the original BFX output. In this case, the Resize settings are active, but you are not able to change the Destination settings (these must remain the same as when you entered BFX from the timeline).

RGB LUT Settings

Similar to the Resize settings, The RGB LUT settings for the BFX Output node are only available if you attach a clip to the pipeline that is a different resolution than the original BFX output.

Saving and Loading Batch or Batch FX Setups

You can save and load Batch or Batch FX setups. When saving a setup, you have the option of saving the complete Batch or Batch FX setup, or only the selected items in the schematic. Use the drop-down list beside the Save button to make a selection before clicking Save. When loading a saved setup, you have the choice of loading (replacing any existing nodes), or load and append.



In Batch or Batch FX, you can also save and load specific node setups. The node load and save buttons appear next to the node name field when a node is selected in the schematic.



Saving Sources with Batch Snapshots

In Batch, instead of saving and loading setups, you also have the option of saving or restoring snapshots. The benefit of using Batch Snapshots is that along with the setup, you also save the sources used by the setup. Batch Snapshots work in a similar manner to Desktop Snapshots, in that they reside in the Workspace Library section of the Media Panel, and you can manage (create, open, or close) multiple snapshots. There are also multiple ways to replace or append a saved snapshot.

To create a Batch snapshot:

1 Enter a name in the name field.



2 Click Snapshot.

A closed snapshot folder is created and can be seen in the Batch Snapshots area in the Workspace Media panel. If you right-click, and select Open Snapshot, you can see a Sources subfolder under the Snapshot entry containing the sources saved with the snapshot.



TIP You can also create your own folders in the Batch Snapshots area to organize snapshots (right-click Batch Snapshots and choose New Folder).

3 The Batch area in the Workspace Media panel also displays the name of the snapshot in parentheses. If you make any changes to the schematic, an asterisk appears next to the name in the Batch area. You can click Snapshot again to overwrite your snapshot with the changes, or create a new snapshot with a different name.

NOTE Saving a Batch snapshot with the same name as an existing saved snapshot allows you to replace the existing snapshot, or bump the snapshot version. Bumping creates a new snapshot, with an incremental version number appended to the name (for example, *comp_V01*). Multiple snapshot versions are automatically grouped into a folder under Batch Snapshots.

As with Media Libraries and Desktop snapshots, you can close Batch snapshots as a memory-saving operation. The contents of a closed snapshot are flushed from memory, but you are able to open a snapshot at any time to view all the sources contained in it. The Open Snapshot and Close Snapshot options are available in the right-click menu for each Batch snapshot in the Workspace Media panel.

TIP You also have the option of creating and saving a Batch setup when you save a Batch Snapshot. Enable Also Save Batch File in the Preferences Timeline FX / BFX section. In this case, the setup is saved in the default Batch folder *~/batch/flame* in a snapshots subfolder. Note that these setup files are not managed by Flame Premium. Therefore, if you delete a Snapshot, the corresponding setup file is not deleted automatically.

Restoring a Batch Snapshot

You can replace or append a batch snapshot in a number a different ways (a snapshot does not need to be in an open state to restore it):

- While in Batch, drag and drop a snapshot to the schematic, and choose whether to replace or append.
- From anywhere in Flame Premium where you can access the Workspace Media panel, drag and drop a snapshot to the Batch Sources folder in the Workspace Media Panel. In this case, a replace is performed once you confirm the operation, and you are switched to the Batch tab automatically.
- From anywhere in Flame Premium where you can access the Workspace Media panel, right-click a Batch snapshot, and select Replace Batch Setup or Append Batch Setup. In this case, you are switched to the Batch tab automatically.
- From anywhere in Flame Premium where you can access the Workspace Media panel, double-click a Batch snapshot, and confirm to replace the current setup. In this case, you are switched to the Batch tab automatically.

NOTE Using any of the above methods while in Batch FX only allows you to append to your existing Batch FX setup. Any Write File or Render nodes in the snapshot are displayed as greyed out nodes in the Batch FX schematic.

Using a Create BFX Clip for Caching or Versioning

Use Create BFX within Batch or Batch FX to create a BFX clip that can be useful for disk caching and give you flexibility for re-use or versioning of complex pipelines.

A BFX clip is a nested group or container that has benefits over Group nodes in Batch or Batch FX.

Creating a BFX Clip

To create a BFX clip, right-click a node in the schematic, and select Create BFX. A new clip node is created that included the node you selected, and all downstream inputs to the node (that is, everything connected to the left of the node). At this point, you can delete the old nodes, and connect the new BFX clip into your pipeline.

Disk Caching Capabilities

You can switch to the Batch or Batch FX Timing view, select your BFX clip in the timeline, and click Render. Working in Batch or Batch FX is now faster, since all the nodes that make up the BFX clip are now rendered. Once you are ready to render your complete Batch or Batch FX pipeline, the BFX clip does not need to be re-rendered if you haven't modified it.

Flexibility

Another benefit of using a BFX clip is the flexibility it offers for versioning or re-using the clip. You can use the offset feature in the Timing view to offset the start time of the BFX clip, or even slide or trim the media and animation keyframes contained within the clip. When you create a BFX clip, it resides in the Batch or

BFX Sources folder in the Workspace Media panel. You can copy this clip into another folder in the Media Library, and re-use it in another timeline segment, or even another project.

Exploding BFX Clips

To return a BFX clip to its original separate nodes, right-click the clip, and select Explode BFX. This action is non-destructive, but it does delete the rendered frames (if you had rendered the BFX clip).

Since you can nest a BFX clip inside another BFX clip, the explode operation only explodes to one level. In the Basic menu for the BFX clip, you have an option to Explode All levels of the clip back to its original nodes.

Using the Timing View to Offset Clips

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

To offset clips:

1. To access the Batch or Batch FX Timing view, click the Timing button.
2. Select the clips and/or segments to offset. If you are offsetting all clips in a group, select one clip or segment in the group and then click Select All Clips in Group. To select all clips on a track, click the clip proxy to the left of the track.
3. Drag left or right in the Offset field, or enter the number of frames by which to offset the clip.



All selections are offset by the same amount.

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

About the Timing View

The Timing View displays the timeline of all Batch or Batch FX clips in one view. This view is especially useful for edits where you want to see the relative position of all clips in time. For example, offset multiple clips or edit one clip in relation to the others.

You can adjust the timing of any number of clips as well as perform basic editing operations such as slipping, sliding, or trimming. You cannot perform editing operations that involve combining clips or inserting frames from one clip into another.

Edit a clip gesturally, or use same buttons and keyboard shortcuts as you would on the timeline.

To access the Timing View, click the Timing button. All clips in the current schematic, including clips that are part of a group node, are represented by a clip proxy and primary video track. Gaps are displayed based on Media settings, which are set in the clip's Basic menu.

You can select multiple clips on different tracks to perform timing operations at the same time. Selected clip proxies and tracks are highlighted by an orange bounding box.

Timing View Settings

Some Timing View settings relating to trimming, slipping, or sliding clips (such as the Ripple and Snap buttons), or to the actual timeline itself (such as the Timeline Options menu) are exactly the same as the main Flame Premium timeline.

The following settings are specific to the Timing View:

Use Start Frame button Click to offset the selected Batch clip so that its source frame matches the timeline's timecode. This allows you to also offset based on header timecode. Not available in Batch FX.

Offset field Displays the number of frames by which selected clips and/or segments are offset. This same setting can be found in the Timing View as well as the Clip Settings. Editable.

Filter box Select the type of clips to display in the Timing View.

Select:	To display:
Show All	All clips in the schematic.
Show Tree	All clips in the selected processing tree.
Show Branch	All clips in the current node branch.
Show Selected	All clips and nodes selected in the schematic.

Select All Clips in Group button Click to select all clips that are part of a group. First select one clip or segment in the group.

Timing View Render box Select an option and click to render selected Timing View clips. Also available as a contextual option when right-clicking a clip node.

Caching Frames

You can avoid reprocessing the same nodes and clips as you navigate in the timebar by specifying which nodes and clips should be cached. Cached frames are stored on the system framestore, increasing system performance and efficiency. Stored cache clips are contained in a _Cache_ library and correspond to the clip IDs in the setups. These cache library files are named according to setup and project name.

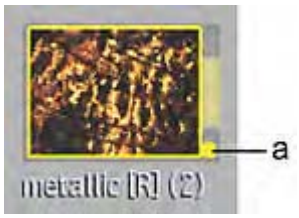
Node and clip caching is enabled in the Cache section of the Batch or Batch FX Preferences menu. The Cache menu also displays the number of rendered frames in the cache for the selected nodes or clips, and the remaining space on the framestore.

Any clip node can be cached as long as it is edited, resized, or has had a LUT conversion applied. Batch setup files include the location of any cached frames, which are stored with the clip ID until removed. The history associated with a clip can also be cached, but not if the clip was rendered in Proxy mode.

Each node and clip has a cache tab on the lower-right corner.

The following steps are not required to actually cache frames. Typically, you use intelligent caching, which automatically caches frames as you move through the timebar. Caching occurs as the frame is displayed and

the node operation is applied. A cached node automatically saves frames as you display—or visit—them and then reloads them each time you visit the frame again. Your workflow speed is increased.



(a) Cache tab



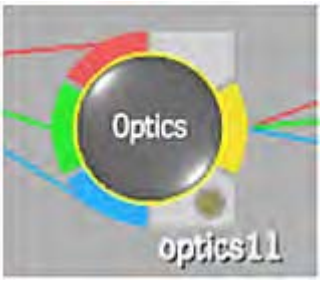




Read and write mode Cache enabled



Read-only mode Frames already cached are accessible

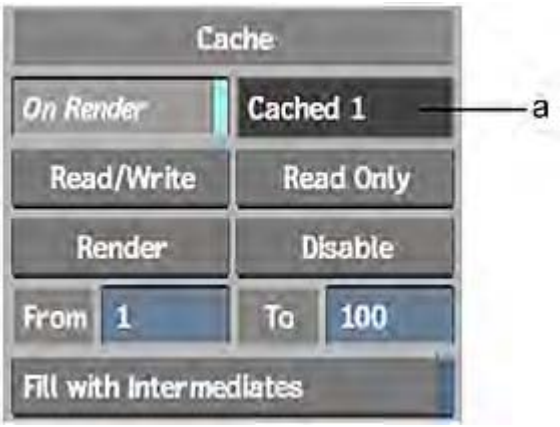


Cache disabled

Click the cache tab to activate any of the following modes.

Mode	Colour	Description
Disabled	Grey	Disables the cache. You can also `Alt`-click to disable and remove cached frames for the selected node.
Read and write	Yellow	Saves and loads data to and from the cache.
Read-only	Amber	Is shown with a yellow outline. No data can be saved to the cache, but rendered frames remain stored. Use this mode to prevent the framestore from getting full.

Changing or cutting an input or any node further along the process tree that has cached frames clears the cache buffer. The Cache section of the Batch or Batch FX Preferences menu (Rendering tab) shows how many frames are cached for the selected node and the number of frames available on the framestore. This allows you to clear the cache of the selected node or all cached frames.



(a) Cached frame counter

To cache frames:

1. Specify the frames you want to render and store in cache using the From and To fields.
2. Click Render.
   The cached frames are stored in the framestore and added to the cached frame counter.

To turn off the cache and remove cached frames, `Alt`-click the cache indicator for the selected node.

To render node caches while rendering:

1. Enable the cache for any nodes or clips whose frames you want to cache.
2. Enable On Render.



3. When you are satisfied with current settings in your process tree, click Render.
   The clip is rendered, and all node caches are rendered. While rendering, the message bar displays the number of frames in the node and the time remaining until the render is complete.

Grouping Nodes

To group nodes:

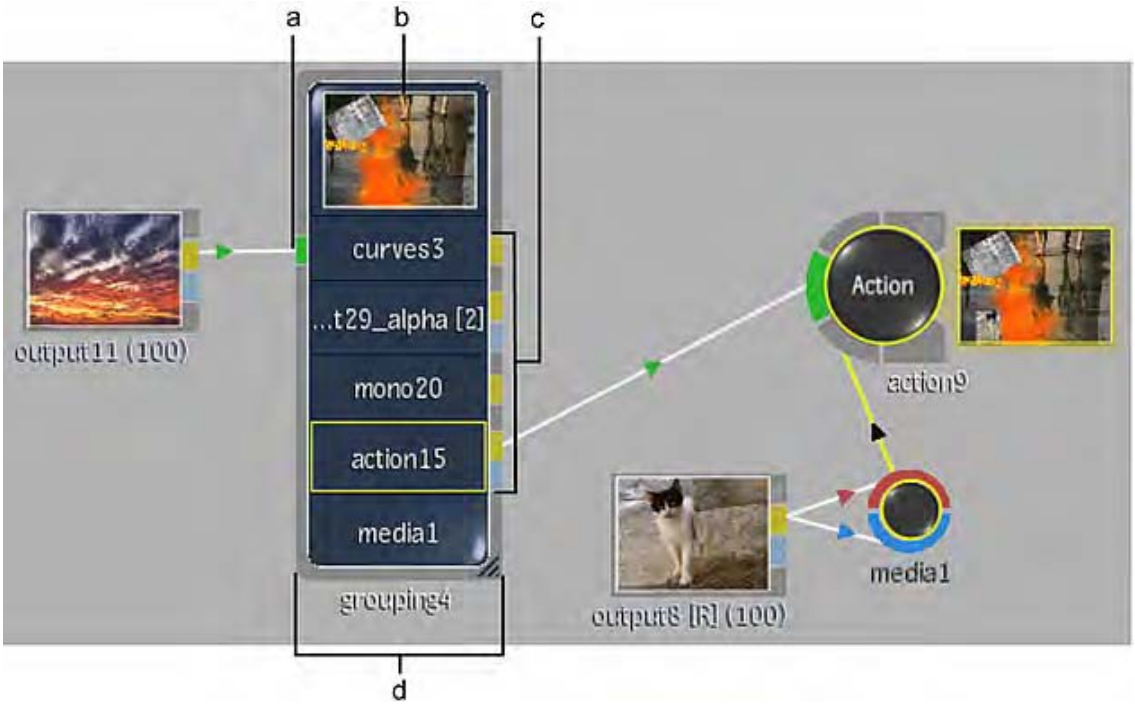
1. `Control`-drag to select the nodes you want to group.

Selected nodes are outlined in white.

2 Right-click the selection, and select Group.

The selected nodes collapse into a Group node. The Group node lists the nodes contained in the group, as well as the non-hidden input tabs and output tabs.

TIP You can also create a group inside a group.



(a) Input connection (b) Proxy window (c) Output connections (d) Group icon

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

4 Click the group node to display group display and editing settings.

To display a proxy of an output in the proxy window in the group node:

1 Do any of the following:

- Click the field in the Proxy list that corresponds to the node whose output you want to display.
- Press the `Shift+up or down arrow` as you navigate through the node's Group List.
- If a node has multiple outputs, press `Ctrl+Shift+up or down arrow` as you navigate through the node's Group List to display all the outputs of a node.

To change the size of the group icon:

1 Drag the lower-right corner of the icon to make it wider or narrower.

Editing a Group

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

To edit a group:

1 Select the group you want to edit.

2 In the Group settings, click Edit.

The nodes in the group appear in the schematic ungrouped, with other schematic nodes grayed out.

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

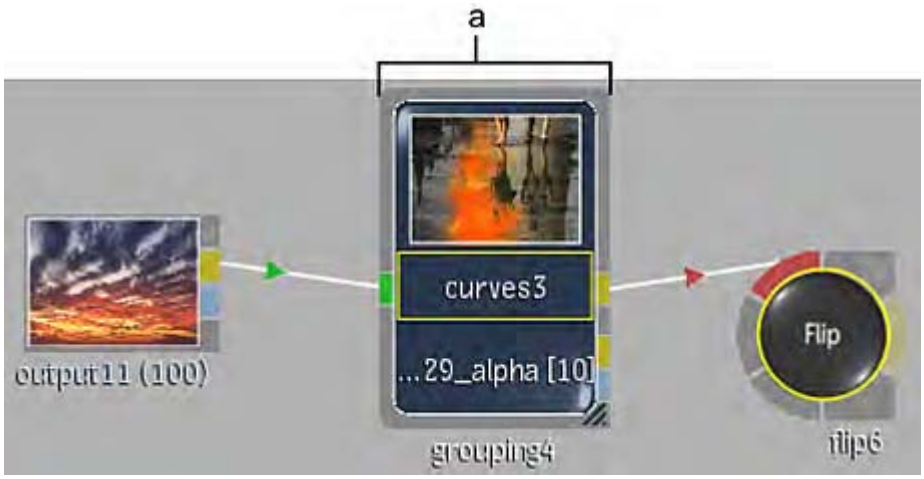
3 Modify the nodes as required.

4 Click Exit Group to return to the previous schematic.

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

About Group Nodes

A group node is a selection of nodes and clips that are collapsed into one node icon. Each node icon displays the group's contents, inputs, and outputs. You can create groups for different branches of your process tree and work on each separately. Groups are useful for simplifying cluttered schematics.



(a) Group node

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

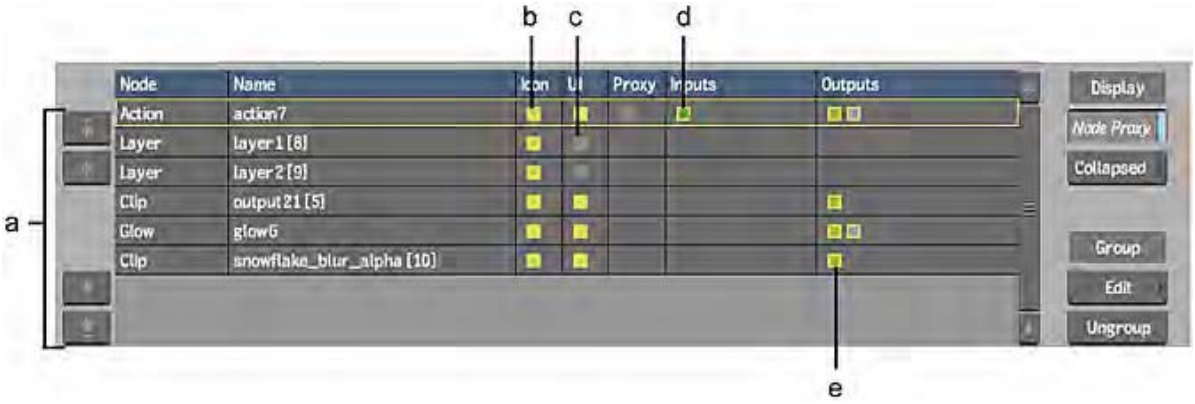
A clip with multiple outputs, (a stereo clip with left and right matte outputs, for example) can be displayed as a group node, allowing all output tabs to be displayed for connection.

Stereo nodes are displayed as group nodes, with inputs and outputs for the left eye and right eye.

Group Settings

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

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



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

Use the Group List menu as follows:

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

Node Proxy button Enable to display a preview proxy of a node in the group icon. To change the proxy, press Shift + up arrow or down arrow as you navigate through the Group List.

Collapsed button Enable to collapse the group icon so that only the preview proxy is displayed.

Edit button Click to display the schematic of the group contents, with all other nodes greyed out. To return to the Group List menu, click EXIT Group.

EXIT Group button Click to return to the Group List menu after editing.

Ungroup button Click to expand the group of nodes to their original positions in the schematic.

Node List box Displays the menu of the selected option. Select List or click the Return to Group button to return to the Group List menu.

Return to Group button Displays the Group List menu.

Mimicking, Copying, and Duplicating Nodes

When working in the Batch or Batch FX schematic, you have a few options to help you quickly get settings, or even complete nodes, to and from similar nodes. See the procedures below for information on how to access each of these options, but as a quick summary, here are the reasons for using each option:

- Mimic Link: You can keep two nodes in the schematic in sync by creating a Mimic Link between them. When you change the settings of one node, they are automatically mimicked in the other node.

  NOTE To mimic nodes, the nodes must be of the same type. For example, you can link between Flip nodes but not between a Flip node and a Colour Correct node. Some nodes, such as Action or Modular Keyer do not support the Mimic Link option. In this case, you can see an error message in the message bar.

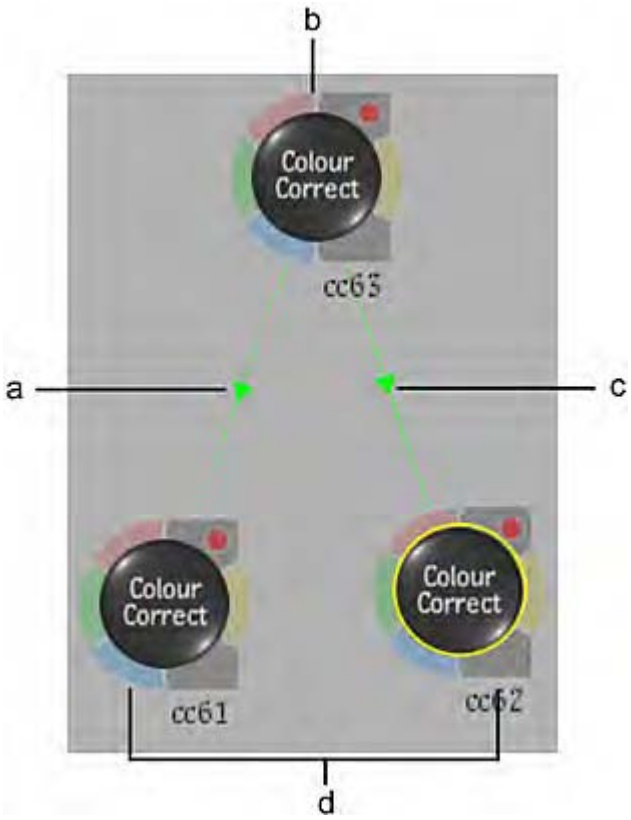
- Duplicate: Create another instance of a node or group of nodes within the same Batch or Batch FX schematic. All node information is duplicated as well.
- Copy: Copy is similar to Duplicate, except that the selected nodes or group of nodes is copied into the clipboard, so you can paste exactly where you want in the schematic. An added bonus of using Copy and Paste instead of Duplicate, is that you can paste a copied node or nodes into a different Batch or Batch FX schematic. You can even copy a Batch node and paste it into a Batch FX schematic or vice-versa.

Using Mimic Link

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

1. From the Edit Mode box, select Mimic Link.
2. In the schematic, drag between nodes of the same type.

   A green-arrowed dotted line indicates that the nodes are linked as duplicates. The direction of the arrow indicates which node is the master.



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

3 Connect each node to a mono clip.

4 Change the settings of any node.

To work with Mimic link using a stereo group node:

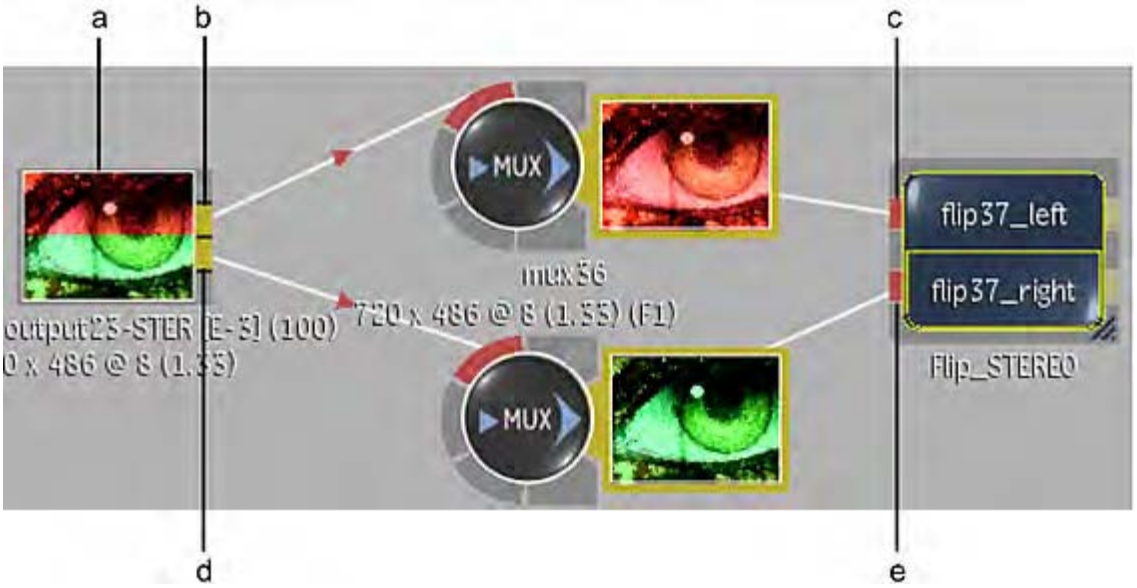
1 With Stereo Mode selected, drag a node from the node bin to the schematic.

A stereo group node containing left eye and right eye inputs appears in the schematic.



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

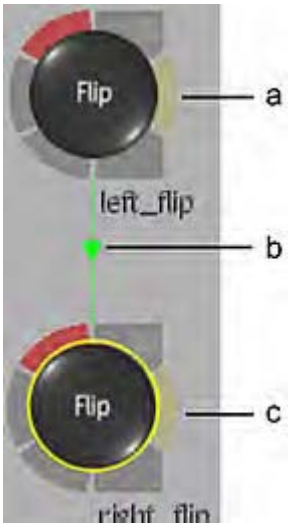
2 Connect the left and right eye inputs of the stereo group node to the clip node's left and right eye outputs, as shown in the following example.



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

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

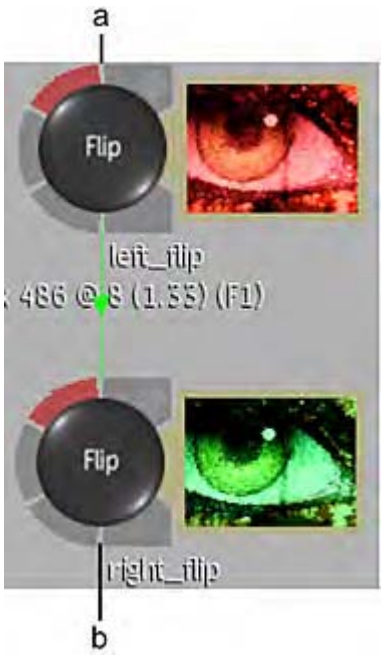
The schematic is dimmed except for the two nodes of the same type as the one you dragged from the node bin. A green-arrowed dotted line indicates that the nodes are linked as mimics.



(a) Master node (b) Mimic link (c) Mimicked node

4 Change the settings of either node.

Notice that the changes made to one node are mimicked on the other node so that both the left and right eye clips are affected simultaneously.



(a) Flip node connected to left eye is edited (b) Settings are automatically mimicked on Flip node connected to right eye

5 Click Exit Group.

Duplicating Nodes

To duplicate nodes:

1 In the schematic, select a node, or group of nodes.

2 Right-click the selection, and choose Duplicate.

A duplicate of the node or nodes (with all node settings) appears in the schematic.

Copying Nodes

To copy nodes:

1 In the schematic, select a node, or group of nodes.

2 Right-click the selection, and choose Copy.

A copy of the node or nodes (with all node settings) is added to the clipboard.

3 Navigate to the location in the schematic (or another Batch or Batch FX schematic) where you want the copied node or nodes to reside, right-click and choose Paste.

A copy of the node or nodes (with all node settings) appears in the schematic.

Customizing the Tools Bin

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

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

To create a tab:

1 Click the plus sign tab.



2 Name the tab in the keyboard that appears.

NOTE You can create a maximum of 6 new tabs.

To copy a node to another bin:

1 Drag the node on top of the destination tab.



2 Release the cursor.

3 Click the destination tab.
The copied node appears in the bin. Nodes are added to the end of a bin in the order copied.

NOTE Nodes cannot be duplicated within the same bin.

To move a node to another bin:

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



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

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

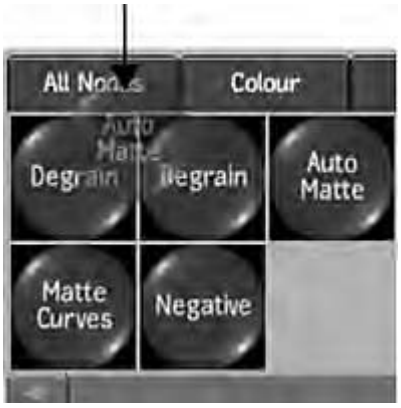
The node is moved from its original location to the destination bin. Nodes are placed at the end of a bin in the order moved.

NOTE Nodes cannot be duplicated within the same bin.

To reorder a node within a bin:

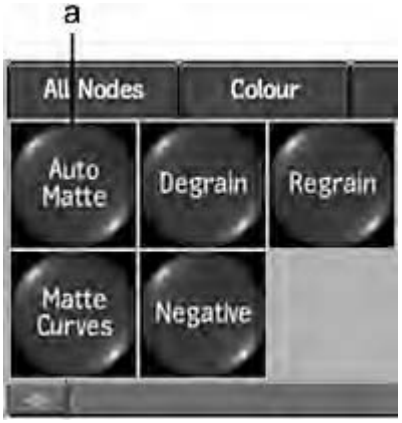
1 Press `Ctrl+Alt` and drag the node to a new location. You can move nodes from one row to another as well as reorganize nodes within a row.

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



2 Release the cursor when it appears where you want the node moved.

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



(a) Reordered node

NOTE Click Sort to reset a tab to alphabetical layout

To delete a bin:

1 Press `Alt+Win` and drag the tab to the bottom of the screen.

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

WARNING There is no undo capability when deleting a bin.

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

To delete a node from a bin:

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

   WARNING There is no undo capability when deleting a node.

To reorder a tab:

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

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

About Batch and Batch FX Node Bins

In Batch and Batch FX, a node is a graphical representation of a function or effect that affects a clip or another node in the process tree. The available nodes are organized in bins that contain all nodes required for building a process tree. The nodes are divided into the following three groups, classified by tabs.

To access the Batch or Batch FX node bins, click the FX Nodes button. To add a node from the node bin to the schematic, double-click a node, or drag a node directly to the schematic. Enable Stereo Mode to add stereo nodes to the schematic.

I/O Bin

Use the nodes in the I/O bin to load clips into Batch or to output rendered clips. Available nodes are Read File, Write File, Render, and MUX.

In Batch FX, the I/O bin includes nodes for importing and working with source files. Available nodes are Read File, MUX, and Back Clip. By default, a BFX Output node is automatically connected to a Batch FX process tree.

Node	Description
Read File	Import clips in any supported formats from any available location. Use to import multi-channel clips such as OpenEXR, or metadata rich clips such as RED clips.
Render	Output clips to the Desktop; metadata such as clip history and timecode is kept.
Write File	Export image sequences in any supported format.
MUX	The MUX node is a schematic tool that helps create cleaner schematics by allowing you to have multiple outputs from one input. It incorporates the hiding of connections to prevent schematic connection overlaps.

Node	Description
Back Clip	The Back clip node enables you to create composites and effects in Batch FX, when using the Generate Adjustment Segment option from the timeline.

You cannot customize the I/O bin.

Tools Bin

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

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

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

Enable the Stereo Mode button below the Tools bin to access only stereo nodes (nodes not available as stereo nodes are greyed out).

TIP Use the scroll bar under the applicable bin to scroll through all available nodes. You can also frame and highlight all the nodes starting with the same letter by pressing the key corresponding to this first letter on your keyboard while the cursor is sitting over a node bin. For example: pressing "K" while hovering the cursor above the All Nodes bin automatically highlights all of the Keyer nodes.

User/Project Bin

The User/Project bin contains custom nodes classified by a User tab and a Project tab. Use this bin to save custom setups per user or project.

Creating Custom Nodes

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

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

To create a custom node:

1. In the schematic, select a node or group of nodes.
2. Drag the selection on top of the User or Project tab.
   The selection is copied to the bin. The original selection remains in the schematic.

To use a custom node:

1. Select a custom node from the User or Project bin. If necessary, scroll through the bin to find the node.
2. Drag the node to the schematic, or double-click the node.
   The same configuration of nodes and clips that was used to create the custom node appears in the schematic.

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

To manage the custom node bins:

1 Select any of the following from the dropdown lists.

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

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

1 Press `Alt+Win` and drag the node to the bottom of the screen.

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

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

Batch and Batch FX Preferences

Click the Batch Prefs or BFX Prefs button to access settings to customize rendering, preferences, and adaptive degradation in Batch or Batch FX.

NOTE Some Batch or Batch FX nodes also have specific preferences related to the functionality of the node. Click Node Prefs to access the preferences for a selected node.

Rendering Tab

Free Frames field Displays the space remaining on the framestore. Non-editable.

Use Proxies button Enable to replace clips with proxies. When enabled, the image window is outlined in amber.

Hardware Anti-aliasing Sample box Select a hardware anti-aliasing sampling level to accelerate edge anti-aliasing with no performance penalty. Available values are dependent on your graphics card.

On Render button Enable to allow node and clip caching.

Current Frame Cache field Displays the current number of cached frames. Non-editable.

Read/Write button Click to set cache to read and write status.

Read Only button Click to set cache to read-only status.

Render Cache button Click to render the cache of selected nodes for the frames defined in the From and To fields.

Disable button Click to turn the caching of selected nodes on or off.

From field Displays the lower limit in the range of frames that can be cached. Editable.

To field Displays the upper limit in the range of frames that can be cached. Editable.

Fill with Intermediates button Enable to fill each node's cache with the intermediate clips contained in the history. If a clip was rendered in proxy mode, the history is not cached.

Preferences Tab

Audio Source box Select the audio context you want to hear when working with another clip. This same setting appears in the Timing View menu.

Text Log button Enable to save rendering status information in a text file, located in /usr/discreet/project/<project home>/batch/log.

Status Webpage button Enable to create three status HTML files, located in /usr/discreet/html.

Update Numerics button Enable to update numerics in fields as the timebar is scrubbed.

Show Negative Frame button Enable to displays negative frame numbers in the Current Frame field when a segment is offset before frame 1. Disable to display only frames from frame 1 onward.

Set to In/Out button Click to have the timebar display frames as defined by the in and out points set on the clip's timeline.

Set to Media Range button Click to have the timebar display the head and tail frames brought in from the clip's timeline.

Clip Info box Select an option to display resolution, frame rate, size, or any combination of information in the schematic.

Transparency field Displays the transparency of currently unselected nodes in the schematic. Editable.

Auto Parent button Enable to allow Parent and Cut options in the schematic while using the Select edit mode.

Auto Insert button Enable to automatically insert a node when dragged between two connected nodes. When disabled, press Shift to auto insert.

Highlight Path button Enable to highlight the path of the currently selected node in the schematic.

Auto Update button Enable to update a node's result proxy automatically.

Update Clip box Choose whether to automatically update all clips or only selected clips.

Clear Buffer button Click to clear the Undo buffer of all previous undo operations.

Check for Updated Versions button Checks each imported Read File .clip file to see if it is the latest version, and updates it if that is not the case. Asks for confirmation before updating the version. Only available when a Read File node is used to import .clip files in the schematic.

Adaptive Degradation Tab

Degrade Mode box Select how to propagate display degradation in the image, according to settings for supported nodes. Nodes in the pipeline respect this degraded image. In this case, press Preview to see your results with full settings on.

Change Node Degradation box Select whether to enable or disable adaptive degradation before clicking Apply To Selected.

Apply To Selected button Click to set the selected nodes to the chosen option in the Change Node Degradation box.

The following two settings appear in the Node Prefs menu for specific nodes that support adaptive degradation:

Active button Enable to activate adaptive degradation display settings. Use to prevent slowdowns in displaying your results when changing supported settings.

Synchronize All button Enable to synchronize all nodes of the same type with the Adaptive Degradation settings of the current node.

Compositing in 3D Space with Action 17

About Action

Action is a multilayer compositing tool for creating complex visual effects. Use Action to animate clips in 3D and add camera, lighting, and shadow effects.

You create effects and animations by manipulating objects in the scene. Objects you work with include surfaces, light sources, axes, particles, shadows, and the camera.

Action Concepts

The following concepts are used throughout the Action topics to describe the workflow and user interface.

Scene The scene is Action's representation of 3D world space. It is where objects are placed and animated. World space has three directions: X (left/right), Y (up/down), and Z (in/out).

Object An object is any element in a scene. An axis, a camera, a surface, a light, or a model can also be referred to as an object.

Media Media consists of a front and matte clip only.

Surface A surface is a special type of 3D geometry onto which media is mapped.

Geometric Object (Models) A 3D geometric object can be something as simple as a cube, sphere, or cone, or as complex as a character. Although some basic geometric objects are included with Flame Premium, most geometry is imported from a 3D application such as Maya and 3ds Max. A model is another way to refer to a geometric object.

3D Models and Text You can import 3ds Max files, FBX format files, Alembic format files, Wavefront files, Inventor files, and Paint geometry into Action. Also, you can create 3D text with custom beveling.

Camera The camera represents the 'eye' you are using to see the scene. The camera is used to “record” the scene in Action. By default, the camera consists of two objects: the camera eye and the point of interest (the coordinates the camera is 'looking' at).

Axis An axis is the element of an object or media that can be manipulated to determine the object's 2D or 3D space, position, and movement.

Schematic The Action schematic is an icon representation of the scene. It shows all the objects in the scene and their relationship with each other. The Action schematic is different from the Batch or Batch FX schematic (which is more of a process tree view).

Particles Action contains a particle system that can simulate environmental effects such as rain, snow, and many other user-defined effects.

Accessing Action From the Tools Tab

When accessing Action from Batch or Batch FX, you have the added benefit of being able to use the Workspace Media panel to add or organize clips and setups. You can also access Action through the Tools tab, allowing you to create modified clips that you can use as a rendered result in the workspace.

To access Action from the Tools tab:

1. From the Tools tab, click Action.
2. Select an option from the Input Mode box.

Select:	To:
Front Back Matte	Select source clips from the workspace. The clips are selected in the order front, back, and then matte.
MultiVersion	Select a multiversion clip. The first frame of the first segment is used to fill the beginning, and the last frame of the last segment is used to fill the end.
MultiVersion Fill Gap	Select a multiversion clip. Each gap is filled with black frames.
MultiVersion SelfKey	Select a multiversion clip. The last version is loaded into the background and the other versions are loaded as fronts. The mattes are created based on each front. Gaps are filled with the last frame from the preceding segment.
MultiVersion Self Gap	Select a multiversion clip with gaps. Each gap is filled with the last frame from the preceding segment. Mattes are created based on each front.
None	Enter Action with no media or back clip. From the Resolution menu that appears, choose a resolution, width, height, pixel aspect ratio, bit depth, scan mode, and frame depth. If Action already contains a setup, entering with the None option keeps the objects present in the scene and creates empty media placeholders in the Media list for each object. Press the `Alt` key when you click the Action button to automatically select the None option. Press `Alt` when you select your destination to open Action with all media and nodes deleted and all parameters (except for resolution) set to default.
Clear All	To reset all parameters (except for resolution), delete all nodes and media, and prompt you to select front, back, and matte clips. Press the `Ctrl+Alt` key when you click the Action button to automatically select the Clear All option.

Although you can use differing resolutions per media, you cannot specify clips of different resolutions to act as the front and matte clip of media (except for 10-bit, 12-bit, and 12-bit unpacked clips, which are interchangeable). An error appears in the message bar if the specified clips are not able to be loaded to the same media.

TIP To standardize some of your clip resolutions, you can resize them before entering .

3 Select the Action front, back, and matte clip or the multiversion video clip. If you selected None from the Input Mode box, proceed to the next step.

If you are selecting clips and not a multiversion clip, you can press `Ctrl` to load multiple fronts and mattes. The `Ctrl` key changes the order of clips to front and matte of media 1, front and matte of media 2, and so on. The back clip and destination are selected after the last media is filled or when the `Ctrl` key is released.

TIP Pay attention to the cursor colours to know which clips you are selecting (front = red cursor, matte = blue cursor, background = green cursor, and destination = white cursor).

4 Select the destination.

Action appears and the selected clips or tracks are loaded into the appropriate media. If Auto Image is enabled in the Action Setup menu, objects are automatically added to the schematic.

The composite of the front, matte, and back clips appear in the image window and their names are listed in the Media list. The Total Frames field defaults to the length of the longest clip loaded.

Loading from a Multiversion Clip

When you load from a multiversion clip, the versions from the selected multiversion clip are loaded from the bottom version to the top version. The last version is loaded as the background and subsequent versions are loaded as matte and front pairs. This means that the last version is loaded as the common back clip, the second to last version is loaded as the matte clip of media 1, the third to last version is loaded as the front clip of media 1, the fourth to last version is the matte clip of media 2, and so on.

NOTE If you select an even number of versions when loading from a multiversion clip, the second to last version selected is loaded as the front clip for its media, and the matte for this media is set to an internal white frame (of the same resolution as the front clip).

Loading from a Multiversion Clip with Selfkey or Self Gaps

When you load from a multiversion clip with Selfkey or Self Gaps, the versions from the selected multiversion clip are loaded from the bottom track to the top version. The last version is the background or back clip and subsequent versions are loaded as front clips for each media. The matte of each media is generated from the front clip. This means that the last version is loaded as the common back clip, the second to last version is loaded as the front clip of media 1, the third to last version is loaded as the front clip of media 2, and so on.

Gaps that are located at the start and end of a version are treated in two ways depending on which multiversion option you choose. If you select MultiVersion, each gap is filled with its preceding non-gap frame. If you select MultiVersion with Gap, each gap is filled with black frames.

About Using Floating Point Images in Action

You can use 16-bit floating point OpenEXR clips in Action. You can use clips of different resolutions on separate media, but you cannot specify clips of different resolutions to act as the front and matte clip of a specific media. Here are some things to keep in mind when working with 16-bit floating point clips:

- Action operations and settings that are supported with 16-bit floating point clips include 3d tracking, textures, displacement mapping, motion blur, blending modes, anti-aliasing, and depth of field.
- You can enable or disable colour clamping when working with 16-bit floating point clips in Action. The Colour Clamping button is located in the Setup menu's Rendering section.
- You can set the output resolution for your Action composite to 16-bit fp in the Frame Depth box located in the Setup menu's Resolution section.

Action Interface Overview

Action is divided into a number of menus and sections designed to help you quickly create your composites.

Changing Views

You can change the orientation to view a scene from many angles. This can help you set up motion paths, light sources, and camera angles more easily.

From the View box, select an option to set the view in the image window.



(a) View box

Most of the View box options have keyboard shortcuts displayed in the expanded list. Some of these keyboard shortcut options are also cycle options. For example, pressing `F5` repeatedly cycles through the animation channels, tracks, and info views.

You can also toggle ` (on the Tilde key) to alternate between the Schematic view and the previous view.

Accessing Action Menus

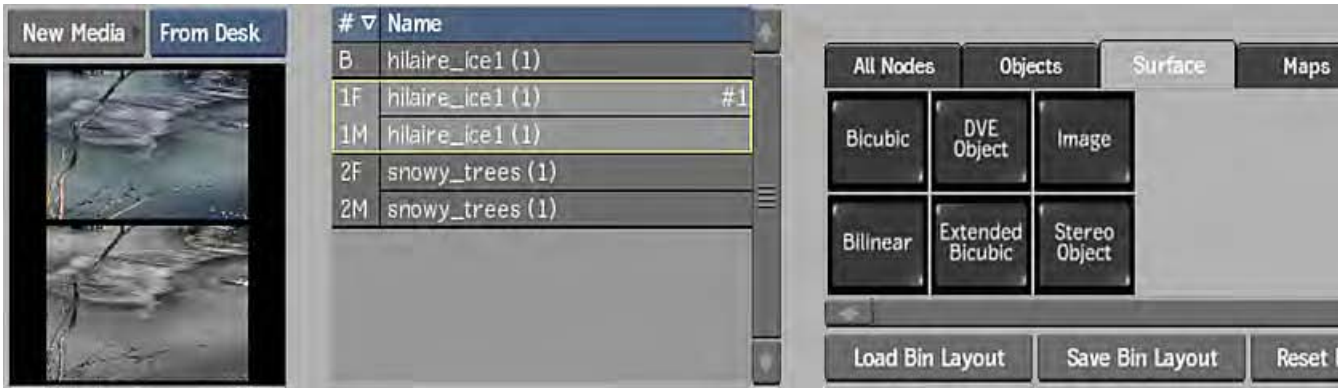
The left side of the Action menu houses a number of buttons that allow you to switch between the various Action sections and menus. Clicking any of these buttons opens the specific Action section, such as the Action Bin or Output menu.



Using the Node Bin

The Action node bin contains nodes classified by tabs. The All Nodes tab contains all Action nodes. The other tabs in the node bin allow you to create and customize bins. The node bin also contains an image proxy that displays a proxy of the currently selected media, as well as a mini Media list.

To access the node bin, click the Action Bin button.



Some nodes are duplicated in other bins according to a preset tab classification. For example, the Light node is found in both the Objects and ReLighting tabs.

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

To add a node from the node bin:

1. If applicable, select media from the mini Media list.
2. Do one of the following:
   - Drag the selected node (or image proxy) from the node bin and place it in the schematic. If the node is an image type node, an axis and an image are created and linked together.

- Drag the node (or image proxy) from the node bin and place it where you want it in Result view. For example, as you drag a Light node into Result view, it becomes active, so you can see its effect on the scene before placing it exactly where you want.
- Double-click a node (or image proxy). Depending on the node, it appears next to the last added object, or is attached to a selected node in the schematic. You do not need to be in Schematic view to add a node in this manner.

TIP You can frame and highlight all the nodes starting with the same letter by pressing the key corresponding to this first letter on your keyboard, while the cursor is sitting over a node bin. For example, pressing "L" while hovering the cursor above the All Nodes bin automatically highlights Lens Flare, Lens Texture, and Light nodes.

Customizing the Node Bin

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

You can customize any tab except the All Nodes tab.

To create a tab:

1. Click the plus sign tab.

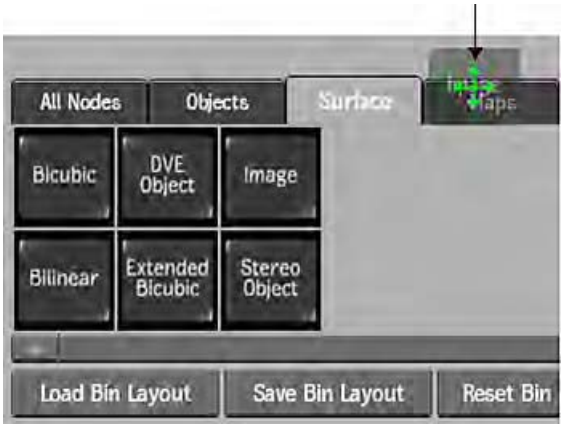


2. Name the tab in the keyboard that appears.

   NOTE You can create a maximum of 6 new tabs.

To copy a node to another tab:

1. Drag the node on top of the destination tab.
2. Release the cursor when it changes to a green crosshair.



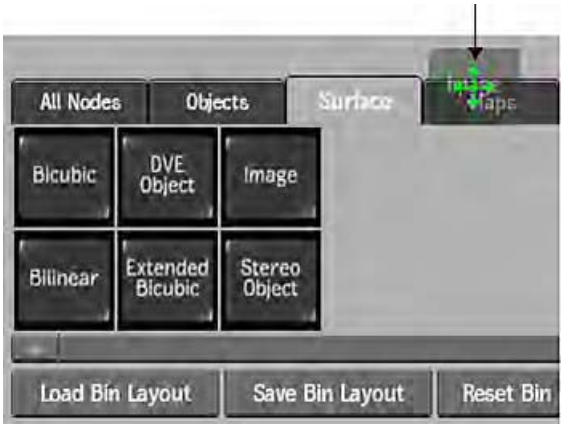
3. Click the destination tab when the standard yellow cursor reappears.

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

   NOTE Nodes cannot be duplicated within the same tab.

To move a node to another tab:

1. Press `Ctrl+Alt` and drag the node on top of the destination tab.
2. Release the cursor when it changes to a green crosshair.



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

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

   NOTE Nodes cannot be duplicated within the same tab.

To reorder a node within a tab:

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

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

To reset a bin to alphabetical layout:

1. With the applicable tab active, click Sort.

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

To delete a tab:

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

   WARNING There is no undo capability when deleting a tab.

   The entire contents of the tab are deleted.

To delete a node from a tab:

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

   WARNING There is no undo capability when deleting a node.

To rename a tab:

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

To reorder a tab:

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

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

To save a bin layout:

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

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

   Layouts are saved per user, not by project.

To load a bin layout:

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

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

Animating with the Channel Editor

Use the Channel Editor to animate the media, axis, surface, light, camera, and other properties of every object in the scene.

To open the Channel Editor and display the Action channels:

1. In Action, click Animation.
2. In the Animation menu, select Channel from the View box.

   The top-level folder in the hierarchy is the Scene, which provides the overall view of the animation. The Scene folder contains the Result Camera and folders of objects in the Action scene. Initially these are the media, motion blur, camera, axis, and image folders.

   If you add an object to the scene, the Channel Editor adds a folder to the channel hierarchy for the new object. For example, if you add a light, a Light folder is added in the channel hierarchy.

Action Channel Editor Reference

Some of the Action folders in the Channel Editor are described as follows.

Camera Animates the camera position and point of interest. It also contains channels for animating camera roll, field of view, and the near and far clipping planes.

When Free Camera is selected, the point of interest channels are replaced by rotation channels in the Channel Editor.

Media Animates media properties such as blur, crop, shadow softness. A Media folder is listed for each media in the scene.

Axis Animates axis properties such as position, rotation, scaling, and shearing.

Image, Bilinear, Bicubic, or Extended bicubic Animates surface properties such as material, offset, and displacement. The Material folder contains a shininess channel and folders for the specular highlight, diffuse, and ambient lighting.

Shadow Animates the shadow colour and shadow transparency.

Light Animates light properties such as intensity, falloff, spread, position, rotation, and colour.

TIP Selecting nodes in the schematic will automatically select the associated channels in the Channel Editor.

Most of the settings in the Animation menu are standard for all tools that support animation, but there are a few animation settings specific to Action, available from any Action menu (on the right side).

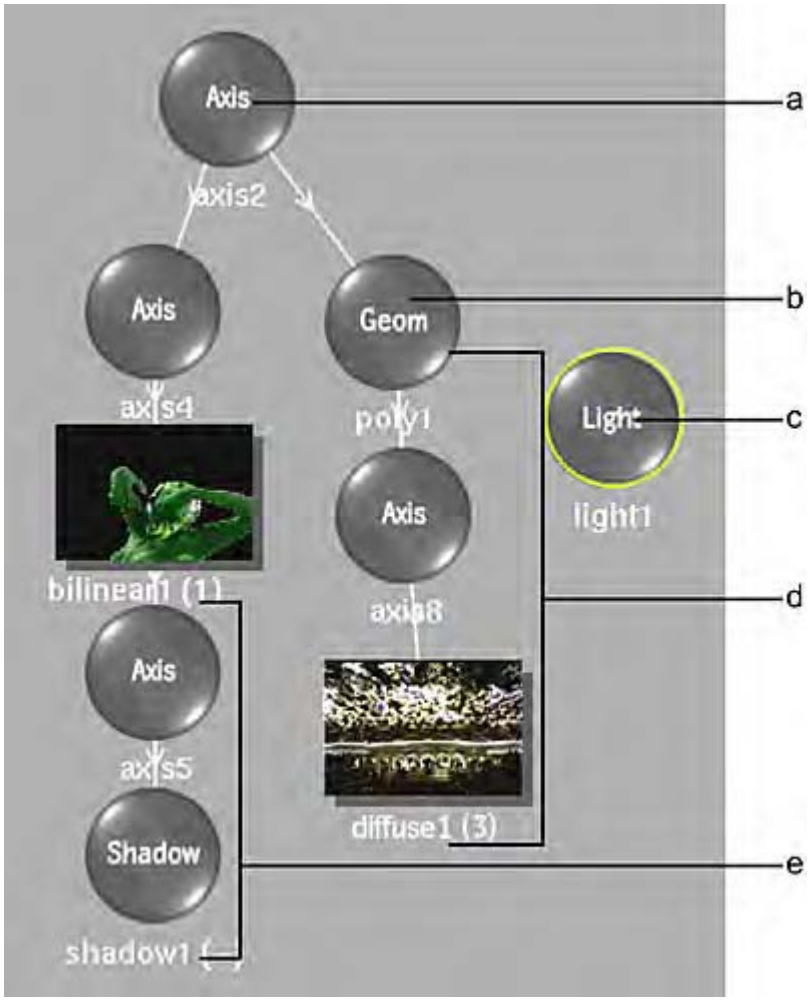
Auto Key button Enable to set a keyframe automatically each time you change a value at any frame.

Set Key button Click to set the current values for the selected channels in the current frame (when Auto Key is disabled).

Delete Key button Click to delete the selected keyframe.

About the Schematic and Menu Tabs

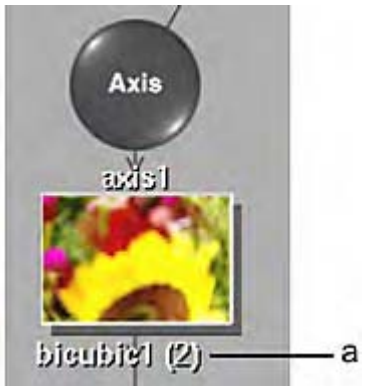
In Action Schematic view, a node exists for all objects in the scene, for example, shadows, lights, and texture maps.



(a) Axis object is the parent of axis 4 and poly1 (b) 3D model (c) Light source (d) Diffuse texture applied to the 3D model (e) Drop shadow of bilinear1

Here are some hints when working in the schematic with some of the various node types.

Surface Nodes (Bilinear, Bicubic, and Extended Bicubic) When you add one of these surface nodes to the schematic, the node name is displayed with a number in parentheses. The number indicates the media applied to the surface. For example, a bicubic object labeled (2) shows that the bicubic uses the clips from Media 2.



(a) Media 2 is used by the bicubic surface.

For more information on the relationship between media and surfaces, see Adding Surfaces (page 458).

Camera Node The camera node appears in Schematic view by default and you can link it to any image. Use the camera node to rotate the camera about its own axis, and parent other nodes including shadow, texture, and geometry nodes.

Shadow and Texture Nodes Shadow and texture nodes each display a single number in parentheses beside the name that indicates the media used for the shadow or texture. For example, a shadow labeled (2) shows that the shadow uses the matte from Media 2.

Source Nodes Source nodes are used as part of an advanced schematic structure that separates the media's matte and front so that each clip can be animated individually. You can also use sources to create complex compositing effects such as nesting.

TIP Adding many nodes and connections can quickly make for a disorganized schematic. To solve this, press `Alt+T` to reorganize the schematic.

Node Settings

Object Node Name field Displays the selected object node name. Editable.

Previous Node button Scrolls to the previous similar node.

Next Node button Scrolls to the next similar node.

The right side of Action also includes other settings to help you work with nodes in the schematic.

Solo button Enable to solo an object or a branch (object and its children), depending on what is selected in the Selection Mode box. See Soloing Objects (page 422).

Selection Mode box Make a selection to copy, delete, hide, or reset.

Clear All button Available in the Action Setup menu. Resets all parameters, and deletes nodes and media as well. Press Alt to bypass confirmation

Reset All button Available in the Action Setup menu. Resets all parameters but does not delete nodes or media. Press Alt to bypass confirmation.

Populating Menu Tabs of Selected Objects

Depending on what type of object is selected in the schematic, the tabs in the Object menu are populated based on different rules, as illustrated in the following examples.

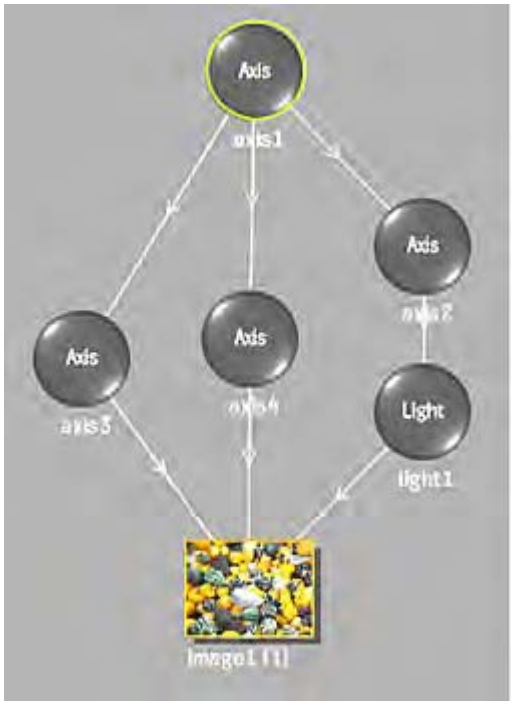
An object's name appears on the Object menu tab as well as beneath its node in the schematic and as a folder in the Channel Editor.

Axis selected

The Axis menu of the selected axis appears on the left side of the Object menu, and a limited number of the children objects' menus appear on the right side of the Object menu. The children objects are identified by scanning the hierarchy of the schematic from top-to-bottom (starting at the selected axis). The hierarchical scanning stops for any given branch when a non-axis object is encountered.

The order of tabs is determined on a per branch basis; that is, all of the tabs of one branch are listed before moving to another branch, starting with the highest levels in the parenting hierarchy (lowest index levels).

For example, in the following schematic, Axis 1 is selected.



The tabs in the Object menu appear as follows. The image1 tab appears in light blue to signify that multiple objects are connected down to it. Use the tabs to switch between menus within the Object menu. The Camera tab appears in orange as the first tab on the right side of the Object menu, and is exempt from the tab population rules.



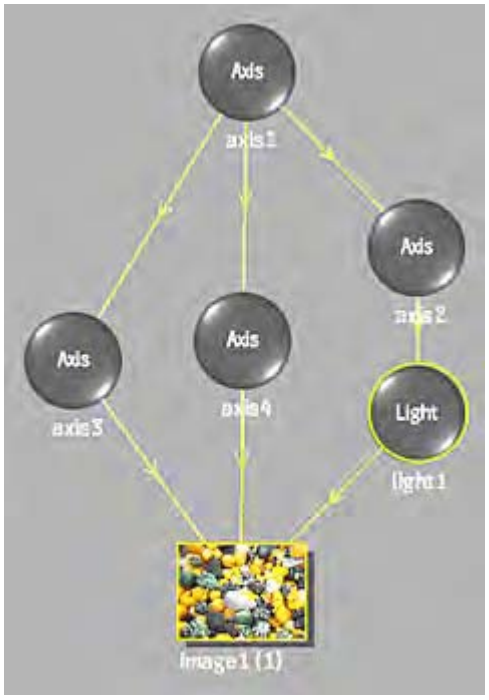
NOTE If there are more than five tabs on either side of the Object menu, use the arrows beside the tab names to navigate to the desired tab.

Other object selected (no “axis attributes”)

These objects include generators, bouncers, deformations, surfaces, texture maps, shadows, geometries, and 3D text. When one of these objects is selected in the schematic, an ascending (bottom-to-top) scanning of branches is performed. The hierarchical scanning stops for any given branch when an axis or object with axis attributes is encountered.

The menu of the selected object appears on the right side of the Object menu, and a limited number of the parent objects' menus appear on the left side of the Object menu.

For example, in the following schematic, Image 1 is selected.



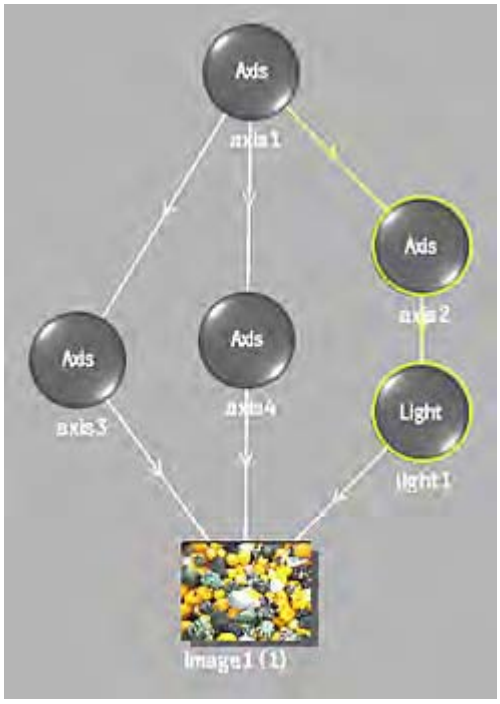
The tabs in the Object menu appear as follows. The image1 tab appears in light blue to signify that multiple objects are connected down to it. Use the tabs to switch between menus within the Object menu. The Camera tab appears in orange as the first tab on the right side of the Object menu, and is exempt from the tab population rules.



Object with “axis attributes” selected

These types of objects include lights, animators, projectors, and cameras. These objects trigger a different scanning behaviour depending on their position within the schematic hierarchy. If the selected object is the first of its branch (top of the hierarchy), it inherits the tab population rules of an axis, that is, descending branch scanning. If the selected object is not the top object of its branch, it inherits the tab population rules of other objects, that is, ascending branch scanning.

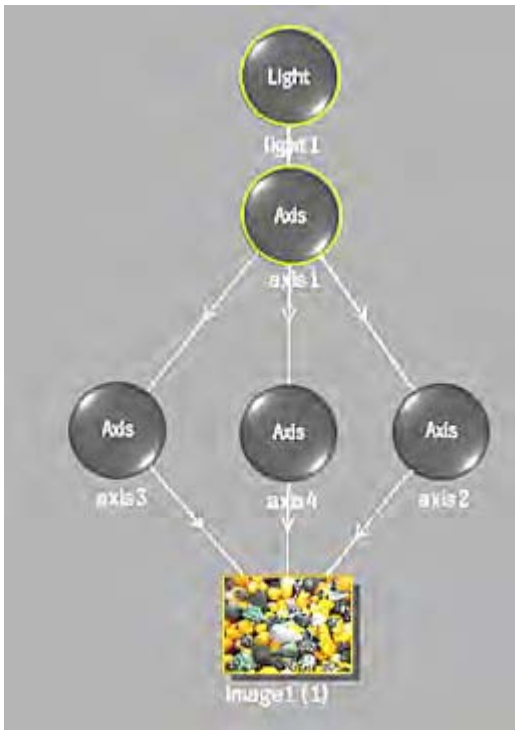
For example, in the following schematic, light 1 is selected.



Since Light 1 is not the top object in its branch, the tabs in the Object menu appear as follows.



In the following schematic, light 1 is moved to the top of the branch.



The tabs in the Object menu appear as follows. The image1 tab appears in light blue to signify that multiple objects are connected down to it. Use the tabs to switch between menus within the Object menu.

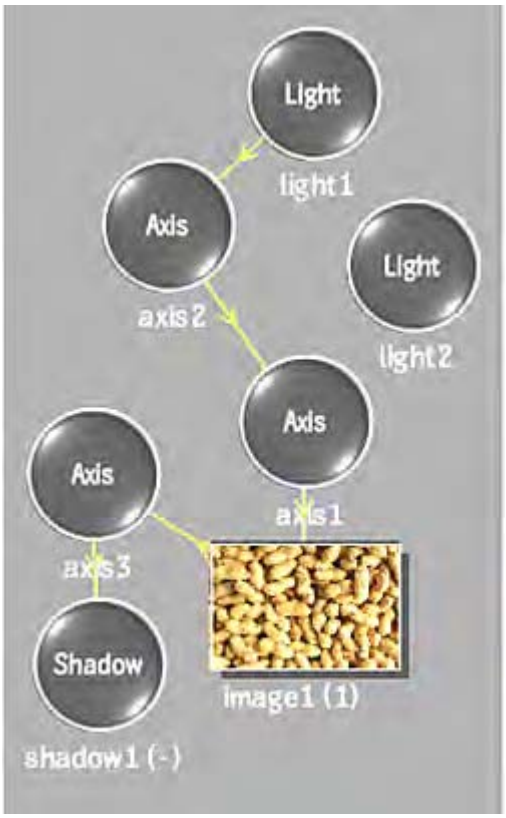


NOTE The Camera tab appears in orange as the first tab on the right side of the Object menu, and is exempt from the tab population rules. If a camera node is selected in the schematic, the special Camera tab does not appear, and the normal tab population rules apply.

Using the Object Menu for Multiple Selections

You can select multiple nodes to change some of the settings in the Object menu for all objects of the same type.

Objects that support multiple selection and changing of field settings are axes, images, shadows, and lights. For example, in the following schematic, you can select all of the nodes.



The tabs in the Object menu reflect the multiple selections.



In the Object menu, you can relatively change any of the available fields. For example, if the X Position for Axis1 and Axis2 was originally 100, and for Axis3 was 200, and you drag the X Position slider under the Mult-axes tab to 50, Axis1 and Axis2 are now set to 150 and Axis3 is set to 250.

NOTE Because the field values for individual objects can be different, the values in the multiple selected fields display as default values in the Object menu. When you select an individual object and view its Object menu, you can see the true field values.

Connecting Action Nodes

In Action, you can create complex animations where movements applied to one node are passed down to all connecting nodes.

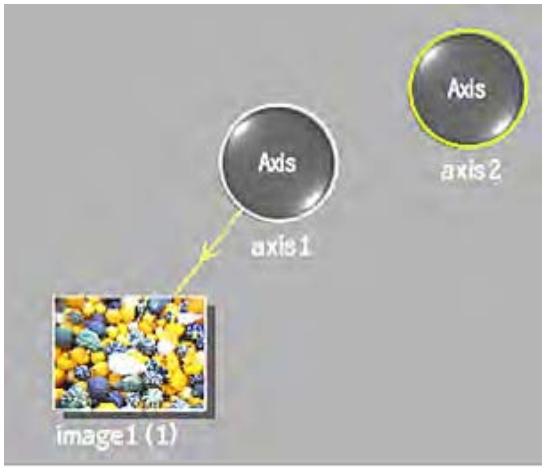
You can add an axis to the scene by itself, then make it the parent of another node. Use this method of parenting additional axes to create complex animations.

For example, create a cube of surfaces by parenting three additional axes to the same surface. Each axis that is parented to a surface places an additional surface in the scene. By changing the position and rotation of each axis, you can create a cube. If you parent the axes with another axis, you can control the position, rotation, scale, and shear of the cube.

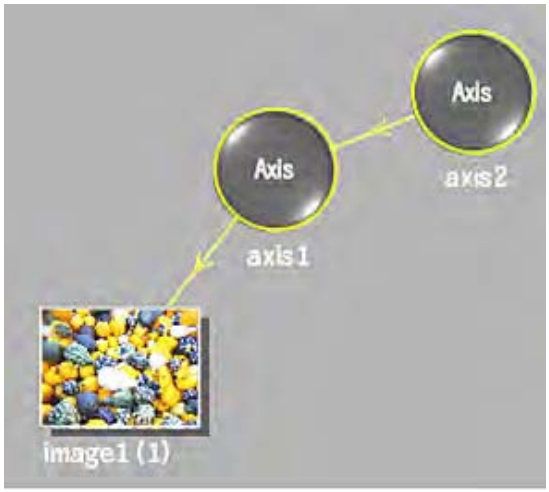
To create a branch:

1. Add an axis (axis2) to the scene.
2. From the View box, select Schematic. The Schematic view should be similar to the *Before* figure.
3. Do one of the following:
   - In the Edit Mode box, select Conenct and drag the cursor from the edge of the parent node to the node that will be its child.
   - Press `Shift` and drag a parent node over another node that will be its child. With this method, the Edit Mode box does not have to be set to Connect mode, and can remain in Select mode, for example.
   - In the Action Prefs menu, enable Auto Parent in the Schematic section, and then in the schematic, drag the cursor from the edge of the parent node to the node that will be its child. With this method, the Edit Mode box does not have to be set to Connect mode, and can remain in Select mode, for example.

   Axis2 becomes the parent of axis1, as shown in the *After* figure.



Before: The schematic shows axis1 as the parent of image1



After: Axis2 is made the parent of Axis1 using Connect mode

Any transformations applied to axis2 are applied to axis1 and its surface (image1). If axis1 has any transformations, they are added to the transformations from axis2. For example, if axis2 is set to 500, 100, 0 and axis1 is set to -50, 20, -30, the positions are accumulated and applied to the surface. In this case, Image1 is positioned at 450, 120, -30.

TIP You can override the transformations passed from a parent to a child by enabling the Free button in the Axis menu.

Inserting a Node Between Connected Nodes

1. Do one of the following:
   - If Auto Insert is enabled in the Setup menu Schematic settings, drag a node and navigate to the link between two connected nodes. The link is highlighted in orange.
   - If Auto Insert is disabled in the Prefs menu Schematic settings (this is the default), press `Shift` then drag a node and navigate to the link between two connected nodes. The link is highlighted in orange.
2. Release the node.

   The node is inserted, and linked to the two previously connected nodes.

Mimicking, Copying, and Duplicating Objects

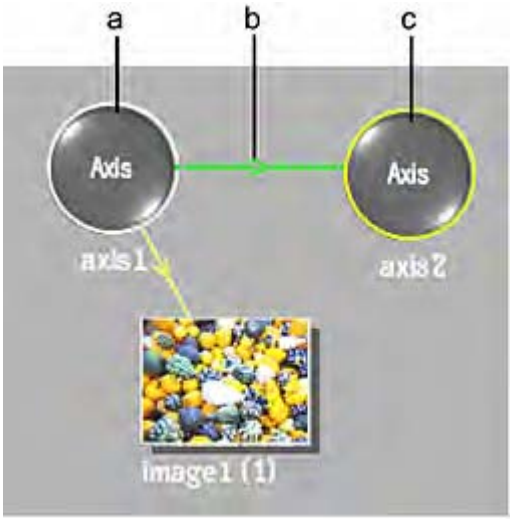
Similar to working in the Batch or Batch FX schematic, you can mimic, copy, or duplicate objects of the same type in the Action schematic.

Using Mimic Link

To create a Mimic link between Action objects:

1. From the Edit Mode box, select Mimic Link.
2. In the schematic, drag between two similar object types, for example Axis to Axis. You can also link different map types, such as from a diffuse map to a reflection map.

   A green arrowed dotted line indicates that the objects are linked as duplicates. The direction of the arrow indicates which object is the master.



(a) Originating object (b) Mimic link (c) Linked object

3. Change the settings of any node.

All of the settings applied to the original object are automatically applied to the linked object. Once objects are linked, any settings applied to either object are applied to both. When linking different map types, only the settings found in the Texture tab are mimicked.

The originating object can link to multiple objects, but only one object can be the originating link.

NOTE To remove the link between mimicked objects, drag the cursor across the green line that joins the two objects. Each object keeps the settings that were applied while they were linked.

Duplicating Objects

To duplicate objects:

1. In the schematic, select a node, or group of nodes.
2. Right-click the selection, and choose Duplicate.

   A duplicate of the node or nodes (with all node settings) appears in the schematic.

Copying Objects

To copy objects:

1. In the schematic, select a node, or group of nodes.
2. Right-click the selection, and choose Copy.

   A copy of the node or nodes (with all node settings) is added to the clipboard.
3. Navigate to the location in the schematic (or another Action schematic) where you want the copied node or nodes to reside, right-click and choose Paste.

   A copy of the node or nodes (with all node settings) appears in the schematic.

Soloing Objects

Use the Solo button to hide all other objects except the selected object. Using Solo is useful for identifying an object in a scene with many objects, without having to hide all of the other objects.

To solo an object:

1. In the schematic, select the object that you want to solo.
2. From the Selection Mode box, choose Selected.
3. Enable Solo.

   All other objects in the scene are hidden, regardless of whether they are set to Hide or not. When Solo is disabled, the Hide settings are restored.

   TIP You can leave Solo enabled and select different objects in the schematic to view each object separately.

To solo a branch:

1. In the schematic, select the parent of the branch that you want to solo.
2. From the Selection Mode box, choose Branch.
3. Enable Solo.

   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.

Grouping or Hiding Objects

You can collapse branches in the schematic into a group to reduce clutter in the schematic. You can also temporarily hide an object or objects in the schematic to see different results, for example.

To group a branch:

1. Select an object with children in the schematic view.
2. Right-click the parent object, and choose Group to collapse the selected object and all its children into a group.

   A blue group node appears in the schematic to represent the entire group.

   NOTE Collapsed groups cannot act as parents, but you can parent nodes to a group.

3. To uncollapse the group, right-click the Group node, and select Ungroup.

NOTE Groups created in the Schematic view are not related to groups created in the Priority Editor.

To hide and object or objects:

1. Select and object or multiple objects in the schematic view.
2. Right-click the object, and choose Hide.

   (hidden) appears after the object name in the schematic.
3. To unhide the object, right-click the object, and select Show.

Setup and Processing Options

Use the Action Node Prefs menu to customize your Action display and to access tools and guides.

When accessing Action through the Tools tab, these options are available from the Setup button in Action.

Use the tabs in the menu to switch between the different options.

Rendering Tab

Resolution Settings

You can set the rendering resolution of clips that are output in Action. For example, if you are working in an NTSC 8-bit project, the default output is NTSC 8 bits. However, if you want to change the output resolution to HD (1920x1080), use the Resolution menu to change the values so the final outcome will be 1920x1080.

You can choose between progressive or interlaced when rendering at video resolution. Action media automatically adjust the rendering mode of each clip. This inherent awareness also makes it possible for Action to properly mix field-based HD / PAL with NTSC and not have dominance problems.



Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

Once you make a selection from the Apply and Scale box, the resolution is set for processing clips, and is remembered between Action sessions.

TIP 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.

Width field Displays the custom width resolution of the clip. Editable.

Height field Displays the custom width resolution of the clip. Editable.

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

Aspect Ratio field Displays the custom render/output aspect ratio. Editable

Frame Depth box Select the render/output frame depth of clips.

Scan Mode box Select the scan mode of clips.

Apply and Scale box Select whether to apply or apply and scale the defined resolution settings. All subsequent processes will use the new render settings.

Select:	To:
Apply	Specify the output resolution.
Apply + Scale	Specify the output resolution and scale a scene to the defined resolution. Use this option if you are working with a low resolution of an Action setup from a previous version of Flame Premium. The scaling is applied to geometries, axes, lights, and cameras as well as their coordinates. Their positions are scaled accordingly. Flame Premium automatically scales textures and images.

NOTE If you do not change the Resolution settings, rendered results and camera settings default to the values set for the current project (set when you created a new project).

Stereo Settings



Reset To Stereo Mode button Enable to clear any previous stereo settings in Action, and reset all stereo settings to their defaults.

In the Stereo mode, these are the default settings when you enter Action with the Front/Back/Matte input option using stereo clips:

- A 3D camera (stereo camera) is created and the default camera is hidden.
- In the 3D Camera menu, the Result Camera is set to the stereo camera.
- In the Output menu, the Mode is set to Stereo and the Camera is set to Result Cam.
- A stereo object is created with the clips you selected.

NOTE If you enter Action using mono clips, the default camera is automatically created. If you need a stereo camera, you must add it manually.

Rendering Settings

Use the Rendering settings to improve the final output quality of your image.



Z-Buffer box Select an option to determine whether the distance from the camera eye is considered to determine the order in which objects are rendered.

Wireframe button Enable to display each surface and 3D model as a wireframe.

This is useful if interaction with Action becomes slow because of many surfaces and 3D models. Convert objects to wireframe when you want to speed up processing or rendering times. Because lighting and textures are not computed in wireframe mode, interaction time and rendering times are reduced.

Shading button Enable to light up the scene using added light sources. When Shading is disabled, no lighting effects appear in the scene; surfaces and 3D models appear flat.

Enable Shading for:

- Light sources
- Ambient or diffuse lighting for surfaces
- Specular highlights for surfaces and 3D models

Clamp Colours button Enable to clamp colour and luminance in the 16-bit floating point processing pipeline.

Anti-Aliasing Settings



Software Anti-Aliasing Sample box Select a software anti-aliasing sampling level.

The jagged lines that often occur along the edges of diagonal or curved lines when processing high-frequency images such as text are the result of aliasing. You can increase or decrease the anti-aliasing sampling level (up to 64 samples). Higher values yield smoother results at the expense of processing time.

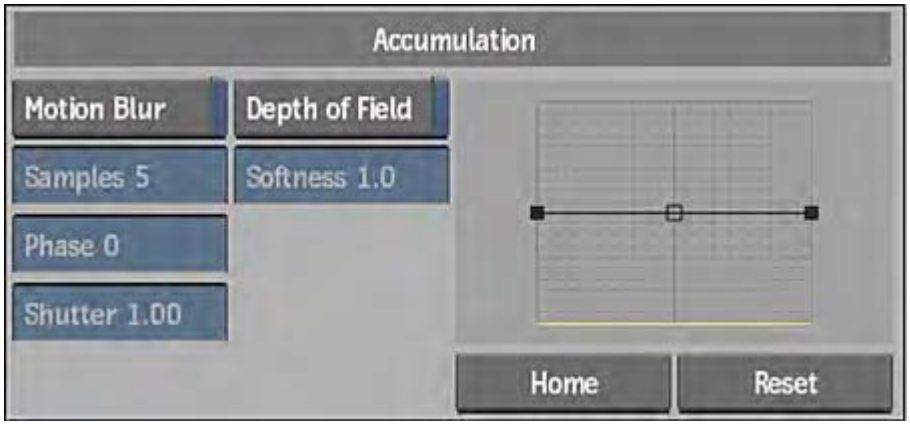
Anti-Aliasing Softness field Displays the softness value of the software anti-aliasing sample. Editable.

Hardware Anti-aliasing Sample box Select a hardware anti-aliasing sampling level to accelerate edge anti-aliasing with no performance penalty. Available values are dependent on graphics card and project graphic bit depth. Hardware anti-aliasing also gives anti-aliasing during normal interaction instead of only while rendering.

You can combine hardware anti-aliasing level with software anti-aliasing to obtain the desired level of image quality. For example, with hardware anti-aliasing set to 4 samples, and with 4 samples of software anti-aliasing selected, your results should be similar to selecting 16 samples of software anti-aliasing, but with a processing time much closer to that of 4 samples. You should experiment with different combinations to determine what works best for you.

Accumulation Settings

Use the Accumulation settings to define motion blur properties.



Global Motion Blur button Enable to use motion blur. Once enabled, specific Action object Motion Blur buttons can be enabled or disabled. See About Motion Blur (page 653) for specific examples of creating motion blurs.

Samples field Displays the quality of motion blur and the depth of field produced by the number of samples taken at each frame. Editable.

Increasing this value causes the processing time to increase linearly and affects the quality of the depth of field. The number of motion blur samples is multiplied by the number of anti-aliasing samples. To reduce the total number of passes made for each frame, reduce the level of anti-aliasing when motion blur is enabled.

Phase field Displays the frame that motion blur is based on (before or after the current frame). Editable.

A value of -100 places the motion blur before while a value of 100 places the motion blur after. A value of 0 is centred, which evenly distributes the motion blur. The default value is 0.

Shutter field Displays the duration of motion blur at each frame (essentially the number of frames that the shutter is open). Increasing this value does not increase the processing time. Editable

---

TIP You can animate the Global Motion blur button, as well as the Phase, Shutter, and Samples fields. They can be found in the Channel Editor under the *motion_blur* folder.

Depth of Field button Enable to use the camera's depth of field.

Depth of Field Softness field Defines the softness of the depth of field. A low value yields a sharp falloff between focused and unfocused regions. Editable.

Motion Blur curve Controls the sample weight over the scope of the motion blur.

Home button Resets the position of the motion blur curve after panning.

Reset button Resets the motion blur curve.

Preferences Tab

Surfaces Settings



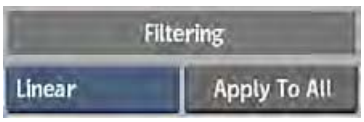
Default Resolution field Displays the default geometry resolution (number of polygons) of surfaces. You can also change the geometry resolution of specific surfaces in the Surface menu. Editable.

The lower the value, the better the resolution and the greater the processing time required to interact with the image. A value of 1 creates one polygon per pixel on a surface, affording accurate displacement and lighting.

Use this field when Exploding Objects and Surfaces (page 762) with a particle generator and when using hardware texture mapping.

Apply Resolution To All button Click to apply the geometry resolution in the Resolution field to all surfaces. Since you can change the resolution of specific surfaces in the Surface menu, you can use this button to re-apply the default resolution to all surfaces.

Filtering Settings



Default Filter box Select the type of filtering to be set as the default when creating surfaces and maps.

Apply To All button Click to apply the default filter to all existing surfaces and maps.

Miscellaneous Settings

Animation Update button Enable to update properties such as position, rotation, and colour in the scene. When disabled, animated objects do not move, but keep the position of their current value. Disable to copy keyframe values from one frame to another.

Play Lock button Enable to update the animation settings according to the frame or timebar position as you move through the clip while keeping it locked at the current frame.

When Play Lock is disabled and you use > or < to play the resulting clip, each frame is loaded and displayed in sequence in the image window.

Front Only button Enable to display only the front media in the Media menu. Disable to display both the front and matte media.

Auto Image button Enable to add an image node and axis automatically in the schematic when new media is added.

Shift-Snap button Enable to use the Shift key as a snap-to-surface modifier key. Hold the `Shift` key when selecting an axis to snap the selected axis to the surface of the underlying geometry. Not available when in Top, Front, or Side views.

Matte Channel box Select which channel is displayed as transparent by default. This can be useful for a multichannel clip to display only the Red channel matte, for example.

Display Settings



Icons box Select object icon (axes, borders, control points) display options. The selected option displays icons only for the object currently selected in the scene.

Icon Transparency field Displays the transparency level for icons in the scene. Editable.

Ruler button Enable to display the ruler in the scene area. Use the arrows at each extremity of the ruler to place the beginning and end of the ruler anywhere in your scene view.



TIP Use the `Shift` key to snap the ruler into place either horizontally or vertically.

Ruler Define button Enable to define real unit measurements, such as feet, metres, or inches, instead of pixels. All camera distance or axes measurements thereafter use the defined scale.

Ruler Length field Displays the length and measuring unit (feet, metres) to use in Action. When you enter a length, click Define again to apply the new scale to the ruler and all pertinent fields, such as position, rotation, and scale. Editable.

TIP Hold the `Ctrl` key and click Define to reset the ruler's scale.

Grid Colour pot Displays the custom colour for the grid. Editable.

Grid box Select the type of grid to display in the scene. Use to position objects in the scene more accurately.

Select:	To:
Grid Off	Disable the grid.
Grid XY	Use a grid constructed on the X and Y planes.
Grid XZ	Use a grid constructed on the X and Z planes. The XZ grid is visible only when the camera is moved from its default position.
Grid YZ	Use a grid constructed on the Y and Z planes. The YZ grid is visible only when the camera is moved from its default position.

NOTE The Action grid is independent from the global grid in the Grids and Guides menu.

Ortho Near field Displays the value of the near view in the image window when using Camera or an orthographic view. Editable.

Ortho Far field Displays the value of the far view in the image window when using Camera or an orthographic view. Editable.

Schematic Settings



Auto Parent button Enable to automatically parent nodes in the schematic. Press Alt and drag a node in the schematic to disable Auto Parent temporarily.

Auto Insert button Enable to automatically insert a node when dragged between two connected nodes. When disabled, press Shift to auto insert.

Snap To Grid button Enable to position objects with precision in the scene. When you move an object in the scene, the object is automatically aligned to the snap grid.

NOTE When enabled, this snap only applies to the grid defined in Action. The grid defined from the Grid & Guides menu remains visible, but its snap is overridden as long as the Action Snap is enabled.

Schematic Transparency field Displays the level of transparency of unselected nodes in the schematic. Editable.

Display Information box Select what clip information is displayed in the schematic.

Proxy Update button Enable to automatically update proxies in the schematic. Interaction is slower when enabled. When disabled, Action updates proxies when you switch views. You can also update proxies by clicking Update.

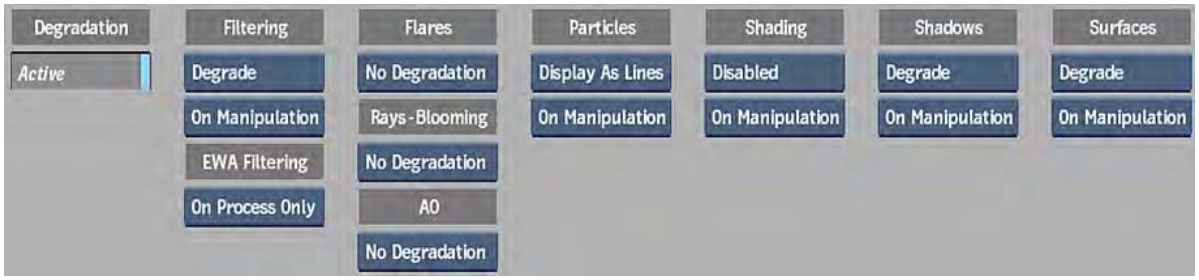
Output Naming Settings

Action Name button Enable to allow newly created outputs in the Output List to inherit the name of the Action node. If disabled, it will use a default output [number] for additional outputs. You also can rename any output manually by clicking the Rename button below the Output List. Once a setup is saved, the output will use the setup name when processed.

Append Type button Enable to append the type of output to the name of a processed file.

Adaptive Degradation Tab

Use these settings to temporarily deactivate taxing operations during interactive manipulations. These settings do not affect the final renders.



Active button Enable to activate adaptive degradation display settings. Use to prevent slowdowns of the image window display when changing Action settings. Press Ctrl+D to activate or deactivate degradation settings.

You'll see an icon displayed at the bottom right corner of the image window when degradation is active in Action.



Synchronize All button Enable to synchronize all Batch FX Action nodes with the Adaptive Degradation settings of the current Action node.

Filtering Degrade box Select whether to degrade filtering selections in the image window when changing any Action settings.

Filtering Update box Select whether to always degrade filtering selections, or on manipulation only.

EWA Filtering box Select whether to apply EWA or EWA+Linear filtering on maps (if selected in the Filter box) only when processing (On Process Only), or all the time when working in Action (Degrade).

Flares Degrade box Select whether or not to degrade Lens Flares.

Rays Degrade box Select whether or not to degrade Rays.

Blooming Degrade box Select whether or not to degrade Blooming.

Particles Degrade box Select a particle quality display setting.

Particles Update box Select whether to always degrade particles selections, or on manipulation only.

AO Degrade box Select an ambient occlusion quality display setting.

Shading Degrade box Select whether to degrade shading selections in the image window when changing any Action settings.

Shading Update box Select whether to always degrade shading selections, or on manipulation only.

Shadows Degrade box Select a shadow quality display setting.

Shadows Update box Select whether to always degrade shadows selections, or on manipulation only.

Surfaces Degrade box Select a surface quality display setting.

Surfaces Update box Select whether to always degrade surfaces selections, or on manipulation only.

Motion Blur Degrade box Select whether or not to degrade Action motion blurs in the pipeline.

Software AA Degrade box Select whether or not to degrade Action software anti-aliasing in the pipeline.

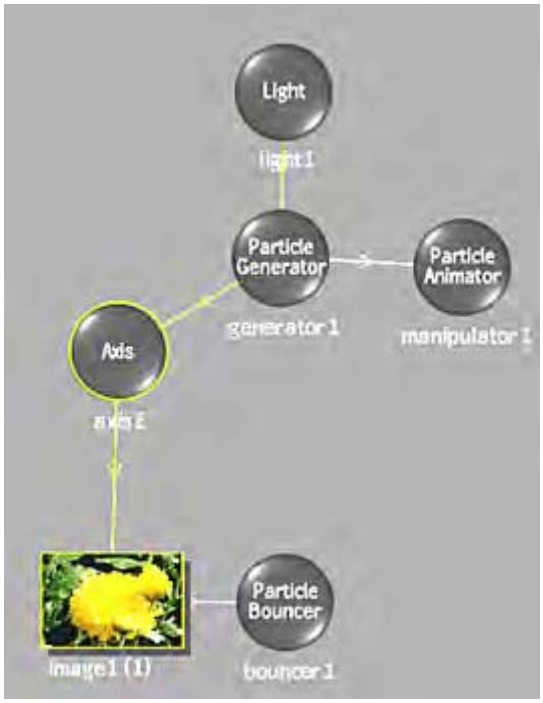
Depth of Field Degrade box Select whether or not to degrade Action depth of field in the pipeline.

NOTE When Adaptive Degradation is active for a Batch FX Action node, you'll see an icon over the node in the schematic, and in the bottom right corner of the image window in result view.

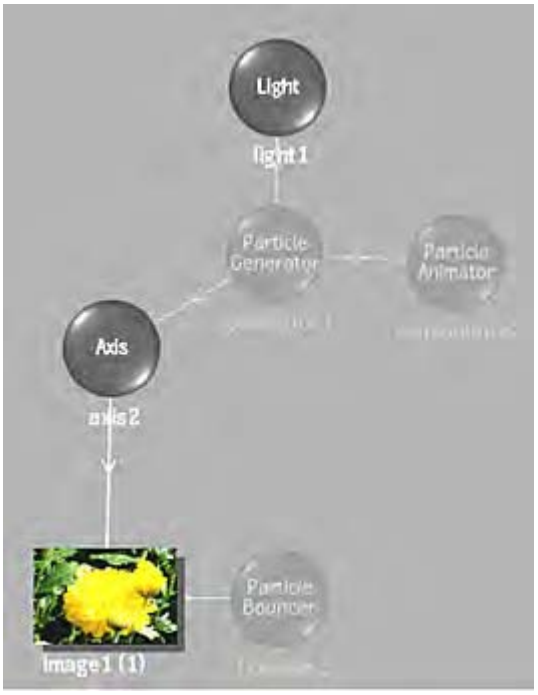
Saving, Loading, and Importing in Action

You can save, load, and delete Action setups, and import Photoshop files into Action.

When importing a setup that contains non-supported objects (such as particles and deformations in Smoke), these objects are displayed as greyed out nodes in the schematic, and greyed out menus in the Object menu. These greyed out objects are read-only objects that can be viewed (in the image window and the Object menu), but not modified or linked to or from. In Channels view, you can view the parent channel for the non-supported object, and in Tracks view, you can slide or stretch the parent channel. If you modify the setup and resave it, you can then open it in the application that supports the objects, and modify all objects.



Schematic of Flame setup



Schematic of same setup opened in Smoke

To save an Action setup:

1. In Batch or Batch FX, select the Action node. Click Save Node (found next to the node name field). If you entered Action through the Tools tab, you can click Save directly in the Action menu.

   The file browser and Save menu appear.

2. From the Save Action box, select the format for saving the setup.

Select:	To save:
All	References to all clips in the Media list.

Select:	To save:
Selected Objects	The selected objects, their media and media settings. Any expressions on channels of selected nodes are baked to the channels before saving, and any Mimic links to or from non-selected nodes are removed before saving. If a node is selected that is mimicked from a non-selected node, though, the animation channels are copied from the node before saving.
Raw	Selected animation channels as a user-readable ASCII file. In the file, each line corresponds to one frame, and each column consists of one frame number and the value of the animation curve at that frame. All saved information starts at frame one. A file with the extension *.raw* is saved in the *.../action* directory by default. At least one channel must be selected in the Channel Editor.
Text	The current text settings, including font, character size, kerning, italics, depth, beveling curve, and text string properties, all of which can be loaded in another Action session. A file with the extension *.atext* is saved in the *.../action* directory by default. A 3D text node must be selected in the schematic.
Preferences	The current Action settings as user preferences. A file with the extension *.pref* is saved in the */usr/discreet/user/<product_name>/<user_name>/action/pref* directory.
Defaults	The current Action preferences as Action's new default settings. To restore Action's factory default settings, select Factory Defaults in the Load menu.
MultiVersion	A multilayer setup for all Action media and save a multiversion clip in the current library.

3 Type a name for the setup file and click Enter.

The name appears in the Name field.

By default, the setup is saved in the */usr/discreet/project/<project_name>/action* directory. Using the file browser, you can save setups to the directory of your choice.

4 Once you have saved an Action setup, you can quickly resave it by clicking Save again.

TIP If you accessed Action from the Tools tab, you can click Revert to revert to the last saved setup. All changes made since the previous Save operation are undone.

To load a setup in Action:

1 In Batch or Batch FX, select an Action node. Click Load Node (found next to the node name field). If you entered Action through the Tools tab, you can click Load directly in the Action menu.

The file browser and Load menu appear.

NOTE When you access the file browser through Load, you also have the option of deleting existing setups.

2 In the Load box, select the format for loading the setup.

Select:	To:
All	Load the clips in the selected setup into their corresponding media. The media in the Media list is replaced with the loaded clips. If a clip cannot be found, Action searches for it and loads it automatically to Action. If the clip still cannot be found, Action displays

Select:	To:
	the missing clip name in red in the Media list. A surface that uses a missing clip appears as an outline in the image area and is shown in red in Schematic view.
No Clips	Load a setup without its clips. The current media in the Media list remains the same.
Add Nodes+Media	Add nodes and media from the setup file. This option appends the schematic from the setup file to the current schematic, and wherever possible, media from the setup file fills empty slots in the Media list. Remaining media from the setup file is appended to the end of the Media list. Enable Load Cameras if you want to include the cameras saved with the setup.
Add Nodes	Add only the nodes from the setup file. This option appends the schematic from the setup file to the current schematic. Enable Load Cameras if you want to include the cameras saved with the setup.
MultiVersion	Load a multiversion setup. Loads front, matte, back, and background video versions from an entire clip. Replaces all media.
Raw	Load raw animation data to a selected channel in the Channel Editor.
Text	Load the text setup files. The text settings are loaded into Action's Text menu.
Preferences	Load a file containing Action preferences.
Factory Defaults	Load original Action default settings. Selecting this option prompts you to confirm that you want to restore factory defaults and returns you to the Action menu.

3 Select the setup you want to load.

NOTE Sample Action setups are provided in the *~/examples/action* directory.

Importing Photoshop Files into Action

You can import Adobe Photoshop® format files into Action without having to convert them into a TIFF or other format. Once imported, the PSD file keeps its inherent layer structure, which can be used or modified within Flame Premium.

You also have the option of automatically importing all the layers at their native resolution. Each layer is parented by an axis that gives it the correct offset in the X and Y axes. Hidden layers are imported, but will remain hidden. Photoshop blend modes are maintained for each layer.

To import a PSD file into Action:

1 From the Action Node bin, double-click the Import node.

The file browser appears.

2 From the Import Type box, select the Photoshop format.



3 Select a *.psd* format file.

4 Click Load.

The Photoshop file is loaded into Action. Each layer is parented by an axis, and blend modes are maintained. The Photoshop files are added to the media list and to the Batch or Batch FX Sources folder (or to the desktop destination you set when entering Action through the Tools tab).

About Rendering Outputs from Action

Use the multi-pass rendering capabilities of Action's Output menu to manage your outputs. Options are available to help you set up and prepare many different types of outputs, much as you would with render passes in a 3D application. You can process multiple outputs at once, from your complete Action scene, to specific Shadow or Z-Depth outputs, for example.

Action Output Workflow

When using the Action Output menu, you usually follow the options from left to right.

Step:	Action:
1	Use the Outputs List (page 435) to add, copy, delete, and rename outputs.
2	Set Output (page 436) and Render (page 438) options per output in the Output List.
3	Select objects (page 439) to output.
4	Render your outputs from Action (using the options in the Render button and dropdown list).

To access the Output menu, click the Output button.

Outputs List Settings

Use the Outputs List to organize the outputs you want to process. You must have at least one output in the Outputs List. By default, a *complete_scene[Comp]* output exists, which is set to process the entire comp and result camera.

When you save an Action setup, your outputs are also saved.



Output List Lists all of the existing outputs. Use the buttons below the list to populate the list with the outputs you want. To select an output to process, click the yellow arrow beside the output (you can click as many outputs as needed).

Add button Adds a new output to the Outputs list. Use the Rename button to change the name to something more meaningful.

TIP Use the Output Naming settings in the Action Setup Preferences (page 427) menu to set default naming conventions for your outputs.

Copy button Creates a copy of a selected output.

Delete button Deletes the selected output from the Outputs list.

Rename button Opens the onscreen keyboard to rename the selected output in the Outputs list.

Output Options

For each output in the Outputs List, you can refine the output parameters.



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

NOTE Selecting the Stereo mode and a stereo camera results in left and right clips when processed. Selecting Stereo mode and any monoscopic camera results in two identical clips. Selecting Left, Right, or Mono results in a single output, regardless of the camera selected.

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

Type box Select an output type. Output and Render options change based on the Output Type.

Select:	To render:
Comp	Selected objects in the scene with their parameters.
Matte	The matte of the selected objects in the scene.
Media Matte	The matte of the media specified in the Media field.
Z-Depth	The Z-depth of selected objects in the scene.
Normal	The normals of selected objects in the scene.
Emissive	The colours of selected objects in the scene.
Shadow	A white image with greyscale regions that represent the shadow cast coverage.
Specularity	The specular map settings of selected objects in the scene.
Reflection	The reflection map settings of selected objects in the scene.
UV	The UV map settings of selected objects in the scene.
AO (Ambient Occlusion)	A grayscale output that is dark in areas that light cannot reach and bright in areas where it can.
Position	World absolute pixel coordinates of a scene or a selection of objects.

Shadow Output Type box Select the type of shadow cast to output.

Comp box Select whether to render the scene over a background or over a colour you choose using the colour picker.

Comp colour pot Displays the colour that the comp is rendered over. Editable.

Threshold field Displays the value at which the alpha is included in the depth of the output.

Matte Background box Not shown. Select whether the object's matte is rendered on top of a black (default) or white background.

Media field Not shown. Displays the number that corresponds to the media in the Media list. Editable.

Light box Not shown. Select which light to include in a shadow output.

Filtering field Not shown. Displays the amount of smoothing in a shadow output. Editable.

Maximum Z Difference field Not shown. Displays the maximum amount of Z depth information to take into account for smoothing a shadow output. Editable.

Bit Depth field Not shown. This locked field displays that this output type is locked to a 16-bit floating point output. Non-editable.

Ambient Occlusion Output Options

Ambient occlusion refers to the blocking of indirect or diffuse light on an object. Ambient occlusion is caused by indirect light's inability to bounce around and illuminate areas that are blocked by a nearby object that absorbs the light rays. These subtle variations in lighting are visual clues for our eyes to detect surface details and distinctions that would otherwise be washed out and unnoticeable. Ambient occlusion adds realism to your scene by adding shadows in crevices, nooks and crannies, and so on. For each surface point, it calculates how much light is blocked by other geometry.



These settings are available for the AO type, and for the Comp type, if Use AO is enabled in Render Options.

Density field Displays the amount of darker areas to include in the ambient occlusion output. Editable.

Tolerance field Displays the starting point at which darker areas are included in the ambient occlusion output. Editable.

Spread field Displays the width of the ambient occlusion effect. Editable.

Blur field Displays the amount of blur applied to the ambient occlusion. Editable.

Falloff field Displays the amount of falloff around the edge of the ambient occlusion effect. Editable.

Precision field Displays the number of steps to take into account on rays cast from the surface of the object. Editable.

Samples box Select an ambient occlusion sampling level. Press Preview to see the result.

Softness field Displays the softness value of the ambient occlusion sample. Press Preview to see the result. Editable.

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

Render Options

Use the Render Options to further refine your output. Different options are available or greyed out depending on the output type you choose in Output Options.



Anti-Aliasing button Enable to include anti-aliasing processing in the output.

Motion Blur button Enable to include motion blur processing in the output.

Transparency button Enable to include the alpha transparency of the objects in the Output when processing this output.

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

Use Emissive button Enable to process an output with the effect of an emissive map in the scene. An emissive map uses colours to simulate a glowing effect within the texture.

Use Ambient button Enable to process an output with the ambient effect.

Use GMask button Enable to process the output with garbage mask effects. Available for all output types except Matte and Media Matte.

Use Gmask box Not shown. Select whether to use the garbage mask transparency or render the GMask colour. Available for Matte and Media Matte output types.

Blending box Not shown. Select whether to use a master blend option (Change Blend) or to use the blend set in the Blending Mode box (Keep Blend). Available for Matte and Media Matte output types.

Blend Mode box Not shown. Select an option to determine the blending mode of the mask when it overlaps with another mask. Available for Matte and Media Matte output types.

Use Specularity button Enable to process an output with the specularity effect.

Use Reflection button Enable to process an output with the reflection effect.

Use AO button Enable to process an output with the ambient occlusion effect.

Selecting Objects to Output

You can select objects to include in your output by using the Object Options and Object List, or the Edit Output mode. Objects that are part of an output are displayed in the Object List. For each selected object, you can decide if you want it to be visible in the final render or not. Non-renderable objects, such as cameras, are always included in outputs.

TIP When you create a new output in the Outputs list, it is empty by default. Enable All Objects to display all objects in the scene.



To select an object:

1. In the Object List, click an object.

   Selected objects are highlighted in the Object List, schematic, and image window.
2. To select additional objects, `Ctrl`-click another object.

   NOTE When you select multiple objects with different visibility settings (Visible or Occluded), Mixed is indicated in the Visibility box.

To output all objects:

1. In the Object Options, enable All Objects.

   All objects in the scene, as well as any object you subsequently create, are included in the selected output.

To set the visibility of an object:

1. Select an object from the Object List.
2. From the Visibility box, select one of the following:

   Occlude Only Renders the object as black, excluding it from the shading effect. The result is a perceptual hole where the object was originally located.

   Visible Renders the object in the final render.

   NOTE When the Output type is set to Normal or Z-Depth, the objects in Occlude Only mode are rendered normally; it will be part of the Normal or Z-Depth output. The Visibility mode is ignored.

To add or remove an object from the Object List:

1. From the Edit Mode box, select Edit Output.

   NOTE In Edit Output mode, All Objects in the Object Options is disabled when you remove objects from the selected output.
2. Add or remove objects from the output by doing one of the following in the schematic:
   - Click an object. The complete tree of the object in the schematic must either be selected or grayed out for it to be included or excluded from the output.

- Press `Ctrl` and drag in the schematic to add or remove multiple objects (or trees) from the output. This acts as a toggle between selected or greyed out (included or excluded from the output).

To set all objects to Visible mode:

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

Object Options

All Objects button Enable to output all objects in the Object List.

Visibility box Select a visibility option for the selected object.

Reset All button Click to set all objects to Visible mode in the Object List.

About Media

You should have a working understanding of Action media and the relationship between media and surfaces to be successful in using Action. Media has the following characteristics:

- Each front and matte clip combination that you load into Action is called *media*.
- The clips you load into Action are listed in the Media list.
- When you first open Action, you load the front clip and matte clip for the first media and a common back clip for all media.
- Media must have a front and matte of the same resolution, but each media can have a different resolution from the other.
- You can work with front only or matte only media. In this case, the empty front or matte is replaced with an internal white frame of the same resolution.
- The same media can be applied to multiple surfaces. Any cropping, blurring, or recolouring that you apply to one instance of media is applied to all the surfaces for that media. For example, if you blur media, all of the surfaces using that media are blurred.

Working With Media in the Media Menu

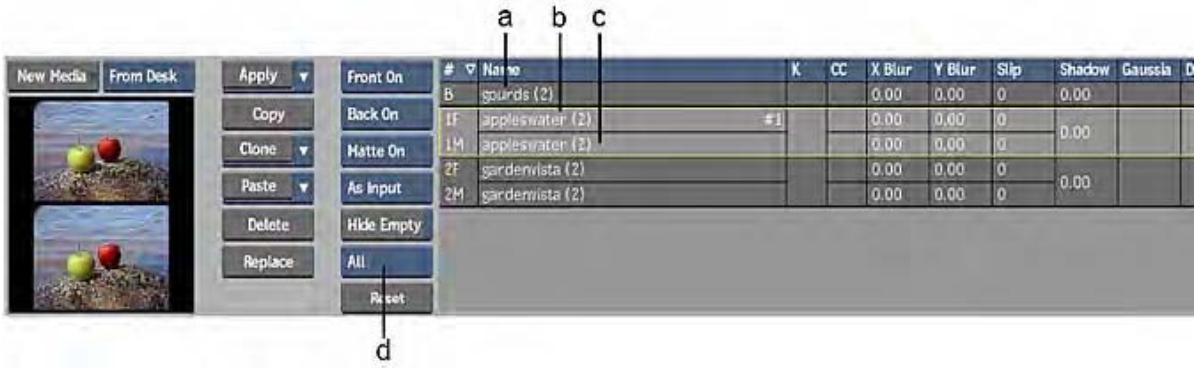
When you open Action, you load the front and matte clips for the first media. You can then load any additional media.

The clips you load for each media appear in the Media list. The back clip you selected to access Action appears in the first row of the Media list and is assigned the letter B in the # column. The second row in the Media list contains the first front and matte clips you selected when accessing Action. The length of each clip appears in parentheses beside the clip name. A new line is added to the Media list each time you add media. For stereoscopic projects, you can apply media to the left and right eye.

You can sort the Media list by list number, front clip name, or matte clip name. As you view clips, you can look at the original front, back, or matte clip individually, as well as the results of colour correcting and keying a matte.

To access the Media menu, click Media in the Action menu. To help identify media, `Alt+click` a surface in the schematic, result, or camera view to automatically select the media in the Media list. If the Media menu is not selected, press `Alt` and double-click the surface to switch to the Media menu, and automatically select the media in the Media list. Also, when you select multiple surfaces, you can press `Alt` to select all

associated media in the Media list. In this case, the first item selected is framed in yellow in the Media list as the current selection.



(a) Back media (b) Front media (c) Matte media (d) Reset box

Media Menu Tips

Use these tips when working with clips and media in the Media menu.

- Different colours and shades are used as visual cues within the Media menu. The current media is outlined in yellow, and the selected media is highlighted in light grey. Selected fields of the media are highlighted in grey.
- `Ctrl`-click fields or media to add to a selection, or `Shift`-click to add a range to a selection. If you `Ctrl`-click a field that is already selected, all selected fields of this type in other selected media become deselected. However, if you continue to press `Ctrl`, you can then click any field, and all fields of the same type in other selected media become selected.
- Click a numeric field to display the calculator, or click and drag to increase or decrease the value in a field.
- `Ctrl+Alt`-click a field to reset the field.
- Double-click the Gaussian, Divide, or Crop field to toggle the effect on or off. Press `Shift` while double-clicking to apply the toggle to all selected media.
- Double-click The K or CC to enter its menu.
- Click a front or matte media to select all the displayed fields.
- Use the `F` keyboard shortcut to toggle the matte media display in the Media list on or off. You can also set this behaviour with the Show Front Only button in the Action Setup menu.
- Resize the column widths in the Media menu by dragging the column dividers.
- `Ctrl`-click to multiselect media for applying to the left or right eye of a stereo object. The media you select first (odd numbers) are applied to the left eye. Even numbered media apply to the right eye.

Adding Media

As the complexity of your effects grows, so will the amount of media you use. You can add unlimited media; media is added to the Media list as they are created. If Auto Image is enabled in the Action Setup menu, an image node and axis are also created in the schematic when new media is added.

If you accessed Action through the Tools tab, the Workspace Media panel is not directly available from within Action. In this case, you'll need to follow these steps to add new media.

To add media from the workspace:

1 Click New Media.

2 From the workspace, select a front and matte clip of the same resolution to load as media.

TIP You can select any number of front and matte clips by holding the `Ctrl` key while selecting clips. Each front/matte pair must have the same resolution, and is added to its own line in the Media list.

NOTE After adding media, if the front or matte can no longer be found in the workspace, the name of the front or matte media is displayed in red in the Media list, and the missing front or matte acts as a white frame in Action.

Media Menu Settings

The Media controls are described as follows.

New Media button Click to open the workspace to add new media. Available when Action is accessed from the Tools tab.

Media List box To add media directly to the Media list, select New Media. To add an indirect layer to the Action node, select New Input. Available when Action is accessed from Batch or Batch FX.

Image Proxy Displays a proxy of the currently selected media.

Apply button Applies the selected media in the Media list to the selected surface in the scene.

Apply dropdown list For stereoscopic clips, select how the selected media in the Media list is applied to the selected surface in the scene.

Copy button Copies media and effects (for example, Blur and Crop settings) of the selected media in the Media list. Use the Paste options to decide how you want to use the copy/paste operation.

Clone button Clones the selected media, or media and effects onto a new media entry in the Media list.

Clone dropdown list Select whether the clone operation clones just selected media, or media and effects.

Paste button Pastes the copied parameters onto the selected media or media entry in the Media list.

Paste dropdown list Select how the paste operation is applied in the Media list.

Select:	To:
Paste Media Only	Paste the copied media onto a selected empty media entry, without any effects (such as Blur or Crop settings).
Paste FX/Media+FX	If pasting onto an existing media entry, pastes only the copied effects. If pasting on an empty media entry, pastes both the copied media and effects.

Delete button Deletes the selected media from the Media list.

Replace button Click to open the area selected in the Media From box to choose media to replace the selected media in the Media list.

Extract button Click to convert the selected media in the Media list from direct to indirect media.

Front Clip box Select an option to edit the front clip's visibility.

Select:	To:
Front On	Display the front clip for the selected media.

Select:	To:
Front Off	Hide the front clip for the selected media.
Front Lock	Lock the selected front clip at the current frame in the timebar.

Back Clip box Select an option to edit the back clip's visibility.

Select:	To:
Back On	Display the back clip.
Back Off	Hide the back clip.
Back Lock	Lock the back clip at the current frame in the timebar.

Matte Clip box Select an option to edit the matte clip's visibility.

Select:	To:
Matte On	Display the matte clip for the selected media.
Matte Off	Hide the matte clip for the selected media.
Matte Invert	Invert a matte. Black areas will be made white, and white areas will become black. **TIP** To invert multiple matte clips, `Ctrl`-click the matte media and select Invert in the Matte Clip box.

TIP Use options in the View box to display selected media in the image window: Media Front (or press `F1`), Media Back (`F2`, and Media Matte (`F3`).

Media Rendering box Select a rendering option per media.

Reset Choice box Select the Media list properties to reset.

Reset button Resets the properties selected in the Reset Choice box.

Keyer field Loads the back, front, and matte clips for the selected media into the Modular Keyer. See Accessing the Colour Corrector and the Modular Keyer from Action (page 448).

NOTE The back clip is loaded into the Modular Keyer for reference only. Any modifications you make to the back clip, such as colour correction, are not applied in Action.

CC field Loads the clip for the selected media into the Colour Corrector. See Accessing the Colour Corrector and the Modular Keyer from Action (page 448).

Xblur and Yblur fields Adjusts the amount of Gaussian or Box blur along the X and Y axes.

Slip field Slips the back, front, or matte clip.

Shadow field Adjusts the softness of a shadow. If the scene contains more than one shadow using the selected media, all shadows are softened by the same amount you specify in this field. See Adding Drop Shadows (page 478).

Gaussian field Uses Gaussian blur on a selected media.

Divide field Divides the front media by its associated matte media.

Crop and Crop Softness fields Crops a front clip or a matte clip for a selected media. Use the Top, Bottom, Left, and Right fields to crop the selected sides. You can also add crop softness.

NOTE For more information on slipping, blurring, dividing, and cropping in the Media list, see Adding Effects to Media (page 445)

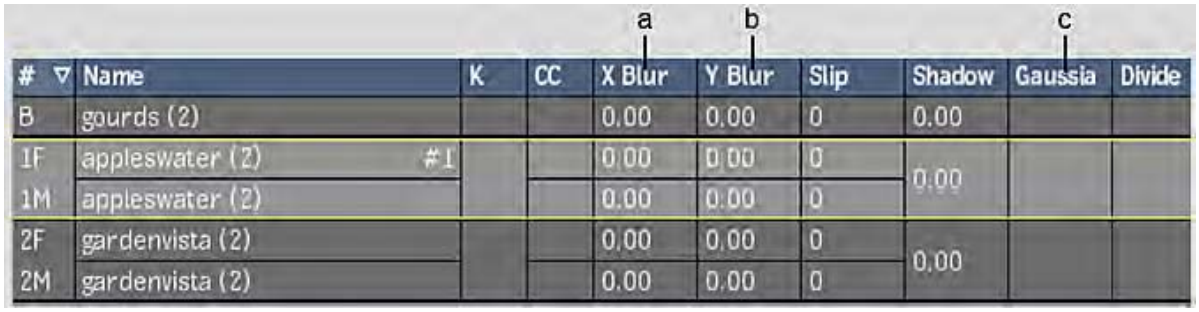
Adding Effects to Media

The Media list allows you to easily add and edit effects to your clips, such as blurs and crops.

Blurring a Clip

To blur the front, matte clip, or back clip for the selected media, use the Xblur and Yblur fields. The Xblur field controls the amount of blurring on the horizontal (X) axis, and the Yblur field controls the amount of blurring on the vertical (Y) axis.

NOTE If you apply the media to more than one surface, all the surfaces are blurred.



(a) Xblur (b) Yblur (c) Gaussian Blur

You can use a Gaussian (Gaussian enabled) or a Box blur (Gaussian disabled):

- Gaussian blur has rounded, smoother edges. It is a better blur for animation because it can be blurred on a subpixel level (0.00).
- Box blur has rectangular, rougher edges.

To blur the front clip of media:

1. In Action, click Media.
2. Select the front media.
3. To use a Gaussian blur, double-click the Gaussian field (otherwise a Box blur is applied).
4. Set the blur using the Xblur and Yblur values, and then negate the values for the Blur fields on the matte media. For example, to blur only the front clip 10 pixels on both axes, set the front Xblur and Yblur fields to 10 and the matte Xblur and Yblur fields to -10.

   TIP `Alt`-drag over the XBlur or YBlur field to change both values proportionally.

To blur the matte clip of media:

1. In Action, click Media.
2. Select the matte media.

3 To use a Gaussian blur, double-click the Gaussian field (otherwise a Box blur is applied).

4 Set the matte Xblur and Yblur fields. Blurring only the matte clip produces a softer edge on the surface of the front clip.

TIP `Alt`-drag over the Xblur or Yblur field to change both values proportionally.

Cropping a Clip

When you crop media, the front and matte clips are cropped together. You cannot crop the back clip. If you applied the media to more than one surface, all the surfaces are cropped.

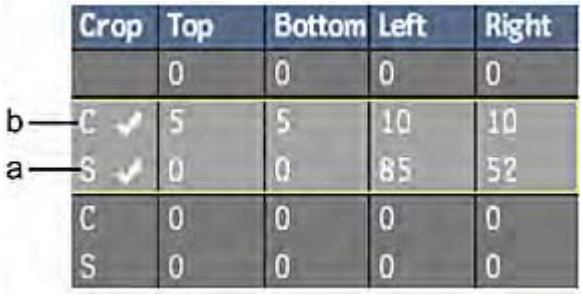
You can animate a crop by changing the size and shape of the Crop box at different keyframes. You can also animate the softness of the Crop box.

NOTE Animating a crop in the Channel Editor does not enable the Crop field in the Media list.

To crop a clip with the Media list:

1 In Action, click Media.

2 Set values for cropping and softness in the Top, Bottom, Left, and Right fields. Once you set a value in one of the Crop or Softness fields, white checkmarks indicate that Crop and Softness are enabled.



(a) Softness area of Crop field (b) Crop area of Crop field

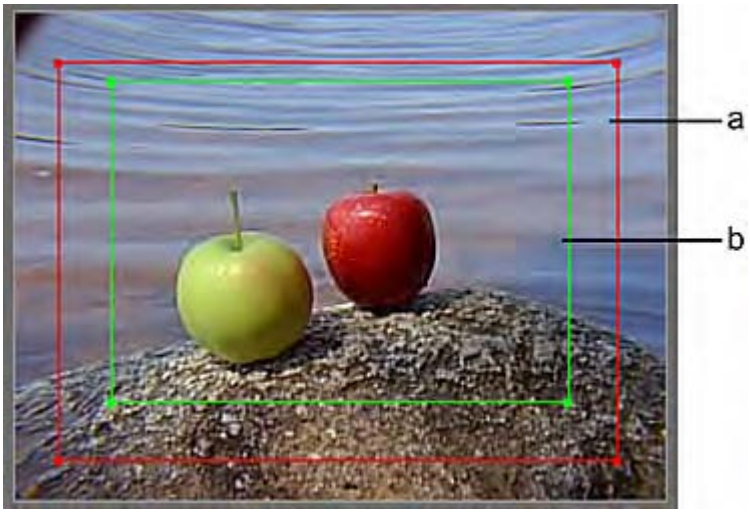
TIP `Alt`-drag over one of the fields to change all four values proportionally.

To crop gesturally in the image window:

1 Double-click the Crop field to enable Crop and Softness.

2 In the View box, select Media Front or Media Matte.

The clip appears in the image window with a red outline, which indicates the Crop box, and a green outline, which indicates the Softness box.



(a) Crop box (b) Softness box

TIP You can use two viewports to view the crop interactively on both the front and the matte clips.

3 Set the corners of the Crop box and Softness box by dragging either the corners or edges of the red and green outlines.

The crop and softness values in the Media list are updated as you drag.

4 Click Result.

The cropped media appears in the image window, and the front and matte clip share the same crop.

5 To mute the Crop and Softness values, double-click the Crop field. To mute only the Softness, double-click the Softness area of the Crop field. When muted, a black checkmark appears in the Crop field.

Eliminating Unwanted Black in Areas of Media Transparency

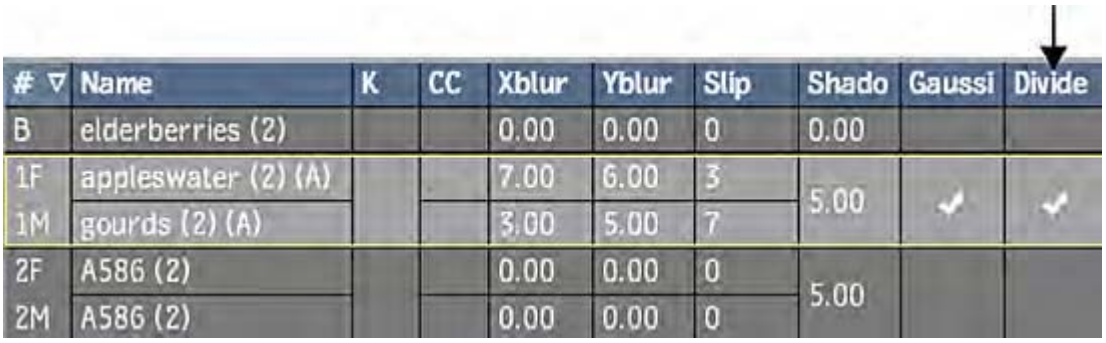
When compositing with images generated by rendering 3D objects over a black background, unwanted black often appears on translucent or anti-aliased areas of the media. Use the Divide tool to divide the front media by its associated matte media, which eliminates the black edges in the composite.

To remove unwanted black in media transparency:

1 In Action, click Media.

2 In the Media list, select the media containing the unwanted black.

3 Double-click the Divide field of the media containing the unwanted black.



# ▽	Name	K	CC	Xblur	Yblur	Slip	Shado	Gaussi	Divide
B	elderberries (2)			0.00	0.00	0	0.00		
1F	appleswater (2) (A)			7.00	6.00	3	5.00	✓	✓
1M	gourds (2) (A)			3.00	5.00	7			
2F	A586 (2)			0.00	0.00	0	5.00		
2M	A586 (2)			0.00	0.00	0			

The selected media's front is divided by its matte, resulting in correct colour in areas of transparency.

Slipping Media

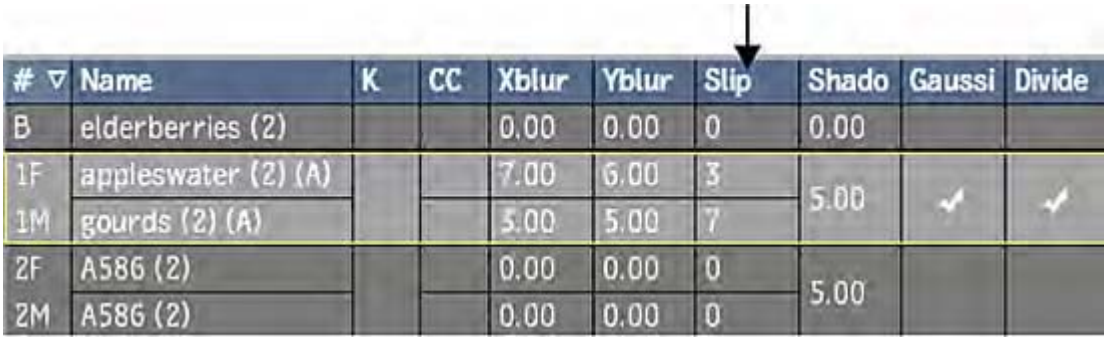
You can animate the Slip value of a media's front, matte or background clip. Do this to begin a clip at a specified frame number, or to produce a freeze frame effect where the first frame is held for a specified number of frames. In addition, Slip values are animatable for all types of clips.

If you want to animate the Slip values, or control the Front Slip and Matte Slip separately, use the Channel Editor. The channels are found in their respective media and are named Slip and slip_matte. The Background Slip channel is found in **Scene ➤ Media ➤ Background ➤ Slip**.

To slip media:

1. In Action, click Media.
2. Change the value of Slip for either the Front or Matte fields, or both, of the media you want to slip.

   TIP `Alt`-drag over the Front or Matte slip field to change both while maintaining the same offset.



# ▽	Name	K	CC	Xblur	Yblur	Slip	Shado	Gaussi	Divide
B	elderberries (2)			0.00	0.00	0	0.00		
1F	appleswater (2) (A)			7.00	6.00	3	5.00	✓	✓
1M	gourds (2) (A)			5.00	5.00	7			
2F	A586 (2)			0.00	0.00	0	5.00		
2M	A586 (2)			0.00	0.00	0			

   A positive value starts the clip at the specified frame. A negative value creates a freeze frame effect where the first frame of the clip is held by the specified number of frames.

To slip a background clip:

1. Change the value of the B (background) Slip field.

Accessing the Colour Corrector and the Modular Keyer from Action

By accessing the Modular Keyer and the Colour Corrector directly from Action, you can key any media or colour correct any front, matte, or back clip without having to exit Action.

When you enter the Colour Corrector or Modular Keyer from Action, you can view the result of your colour correction or key as it would appear in Action by selecting Context from the View box. The Context view is interactive; as you make changes, the Action result is updated in the image window.

Use Result view while keying or colour correcting. In this way, the number of Action media has no impact on system performance. You can tweak the keying and colour correction setup using Context view.

While you work in the Colour Corrector or Modular Keyer, you do not have to click Render; the modifications are automatically applied to the media in Action. If you enter the Colour Corrector from Action, scrub the timebar to view the Result.

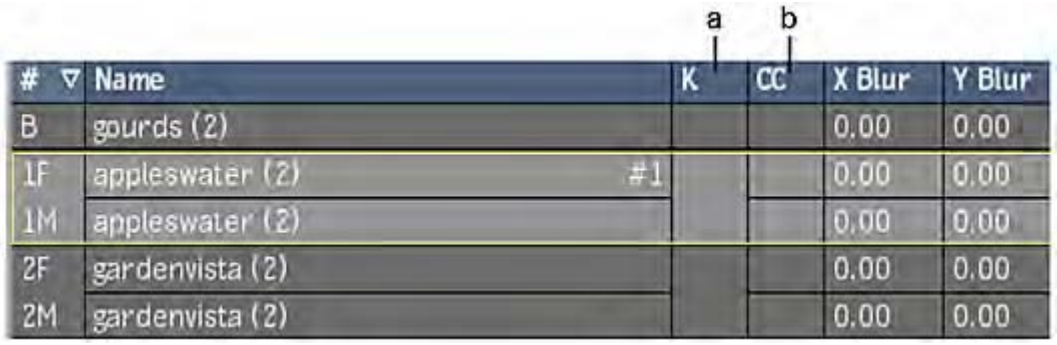
When loading Action setups that contain CC or Keyer effects from a previous version, you may experience a slight loss of quality when viewing proxies. To avoid this loss of quality, select Proxy Full from the Action Setup menu.

Also, when loading Action setups through Batch or Batch FX that contain CC or Keyer effects from a previous version, consider the following:

- If you enabled the Flame Reactor rendering engine when you created your project, make sure the media in the Setup is 8 or 16 bit, as the Flame Reactor engine only supports 8-bit and 16-bit floating point media.
- If you enabled the Classic Rendering engine when you created your project, the media in the Setup can be 8, 10, or 12 bit, but not 16-bit floating point, as the classic keyer and colour corrector do not support 16-bit floating point media.

To access other tools from Action:

1. In Action, click Media.
2. Double-click the corresponding field for your chosen media and tool. For example, to colour correct the matte clip, double-click the matte CC field.



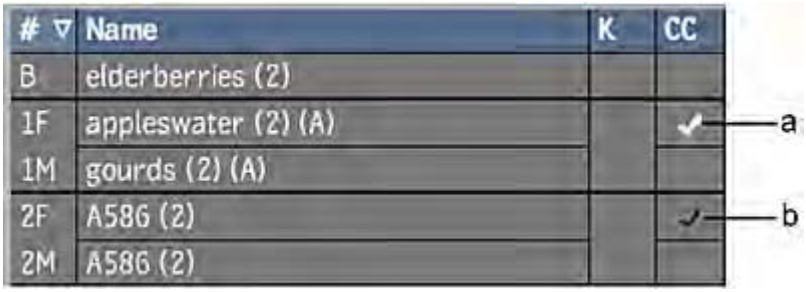
(a) Modular Keyer (b) Colour Corrector

3. As you work in the tool, select Context from the View box to preview the result.
4. Click Return to return to Action.

Muting Media Effects

You can mute Keyer and Colour Corrector media effects in the Action Media menu. Muting effect media can be helpful when you want to see what your Action setup looks like without the Keyer or Colour Corrector settings, but not lose any of the settings of the muted media effect.

To mute an effect, press `Alt` and click the K or CC media. The check mark in the field turns black to signify the media effect is muted. To unmute the media effect, press `Alt` and click the field again.



(a) Active media effect (b) Muted media effect

About Using the Modular Keyer from Action

There are some minor differences when you access the Modular Keyer from Action instead of from the Batch or Batch FX, or from the Tools tab. This section describes features specific to using the Modular Keyer in Action.

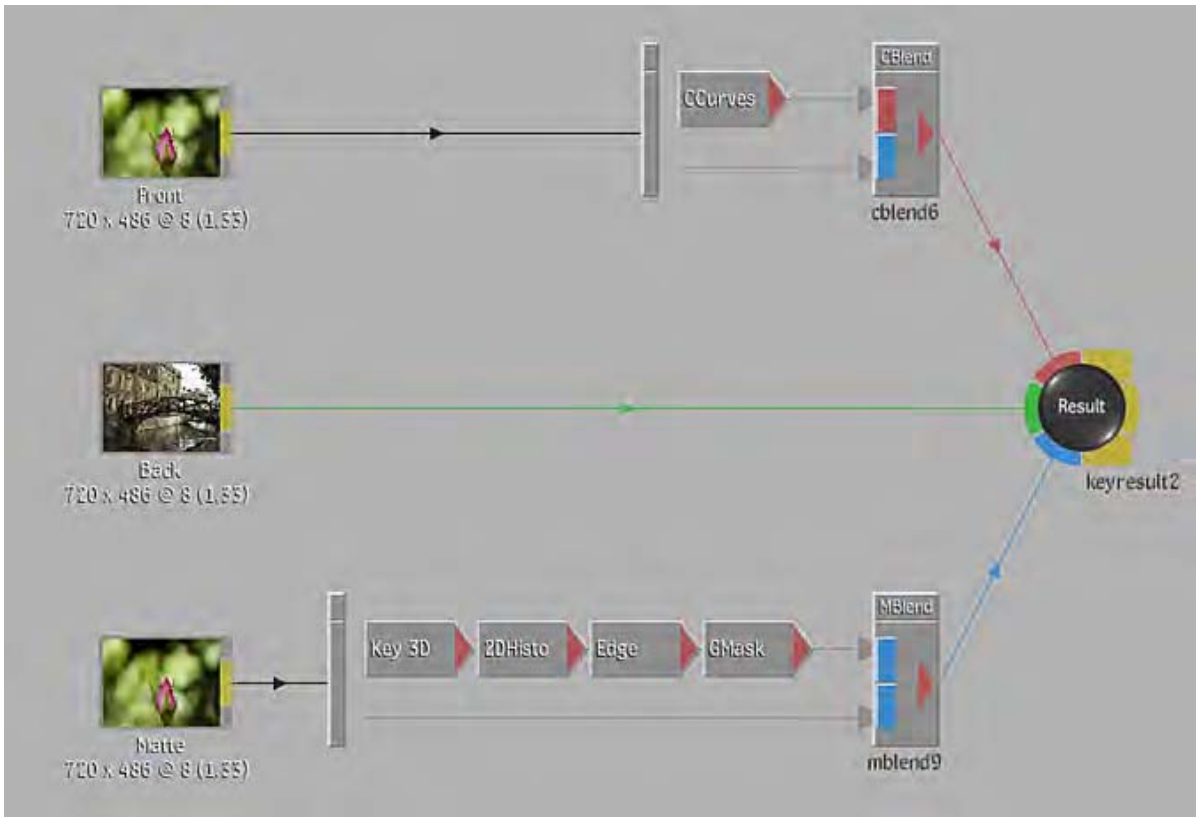
When you load media into the Modular Keyer, the front and matte clips for the selected media are loaded with the back clip and appear at the beginning of the processing pipeline. The front clip is used as the Front clip and the matte clip is used as the Key In clip for the processing pipeline.

The back clip is used as the Back clip in the processing pipeline and is loaded as a reference only. Any modifications you make to the Back clip in the Modular Keyer are not used when you return to Action.

NOTE If the Action back clip is a different resolution than the front clip, a Resize node is added to the Back pipeline when entering the Modular Keyer.

To view the results of your Modular Keyer work as it would appear in Action, select ActionRes from the View box.

The following example shows media loaded into the Modular Keyer from Action.



All branches in the processing pipeline are connected to the Result node. When you return to Action from the Modular Keyer, the following information is used:

- The colour-corrected front clip (attached to the red tab of the Result node)
- The matte clip (attached to the blue tab of the Result node)
- The matte curves for the Result node (To access the matte curves, click the Result node.)

NOTE The back clip is for reference only. Any modifications you make to the back clip, such as colour correction, are not applied in Action.

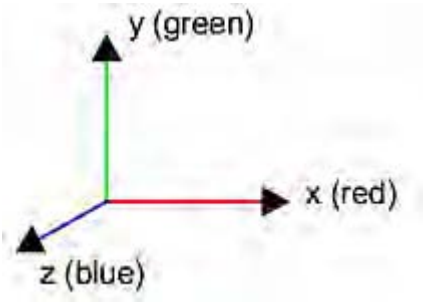
About Axes

An axis is the element of an object or media that can be manipulated to determine the object's 2D or 3D space, position, and movement.

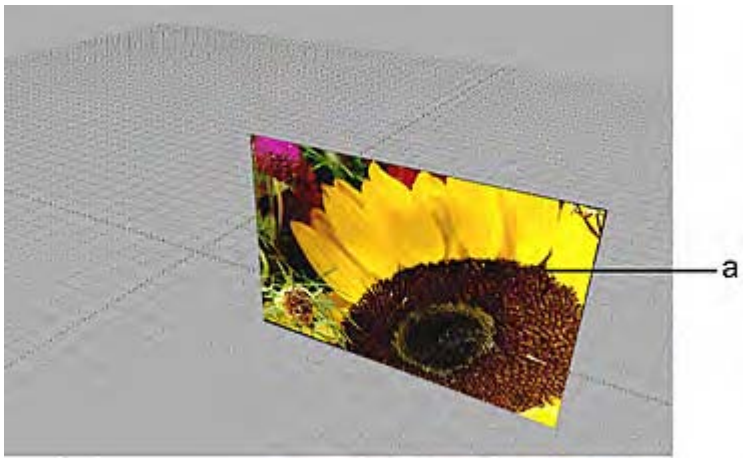
Use the scene's X,Y, and Z coordinate system to place each surface in the scene, or to rotate, scale, and shear surfaces. You move, rotate, and animate objects directly in the scene and use the camera to record the scene. The part of the scene that the camera, or frustum, looks at is what gets rendered.

Manipulating an Object's Axis

When you add certain objects to the scene, such as models or images, they are added with their own axes. Each axis is used to place its respective object in the scene. An axis is represented by the following icon.



All transformations that you apply to the selected axis are applied to the objects connected to the axis. For example, if the position of a surface's axis is set to 500, 100, 0, then its surface is placed at 500 on the X-axis, 100 on the Y-axis, and 0 on the Z-axis.



(a) Surface is placed in the scene at 500, 100, 0

You can also add an axis manually to the scene from the node bin.

Action Views

Front, top, and side view are orthographic views used to position an object's axis in the scene. These views are useful for viewing the scene from another angle than the camera's. See Setting the Camera and Orthographic Views (page 650).

Selecting an Axis

You can select an axis in any of the following ways:

- Click directly on the axis in the scene.

  TIP If you have numerous overlapping axes in the scene, press and hold Q while clicking to cycle through each axis until the axis you need is selected.

- Go to Schematic view and click the node for the axis.
- Display the Channel Editor and select the Axis folder or one of its channels.
- Use the Prev and Next buttons in the Axis menu to select the previous or next axis.

Moving and Rotating Axes in the Scene

You can move and rotate an axis directly in the scene. You can do this when viewing the scene in Camera, Top, Side, or Front view. See Setting the Camera and Orthographic Views (page 650).

The selected mode remains in effect until you select a different mode. To select a mode, use the Edit Mode box.



(a) Edit Mode box

Moving an Axis in the Scene

You can move an axis in the scene, as well as objects that do not have exclusive axes, such as lights, particle generators, and manipulators.

To move an axis directly in the scene:

1. In the Edit Mode box, select Move.
2. Select the axis you want to move and drag it to a new position.

   If the Axis menu is displayed while you move an axis, you can see the Position X, Y, and Z fields update after the axis is placed in its new position. You can also change the X, Y, and Z position by dragging any of the position fields in the Axis menu to move the axis.

TIP Enable Shift-Snap (Setup > Preferences), and then hold the Shift key when selecting an axis to snap the selected axis to the surface of the underlying geometry. Not available when in Top, Front, or Side views.

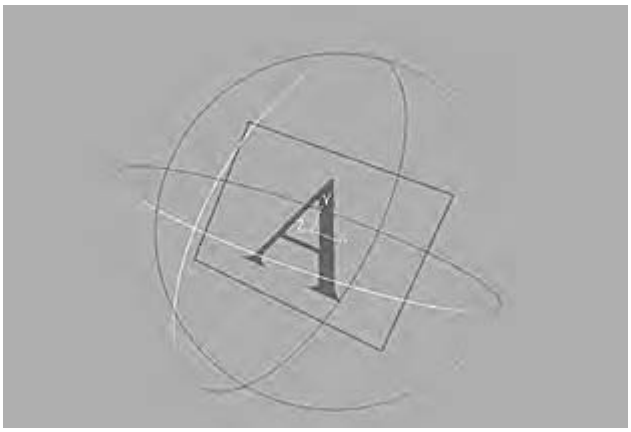
Rotating an Axis in the Scene

You can rotate an axis in the scene, as well as objects that do not have exclusive axes such as lights, particle generators, and manipulators.

To rotate an axis directly in the scene:

1. From the Edit Mode box, select Rotate.
2. Select the axis or the object you want to rotate.

   A trackball appears in the scene. The trackball consists of three concentric rings. Each ring is used to rotate the object and its axis around one of the three axes.



3. Click and drag one of the trackball rings.

   The axis and its object rotate in the image window. If the Axis menu is displayed while you rotate, you can see the Rotation fields update after the axis is rotated to its new position. You can also rotate the axis by dragging any of the Rotation fields in the Axis menu.

Changing the Plane

When you move an object in the scene, the object moves on a 2D plane. By default, an axis is moved gesturally on all three planes. You can change the orientation of the plane by selecting an option in the Plane box.

For example, if the camera is pointed toward the Y plane and you want to move an axis along the X and Z planes, change the orientation to Plane XZ.

Select:	To move objects:
Plane XY	On the X or Y plane, but not on the Z plane.
Plane XZ	On the X or Z plane, but not on the Y plane.
Plane YZ	On the Y or Z plane, but not on the X plane.
Plane Off	On the X, Y, or Z plane. The plane is oriented to face the camera.

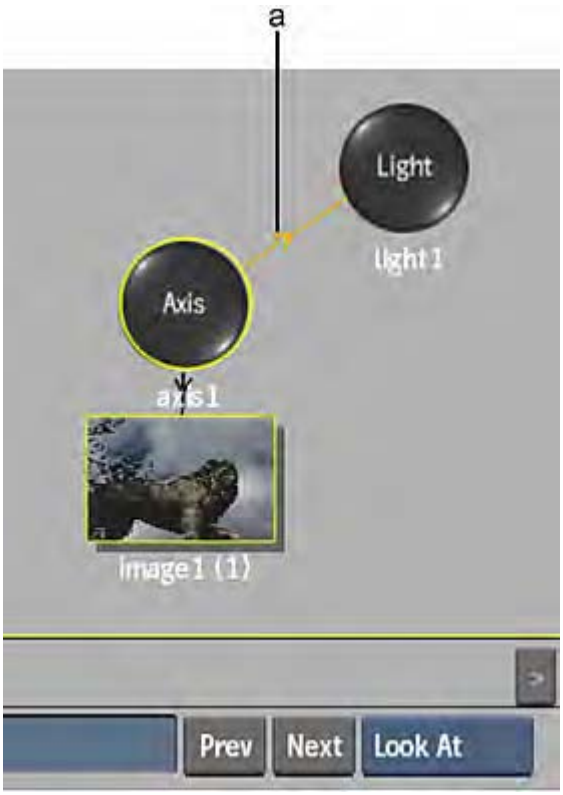
NOTE The different planes can only be used in Camera view. You see the change in the orientation of the plane only when you move the camera away from its default position.

Applying an Axis Look-at Connection

You can attach a look-at connection between the axis and another object in your scene. The axis then rotates to face the look-at object, no matter where it is positioned. You attach a look-at connection in the schematic between the Axis node and any object with axis characteristics (Axis, Camera, Light ,Projector, Particle Animator).

To apply a look-at connection:

1. Select Look At in the Edit Mode box.
2. In the schematic, drag from the Axis node to an object with axis characteristics.
   The selected object is connected to the Axis node by an orange dotted line with an arrow.



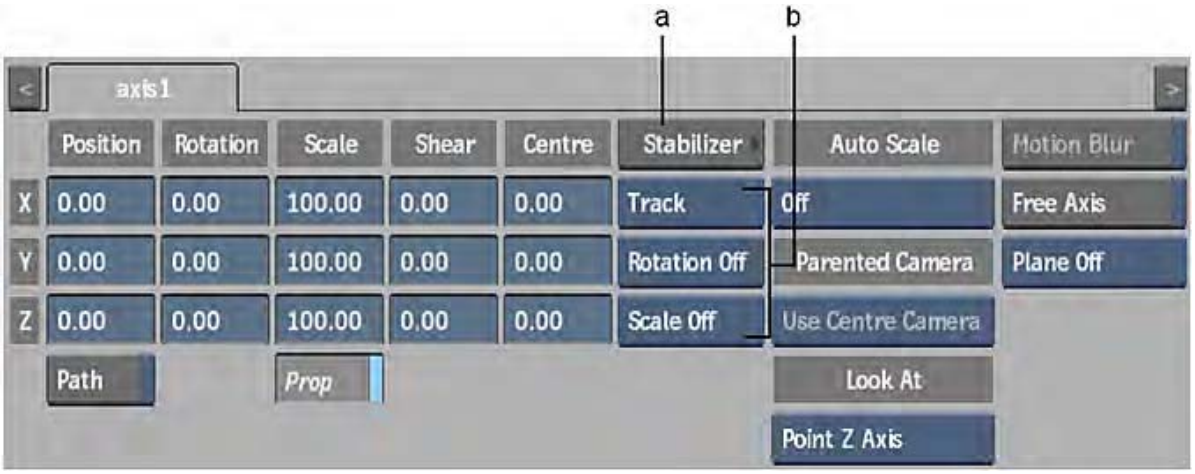
(a) Look-at connection

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



Applying Tracking Data to an Axis

To remove jitter, or track the movement of a feature in the back clip, you can apply tracking data to an axis using the Stabilizer. You can access the Stabilizer directly from the Axis menu in Action.



(a) Stabilizer button (b) Tracking Data controls

Stabilizing an Axis

You can import stabilizing data from the Stabilizer to lock the position of the selected axis to a reference point on the front clip in Action. Any movement in the front clip is matched by the axis. See Stabilizing a Clip from Action (page 874).

Tracking a Feature in the Back Clip

You can track the movement of a feature in the back clip and apply the tracking data to the selected axis in Action. The axis follows the movement of the feature in the back clip. You can also track the rotation or change in size of the feature. See One-Point and Two-Point Tracking (page 878).

TIP To help you view your tracking result without any objects blocking the view, enable the Context button in the Stabilizer menu. While similar to a result view, the context view allows you to see all of the Action scene, except for the selected node (and any children of the selected node).

Axis Menu Settings



X Position field Displays the position of the X axis. Editable.

Y Position field Displays the position of the Y axis. Editable.

Z Position field Displays the position of the Z axis. Editable.

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

X Rotation field Displays the rotation of the X axis. Editable.

Y Rotation field Displays the rotation of the Y axis. Editable.

Z Rotation field Displays the rotation of the Z axis. Editable.

X Scale field Displays the scale of the X axis. Editable.

Y Scale field Displays the scale of the Y axis. Editable.

Z Scale field Displays the scale of the Z axis. Editable.

Proportional button Enable to change the fields proportionally.

X Shear field Displays the shear of the X axis. Editable.

Y Shear field Displays the shear of the Y axis. Editable.

Z Shear field Displays the shear of the Z axis. Editable.

X Centre field Displays the centre of the X axis. Editable.

Y Centre field Displays the centre of the Y axis. Editable.

Z Centre field Displays the centre of the Z axis. Editable.

Stabilizer button Opens the Stabilizer menu to apply stabilizing data to an axis. See Applying Tracking Data to an Axis (page 455).

Stabilizer box Select Track or Stabilize, then click the Stabilizer button.

Tracking Rotation box Select whether tracking rotation is On, Off, or Inverted.

Tracking Scale box Select whether tracking scaling is On, Off, or Inverted.

Autoscale option box Select whether to autoscale when Position settings are changed relative to the camera (On Position Changes), or when Position, Rotation, Shear, or Centre settings are changed (On All Transforms).

Select:	To:
Off	Not use autoscaling on the image.
On Position Changes	Preserve the projected size of the image by automatically scaling when changing any Position parameters that affect the depth of the image relative to the camera.
On All Transforms	Preserve the projected size of the image by automatically changing Position, Scale, and Centre parameters (when any Position, Rotation, Shear, or Centre parameters are changed).

Parented Camera box Select which camera is used for the parenting offset when the immediate parent is a stereo camera. Choose left, right, or center camera. Active only when there is exactly one parent that is a stereo camera, or if a look-at connection is attached between the axis and the stereo camera.

Point Axis box Select which axis is pointed to the attached look-at object. Available only when objects are attached with a Look At connection. See Applying an Axis Look-at Connection (page 454).

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

Free Axis button Enable to ignore transformations from parent axes.

Plane box Select an orientation for the plane (in Camera view). See Changing the Plane (page 453).

About Surfaces

You use surfaces to display media in the scene and then composite front and matte clips with a common back clip.

You should have a working understanding of Action media and the relationship between media and surfaces to be successful in using Action.

A surface is used to place a media in the scene. To use media that you load into Action, you must add a surface to the scene for that media. A surface has the following characteristics:

- A surface type can be flat, bilinear, bicubic, or extended bicubic.
- The same media can be applied to multiple surfaces. Any cropping, blurring, or recolouring that you apply to one media is applied to all the surfaces for that media. For example, if you blur a media, all of the surfaces using that media are blurred.

The first time you open Action during a session, an image surface is added to the scene using the first media (if Auto Image is enabled in the Action Setup menu). You must add a surface for each additional media before its clips can be used in the scene. You can then add textures and lighting effects, or change the surface's properties such as its shape, transparency, and specular highlight. Once added, the media appears in the Media list.

By default, a Flat surface is added. You can change the surface type in the Shape box of the Surface menu. See Changing the Shape of a Surface (page 465).

You control the position of the surface using axis, rotation, scale, shear, and other attributes related to its placement. See Manipulating an Object's Axis (page 451).

Adding Surfaces

To add a surface:

1. In the Media list, select the media containing the front and matte that you want to add to the scene.
2. Do one of the following:
   - Drag an image node from the node bin and place it in the schematic. An axis and an image are created and linked together.
   - Drag an image node from the node bin to the Result view, so you can see its effect on the scene before placing it exactly where you want.
   - Double-click an image node. The node appears next to the last added object. You do not need to be in Schematic view to add a node in this manner.

   The surface is added to the scene with its own axis. The selected media in the Media list is automatically applied to the surface.

   Notice that when media is applied to the image surface, the front and matte clip are combined. You can turn off the matte in the Matte Clip box in the Media menu to show the entire front clip.
3. If you later decide to change the media on the surface, select the surface in the schematic, then select the new media in the Media list and click Apply.

Modifying Surfaces

The Surface menu includes properties common to all surfaces, and specific controls for bilinear and bicubic surfaces. You can change a surface's shape, position, and transparency, as well as apply lighting effects. You can also apply four-point tracking data to a bilinear surface.

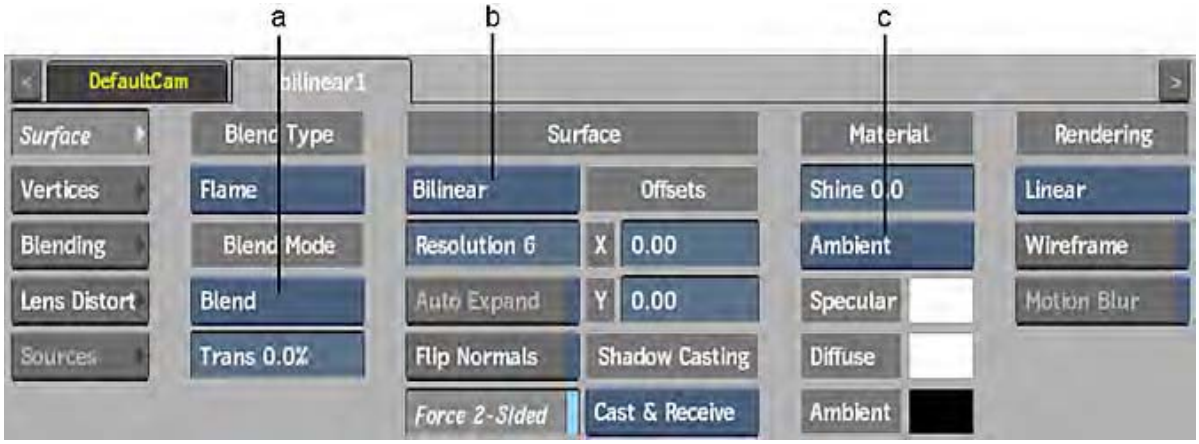
To access the Surface menu:

1. Double-click the selected surface in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 415).

   The Object Image menu appears with the Image controls for the selected surface displayed on the right side of the menu. Some of the tabs for the Surface menu are contextual, and are available depending on the circumstance.

Surface Settings

The Surface tab is available at all times, and is divided into a number of sections.



(a) Blend Mode box (b) Shape box (c) Lighting box

Blend Type Section

Blend Type box Select whether to use Flame or Photoshop blend modes.

Blend Mode Section

Blend Mode box Select how the front and matte clips or the front and back clips are combined. See Surface Blending Modes (page 467).

Transparency field Displays the transparency of the image. Enter 0 for a completely opaque image, or 100 for a completely transparent image.

Surface Section

Shape box Select a shape for the selected surface. For Stereo Objects, you are limited to using a Flat surface. See Changing the Shape of a Surface (page 465).

Surface Resolution field Displays the geometry resolution of the selected surface. By default, the value is equal to the default Action Resolution setting (in the Rendering section of the Setup menu).

The lower the value, the better the resolution and the greater the processing time required to interact with the image. For example, a value of 1 on an NTSC image creates a mesh with 720 horizontal and 486 vertical subdivisions on the selected surface, affording accurate displacement, normals, and lighting.

Auto Expand button Enable to automatically resize the surface when an attached diffuse map's Axis settings are changed. Only available for Flat surfaces, and when a diffuse map is parented from the surface. Always available for stereo objects.

Flip Normals button Enable to flip the selected surface normals to light the back side of a surface.

Force 2-Sided button Enable to have lights in the scene light both sides of the surface (when shading is turned on).

Surface Offset X field Defines the level of offset for a surface along the X axis. Editable.

Surface Offset Y field Defines the level of offset for a surface along the Y axis. Editable.

See Offsetting a Surface (page 467).

Shadow Casting Section

Shadow Casting box Select how the selected image object will be affected by a Shadow Cast object in the scene. See Surface and Geometry Shadow Casters (page 508).

Material Section

Shine field Displays the level of shine for the specular highlights (there are no specular highlights when Shine is set to 0). Editable.

Lighting box Select Ambient or Diffuse lighting so that the surface can reflect incidental light. See Applying Incidental Light Reflection (page 501).

Specular Highlights colour pot Displays the colour of the surface highlights. Editable.

The specular highlight is visible only if Shading is enabled in the Setup menu and if shine is greater than 0. See Adjusting Specular Highlights (page 501).

Diffuse colour pot Displays the diffuse colour. If Diffuse is selected in the Lighting box, the Ambient colour takes on the same shade as the selected Diffuse colour. Editable.

Ambient colour pot Displays the ambient colour. Available when Ambient is selected in the Lighting box. Editable.

Rendering Section

Filter box Select the type of filtering to apply to the surface.

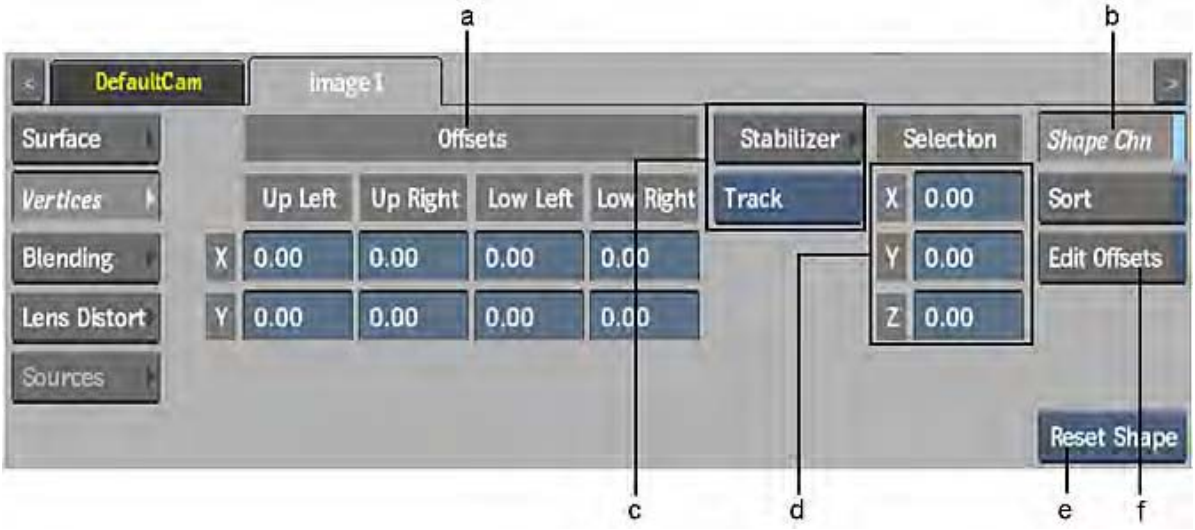
Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

Wireframe button Enable to convert the selected surface to a wireframe representation (lighting and transparency properties are kept).

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

Vertices Settings for Bilinear Surfaces

For bilinear surfaces, there are extra controls available in the Vertices tab.



(a) Offsets fields (b) Shape Channel button (c) Stabilizer controls (d) Vertex Translation fields (e) Reset Selection box (f) Edit Offsets button

Upper Left X Offset field Displays the position of the upper left offset point along the X axis. Enable Edit Offsets, and use Media Front view to see the offset point. Editable.

Upper Left Y Offset field Displays the position of the upper left offset point along the Y axis. Enable Edit Offsets, and use Media Front view to see the offset point. Editable.

Upper Right X Offset field Displays the position of the upper right offset point along the X axis. Enable Edit Offsets, and use Media Front view to see the offset point. Editable.

Upper Right Y Offset field Displays the position of the upper right offset point along the Y axis. Enable Edit Offsets, and use Media Front view to see the offset point. Editable.

Lower Left X Offset field Displays the position of the lower left offset point along the X axis. Enable Edit Offsets, and use Media Front view to see the offset point. Editable.

Lower Left Y Offset field Displays the position of the lower left offset point along the Y axis. Enable Edit Offsets, and use Media Front view to see the offset point. Editable.

Lower Right X Offset field Displays the position of the lower right offset point along the X axis. Enable Edit Offsets, and use Media Front view to see the offset point. Editable.

Lower Right Y Offset field Displays the position of the lower right offset point along the Y axis. Enable Edit Offsets, and use Media Front view to see the offset point. Editable.

TIP In Media Front view, you can also move the offset points directly in the image window. The offset fields in the Vertices menu are updated accordingly.

Stabilizer button Opens the Stabilizer menu to apply stabilizing data to a surface or surface offsets.

Stabilizer box Select Track or Offsets, then click the Stabilizer button.

Vertex Position X field Defines the position of the bilinear or bicubic vertices on the X axis. Editable.

Vertex Position Y field Defines the position of the bilinear or bicubic vertices on the Y axis. Editable.

Vertex Position Z field Defines the position of the bilinear or bicubic vertices on the Z axis. Editable.

Shape Channel button Enable to use the Shape channel in the Channel Editor. Disable to use individual Vertex channels in the Channel Editor. See Reshaping Using the Channel Editor (page 468).

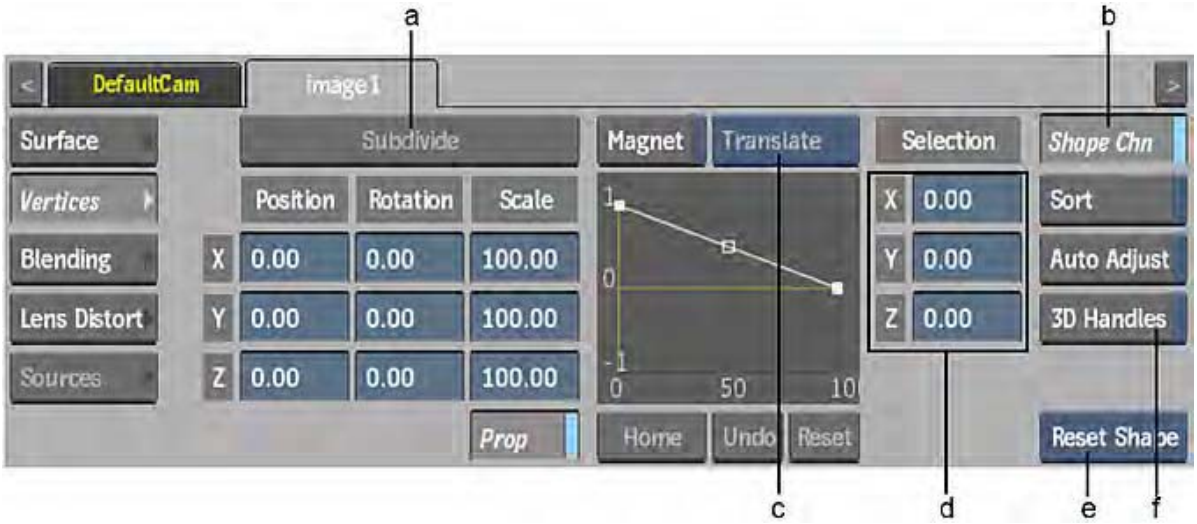
Sort button Enable to sort non-flat surfaces where overlapping transparent regions are causing artefacts.

Edit Offsets button Enable to move the bilinear anchor points away from the entire image (used with four-point tracking). Use Media Front view to see the offset points. See Editing Tracker Offsets of a Bilinear Surface (page 471).

Reset Selection box Select whether to reset surface handles to their default position (Reset Shape) or reset selected points (Reset Select) on a bilinear or bicubic surface.

Vertices Settings for Bicubic and Extended Bicubic Surfaces

For bicubic and extended bicubic surfaces, there are extra controls available in the Vertices tab.



(a) Subdivide button (b) Shape Channel button (c) Magnet Transformation box (d) Reference Point fields (e) Reset Selection box (f) 3D Handles button

Subdivide button Click to subdivide the extended bicubic surface into more sections (can be clicked multiple times). See Subdividing an Extended Bicubic (page 473).

X Position field Positions the selected surface points along the X axis. Editable.

Y Position field Positions the selected surface points along the Y axis. Editable.

Z Position field Positions the selected surface points along the Z axis. Editable.

X Rotation field Rotates the selected surface points along the X axis. Editable.

Y Rotation field Rotates the selected surface points along the Y axis. Editable.

Z Rotation field Rotates the selected surface points along the Z axis. Editable.

X Scale field Scales the selected surface points along the X axis. Editable.

Y Scale field Scales the selected surface points along the Y axis. Editable.

Z Scale field Scales the selected surface points along the Z axis. Editable.

Proportional Scale button Enable to scale the surface points proportionally.

Magnet button Enable to transform a range of extended bicubic surface points. Use in conjunction with the Magnet Transformation box. See Transforming Multiple Points (page 475).

Magnet Transformation box Select a transformation type to use when Magnet is enabled.

Magnet Curve Editor Displays the weighted polarity from the centre to the edge of the magnet.

Magnet Curve Home button Resets the position of the magnet curve after panning.

Magnet Curve Undo button Undoes a change to the Magnet Curve Editor.

Magnet Curve Reset button Resets the Magnet Curve Editor.

Vertex Position X field Defines the position of the bilinear or bicubic vertices on the X axis. Editable.

Vertex Position Y field Defines the position of the bilinear or bicubic vertices on the Y axis. Editable.

Vertex Position Z field Defines the position of the bilinear or bicubic vertices on the Z axis. Editable.

Shape Channel button Enable to use the Shape channel in the Channel Editor. Disable to use individual Vertex channels in the Channel Editor. See Reshaping Using the Channel Editor (page 468).

Sort button Enable to sort non-flat surfaces where overlapping transparent regions are causing artefacts.

Auto Adjust button Enable to scale adjacent tangents automatically. Auto Adjust creates a smooth curve between points in the deformation. If you want to work on a specific area of the image without affecting other tangents, disable Auto Adjust.

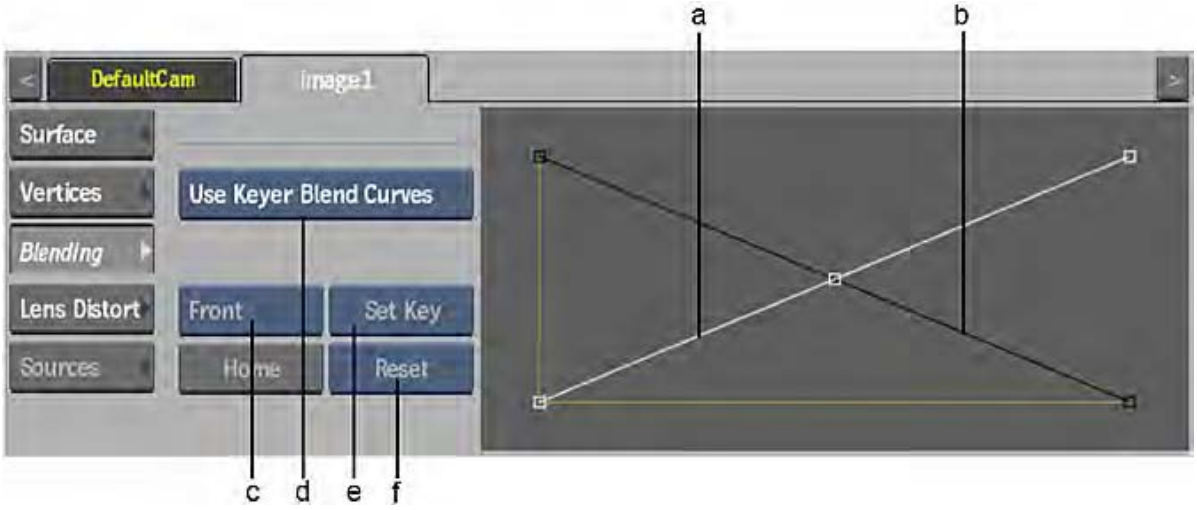
NOTE If you move a tangent explicitly, it is not affected by Auto Adjust. Click Reset Select or Reset Shape so that the tangents will be affected by the Auto Adjust mode.

3D Handles button Enable to allow Z buffering of the vertices. By default, the vertices are always visible, regardless of their position in Z space in relation to other media.

Reset Selection box Select whether to reset surface handles to their default position (Reset Shape) or reset selected points (Reset Select) on a bilinear or bicubic surface.

Blending Settings

Use the settings in the Blending tab to adjust blending curve for each surface.



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

Blend Curves option box Select whether to work with the surface blend curves or view the existing keyer blend curves.

Matte box Select the matte curve you want to adjust.

Keyframe Option box Select an option to Set, Delete, or Reset keyframes. If Auto Key is enabled, a keyframe is added automatically when you adjust the blending curve.

Home button Click to reset the curve view.

Reset Selection box Select whether to reset the selected curve (Reset) or all curves (Reset All) to their default settings.

For more information on using the blending curves, see Applying Blending Curves per Surface (page 470).

Lens Distort Settings

Differences in camera lenses or perspective irregularities cause lens distortion that results in skewed angles. Use the settings in the Lens Distort tab to rectify or simulate these types of distortions in your images.



Distortion box Specify whether you want perform a lens distortion or rectification.

Distort Corrections box Select whether to apply radial corrections manually in this menu, or use the automatic settings derived from the selected analyzer.

Distort Analyzer box Select which analyzer is used to provide automatic radial corrections.

Auto Expand button Enable to automatically resize the surface when an attached diffuse map's Axis settings are changed. This same button appears in the Surface tab.

Centre X field Displays the horizontal position of the centre of the lens. Editable.

Centre Y field Displays the vertical position of the centre of the lens. Editable.

Magnitude field Displays the magnitude of radial distortion or rectification. Editable.

Adjustment field Displays the level of secondary adjustment of radial distortion or rectification. Editable.

Anamorph field Displays the ratio of radial distortion or rectification along the X or Y axis. Editable.

Sources Settings

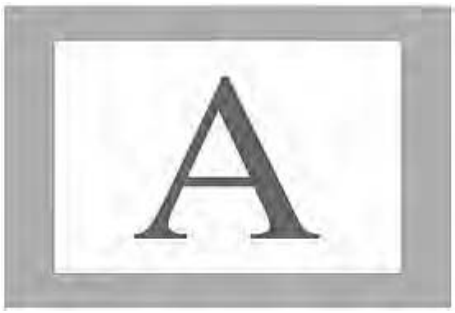
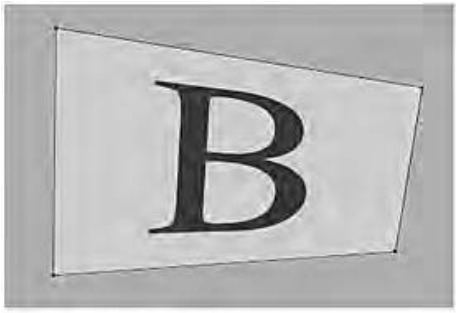
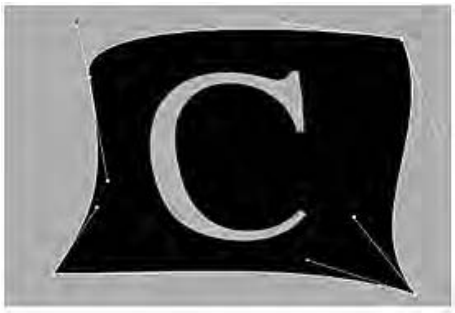
If a source node is parented above a surface node in the schematic, the Source tab is available. These settings allow you to change source settings on a per surface basis, independent of the source type set in the Source menu for the parent source node or nodes. See Replacing the Front or Matte Clip (page 701).

Changing the Shape of a Surface

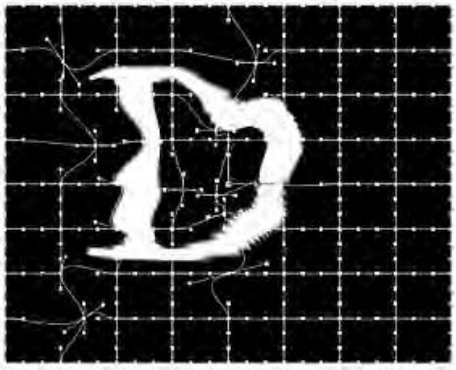
When you first activate media or access Action, the media's surface is set to Flat by default. In Action, you can also represent a clip using custom bilinear or bicubic surfaces.

To change the shape of a surface:

1. Double-click the surface in the Schematic view.
2. Select the shape of the surface from the Shape box.

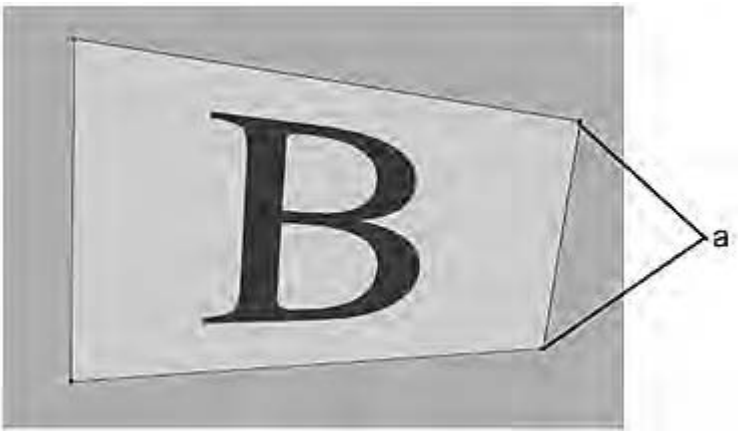
Flat You cannot change its shape because a flat image does not have vertices. You can, however, scale and shear a flat image using its axis. See Manipulating an Object's Axis (page 451). This is the simplest surface. It is added to the scene by default the first time you enter Action with a Front, Back, and Matte. When working with a Stereo Object, Flat is the only shape available.	
Bilinear A bilinear surface has four vertices: one for each corner. The vertices are joined using linear interpolation (straight lines). You can animate the shape of a bilinear surface by changing the position of the corners.	
Bicubic A bicubic surface has four vertices: one for each corner. The vertices are joined using bicubic interpolation (curved lines). Each corner has two additional tangent handles used to adjust the curve of the line between points. You can animate the shape of a bicubic surface by changing the position of the corners and moving the tangent handles to adjust the curve between corners.	

Extended Bicubic An extended bicubic surface has four vertices like a bicubic surface, but the sections of the extended bicubic surface can be subdivided up to eight times to increase the number of vertices. The vertices are joined using bicubic interpolation (curved lines). You can animate the shape of a bicubic surface by changing the position of the corners and moving the tangent handles to adjust the curve between corners. See Warping an Extended Bicubic Surface (page 472).

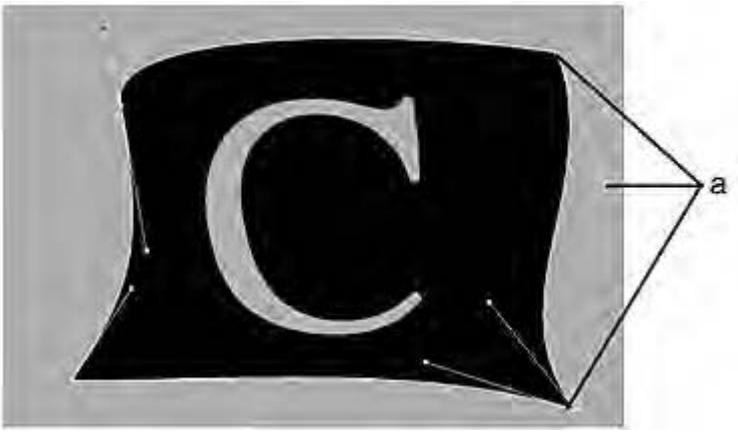


3 Click the vertex you want to edit and drag it to its new position.

You use the vertices—or handles—on bilinear and bicubic surfaces to change a surface's shape. You can move, rotate, shear, and scale a surface using the Axis (page 456) menu.



(a) Vertices on a bilinear surface



(a) Vertices and tangent handles on a bicubic surface

To move a handle on a bilinear or bicubic surface, click the handle that you want to edit. The selected handle appears in red. Use the cursor to drag the handle to its new position.

Each corner has two tangent handles. Lengthen or move the tangents to change the shape of the bicubic between corners.

All of the animation modifications you make to a shape are saved. If you decide to change a shape, then change it back to the original shape, your modifications are not lost.

Surface Blending Modes

Use the Blend Mode box to select how the front and matte clips or the front and back clips are combined. For example, use Screen for blending a fire or a lightning bolt shot on black that you want to composite with a different background.

You can select between a number of Flame or Photoshop blend modes by making a selection in the Blend Type box. Some of the blend modes are similar (or identical) between the different types, but are repeated with the mode names you are used to working with.

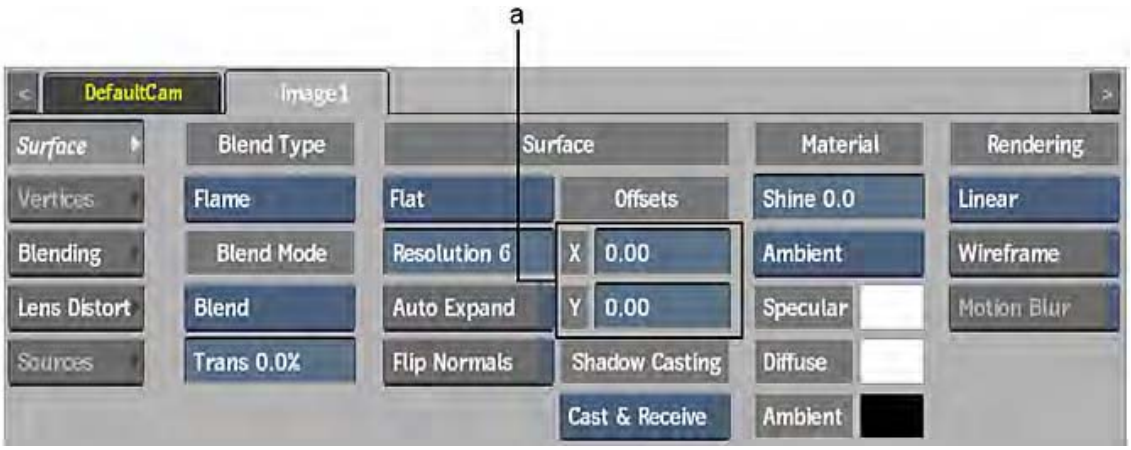


Offsetting a Surface

By default, a surface's axis is at the centre of the surface. Use the Surface Offset X and Y fields to offset a surface along the X-axis or Y-axis. All rotations, scaling, and shearing applied to a surface are applied about its axis. The location of the axis is indicated by the axis icon in the scene.

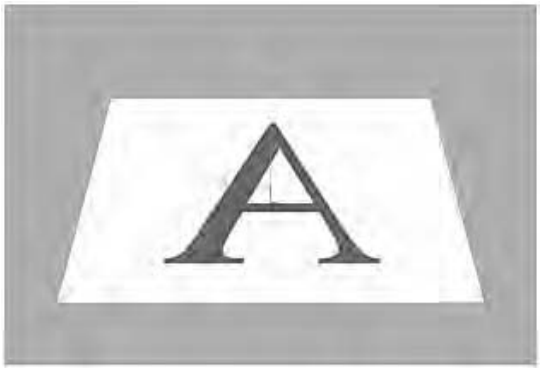
To offset a surface:

1. Select the surface you want to offset.
2. In the Surface menu, change the Surface Offset X and Surface Offset Y fields accordingly.

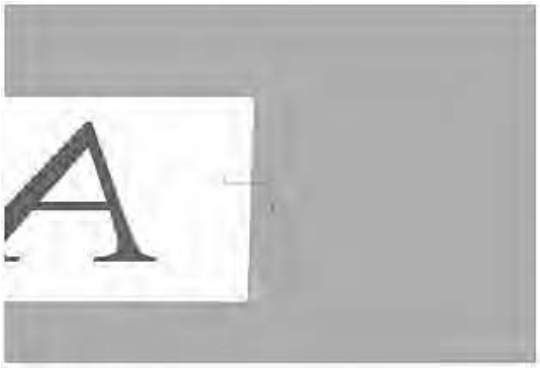


(a) Surface Offset fields

The following figure illustrates the difference between rotating a surface that has not been offset and a surface offset using the Surface Offset X field.



Rotated image with no offset. Notice that the axis is in the centre of the image.



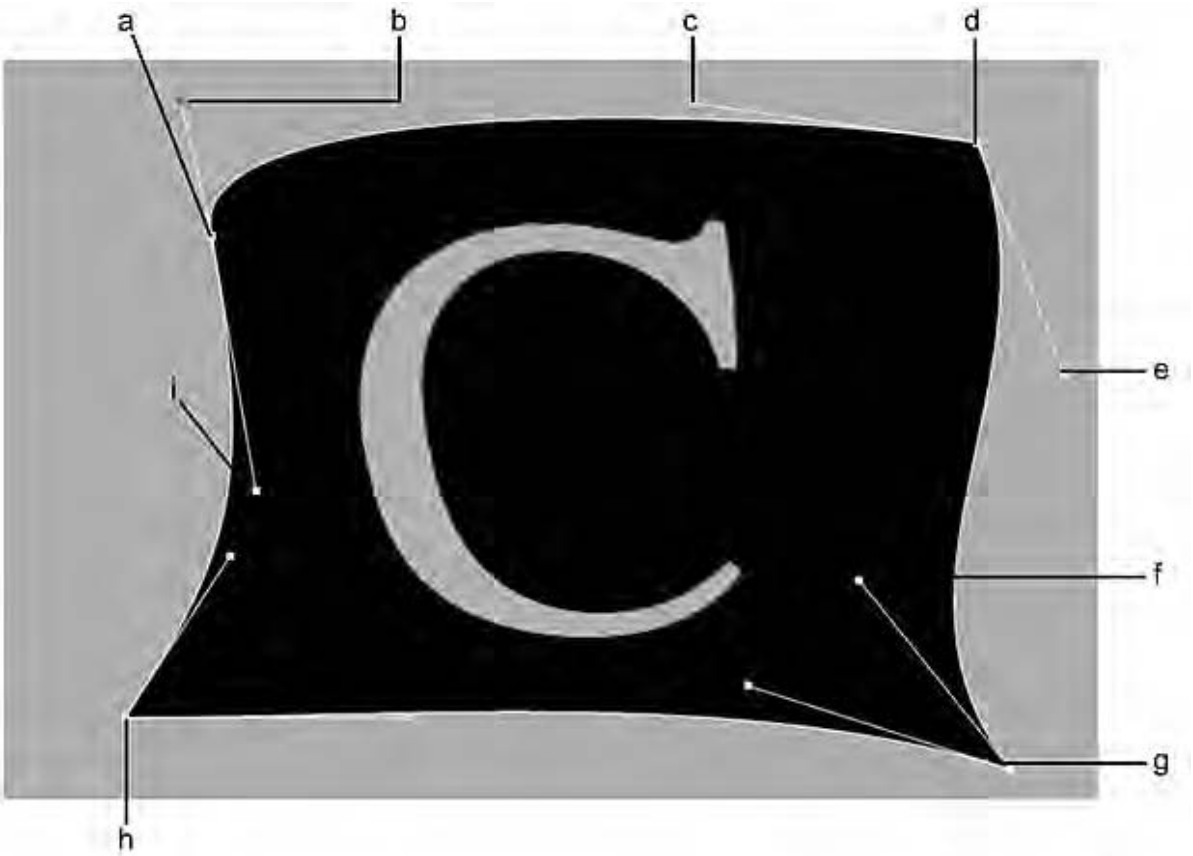
Rotated image with an offset along the X-axis. Notice that the axis is offset from the image centre.

Reshaping Using the Channel Editor

You can select whether you want to create an animation for bilinear, bicubic or extended bicubic surfaces using the Shape channel or individual Vertex channels in the Channel Editor. The Shape channel shows when the shape of the surface changes during the animation. Each time you move a surface handle, a shape key is added at the current frame, provided Auto Key is enabled in the Setup menu.

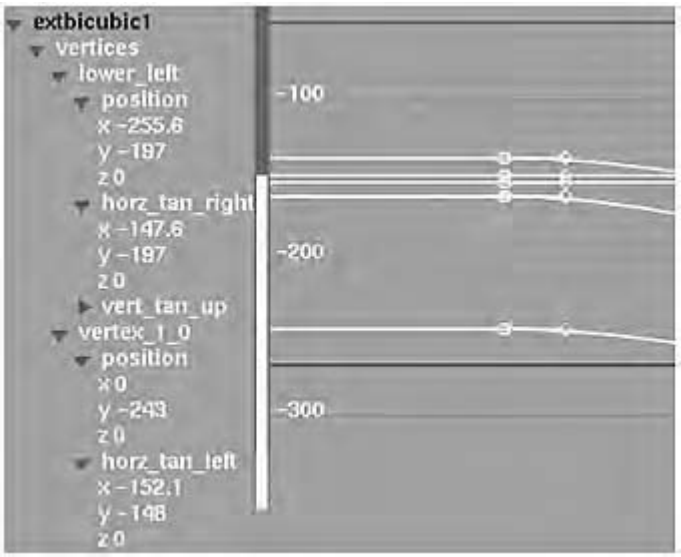
Vertex channels correspond to the vertices—or handles—that appear on the four corners of bilinear, bicubic, and extended bicubic surfaces. The channel names for each corner are: upper_left, upper_right, lower_left, and lower_right, and are listed in the channel hierarchy each with a position x,y,z.

Each corner vertex has two tangent handles—the tangent handles for upper_left are named horz_tan_right and vert_tan_down.



(a) upper_left (b) horz_tan_right (c) horz_tan_left (d) upper right (e) vert_tan_ down (f) vert_tan_ up (g) lower_right (h) lower_left

Extended bicubics have additional vertex channels that appear in the Channel Editor for subdivided vertex channels. These channels appear only when you create a keyframe for the channel and its value changes. These vertex channels are named vertex_0.1, vertex_1.0, vertex_2.1, and so on, according to their position on the surface. Click the vertex on the surface to highlight its channel in the channel hierarchy. See Warping an Extended Bicubic Surface (page 472).



Blending Curves

You can adjust the blending curves of each surface separately. The blending curve is similar to the Keyer luminance blending curve, but you can adjust it per surface.

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

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

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

Result = F * FrontLUT + B * BackLUT

where:

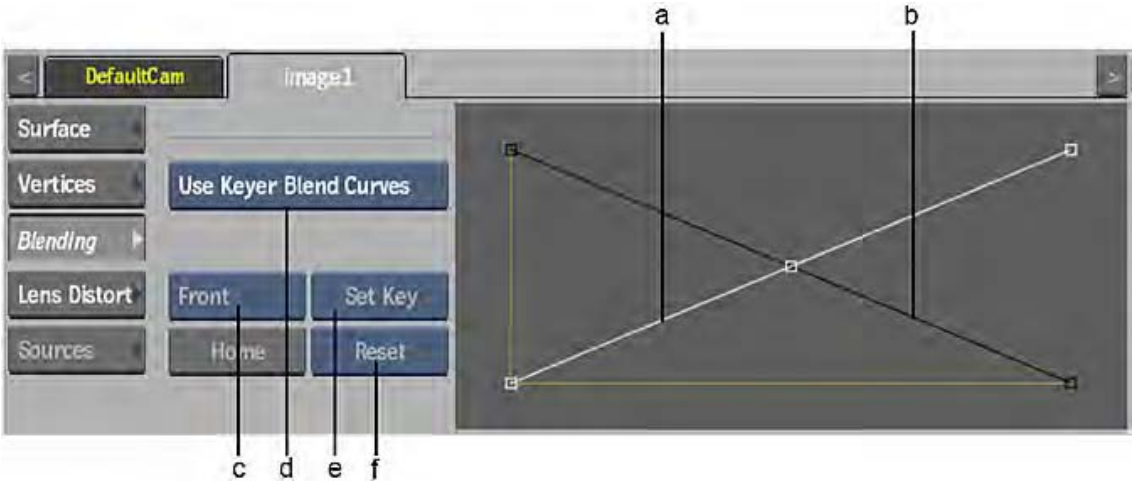
- F = the R, G, and B values of the front image
- B = the R, G, and B values of the back image
- FrontLUT is the front matte pixel value, re-mapped according to any luminance curve change made in the blending curve. The value is expressed as a decimal, where, for example:
  - in 8-bit mode, 0 = 0, 127.5 = 0.5, and 255 = 1
  - in 12-bit mode, 0 = 0, 2047.5 = 0.5, and 4095 = 1
  - 16-bit floating point images, the values are represented on a logarithmic scale between 0 and 1.
- BackLUT is the back matte pixel value, re-mapped according to any luminance curve change made in the blending curve. The value is expressed as a decimal, as is the FrontLUT.

Applying Blending Curves per Surface

To adjust the blending curve:

1 In the Surface menu, click the Blending tab.

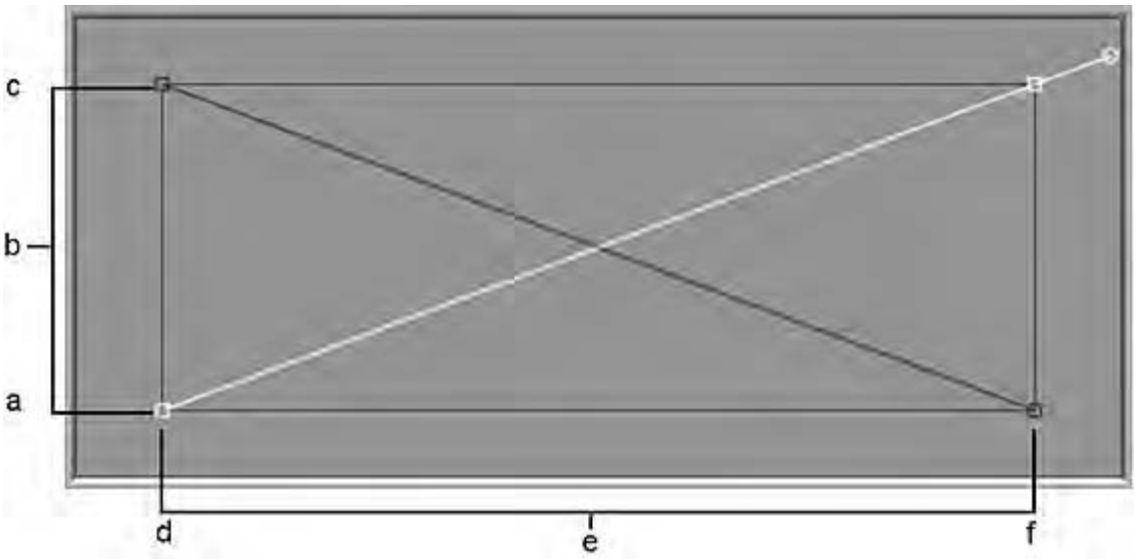
The Blending Curve menu appears.



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

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

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



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

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

4 Use the options in the Keyframe option box to Set, Delete, or Reset keyframes. If Auto Key is enabled, a keyframe is added automatically when you adjust the blending curve.

5 Use the Blend Curves option box to switch between the surface blend curves and the keyer blend curves. This gives you a good comparison to luminance curve work you have already done in the Keyer. When Use Keyer Blend Curves is selected, all of the other Action blend curve settings are unavailable.

Editing Tracker Offsets of a Bilinear Surface

Enable the Edit Offsets button to change the way the image is applied to a bilinear surface. For example, when you enable this button, an offset point appears for each of the surface's four corners (if Media Front is selected in the View box), then you use the Offsets controls to edit the offsets to match the corners of your square. View the result by disabling Edit Offsets.



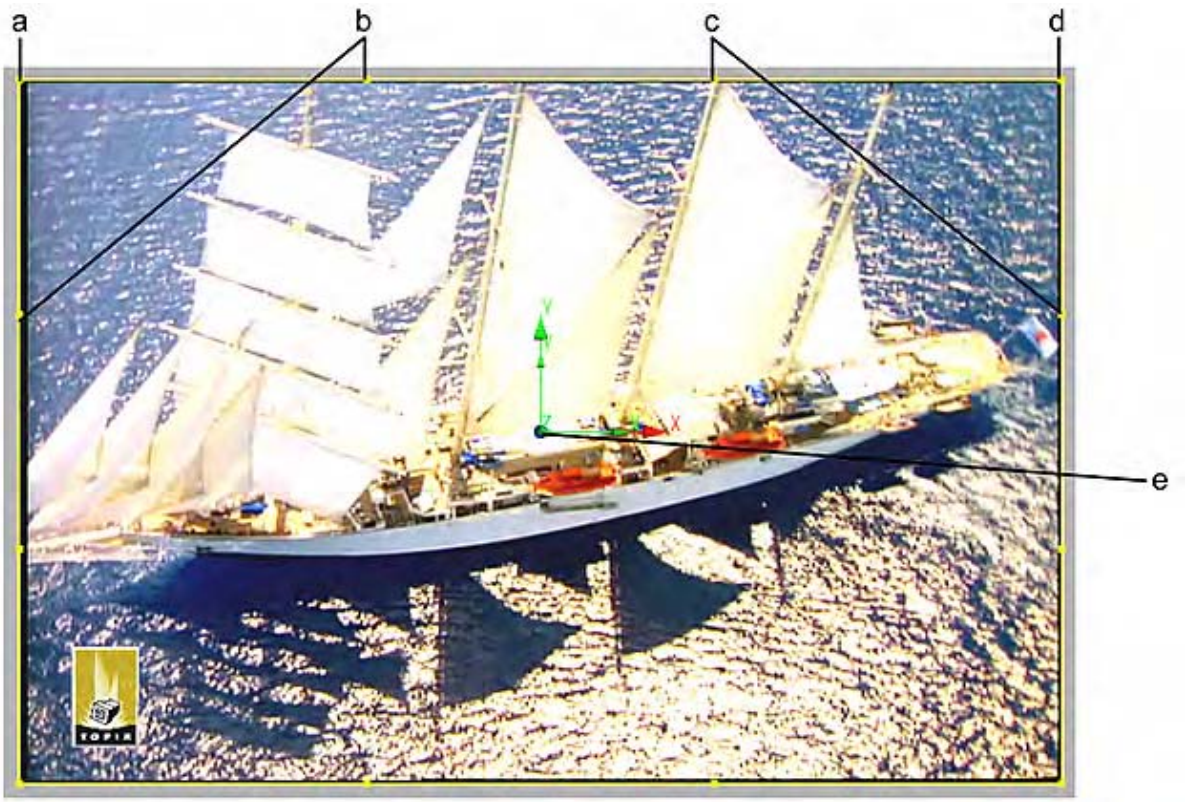
You can then apply stabilizing data to a surface or surface offsets. See Four-Point Tracking (page 880).

Warping an Extended Bicubic Surface

Use an extended bicubic surface to warp specific areas of a surface. An extended bicubic surface can be subdivided into many sections, which are controlled by vertices. The vertices allow you to bend and contort the surface. The vertices are joined using bicubic interpolation—curved lines.

You can animate the shape of a bicubic surface by changing the position of the vertices and moving the vertex tangent handles. See Reshaping Using the Channel Editor (page 468).

By default, the extended bicubic surface has four vertices and eight tangents. You can translate, scale, or rotate these points and tangents individually or as a group. Subdividing the bicubic surface increases the number of vertices in the image and allows for more precision.



(a) vertex 1 (b) tangents of vertex 1 (c) tangents of vertex 2 (d) vertex 2 (e) Reference point

Image courtesy of Topix

Extended bicubics have three types of points: vertices, tangents, and the reference point. Vertices and tangents are collectively called *surface points*.

Vertices lie on the surface and its tangents determine the curvature of the deformation at each vertex. The reference point indicates the axis of origin for applying rotation and scaling to surface points.

Use the surface points to control the deformation of the surface. Each vertex has tangent handles. By moving the tangents and their handles, you can warp specific areas of the surface. You can translate, scale, or rotate these points and tangents individually or as a group.

TIP If a tangent point is hidden behind a vertex or reference point, press Q and click the point in the image window to select the tangent.

Tangent handles have three modes.

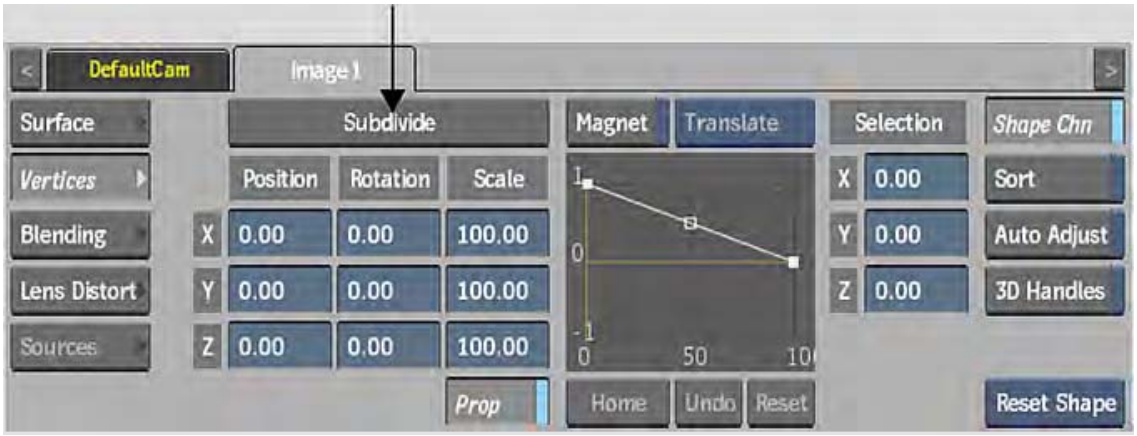
This mode:	Indicates:
■	The handles are not broken. If you move this handle, the adjacent handle will move in the opposite direction and by the same magnitude (length).
□	The handles are broken. If you move this handle, no other handles are affected.
○	The handles are geometrically continuous. If you move this tangent, the adjacent handles will move in the opposite directions but maintain its magnitude (length).

Subdividing an Extended Bicubic

Subdivide the bicubic surface up to eight times to increase the number of vertices and tangents. Then translate the vertices for a smaller region of the image for more precision. You should warp the surface with a small number of subdivisions to obtain the best results. After applying some deformations to a large portion of the surface, subdivide the surface further and perform deformations on a more localized region of the surface.

To subdivide extended bicubics:

1 From the Extended Bicubic Vertices menu, click Subdivide.



You can click the subdivide button up to eight times to further subdivide the surface.

2 Transform the tangents to achieve the effect you want.

To move the reference point:

1 Select the reference point.

The reference point turns red when selected.

2 Drag the reference point to a new location, or use the X, Y and Z fields to assign a new coordinate for the reference point.

To rotate multiple surface points:

1 Set the reference point values you want to use as the axis of origin using the X, Y, Z Reference Point fields.

2 Select multiple surface points by pressing `Ctrl` and dragging to select the surface points.

3 From the Edit Mode box, select Rotate.

A 3D trackball appears on the reference point.

4 Use the 3D trackball to rotate the reference point and selected points.

All selected points rotate around the reference point.

To scale multiple surface points:

1 Set the reference point values you want to use as the axis of origin using the X, Y, Z Reference Point fields.

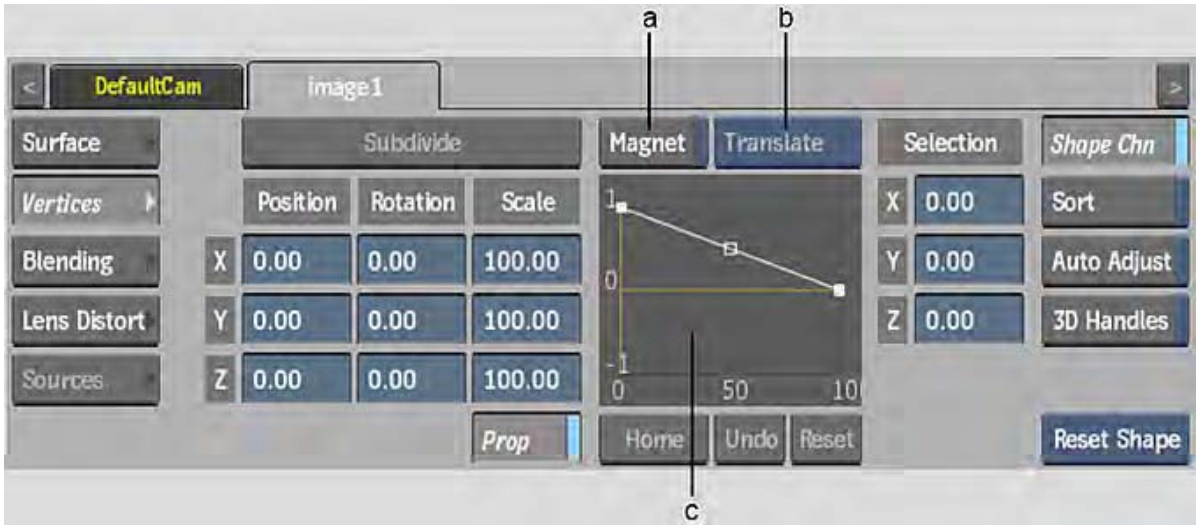
2 Select multiple surface points by pressing `Ctrl` and dragging to select the surface points.

3 From the Edit Mode box, select Scale.

4 Drag in a direction in the image window to scale accordingly.

About Transforming Multiple Points

Use the magnet to transform a range of extended bicubic points. Use the magnet when there are many surface points as a result of more than one subdivision. In the Extended Bicubic menu, the magnet controls include the Magnet button, the Magnet Transformation box, and the Magnet Curve Editor.



(a) Magnet button (b) Magnet Transformation box (c) Magnet Curve Editor

There are two ways to transform points with the magnet. You can use the magnet to select a range of points and transform them gesturally. Or, you can use the invisible magnet in conjunction with the Translation, Rotation, and Scale fields to transform selected points numerically.

Before you can use the magnet, you must set the polarity of the magnet and the magnet area of focus. The magnet's area of focus is determined by where you click the surface and the polarity of the magnet is set by the Magnet Curve Editor. The radius is determined by the distance from the centre to the farthest selected point.

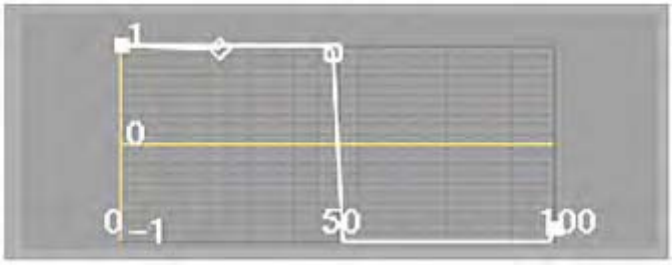
Transforming Multiple Points

Setting the Magnet Polarity

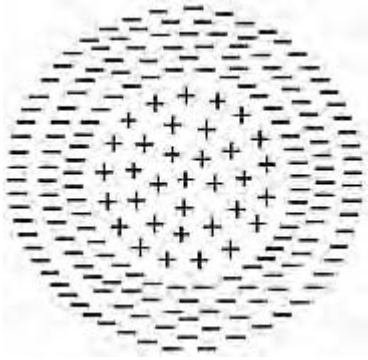
Use the Magnet Curve Editor to define the weighted polarity from the centre to the edge of the magnet. Points are either attracted or repelled depending on their location in the magnet and the shape of the magnet curve.

The area of the magnet is plotted on the X-axis where 0 is the centre of the magnet and 100 is the edge of the magnet. The polarity is plotted on the Y-axis where 1 is maximum positive strength and -1 is maximum negative strength.

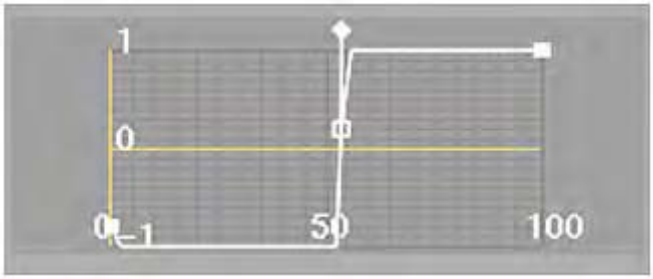
A positive polarity value attracts nearby points. A negative polarity value repels nearby points. The following example illustrates a magnet that has a strong positive polarity near the centre and strong negative polarity near the edges.



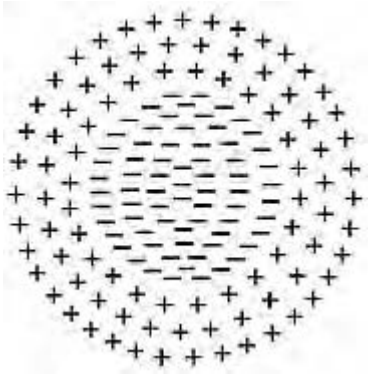
Magnetic Curve Editor



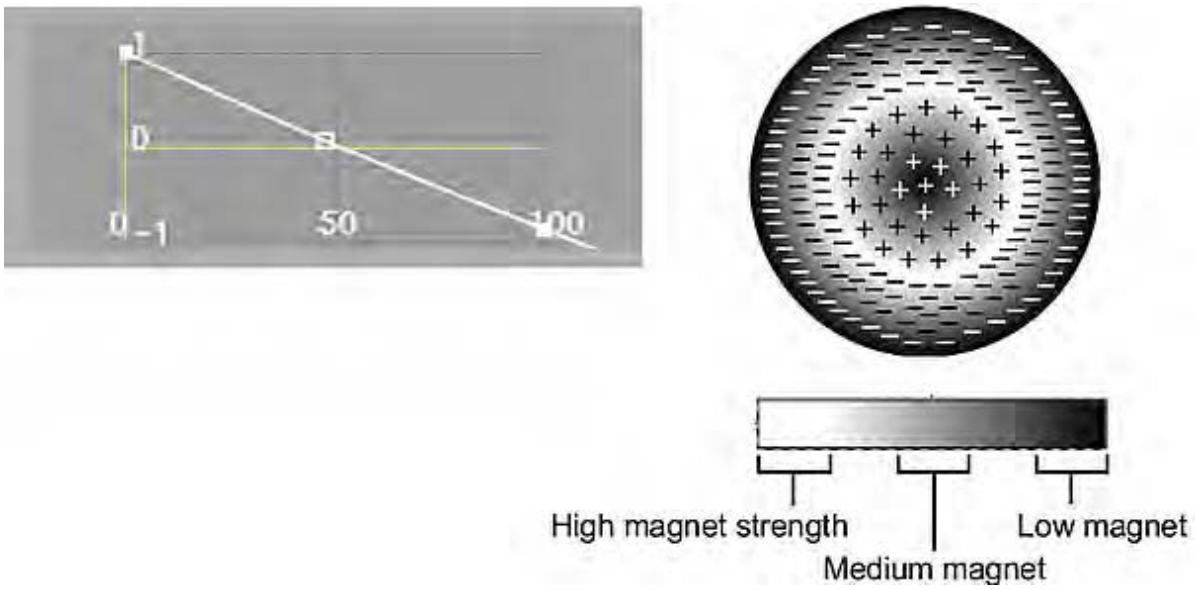
The following example illustrates a magnet that has a strong negative polarity near the centre and a strong positive polarity near the edges.



Magnetic Curve Editor



The following example illustrates a magnet whose positive polarity decreases from the centre and whose negative polarity increases toward the edges.



When you transform points, the direction and size of the transformation is determined by the weighted polarity of the magnet at the location of the points.

Using the first example, if you use the magnet to translate the points on the X-axis by +200, the points in the centre of the magnet move in the positive direction on the X-axis—they are attracted to the magnet—while the points near the edge of the magnet move in the negative direction on the X axis—they are repelled by the magnet.

To change the polarity of the magnet:

1. In the Extended Bicubic Vertices menu, click Reset to return the magnet curve to its default.
2. Click the left-most handle on the curve and drag to define the polarity for the centre of the magnet.

Drag the point:	**To:**
Up	Increase the positive polarity.
Down	Increase the negative polarity.
To the middle	Assign no polarity.

3. Click the right-most handle on the curve and drag to define the polarity for the edge of the magnet.

Drag the point:	**To:**
Up	Increase the positive polarity.
Down	Increase the negative polarity.
To the middle	Assign no polarity.

4 Click the middle handle on the curve and drag to define the transition of polarity from the centre to the edge of the magnet.

Transforming Points

Use Magnet mode to transform points gesturally. The magnet's area of focus is determined by where you click the surface and the radius of the magnet is set by the Magnet Curve Editor. The radius is determined by the distance from the centre to the farthest selected point.

To transform points:

1 Use the Magnet Curve Editor to determine the polarity for the magnet.

2 From the Edit Mode box, select Magnet.



The magnet appears as a red circular outline.

3 To resize the magnet, press `Ctrl+S` and drag left or right.

4 From the Magnet Transformation box, select the type of transformation.

Select:	To:
Translate	Translate the selected points along the X-, Y-, or Z-axis.
Rotate	Rotate the selected points about the centre of the magnet.
Scale	Scale the selected points about the centre of the magnet.

5 Click the points you want to transform in the image window.

6 Drag in the image window to apply the transformation to the selected points.

Transforming Unselectable Points

Use the invisible magnet to transform points you cannot select with the magnet. The centre of the magnet is determined by the centre of the selection and the radius is determined by the distance from the centre to the farthest selected point. Any transformations to the selected points are affected by the magnet.

To transform unselectable points:

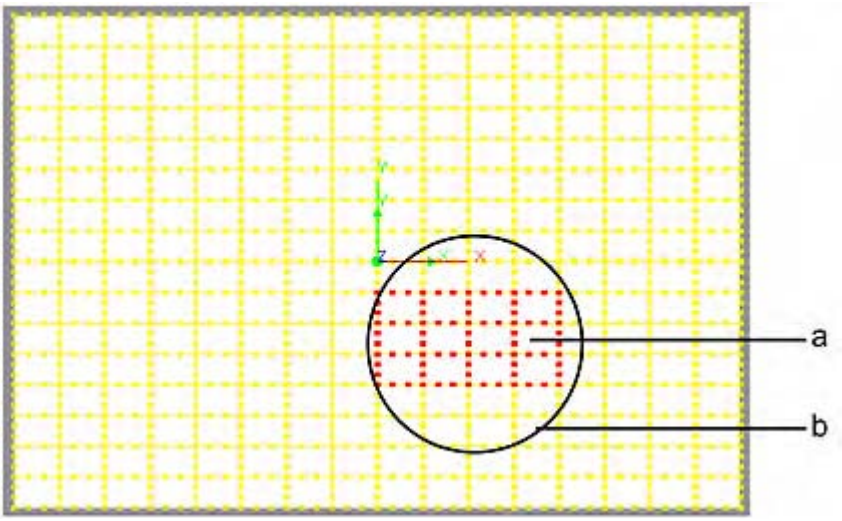
1 Use the Magnet Curve Editor to determine the polarity for the magnet.

2 From the Edit Mode box, select Move or Select.

3 Hold `Ctrl` and drag to select a range of points.

4 In the Extended Bicubic Vertices menu, enable Magnet.

The magnet is not visible, but is illustrated here to show how the points will be affected when you transform them.



(a) Selected Points (b) Magnet (not visible)

5 From the Magnet Transformation box, select the transformation type.

Select:	To:
Translate	Translate the selected points along the X-, Y-, or Z- axis.
Rotate	Rotate the selected points about the centre of the magnet.
Scale	Scale the selected points about the centre of the magnet.

6 Change the values in the Translation, Rotation, or Scale fields.

Adding Drop Shadows

In Action, drop shadows are cutouts based on a surface's matte. A drop shadow can be fully opaque or slightly transparent to simulate a real shadow.

Shadows cast by lit objects are also supported in Action (see Casting Shadows (page 506)).

To add a drop shadow:

1 Select the surface that you want to use.

2 Do one of the following:

- Drag the shadow node from the node bin and place it in the schematic. An axis and a shadow are created and linked together.
- Drag the shadow node from the node bin to the Result view, so you can see its effect on the scene before placing it exactly where you want.
- Double-click the shadow node. An axis and a shadow are created and linked together. You do not need to be in Schematic view to add a node in this manner.

If a surface node is selected in the schematic, the axis of the shadow node is automatically connected to the surface.

NOTE A shadow has its own parent axis. It can be moved, rotated, scaled, and sheared independently of its parent surface. Because a shadow is also the child of the surface, moving the surface axis also moves the shadow.

Adjusting Shadow Softness

You can adjust the softness of a shadow using the Shadow field in the Media menu. Use a value of 0.00 to have a distinct shadow. Increase the value to soften the shadow.

Shadow softness is a property of the media. If you add more than one shadow for the same media, all shadows are softened by the same amount.

Adjusting Shadow Colour and Transparency

Adjust shadow colour and transparency in the Shadow menu of the Object menu.

To adjust the colour and transparency of a shadow:

1 Double-click a shadow node to display the Shadow menu.



2 Make sure that the Shadow button is set to Shadow On.

NOTE You can turn the shadow on or off from the Shadow button. This button is independent of the Hide option.

3 To change the colour of the shadow, do one of the following:
- Enter colour values in the three colour fields in the Shadow menu.
- Click the colour pot below the fields. Use the colour picker to pick the shadow colour.

4 To adjust the transparency of the drop shadow, use the Transparency field. When this field is set to 100, the shadow is completely transparent. When set to 0, the shadow is completely opaque.

TIP You can lock the shadow's transparency to the transparency of its parent surface using the Transparency Lock button. Once a shadow transparency is locked to its parent surface, changes to the surface transparency also affect the shadow's transparency. You can then adjust transparency of the shadow as an offset of the surface's transparency.

Using Shadow Mix

Use Shadow Mix to render shadows and surfaces according to their order in the Priority Editor, independently of their position in Z-space. This option makes it possible to have the shadow of one media on top of the shadow of another media. Shadow Mix is an option in the Z-Buffer box in the Rendering section of the Setup menu.



Shadow Menu Settings



Red Shadow field Displays the red shadow value. Editable.

Green Shadow field Displays the green shadow value. Editable.

Blue Shadow field Displays the blue shadow value. Editable.

Shadow colour Pot Displays the colour of the shadow. Editable.

Shadow Transparency Lock button Locks a shadow's transparency to the transparency of its parent surface.

Shadow Transparency field Displays the transparency level of the drop shadow. Editable.

Shadow button Toggles the shadow on or off.

Reordering Surfaces

When you add a surface (flat image, bilinear, or bicubic) or geometry (3D text or model), the surface or geometry appears in front of all other objects in the scene. These overlapping objects create a stack and an order of priority as one object is drawn in front of the other.

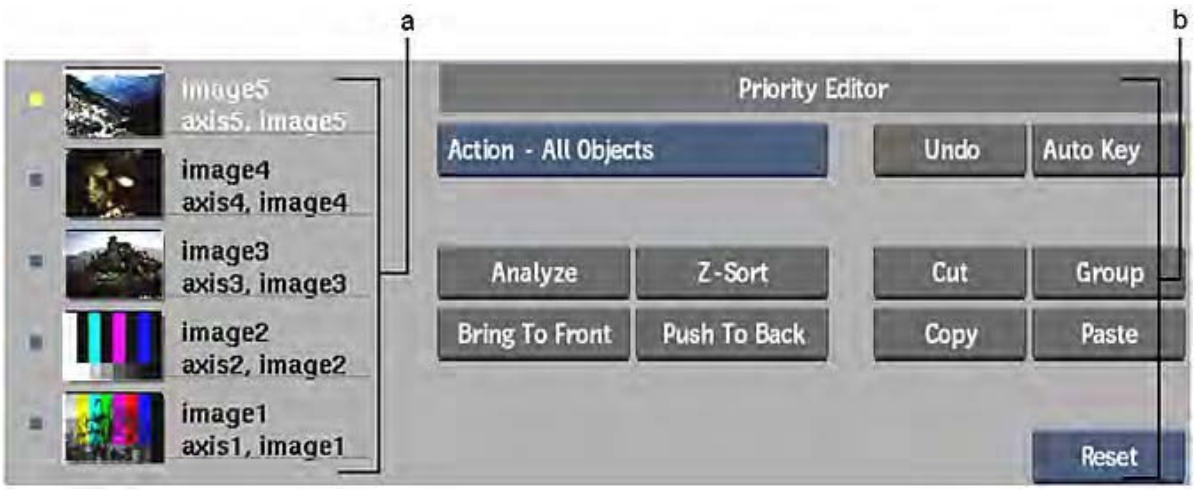
You can change the drawing order of surfaces, 3D models, and other objects using the Priority Editor.

Accessing the Priority Editor

Each entry in the Priority Editor indicates the object's name, the order of the object in the stack, and the location of the object in its branch. There is also a proxy showing what the object looks like in Schematic view.

To access the Priority Editor, click the Priority button, or swipe the bar at the bottom of the Media or Object menu.

NOTE If a source node is selected in the schematic, the Priority Editor that is displayed is that of the source node (the source node name is prepended to the branch name).



(a) Drawing order of objects in the scene (b) Priority Editor controls

Use the Priority Editor controls to analyse the scene for changes in rendering priority, to change the rendering priority, and to cut, copy, and paste priority information from one frame to another.

Changing the Drawing Order

The objects in the scene are shown according to z-depth, while still being drawn in the order of their priority: highest priority at the top of the list to the lowest priority at the bottom. The lowest level object is drawn first, the second to lowest level object is drawn next, and so on, until the top of the list.

To change the drawing order of objects:

1. In the Priority Editor, select the object.

   The selected object name is highlighted in yellow.
2. Click the Push To Back button to move the selected object one position lower in the stack, or click the Bring To Front button to move the selected object one position higher in the stack.

   When you move an object in the stack, a keyframe is added to the timebar.
3. Click Preview to ensure that the drawing order is correct.

   Occasionally, a matte may obscure another object if the drawing order is unordered. If this happens, move the surface up and down in the stack until the drawing order is corrected, or click Z-Sort.

TIP You can also select and drag a media (or multiple media using `Ctrl`) while holding the `spacebar`.

Grouping Media

Select objects in the Priority Editor and group them to edit or affect several media at once. You can analyse or Z-Sort groups as you would single media.

Groups are named according to their contents. For example, if a group is created from Image 3 and Image 32, the Group's label will be “Image 2 Image 32”.

It is not possible to add a scene object to an existing group. To do this, you must ungroup the objects, reselect those you want to include in the group, and click Group.

NOTE Single element groups cannot be created.

To create groups in the Priority Editor:

1. Select multiple scene objects in the Priority Editor using either `Alt`-click for individual objects or `Shift`-click to select a range.
2. Click Group in the Priority Editor menu.

   Media is minimized to a single line in the Priority Editor. To expand the group and view its contents, click the arrow to the left of the group.



3. Scene objects can be ungrouped by clicking Group once again.

Analyze, Z-Sort, and the Z-Buffer

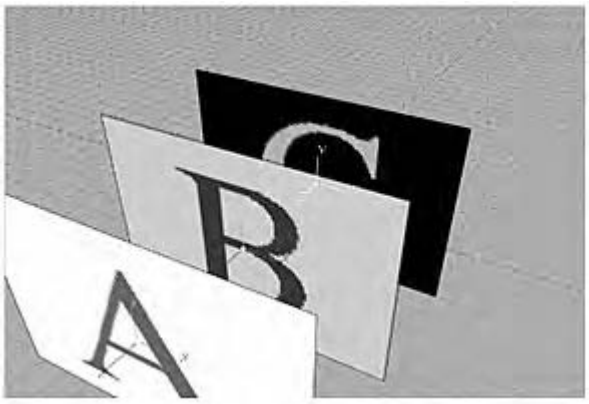
When you use Analyze or Z-Sort, the objects in the scene are compared using their position in the scene and not the individual pixels of a shape or model. The Priority Editor uses distance between the camera and the nearest and farthest points on the objects in the scene to determine the sorting order.

Since the Priority Editor is geometry based and not polygon based, you cannot properly order objects or groups that intersect due to their rotation, scale, or shear values. The Z-buffer uses the Z-value of each pixel for the sorting order. Use the Priority Editor in conjunction with the Z-Buffer box in the Setup menu Rendering section.



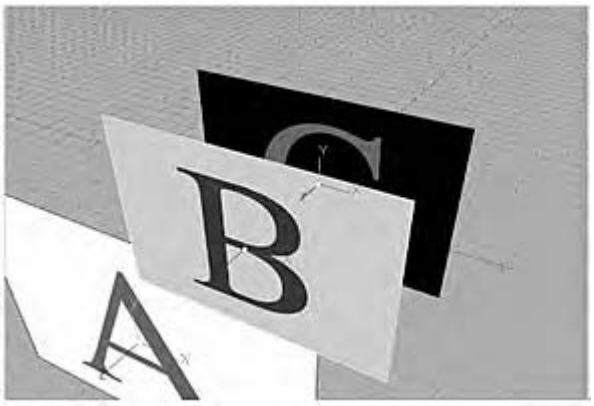
(a) Z-buffer box

When Z-buffer is on, objects and groups are arranged according to their distance from the camera eye. Since the camera is pointed towards the Z-axis by default, objects and groups are arranged according to their location on the Z-axis. In the following example, Image A is at Z position 200, Image B is at Z position 100, and Image C is at -50. When Z-buffer is on, Action draws these images as shown.



If you move the camera, objects are sorted according to the axis the camera is pointed towards.

When Z-buffer is off, the distance of objects from the camera eye is not considered. Objects are drawn in the order shown in the Priority Editor. To follow the previous example, you can turn off Z-buffer and change the priority of Image B so that it is drawn on top of both Image A and Image C. The Z position of these images is therefore ignored.



Shadow Mix, like Z-buffer, arranges objects according to their distance from the camera eye. Use Shadow Mix so that each shadow is rendered in the correct Z order with its corresponding surface.

Creating Keyframes Using the Priority Editor

When you edit media priorities in the Priority Editor, keyframes are not created by default. To create keyframes from the Priority Editor, the Auto Key button must be enabled. By default, the Auto Key button is set to off so as not to create keyframes when reordering media.

To create keyframes for a media priority change:

1. Display the Priority Editor by clicking the Priority button, or swipe the bar at the bottom of the Media or Object menu.
2. Click Auto Key to enable auto-keyframing.



3 Reorder priorities at different points in the timebar. Keyframes are automatically created as priorities are edited.

NOTE Stop all priority auto-keying anytime by disabling the Auto Key button.

Priority Editor Settings

Action Layers Selection box Based on the object selection in the Action scene, select an option to determine which Priority Editor to display.

Analyze button Click to analyze the entire scene for changes in rendering priority.

Z-Sort button Click to analyze the current frame for changes in rendering priority.

See Analyze, Z-Sort, and the Z-Buffer (page 482).

Bring To Front button Click to move the selected object one position higher in the stack, in front of the next higher object.

Push To Back button Click to move the selected object one position lower in the stack, behind the next lower object.

Undo button Undoes the last Priority Editor operation, except for Z-Sort or Analyze.

Use Action's Undo List to view a list of recent operations and revert to a prior state. Select an item in the list to return to that state. All actions that occurred after the selected item are undone.

Auto Key button Enable to create keyframes when reordering media. See Creating Keyframes Using the Priority Editor (page 483).

Group button Click to create groups of selected media (use this to analyze, Z-Sort, or manage several lines of media at once). See Grouping Media (page 482).

Cut button Click to cut priority information between frames.

Copy button Click to copy priority information between frames.

Paste button Click to paste priority information between frames.

NOTE A mark indicates when a change in priority occurs. If you copy and paste a mark that does not change the priority, no mark appears on the timebar.

Reset box Select whether to reset priority information for the current frame or for all frames.

About Stereoscopic Workflow in Action

A stereoscopic workflow in Action allows you to create stereo composites using 3D and stereo elements. With the stereo camera rig (the 3D camera), you can access stereo rigs and monoscopic cameras. With support for multiple outputs, you can experiment with any number of passes, including left and right scene output, as well as normals, Z-Depth, matte, media matte, and composition outputs.

When working in a stereoscopic compositing workflow in Action, there are three essential elements: a stereo camera, a stereo object for viewing and adjusting the result, and outputs. The following table outlines the stereoscopic workflow in Action.

Step:	Refer to:
1. Bring stereo clips into Action.	Starting a Stereoscopic Session (page 485).
2. Work with the stereo camera and stereo object to make any adjustments to the scene.	About the FBX Camera (page 657) and Working with the Stereo Scene (page 486).
3. Output various passes of your work.	About Rendering Outputs from Action (page 435).

Starting a Stereoscopic Session

You can start a stereoscopic session in Action by loading stereo clips and using the Stereo Startup mode. This creates a stereo camera. The output is set to the stereo camera, a stereo object is created for visualizing the scene, and the clips are placed on separate lines in the Media list.

NOTE A stereo clip cannot be loaded in Action if it is selected as the Back or MultiVersion input, or if it is selected to replace a clip in the Media list.

When using a stereo clip, its left and right eye tracks are automatically split and placed on individual lines in the Media list. Also, new clips appear in the workspace. They retain the name of the original clips, and are appended with a "_Left" or "_Right" suffix. It is important to save these clips, as they are used for loading an Action setup or loading the previous Action session.

NOTE It is not possible to select a mono clip as the Matte input if the Front input is stereo, and vice versa.

To start a stereo session from the Tools tab:

1. Click Action.
2. From the Input Mode box, select one of the following:
   - Clear All to delete all media and objects from the previous setup. Select stereo clips for the front and matte input. This is the Stereo Startup mode.
   - Front/Back/Matte to enter Action using the previous setup. Select stereo clips for the Front and Matte input, and a mono clip for the Back input.

   NOTE If you enter the Action module using mono clips, the default camera is automatically created. If you need a stereo camera, you must add it manually.
3. Select the destination.

If you selected Clear All and selected stereo clips for the Front and Matte input before entering Action (Stereo Startup mode), the following is created:

- In the schematic, a 3D camera (stereo camera) is created and the default camera is hidden.
- In the Output menu, the Mode is set to Stereo and the Camera is set to Result Cam.
- In the Camera menu, the result camera is set to the 3D stereo camera.
- A stereo object is created with the clips you selected.
- In the Media list, the Front and Matte inputs are split into separate lines for the right and left eye.

NOTE If you entered Action using the Front/Back/Matte option and want to clear all previous settings and use the Stereo Startup mode, click the Reset To Stereo Mode button in the Action Setup menu. You will now have to bring in new clips.

When you exit Action, the workspace contains left and right clips for the Front and Matte inputs for a total of four new clips.

Working with the Stereo Scene

When working in a stereoscopic workflow, the stereo object lets you visualize the scene. The stereo object lets you composite stereo sources in Action, ensuring the stereo effect of the stereo source is preserved during the compositing process. You can combine stereo objects with 3D geometry, such as 3D text or FBX models within the same scene.

The stereo object is a single image surface which contains two diffuse maps (left and right) for handling stereo sources. When working in a stereoscopic workflow in Action, you must handle the left and right eye as separate media in the Media list. When a stereo object is filmed by a stereo camera, a link is created between the left image of the object and the left camera of the stereo rig. This is the same for the right image and right camera. The link ensures that left-eye material is only visible through the left camera, and likewise, right-eye material is handled with the right camera.

NOTE Entering Action with a stereo clip will automatically create a stereo object with the stereo clip used as the left and right material.

To add a stereo object to a scene:

1. In the Media list, select the media for the left eye. Press `Ctrl` and select media for the right eye.

   NOTE By default, the first clip you select is the media for the left eye. You can select multiple pairs of left-right media. Odd numbered selections are considered as left media and even numbered selections are the right media when creating stereo objects.

2. Create a stereo object for the media by doing one of the following:
   - Drag the Stereo Object node from the node bin and place it in the schematic.
   - Drag the Stereo Object node from the node bin and place it where you want it in the Result view.
   - Double-click the Stereo Object node. You do not need to be in Schematic view to add a node in this manner.

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

   NOTE The viewport must be set to Result.

4. To open the Stereo Object menu, double-click the StereoObject node in the schematic.

Stereo Object Settings

Stereo Object Axis Settings

Use the Axis tab of the Stereo Object menu to position, rotate, scale, and shear an axis, as well as adjust the convergence, parenting, and autoscaling of stereo objects.



The Position, Rotation, Scale, and Centre settings are the same as in the Axis menu. The following Axis settings are specific to the Stereo Object:

Autoscale box Select whether to autoscale when settings are changed relative to the camera.

Select:	To:
Auto Off	Not use autoscaling on the image.
Auto Z	Link the Position Z parameter with Convergence. This allows you to see how a change in the Convergence value makes the object appear to move closer or farther from the camera. This preserves the same visual aspect while scaling the textures up or down accordingly. **NOTE** The result camera must be set to the stereo camera.
AutoScale	Change the left and right texture parameters when the Position Z or a Scale parameter is changed. When you create a stereo object, it is automatically oriented towards the camera and automatically scales. That is, the apparent size of the images scale to compensate when you move the stereo object along the Z axis. If you move a stereo object away from the camera, it grows, and vice versa. **NOTE** The result camera must be set to the stereo camera.

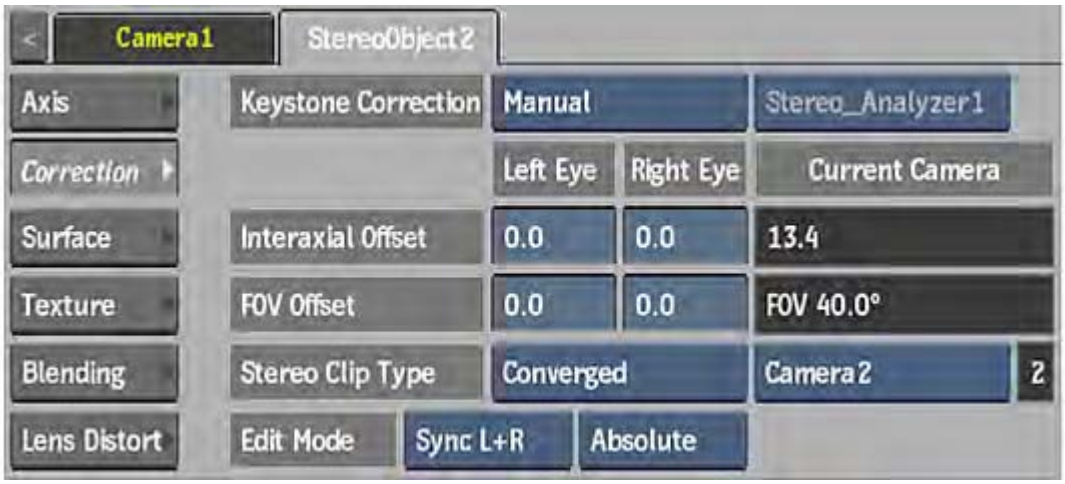
Convergence field Displays the equal amount of convergence in opposite directions horizontally, between left and right images. Positive values make the image appear farther from the camera. Editable.

Parent To button Enable to associate a stereo object with the camera selected in the Parent Camera box. Enabled by default to force the stereo object to face the 3D camera at all times when the camera is moved around.

Parent Camera box Select which camera in the scene is associated with the stereo object. Setting found in the Axis and Correction tabs.

Stereo Object Correction Settings

Use the Correction tab of the Stereo Object menu to indicate how the footage was shot, and make interaxial offset and FOV offset adjustments to the left/right images.



Keystone Correction box Select whether to apply keystone corrections manually in this menu, use the automatic settings derived from the selected analyzer, or inherit the Stereo Mode settings from the stereo camera (Pass Through).

In most cases, a keystone correction is needed if the stereo types of the camera and the stereo object do not match, but there may be cases when you want to sync the two.

Correction Analyzer box Select which analyzer is used to provide automatic keystone corrections.

Interaxial Offset Left Eye field Displays the offset distance between the left eye compared to the 3D camera. Editable.

Interaxial Offset Right Eye field Displays the offset distance between the right eye compared to the 3D camera. Editable.

Interaxial Offset Current Camera field Displays the interaxial offset of the selected camera. Non-editable.

FOV Offset Left field Displays the offset field of view between the left eye compared to the 3D camera. Editable.

FOV Offset Right field Displays the offset field of view between the right eye compared to the 3D camera. Editable.

FOV Offset Current Camera field Displays the offset field of view of the selected camera. Non-editable.

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

Parent Camera box Select which camera in the scene is associated with the stereo object. Setting found in the Axis and Correction tabs.

Camera Number field Displays the number of the selected camera. Non-editable.

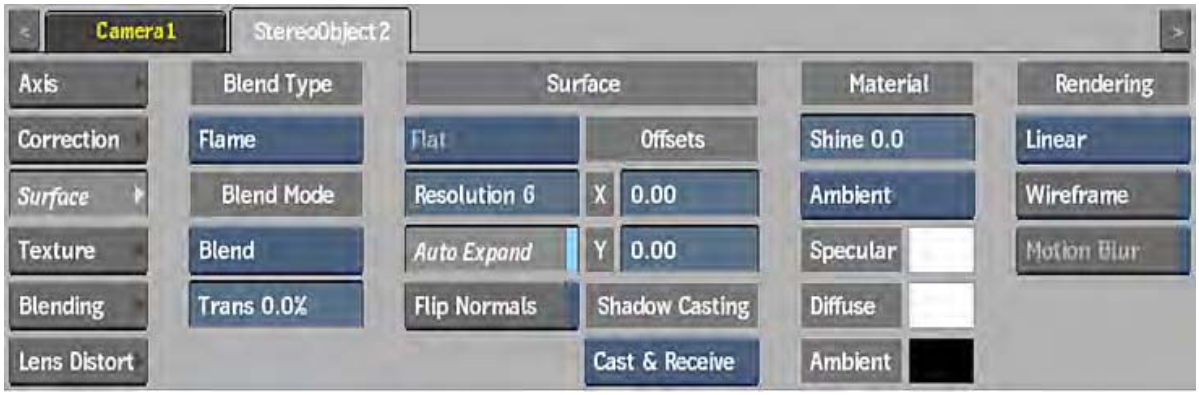
Edit Mode box Select whether to sync the left and right eye correction values.

When synced, a change to one value affects the other value. The left eye position is a combination of the camera's left eye position and the left interaxial offset; the same applies for the right eye. The effective FOV for the correction is the sum of the camera's FOV and the FOV offset.

Sync Type box Select whether the relationship between the left and right eye correction values is absolute or relative. Available when Sync L+R is selected in the Edit mode box.

Stereo Object Surface Settings

Use the Surface tab of the Stereo Object menu to adjust surface and shading properties of the stereo object.



The Surface tab settings are the same as in the Object Image menu for non-stereo objects. See Surface Settings (page 458).

Stereo Object Texture Settings

Use the Texture tab of the Stereo Object menu to set the built-in diffuse maps (as well as any added texture maps) of a stereo object.



X Position field Displays the position of the X axis. Editable.

Y Position field Displays the position of the Y axis. Editable.

Z Position field Displays the position of the Z axis. Editable.

X Rotation field Displays the rotation of the X axis. Editable.

Y Rotation field Displays the rotation of the Y axis. Editable.

Z Rotation field Displays the rotation of the Z axis. Editable.

X Scale field Displays the scale of the X axis. Editable.

Y Scale field Displays the scale of the Y axis. Editable.

Z Scale field Displays the scale of the Z axis. Editable.

Proportional button Enable to change the fields proportionally.

X Shear field Displays the shear of the X axis. Editable.

Y Shear field Displays the shear of the Y axis. Editable.

Z Shear field Displays the shear of the Z axis. Editable.

X Centre field Displays the centre of the X axis. Editable.

Y Centre field Displays the centre of the Y axis. Editable.

Z Centre field Displays the centre of the Z axis. Editable.

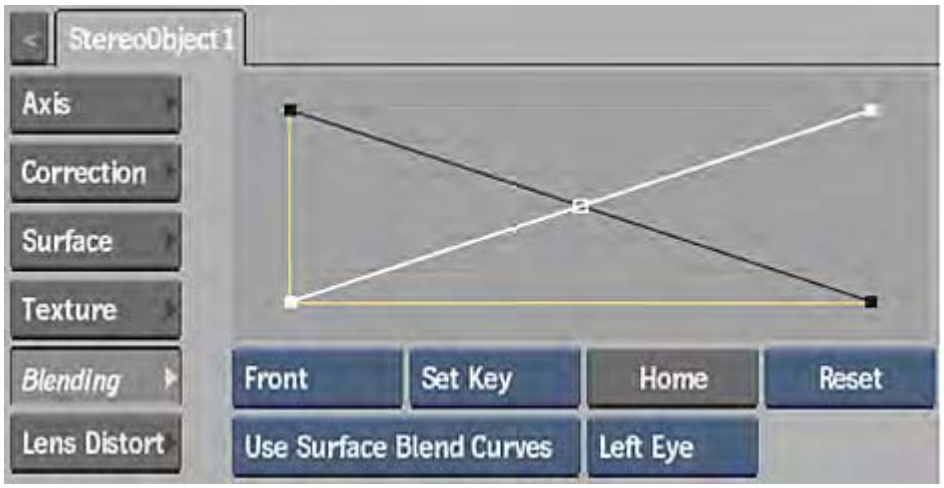
Edit Mode box Select whether to sync the left and right eye texture values.

Sync Type box Select whether the relationship between the left and right eye texture values is absolute or relative. Available when Sync L+R is selected in the Edit mode box.

Texture Eye box Select whether to apply changes to the left or right eye and display the values of the left or right texture.

Stereo Object Blending Settings

Use the Blending tab of the Stereo Object menu to adjust the blending curves of each stereo surface separately. The blending curve is similar to the Keyer luminance blending curve, but you can adjust it per eye.



You can switch between the stereo surface blend curves and the keyer blend curves. This gives you a good comparison to luminance curve work you have already done in the Keyer.

The Blending tab settings are the same as in the Object Image menu for non-stereo objects. See Applying Blending Curves per Surface (page 470). One extra setting is available for stereo objects, allowing you to select which eye to apply blending curves.

Eye Selection box Select which eye to apply the blending curves. Select Both Eyes to apply the same blending settings to both eyes. In this case, the settings of the last selected eye apply to both eyes.

Stereo Object Lens Distort Settings

Differences in camera lenses or perspective irregularities cause lens distortion that results in skewed angles. Use the settings in the Lens Distort tab to rectify or simulate these types of distortions in your stereo images.



The Lens Distort tab settings are the same as in the Object Image menu for non-stereo objects. See Lens Distort Settings (page 464).

About the Perspective Grid

Use the Perspective Grid node to help you with perspective alignments in your Action scene. The perspective grid creates a rectangle on a plane within the perspective of the clip. The accuracy of the plane determines whether any adjustments are properly scaled and oriented in your clip. Objects can be placed anywhere on this plane. Aligning objects to the perspective grid has multiple benefits:

- You can perform a 4-corner pin with a surface or geometry that respects the perspective.
- Aligning the grid establishes the orientation of the plane in 3D space and the Field of View (FOV) of the camera that shot the scene.
- Because the Perspective Grid node has an axis-like behaviour, objects attached to it inherit the grid's 3D transformations.

Working with FOV

You can animate the perspective grid by positioning corner points in other frames. To create an animated transform, place the grid in the first frame, then Lock Camera in other key frames. You would also enable the Lock Camera if the camera FOV does not change over the clip.

NOTE If you enable Allow Adjustments, Action will solve for the FOV in each frame on which you edit the corners.

To use the existing value in the camera and not solve for it, regardless if you animate the perspective grid, you should lock Camera before positioning in the first frame.



NOTE When the FOV of the camera is computed (not locked) the camera is repositioned in order to frame the media.

Aligning an Object with a Perspective Grid

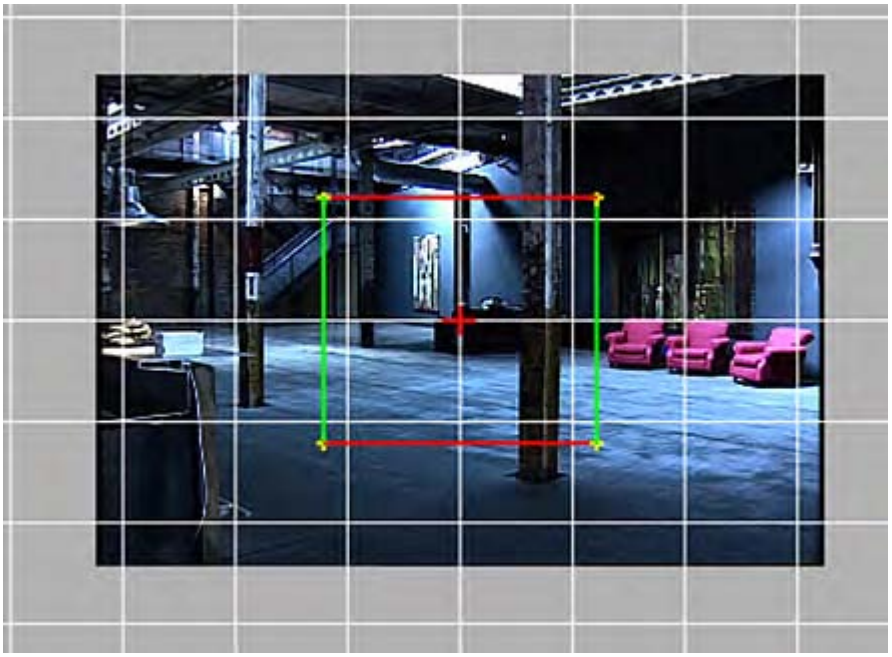
To align an object with a perspective grid:

1. Click Media.
2. In the Media menu, select the media you want to use for the perspective grid (you can change the media later in the Perspective Grid menu).

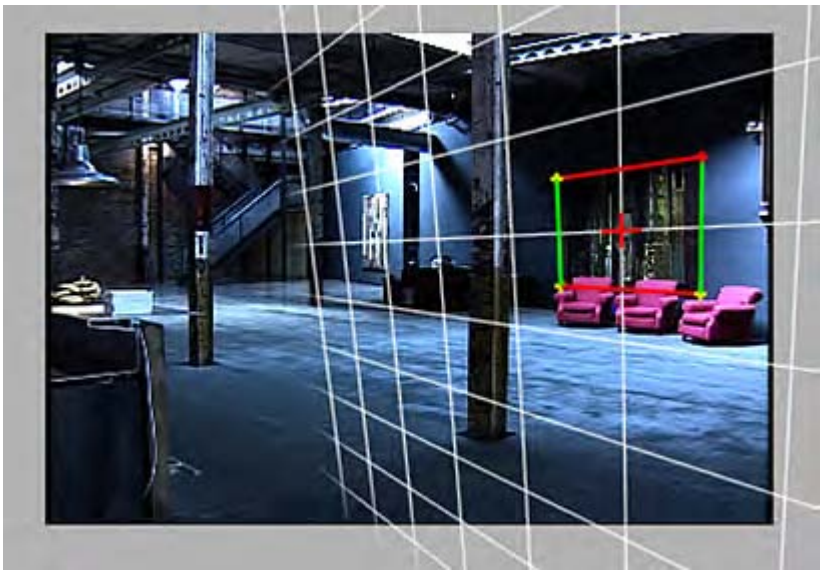
   This media becomes the reference for the perspective grid alignment.
3. Do one of the following:
   - Drag the Perspective Grid node from the node bin and place it in the schematic.
   - Drag the Perspective Grid node from the node bin and place it where you want it in Result view.
   - Double-click the Perspective Grid node. You do not need to be in Schematic view to add a node in this manner.

   The Perspective Grid object is added to the schematic. In the Schematic view, the number in brackets next to the name of the node indicates the media used for the perspective grid reference.

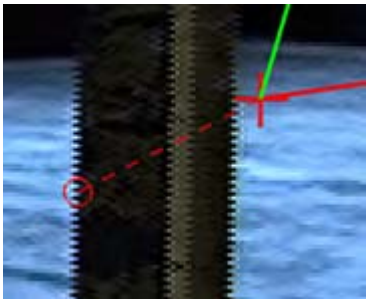
   To specify different perspective grid reference media, select the media in the Media menu, then click Apply, or enter a different media number in the Use Media field of the Perspective Grid menu.
4. With the Perspective Grid object selected in the schematic, select Object view from the View box to see the grid and the selected media.



5 While in Object view, drag each of the four corners of the rectangle to the desired location. The grid automatically aligns to the new perspective.



NOTE When dragging one of the corner points of the rectangle, if your desired location cannot be computed (for example, if a point goes past another point on the plane), the location is remembered with a dotted line and red circle. Once you move other corners, your original location may now become viable, and the rectangle and grid align properly.



- If Allow Adjustments is selected in the Lock Camera box , the target camera FOV and position adjusts to fit the perspective set by the perspective grid. If Lock Camera is selected in the Lock Camera box, the perspective grid does not adjust the target camera.

6 Once you are satisfied with the alignment of the perspective grid rectangle, parent an object to the Perspective Grid node. The object inherits the perspective transformation of the Perspective Grid node. Switch to Result view (F4) to see the complete scene.

NOTE You can also manipulate the perspective grid in the Result view, provided that Lock Camera is not enabled in the Perspective Grid menu, or the perspective grid camera is the same as the Result camera.

Perspective Grid Menu Settings



Lock Camera box Select Allow Adjustments to allow adjustments of the Target camera FOV and position. Select Lock Camera to lock camera FOV and position.

Perspective Camera box Specify which camera to view the perspective grid in Object view, and for modifying its FOV when working with the perspective grid.

Perspective Camera field Displays the active perspective camera number. Non-editable.

Stereo Camera Perspective box Select whether to use the left or right camera from a stereo camera rig when working with the perspective grid.

FOV field Displays the field of view of the active perspective camera. When adjusting the perspective grid, the FOV of the target camera updates automatically (unless Lock Camera is enabled). Non-editable.

Width field Displays the width of the four-point perspective grid rectangle. Non-editable.

Height field Displays the height of the four-point perspective grid rectangle. Non-editable.

Lock Size button Enable to lock the width and height of the perspective grid rectangle. For example, use to lock the dimensions of the grid when animating it, if the feature size you aimed for in the image doesn't change in time.

Use Media field Displays the number of the media associated with the perspective grid. You can change the media number in this field, or select a media entry in the Media List and click Apply.

Z Offset field Displays the amount of offset to apply along the Z axis to make the grid larger or smaller while maintaining the visual result.

Display Grid button Enable to display the grid lines in Result or Object view.

Grid Colour pot Displays the colour of the grid lines. Click to change the colour.

Magnifier button Enable to display the magnifier while dragging a corner of the perspective grid rectangle in Object view.

Zoom field Displays the zoom factor of the magnifier. Editable up to 5x.

Reset button Resets the perspective grid settings, but does not reset the active perspective camera settings.

About Lights and Lighting Effects

Objects are lit up in the scene according to the number, position, direction, and colour of light sources, as well as the rotation and spread of each light source.

You can add up to 64 active light sources to a scene. You can also control each light individually. By default, the light you add to the scene is applied to all surfaces. However, you can also apply a light source to specific surfaces.

Once lights are added to your scene, you can use them to add other effects to enhance the scene, such as cast shadows, lens flares, rays, and blooming effects.

You can also use image-based lighting (IBLs) to light objects in the scene.

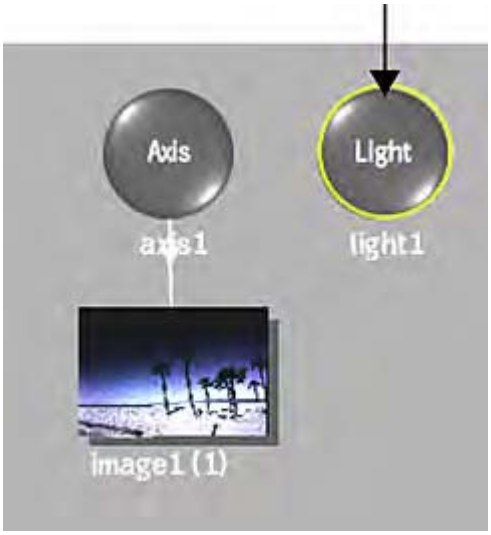
Adding a Light Source

When you add a light to a scene, it is applied to all objects. Before adding another light, you may want to position the light and modify its attributes. In the Lights menu, you can set lighting properties such as position, orientation, spread, falloff, and colour.

To add a light to the scene:

1. Do one of the following:
   - Drag the light node from the node bin and place it in the schematic.
   - Drag the light node from the node bin to Result view, so you can see its effect on the scene before placing it exactly where you want.
   - Double-click the light node. The node appears next to the last added object. You do not need to be in Schematic view to add a node in this manner.

   A light is added to the scene. An icon representing the light source is added to the schematic.

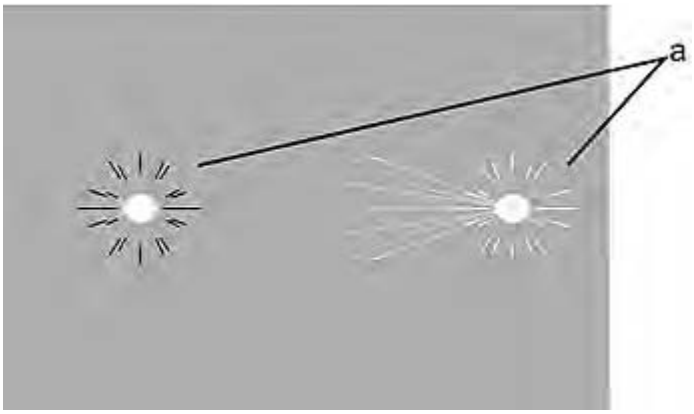


Unlike many objects, a light is added without an axis. To set the position and rotation of a light source, use the Position and Rotation fields in the Light menu.

2 To display the Light menu, double-click the selected light in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 415).

3 Enable the Active button in the Light menu to activate the light source.

The light icon appears in the scene.



(a) Light sources in the scene.

Importing Lights

In Action, you can import 3D data saved in the FBX format. FBX files may have lights and their data included, and you can use these same lights in Action. Supported light types from FBX files are Area, Directional, Ambient, Point, or Spot. See Importing the FBX Format (page 536).

Selecting a Light Source

You can select a light source in the following ways:

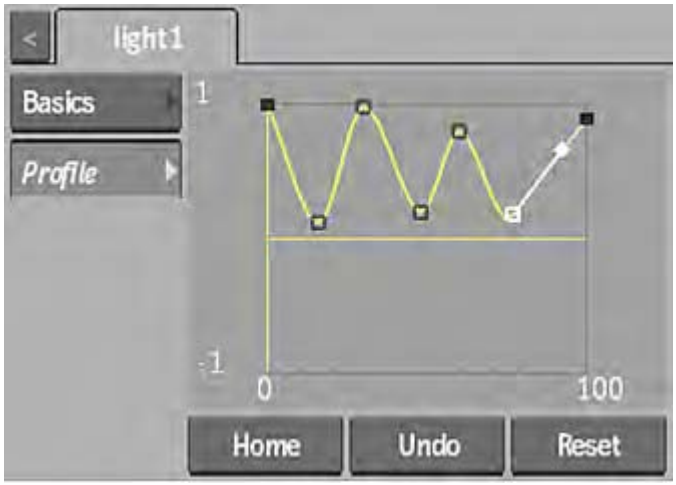
- Click the light source in the scene.
- In Schematic view, click the icon corresponding to the light source. All light sources initially appear at the same X, Y, and Z position in the scene (0, 0, 0). If you add two light sources, for example, you need to move one light source in order to see the other.

- Select a Light folder in the Channel Editor.
- Use the Prev and Next buttons to select the previous or next light source.

Using the Light Bevel Curve

The bevel curve allows you to create unique lighting effects in conjuntion with the Spread field in the Light Basics tab. Since a spread value of 90 or less creates a spotlight, you can then create a bevel curve to act as a multiplier of the spotlight intensity.

For example, a curve such as this.



Results in a lighting ripple-like effect.



TIP To create black holes in your lighting effect, use values below zero on the Bevel curve.

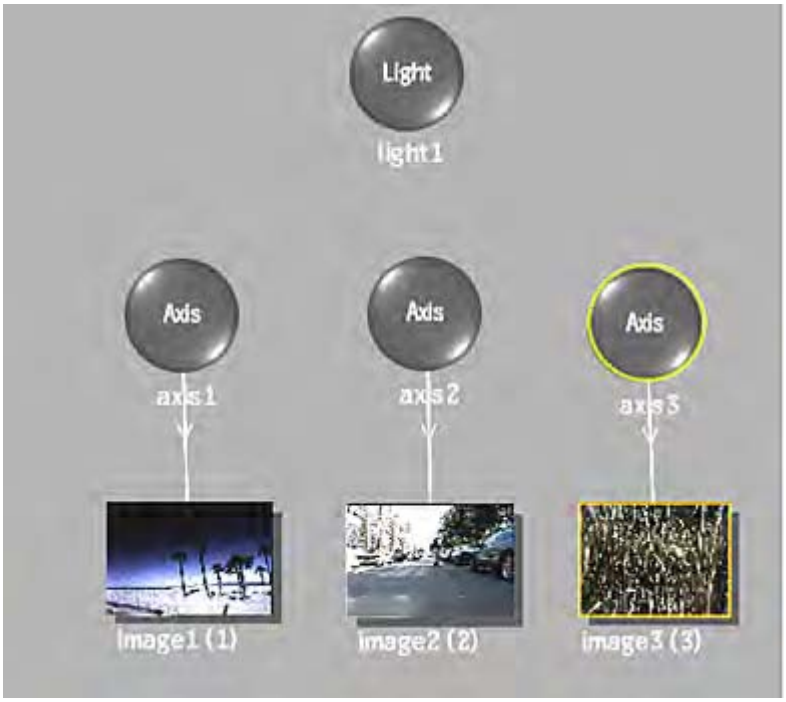
Applying Selective Lighting

When you add a light source to a scene, the light is applied to all surfaces. You may want a light source to only illuminate an individual or specific group of surfaces, or prevent a light source from illuminating an individual or specific group of surfaces.

To apply selective lighting:

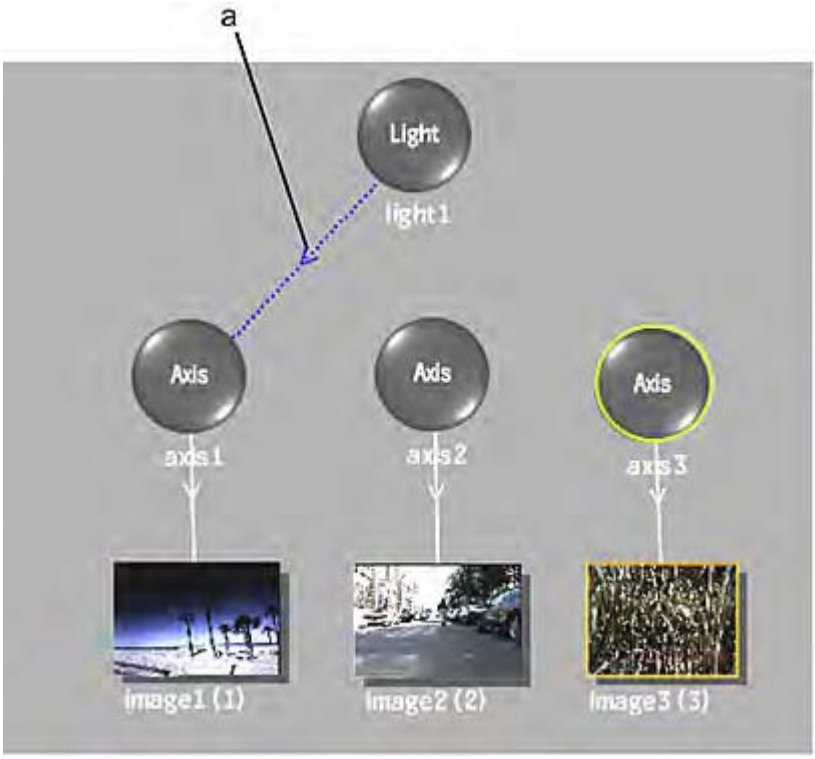
1. Add multiple surfaces to the scene.
2. Add a light to the scene.

   All surfaces are illuminated.



3. Select Lighting from the Edit Mode box.
4. To illuminate only a selected surface, click the light node, and drag it to an axis or image you want illuminated.

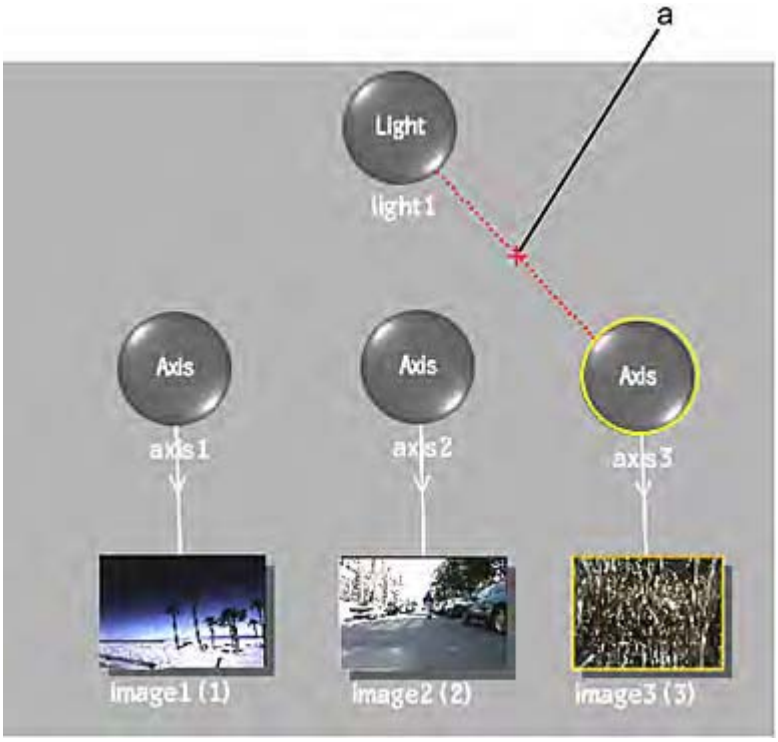
   The selected object is connected to the light source by a blue dotted line with an arrow, and only the selected surfaces are illuminated.



(a) Light inclusion link

5 To exclude a surface, hold the `Alt` key while clicking and dragging from the light source to the surface you do not want illuminated.

Excluded surfaces are connected to the light source by a red dotted line with an "X", and they are not illuminated. In the following example, all surfaces are illuminated, except for image3, which is excluded.

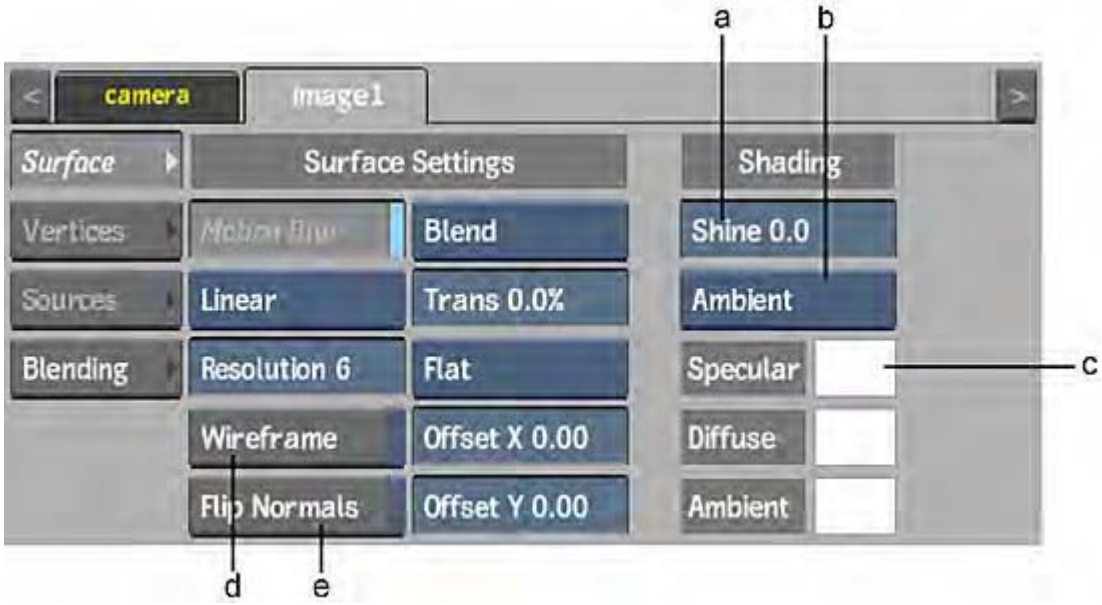


(a) Light exclusion link

NOTE To remove the inclusion or exclusion link, while still in Lighting mode, click and drag over the line that connects the light source to the axis or image.

Modifying Surface Lighting

After you set the parent-to-child relationships between lights and surfaces, you can set specific surface lighting properties. You control a surface's lighting and adjust the surface's specular highlight in the Surface menu. To access the Surface menu, double-click the selected surface in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).



(a) Shine field (b) Lighting box (c) Specular Highlight colour box (d) Flip Normals button (e) Wireframe button

Surface Controls Settings

The Surface controls that relate to lights are described as follows.

Motion Blur button Enable to exclude the selected object from the global motion blur effect. See Blurring a Single Object (page 654).

Wireframe button You can remove some lighting effects for a selected surface by converting surfaces to a wireframe depiction of the surface. See Converting to Wireframe (page 502).

Flip Normals button Flips the normals of the surface so that light is applied to the opposite side of the surface. See Flipping a Surface's Normals (page 503).

Shine field Displays the intensity of the specular highlight. See Adjusting Specular Highlights (page 501).

Lighting box Applies diffuse or ambient lighting to a surface. See Applying Incidental Light Reflection (page 501).

Specular Highlight colour box Click the colour pot and use the colour picker to change the specular colour.

The specular highlight is visible only if Shading is enabled and Shine is greater than 0. See Adjusting Specular Highlights (page 501).

About Specular Highlights

A specular highlight is a reflection of a light source. The position of the specular highlight depends on the position and number of light sources surrounding a surface and the angle of the camera.

Use the Shine field to change the intensity of the specular highlight. When the Shine value is set to 0, the specular highlight is disabled. To change the size of the specular highlight, use the Falloff field in the Light menu.



This surface is lit using a Falloff of 30 and a Spread of 27. The surface's Shine is set to 10.



This surface is lit using a Falloff of 20 and a Spread of 27. The surface's Shine is set to 1.

By default, the specular highlight is the same colour as the light source. You can change the colour of the specular highlight by changing the specular colour values.

The specular colour is the colour of light that is reflected by the surface. For example, if the specular colour is red and the light source is white, the specular highlight is red. If the specular colour is yellow and the light source is red, the highlight is orange.

Adjusting Specular Highlights

To adjust specular highlights:

1. Add and position a light source in the scene.
2. In the Light menu or the Rendering section of the Action Setup menu, enable Shading.
3. In the Surface menu, use the Specular Highlight colour pot to set the specular colour.
4. Set a value in the Shine field. A small Shine value produces an intense highlight while a large Shine value produces a dim highlight.

Applying Incidental Light Reflection

Set how a surface reflects incidental light by applying ambient or diffuse lighting. The actual colour of the reflection depends on both the colour value of each pixel and the colour of the incidental light. The intensity of the reflection depends on the orientation of the light source relative to the surface; it is greatest where the incident light strikes the object perpendicular to its surface. The intensity of the reflection is independent of the camera eye position.

When you turn shading on, you do not have to enable a light source to see the lighting effect, as a default infinite light source supplies ambient light at 20% intensity. The infinite light source is located behind the camera eye and cannot be moved. As soon as you add a light source, the infinite light source is replaced by the new light source.

To apply incidental light reflection:

1. In the Light menu or the Rendering section of the Action Setup menu, enable Shading.
2. In the scene, select the surface to receive incidental light reflection.
3. In the Surface menu, make a selection from the Lighting box.



Select:	To reflect incidental light:
Ambient	To all parts of a surface that are not directly illuminated.
Diffuse	Equally in all directions, producing a flat reflection on the object.

4. Optional: Adjust the colour of the incidental light using the ambient or diffuse RGB channels.

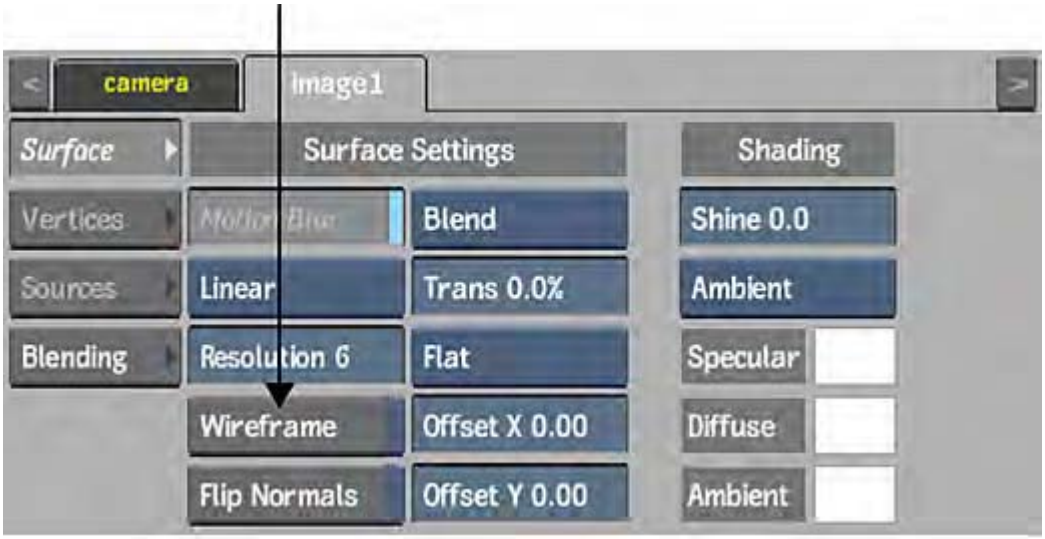
   To display the RGB channels, click Animation to display the Channel Editor. Expand the surface's folder (it should already be selected), expand its Material folder, then expand the Ambient or Diffuse folder.

Converting to Wireframe

You can remove some lighting effects for a selected surface by converting surfaces to a wireframe depiction of the surface. When Wireframe is enabled, specular values and any applied textures are replaced with a wireframe view of the surface. When used on an image, the surface will adopt a screen-like look.

To convert to wireframe:

1. In the scene, select the surface to convert to wireframe.
2. In the Surface menu, enable Wireframe.



The selected surface becomes a wireframe surface.

Flipping a Surface's Normals

When you flip a surface's normals, light is applied to the opposite side of a surface. Use this feature to create a two-sided shaded surface.

To create a two-sided surface:

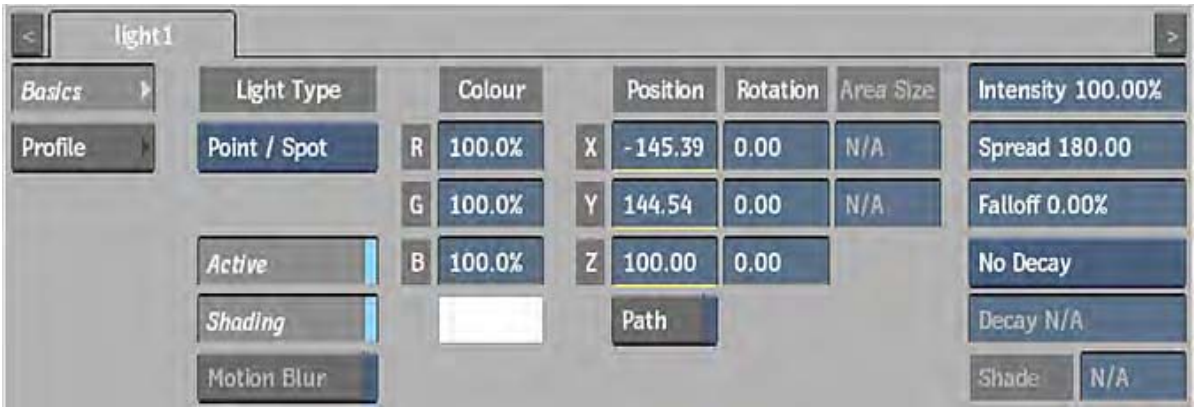
1. Add a surface and a light source.
2. In the Light menu or the Rendering section of the Action Setup menu, enable Shading.
3. Copy the surface and its axis in Schematic view. See Mimicking, Copying, and Duplicating Objects (page 421).
4. Select the surface of the copied branch.
5. In the Surface menu, enable Flip Normals.



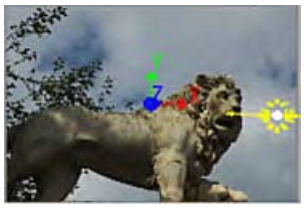
To control both surfaces, parent them by a new axis and use this axis to rotate, scale, shear, and move the two surfaces. There may be a priority problem causing one surface to be drawn over the other. To correct this problem, use the Priority Editor to animate the drawing priority of surfaces or change the Z position of one surface by one pixel. See Reordering Surfaces (page 480).

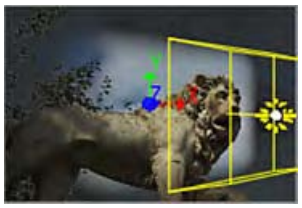
Light Menu Settings

The Light menu settings are described as follows.



Light Type box Select the type of light to apply to the scene.

Light Type:	Description:	Example:
Point / Spot	A point light radiates light uniformly in all directions. A spotlight radiates a cone of light centred along the spotlight direction. Use the Spread field to change the spread angle. A spread of 90 or less creates a spotlight.	
Directional	A directional light shines evenly in one direction only. The light icon in the scene displays an arrow showing the direction of the light. Use a directional light to simulate a very distant point light source (for example, the sun as viewed from the surface of the Earth).	
Ambient	An ambient light shines in two ways—some of the light shines evenly in all directions from the location of the light (similar to a point light), and some of the light shines evenly in all directions from all directions (as if emitted from the inner surface of an infinitely large sphere). Use the Shade field to set the percentage on ambient light applied.	

Light Type:	Description:	Example:
Rectangle Area	A rectangle area light is similar to a point or spot light light except that it produces a hotspot based on the shape of the rectangle. Use the area size fields to set the size of the rectangle.	
Ellipse Area	An ellipse area light is similar to a point or spot light light except that it produces a hotspot based on the shape of the ellipse. Use the area size fields to set the size of the ellipse	

Active button Enable to turn the selected light source on.

Shading button Enable to light up the scene using added light sources. When Shading is disabled, no lighting effects appear in the scene; surfaces and 3D models appear flat.

Enable Shading for:

- Light sources
- Ambient or diffuse lighting for surfaces
- Specular highlights for surfaces and 3D models

This same button appears in the Rendering section of the Action Setup menu.

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

Red Light field Displays the red value of the selected light. Editable.

Green Light field Displays the green value of the selected light. Editable.

Blue Light field Displays the blue value of the selected light. Editable.

Light colour pot Displays the colour of the light source. Editable.

X Position field Displays the position of the selected light along the X axis. Editable.

Y Position field Displays the position of the selected light along the Y axis. Editable.

Z Position field Displays the position of the selected light along the Z axis. Editable.

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

X Rotation field Displays the rotation of the selected light along the X axis. Editable.

Y Rotation field Displays the rotation the selected light along the Y axis. Editable.

Z Rotation field Displays the rotation the selected light along the Z axis. Editable.

Width field Displays the width of an area or directional light. Editable.

Height field Displays the height of an area or directional light. Editable.

Light Intensity field Displays the intensity of the selected light. Editable.

Light Spread field Displays the spread angle. A value of 90 or less creates a spotlight. Editable

Light Falloff field Displays the amount of falloff around the edge of the light source (also changes the size of the specular highlight). Editable.

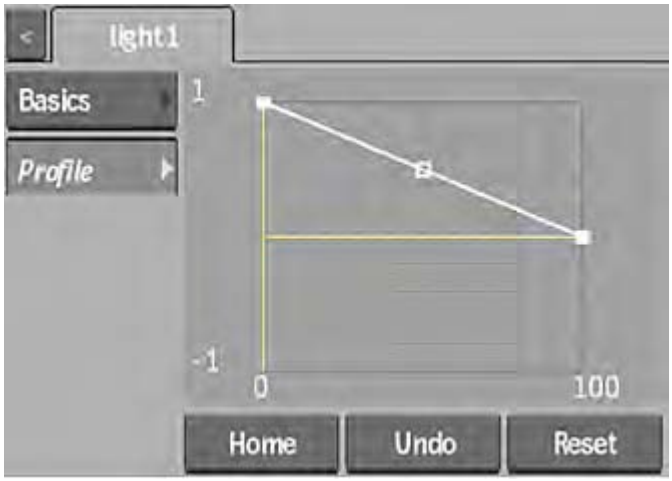
Decay Type box Select the type of decay to apply to the light source.

Decay field Displays the rate at which light decreases for the chosen decay type. Editable.

Shade field Displays the percentage of ambient light applied. Editable.

Light Profile Settings

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



Light Bevel curve Adds a specific profile to the selected light. Use the options in the Edit Mode box to add, select, delete, or move keyframes on the bevel curve. The bevel curve behaves in much the same way as an animation curve in the Channel Editor. See Using the Light Bevel Curve (page 497).

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

Undo button Undoes the last set of Bevel curve operations.

Reset button Resets the Bevel curve.

Relighting: Casting Shadows

You can create realistic 2D and 3D shadows in your Action scene by using lights to cast and receive shadows, as well as self-shadow (an object can cast a shadow on itself).



After adding and setting up shadow casts in Action, you can output the shadow by itself, or as part of your overall composition. A shadow pass is a white image with greyscale regions that represent the shadow coverage. See Output Options (page 436) for information on the specific shadow output settings.

Action also supports drop shadows (see Adding Drop Shadows (page 478)).

Adding a Shadow Cast to a Light

Lights in the scene are able to cast shadows. You can parent a Shadow Cast object to multiple lights at once. This allows you to control the overall attributes of the shadow (for example, colour, softness, and transparency).

To add a shadow cast to the scene:

1. Add and position a light to your scene.
2. Do one of the following:
   - Drag the shadow cast node from the node bin and place it in the schematic.
   - Double-click the shadow cast node.

   If there is only one light in the scene, the shadow cast node is automatically connected to it. If there are several lights in the scene, select the light in the schematic that you want parented to the shadow cast before added the shadow cast node. Otherwise, you can parent the shadow cast node to the light or lights manually in the schematic.
3. To display the Shadow Cast menu, double-click the shadow cast object in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 415).

Surface and Geometry Shadow Casters

By default, all objects in the scene cast and receive shadows from a light attached to a Shadow Cast object. You can control this on a per object basis in the Object Surface or Geometry menu.

The Shadow Casting box in the Surface or Geometry menu allows you to select how the each image or geometry object is affected by a Shadow Cast object in the scene.

Select:	For the Surface or Geometry to:
Cast & Receive	Cast and receive a shadow (this is the default).
Receiver	Receive, but not cast a shadow.
Caster	Cast, but not receive a shadow.
Shadow Only	Not be displayed, but the shadow is displayed.
Off	Not cast or receive a shadow.

Shadow Cast Menu Settings

The Shadow Cast menu settings are described as follows. Shadow types are dependant on your graphics card, so you may not see all of these settings.



Shadow Output Type box Select the type of shadow to output. This setting is repeated in the Output Options section of the Output menu.

Source Shadow Type box Select the type of shadow cast for the source node. This setting is available in the Shadow Cast menu only if the shadow cast node is a child of a source node, and is repeated in the Source menu.

Shadow Type box Select a mapping type for the shadow.

Shadow Type:	Description:
3D Hard Shadows	Use 3D Hard Shadows for higher precision shadows when lights are close to the objects in the scene. These are best for hard edge shadows with penumbra effects, but can be slower.

Shadow Type:	Description:
3D Soft Shadows	Use 3D Soft Shadows if you want faster soft shadows with lights further away from your objects. These shadows offer explicit control over softness.
2.5D Shadows	2.5D Shadows work best on transparent or semi-transparent objects, for example a shadow projected on a wall by smoke.

NOTE Some of the settings in the Shadow Cast menu differ based on the shadow type you choose.

Resolution box Select a resolution to determine the quality of the selected mapping type.

Anti-Aliasing Sample box Select an anti-aliasing sampling level for the shadow cast.

NOTE Depending on your graphics card and the size of your Action scene, shadows may not appear or render properly with higher anti-aliasing sampling levels.

Anti-Aliasing Softness field Displays the softness value of the anti-aliasing sample for the shadow cast.

NOTE Jitter on 3D shadows is inherent to the shadow map technique. However, in most situations it can be go completely unnoticed. Some lighting situations which cause stretched shadows, such as lights close to the horizon, are very likely to create visible jittering. To alleviate jittering problems, you can try to use higher resolution, anti-aliasing, and anti-aliasing softness levels. In these cases, interactive manipulations in the image window may become taxing; therefore, you should activate Adaptive Degradation (page 430) for Shadows.

Red Colour field Displays the amount of red in the shadow (based on the colour of the attached light). Editable.

Green Colour field Displays the amount of green in the shadow (based on the colour of the attached light). Editable.

Blue Colour field Displays the amount of blue in the shadow (based on the colour of the attached light). Editable.

Shadow Colour pot Displays the colour of the shadow (based on the colour of the attached light). Editable.

Proportional button Enable to change the Red, Green, and Blue colour fields proportionally.

Transparency field Displays the transparency level of the shadow. Editable.

Decay Type box Select the type of decay to apply to the shadow.

Decay field Displays the rate at which the shadow decreases for the chosen decay type. Editable.

Softness field Displays the softness of a shadow. Editable.

Density field Displays the amount of dark values to include in the softness of a 3D soft shadow. Editable.

Dark Threshold field Displays the amount of light values to include in the softness of a 3D soft shadow. Editable.

Focus field Displays the softness of the 2.5D shadow based on the distance from the light. Objects closer to the focus distance are less blurred. Editable.

Colour Bleed field Displays the amount of colour bleed in the 2.5D shadow from semi-transparent objects in the scene. Editable.

Flattening Mode box Select a flattening mode for the 2.5D shadow. Most of the time, Best Fit gives the best quality, but if you see clipping artefacts in the shadow, try one of the other modes.

Penumbra field Displays the softness of a 3D hard shadow. Editable.

Sampling Mode box Select a softness sampling mode for the 3D hard shadow.

Filter Samples field Displays the amount of filter samples to take into account when creating softness (X xY) for a 3D hard shadow. Available when Regular sampling is chosen in the Sampling Mode box. Editable.

Caster Samples field Displays the amount of shadow caster samples to take into account when creating softness (X xY) for a 3D hard shadow. Editable.

Auto Near/Far button Enable to automatically set the near and far parameters for the 3D shadow based on the objects in the scene.

Near field Displays the near distance of the start of the 3D shadow. Editable.

Far field Displays the far distance of the end of the 3D shadow. Editable.

Matte Threshold field Displays the value at which the alpha casts a 3D shadow. Editable.

Overlap field Displays the amount of overlap from the light source. Increase to remove imperfections in the 3D shadow. Editable.

Relighting: Lens Flares

Use lights in your scene to generate procedural lens flares with built-in 3D occlusions.



With lens flares in Action, you can control:

- How lights change as they move behind 2D or 3D layers.
- How flares behave when a light exits or enters the camera field of view.

Lens Flares in Action are comprised of a Lens Flare object, attached to one or more Border FX objects, and any number of texture components, such as irises, streaks, and glows. To help you get accustomed to working with lens flares, a preset with typical settings is loaded when you first add a lens flare object to your scene.

You can attach multiple lens flares to a light, and multiple lights can attach to a lens flare.

TIP Use a GMask link to connect a GMask directly to the Lens Flare node in the schematic. If you then select Use As Occluder in the Post Processing box in the GMask menu, you can create interesting effects by having the GMask occlude only the Lens Flare effect.

Using Lens Flare Presets

A number of lens flare presets are included in Action, simulating the light refraction of certain camera lenses, as well as some creative flare effects, such as car lights. These presets can help you add complex lighting effects with just a few clicks.

To add a lens flare preset:

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

     The file browser opens.
2. From the Preset Type box, select Lens Flare.



   The Lens Flare Preset file browser appears, pointing to the default location of the presets: *usr/discreet/<product home>/lensflare/presets*.
3. Optional: Enable Scale to Action Resolution to load the preset in the current Action resolution.
4. Optional: Select which rendering settings to enable or disable in the preset (Z-Buffer, Shading, Polygon Resolution, and Colour Clamping).

   NOTE These settings are enabled by default, and by disabling any of them, you may not see the intended results in the preset.
5. Select the lens flare preset you want to load. Hold `Ctrl` and click to select multiple presets.

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

   The lens flare preset is then appended to your Action scene. You may need to move the light in the image to see the full lens flare effect.

Presets Browser Settings

Preset Type box Select the category of presets to display in the browser.

Scale to Action Resolution button Enable to load the preset in the current Action resolution.

Z-Buffer button Enable to load the Z-buffer rendering settings of the preset.

Shading button Enable to load the shading rendering settings of the preset.

Polygon Resolution button Enable to load the rendering resolution settings of the preset.

Colour Clamping button Enable to load the resolution colour clamping settings of the preset.

Adding a Lens Flare to a Light

In Action, a lens flare is a child of a light. Multiple lights can be parented to the same Lens Flare object.

To add a lens flare to the scene.

1. Add and position a light to your scene.
2. Do one of the following:
   - Drag the lens flare node from the node bin and place it in the schematic.
   - Double-click the lens flare node.

   A default lens flare (including a Lens Flare object, Border FX object, and multiple component texture objects) is added to the schematic. In Result view, you can see the lens flare (you may need to move the light in the scene to see the full lens flare effect).

   If there is only one light in the scene, the lens flare node is automatically connected to it. If there are several lights in the scene, select the light in the schematic that you want parented to the lens flare before added the lens flare node. Otherwise, you can parent the lens flare node to the light or lights manually in the schematic.
3. To add a texture component to a lens flare, select the Border FX object in the schematic, then double-click a texture node in the Relighting tab of the node bin.
4. To display the Lens Flare menu, double-click the lens flare object in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 415).

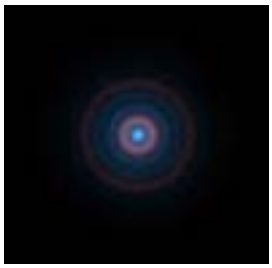
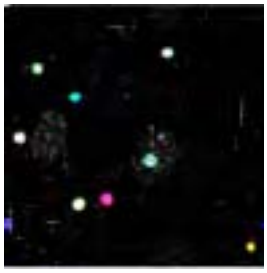
You can also add a Lens Flare using the Presets node. See Using Lens Flare Presets (page 512). In this case, a light is automatically added as part of the preset.

Using Texture Components With Lens Flares

You must add texture components to the Border FX object of a lens flare to see a result. These components are textures that are attached to a Lens Flare Border FX object (or optionally, directly to a Rays or Blooming object). The Lens Flare Object menu has global settings for the complete lens flare, but each component texture has its own menu to control settings particular to it.

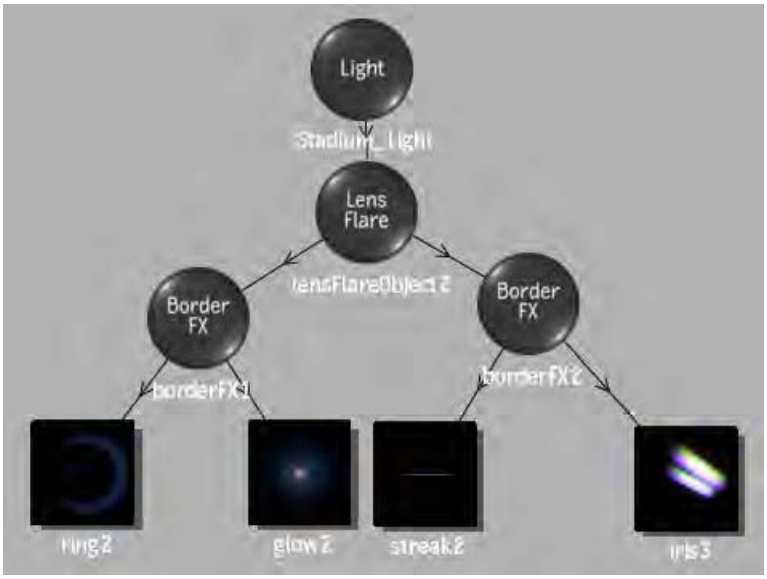
Each type of component can be added multiple times, each with its own settings. The following are the types of texture components you can add:

Component:	Description:	Example:
Glint Texture	Star-like texture.	

Component:	Description:	Example:
Glow Texture	Glowing loops of different colours.	
Iris Texture	Multiple shapes, such as polygons, discs, orbs, or caustics.	
Lens Texture	Lens "defects" such as hair, scratches, or fingerprints.	
Ring Texture	Rings with multi-coloured loops.	
Streak Texture	Lines streaking across the image.	

In the Action schematic, the components are attached to a Border FX object that is itself attached to the Lens Flare object. You can use multiple Border FX objects to achieve the lens flare look you want. For example,

attach a ring and glow to one Border FX object, while a streak and iris are attached to another Border FX object. In this case, you can control how the lens flare behaves when it reaches the borders of your image differently for the components under each Border FX object.



TIP You can also attach multiple Border FX objects to the same texture component, to compare different Border FX settings, for example. In this case, you can use the Action Hide button to hide each selected Border FX node to see the different results.

Re-Texturing Components

Since the components are textures, you can also re-texture a component using a Diffuse Map with your own texture media applied. To do so, select the component in the schematic, then select the media you want to use from the media list, and double-click the Diffuse Map node in the node bin. In this case, the Pattern settings in the component menu are not applicable, though you can still use the settings in the Basics and Border FX tabs of the component menu, as well as the Diffuse menu settings.

You can also use lens flare components on their own as Substance Texture presets, as a *LENSFLARE* subfolder is available in the preset browser. See Substance Textures (page 579) for more information.

Lens Flare Menu Settings

The Lens Flare Object menu lets you control settings for the complete lens flare effect. The Border FX object and each attached component texture also have their own specific menus.

Basics Tab



Position X field Displays the screen space position along the X axis of the Lens Flare pivot point. Editable.

Position Y field Displays the screen space position along the Y axis of the Lens Flare pivot point. Editable.

Position Z field Displays the screen space position along the Z axis of the Lens Flare pivot point. Editable.

Red Colour field Displays the amount of red in attached flare components (based on the colour of the attached light). Editable.

Green Colour field Displays the amount of green in attached flare components (based on the colour of the attached light). Editable.

Blue Colour field Displays the amount of blue in attached flare components (based on the colour of the attached light). Editable.

Flare Colour pot Displays the colour of the attached flare components (based on the colour of the attached light). Editable.

Global Intensity field Displays the intensity of the attached flare components (multiplied by the intensity of the attached light). Editable.

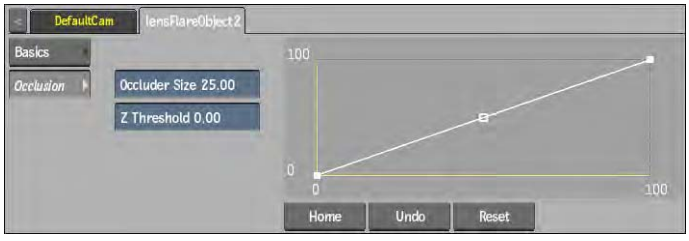
Global Scale field Displays the scale of the attached flare components. Editable.

Global Position field Displays the position of attached flare components in relation to the light (0%) and pivot point (100%). Editable.

Hue Shift field Displays the RGB offset applied to the edges of attached flare components. Editable.

Spread field Displays the amount of refractive distortion applied to the edges of attached flare components. Editable.

Occlusion Tab



Occlusion curve Displays the occlusion profile of the lens flare. Use to set the behaviour of the lens flare components when behind other objects in the scene.

Occluder Size field Displays the size of fade in/out of occluded lens flare components. Editable.

Z Threshold field Displays the transparency value at which objects start occluding the flare. Editable.

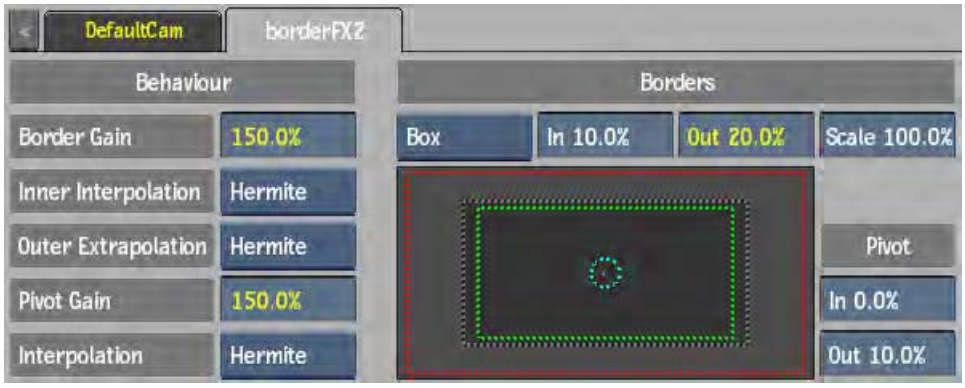
Home button Resets the bevel curve viewer to show the whole curve.

Undo button Undoes bevel curve operations.

Reset button Resets the bevel curve.

Border FX Menu Settings

Use the Border FX settings to control how the lens flare behaves when it reaches the borders of your image. A visual representation of the inner and outer borders, as well as the pivot area is displayed in the menu.



Border Gain field Displays the amount of gain to apply to the attached lens flare components when the light reaches the defined border. Editable.

Inner Interpolation box Select an interpolation type to define the transition between the border gain and the inner border.

Outer Extrapolation box Select an extrapolation type to define the transition between the border gain and the outer border.

Pivot Gain field Displays the amount of gain to apply to the attached lens flare components when the light reaches the defined pivot area. Editable.

Pivot Interpolation box Select an interpolation type to define the transition between the pivot gain and the pivot point.

Border Mode box Select whether Border FX settings are applied vertically, horizontally, or in both directions.

Inner Border Margin field Displays the position of the inner border. Editable.

Outer Border Margin field Displays the position of the outer border. Editable.

Border Scale field Displays the scale of the border that controls where the lens flare effect occurs. Editable.

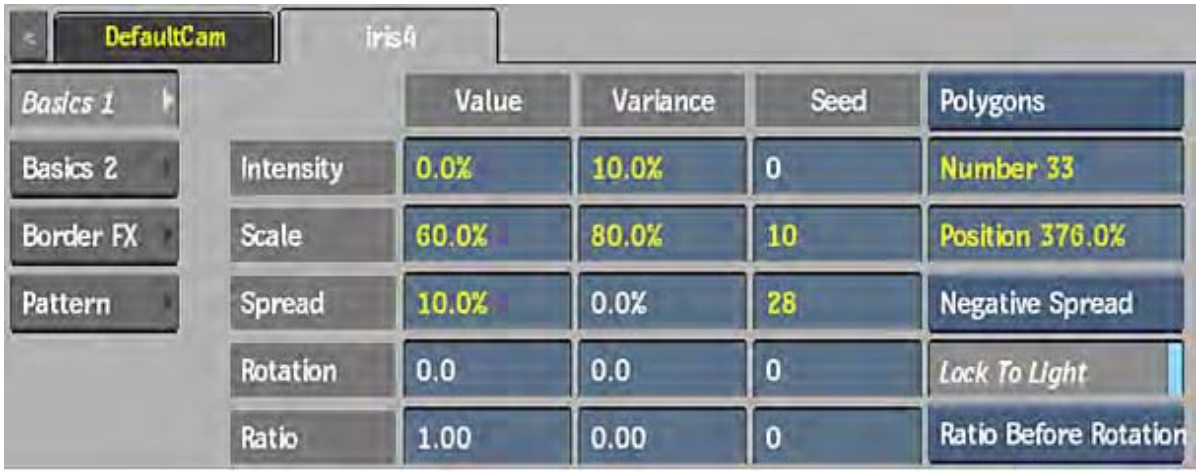
Pivot Radius In field Displays the inner radius of the pivot point. Editable.

Pivot Radius Out field Displays the outer radius of the pivot point. Editable.

Component Menu Settings

Each flare component has its own menu to control settings particular to the component. Some of these settings are common among the different component types, while some are specific to the component type.

Basics 1 Tab



Intensity Value field Displays the brightness of the component. Editable.

Intensity Variance field Displays how much the intensity varies. Available when the Number field value is greater than 1. Editable.

Intensity Seed field Displays the random intensity seed value. Available when the Number field value is greater than 1. Editable.

Scale Value field Displays the size of the component. Editable.

Scale Variance field Displays how much the scale varies. Available when the Number field value is greater than 1. Editable.

Scale Seed field Displays the random scale seed value. Available when the Number field value is greater than 1. Editable.

Spread Value field Displays the position of components in relation to each other. Available when the Number field value is greater than 1. Editable.

Spread Variance field Displays how much the spread varies. Available when the Number field value is greater than 1. Editable.

Spread Seed field Displays the random spread seed value. Available when the Number field value is greater than 1. Editable.

Spread option box Select a behaviour for the spread settings: Centre (equal), Positive (light to pivot direction), or Negative (pivot to light direction).

Rotation Value field Displays the level of rotation of the component. Editable.

Rotation Variance field Displays how much the rotation varies. Available when the Number field value is greater than 1. Editable.

Rotation Seed field Displays the random rotation seed value. Available when the Number field value is greater than 1. Editable.

Ratio Value field Displays the aspect ratio of the component. Editable.

Ratio Variance field Displays how much the ratio varies. Available when the Number field value is greater than 1. Editable.

Ratio Seed field Displays the random ratio seed value. Available when the Number field value is greater than 1. Editable.

Iris Shape box Select the shape of iris component. For other component types, this box is grayed out and displays the type of component.

Number field Displays the amount of components. Editable.

Position field Displays the offset applied to the component in relation to the light (0%) and pivot point (100%). Editable.

Lock To Light button Enable to lock the orientation of the component to the light.

Order box Select whether ratio is applied before rotation or vice-versa.

The Lens Texture component Basics 1 tab only has the following two settings (not shown):

Overall Brightness field Displays the brightness level of the Lens Texture component. Editable.

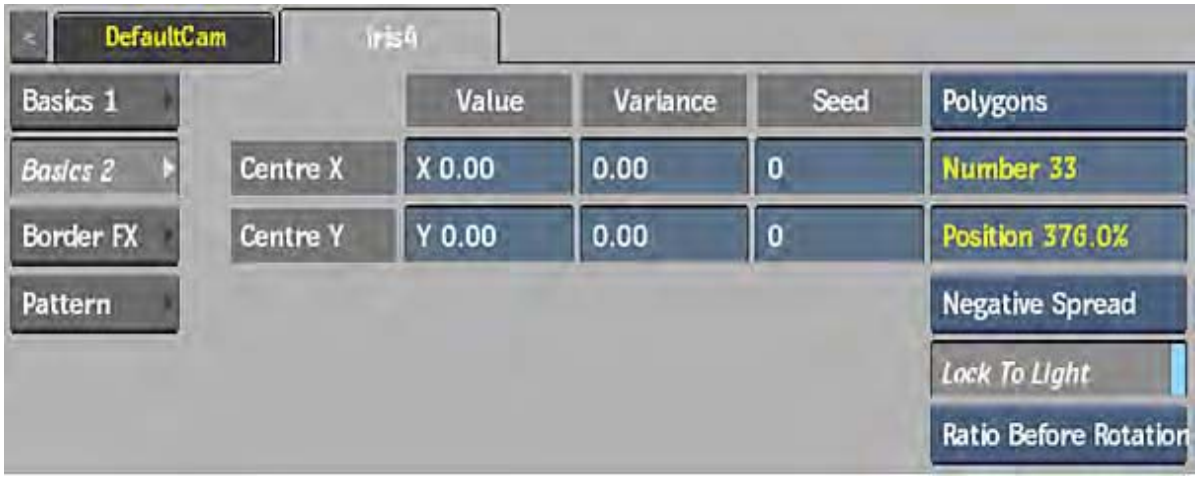
Inner Brightness field Displays how much of the lens is revealed. Editable.

The Glint component Basics tab also has the following two settings (not shown):

Overall Speed field Displays the rate at which the Glint animation plays.

Time Offset field Displays the start point of the Glint animation. With a value of 0, the animation starts at frame 1. With a value of 100, the animation begins as if it has been generating for 99 frames. You cannot animate this field.

Basics 2 Tab



Centre X Position field Displays the position of scaling and rotation of the component along the X axis. Editable.

Centre X Variance field Displays how much the centre position varies along the X axis. Available when the Number field value is greater than 1. Editable.

Centre X Seed field Displays the random centre X seed value. Available when the Number field value is greater than 1. Editable.

Centre Y Position field Displays the position of scaling and rotation of the component along the Y axis. Editable.

Centre Y Variance field Displays how much the centre position varies along the Y axis. Available when the Number field value is greater than 1. Editable.

Centre Y Seed field Displays the random centre Y seed value. Available when the Number field value is greater than 1. Editable.

Border FX Tab



The component Border FX settings allow you to choose which how the component interacts with the parent lens flare border settings.

Border Intensity button Enable to take into account intensity values of the parent relighting object.

Intensity Gain field Displays the amount of gain applied to the intensity. Editable.

Intensity Variance field Displays how much the intensity varies. Editable.

Border Scale button Enable to take into account scale values of the parent relighting object.

Scale Gain field Displays the amount of gain applied to the scale. Editable.

Scale Variance field Displays how much the scale varies. Editable.

Border Rotation button Enable to take into account rotation values of the parent relighting object.

Rotation Gain field Displays the amount of gain applied to the rotation. Editable.

Rotation Variance field Displays how much the rotation varies. Editable.

Border Centre button Enable to take into account centre values of the parent relighting object.

Centre Gain field Displays the amount of gain applied to the centre. Editable.

Centre Variance field Displays how much the centre varies. Editable.

Border Ratio button Enable to take into account ratio values of the parent relighting object.

Ratio Gain field Displays the amount of gain applied to the ratio. Editable.

Ratio Variance field Displays how much the ratio varies. Editable.

Border Spread button Enable to take into account spread values of the parent relighting object.

Spread Gain field Displays the amount of gain applied to the spread. Editable.

Spread Variance field Displays how much the spread varies. Editable.

Pattern Tab(s)



Use the pattern settings to control the texture before it is applied to the lens flare or ray. The settings in the Pattern tabs vary depending on the component type. You can get a quick description of each setting by viewing its tooltip.

Relighting: Rays

Use rays to simulate volumetric effects. You can attach a Rays object to a light in Action.



You can position rays behind 2D or 3D objects to generate effects, although rays are still visible even without an object to outline them. When semi-transparent objects are positioned in front of the light, it is possible to generate volumetric rays that use the colour of the object.

TIP Use a GMask link to connect a GMask directly to the Rays node in the schematic. If you then select Use As Occluder in the Post Processing box in the GMask menu, you can create interesting effects by having the GMask occlude only the Rays effect.

Adding a Rays Object to a Light

In Action, a ray is a child of a light. Multiple lights can be parented to the same Rays object.

To add a ray to the scene.

1. Add and position a light to your scene.
2. Do one of the following:
   - Drag the rays node from the node bin and place it in the schematic.
   - Double-click the rays node.

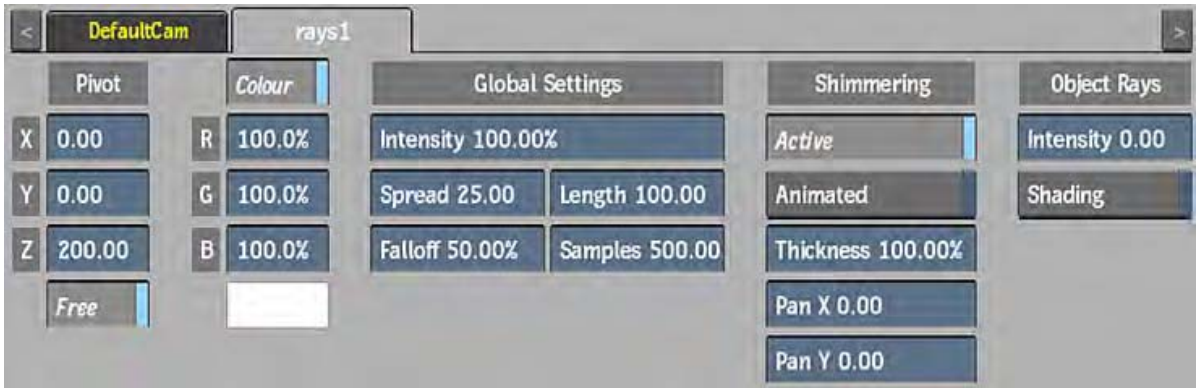
   A Rays object is added to the schematic. In Result view, you can see the ray.

   If there is only one light in the scene, the rays node is automatically connected to it. If there are several lights in the scene, select the light in the schematic that you want parented to the rays node before added the rays node. Otherwise, you can parent the rays node to the light or lights manually in the schematic.
3. To display the Rays menu, double-click the Rays object in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 415).

Similar to the workflow of the Lens Flare, you can add texture components (page 513) to a rays node to enhance the effect.

Rays Menu Settings

The Rays Object menu lets you control settings for the complete rays effect. If you have added component textures to the Rays effect, each component texture also has its own menu that is specific to the component (see Component Menu Settings (page 517)).



Position X field Displays the screen space position along the X axis of the Rays point-of-interest. Unavailable if Free is disabled. Editable.

Position Y field Displays the screen space position along the Y axis of the Rays point-of-interest. Unavailable if Free is disabled. Editable.

Position Z field Displays the screen space position along the Z axis of the Rays point-of-interest. Editable.

Free button Enable to ignore transformations from the parent light object. When disabled, the ray is affected by the parent light's position, rotation, spread, and falloff settings.

Colour button Enable to add the ray colour to the colour bleed applied to semi-transparent 3D objects placed in front of the attached light.

Red Colour field Displays the amount of red in the effect and in any attached components (based on the colour of the attached light). Editable.

Green Colour field Displays the amount of green in the effect and in any attached components (based on the colour of the attached light). Editable.

Blue Colour field Displays the amount of blue in the effect and in any attached components (based on the colour of the attached light). Editable.

Colour pot Displays the colour of the effect and any attached components (based on the colour of the attached light). Editable.

Intensity field Displays the intensity of the effect and any attached components (multiplied by the intensity of the attached light). Editable.

Spread field Displays the shape of the ray cone. Editable.

Falloff field Displays the amount of smoothness applied to the borders of the ray cone. Editable.

Length field Displays the amount of softness applied to the ray. Editable.

Samples field Displays the quality of the ray based on the radial distance to the attached light. Editable.

Active button Enable to use shimmering settings to modulate the ray with noise particles.

Animated button Enable to automatically apply a noise effect to the shimmer at each frame.

Thickness field Displays the thickness of the shimmer noise particles. Editable.

Pan X field Displays the amount of movement of the shimmer noise particles along the X axis. Editable.

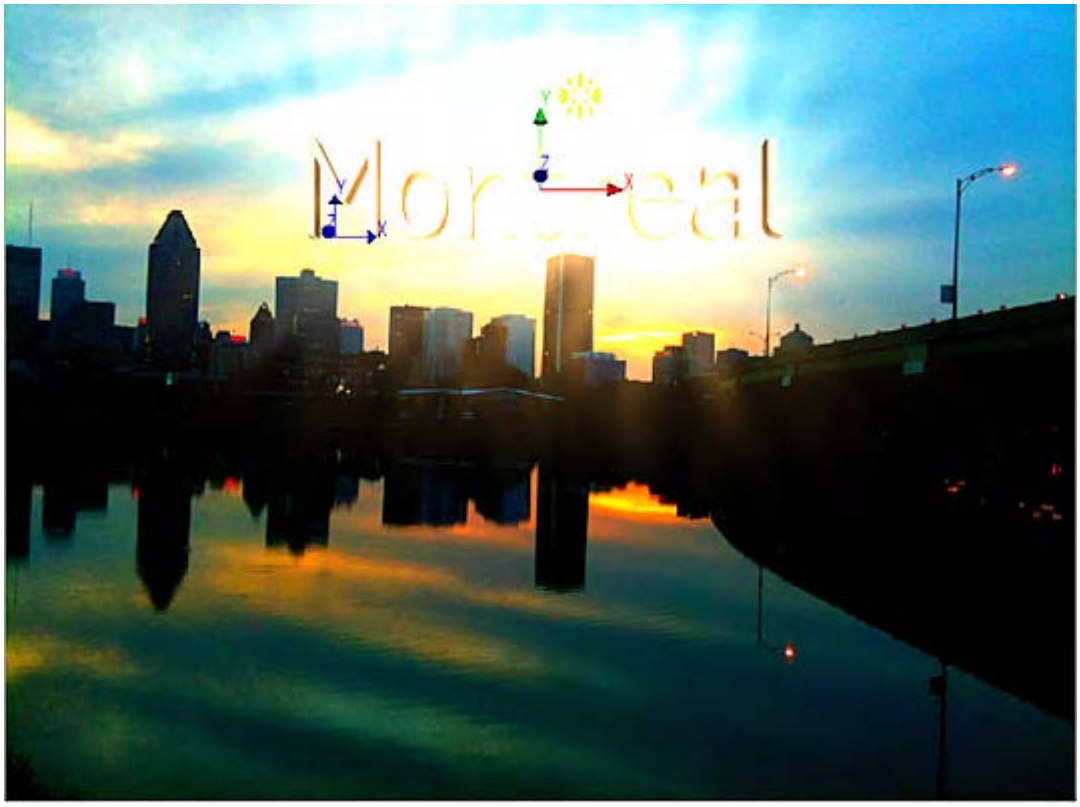
Pan Y field Displays the amount of movement of the shimmer noise particles along the Y axis. Editable.

Object Intensity field Displays the amount of ray colour bleed applied to semi-transparent 3D objects placed in front of the attached light. Editable.

Shading button Enable to allow the ray to inherit the shaded colours of a 3D object as it passes through the object.

Relighting: Blooming

Attach a Blooming node to a light in the scene to help define highlight areas that generate a glowing effect. You can add textures to stamp a blooming node with particular patterns, such as streaks and glints.



Blooming can affect surfaces and geometries in your Action scene, and you can also use lighting links from the attached lights to selectively include or exclude blooming from objects in the scene.

Since Blooming works in Action screen space, the effect is not limited to the surface or geometry in your scene. Note however, that while blooming can occur outside of the Action scene while you are viewing the effect in the image window, in this case you may not see the full blooming effect when you preview or process in Action. You may notice this also if you zoom in or out while viewing in the image window.

TIP Use a GMask link to connect a GMask directly to the Blooming node in the schematic. If you then select Use As Occluder in the Post Processing box in the GMask menu, you can create interesting effects by having the GMask occlude only the Blooming effect.

Adding a Blooming Object to a Light

In Action, a blooming node is a child of a light. Multiple lights can be parented to the same Blooming object.

To add a bloom to the scene.

1. Add and position a light to your scene.
2. Do one of the following:
   - Drag the blooming node from the node bin and place it in the schematic.
   - Double-click the blooming node.

   A Blooming object is added to the schematic. In the Result view, you can see the bloom effect.

   If there is only one light in the scene, the blooming node is automatically connected to it. If there are several lights in the scene, select the light in the schematic that you want parented to the blooming

node before adding the blooming node. Otherwise, you can parent the blooming node to the light or lights manually in the schematic.

3 Optional: To add a texture component, select the blooming object in the schematic, then double-click a texture node in the Relighting tab of the node bin.

4 To display the Blooming menu, double-click the Blooming object in the schematic, or follow the tab population rules for the Object menu.

Blooming with Texture Components

Similar to the workflow of the Lens Flare, you can add texture components to a blooming node to enhance the effect. When using blooming in Stamping mode, texture components must be attached to see any result.

The Stamping tab of the Blooming menu has global settings for all attached components, but each component texture has its own menu to control settings particular to it.

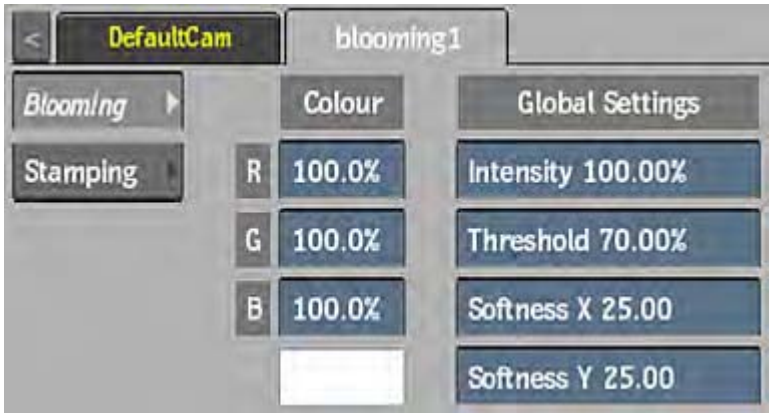
Be aware of the following when working with texture components attached to a blooming node:

- Each type of component can be added multiple times to a blooming object, each with its own settings.
- Some settings in the Basic tab of the component menu do not affect stamping, such as Number, Position, Variance, and Seed. Only one instance of the texture pattern is used for stamping, so these settings have no effect.
- You can use the Border FX settings for each component to decide how the luminance of the objects that the blooming is affecting modulates the Basics settings (Intensity, Scale, Rotation, and Ratio) of the texture pattern. Centre and Spread are not used for blooming.
- You can re-texture a component using a Diffuse Map with your own texture media applied. To do so, select the component in the schematic, then select the media you want to use from the media list, and double-click the Diffuse Map node in the node bin. In this case, the Pattern settings in the component menu are not applicable, though you can still use the settings in the Basics and Border FX tabs of the component menu, as well as the Diffuse menu settings.

Blooming Menu Settings

The Blooming menu is divided into two tabs, each with its own type of blooming effect, which can work independantly, or in combination with each other.

Blooming Tab



Red Colour field Displays the amount of red in the effect and in any attached components (based on the colour of the attached light). Editable.

Green Colour field Displays the amount of green in the effect and in any attached components (based on the colour of the attached light). Editable.

Blue Colour field Displays the amount of blue in the effect and in any attached components (based on the colour of the attached light). Editable.

Rays Colour pot Displays the colour of the effect and any attached components (based on the colour of the attached light). Editable.

NOTE The same colour settings are also found in the Stamping tab.

Intensity field Displays the intensity of the effect and any attached components (multiplied by the intensity of the attached light). Editable.

Threshold field Displays the minimum luminance value at which blooming occurs. Editable.

Softness X field Displays the amount of softness along the X axis of the blooming effect. Editable.

Softness Y field Displays the amount of softness along the Y axis of the blooming effect. Editable.

Stamping Tab

For stamping settings to have any effect, you need to attach one or more texture components to the blooming node. In this case, stamping settings are global for all attached components, and each component texture has its own menu that is specific to the texture.



Red Colour field Displays the amount of red in the effect and in any attached components (based on the colour of the attached light). Editable.

Green Colour field Displays the amount of green in the effect and in any attached components (based on the colour of the attached light). Editable.

Blue Colour field Displays the amount of blue in the effect and in any attached components (based on the colour of the attached light). Editable.

Rays Colour pot Displays the colour of the effect and any attached components (based on the colour of the attached light). Editable.

NOTE The same colour settings are also found in the Blooming tab.

Stamping Intensity field Displays the global stamping intensity of all attached texture components. Editable.

Stamping Threshold field Displays the minimum value at which stamping occurs for all attached texture components. Editable.

Stamping Softness X field Displays the softness along the X axis for all attached texture components. Editable.

Stamping Softness Y field Displays the softness along the X axis for all attached texture components. Editable.

Stamping Attenuation field Displays the smoothing level of the blooming effect. Use to fade out regions that have too much blooming. Editable.

Texture Minimum Size field Displays the minimum size of all attached texture components. Editable.

Texture Scale field Displays the size of all attached texture components. Editable.

Texture Rotation field Displays the level of rotation of all attached texture components. Editable.

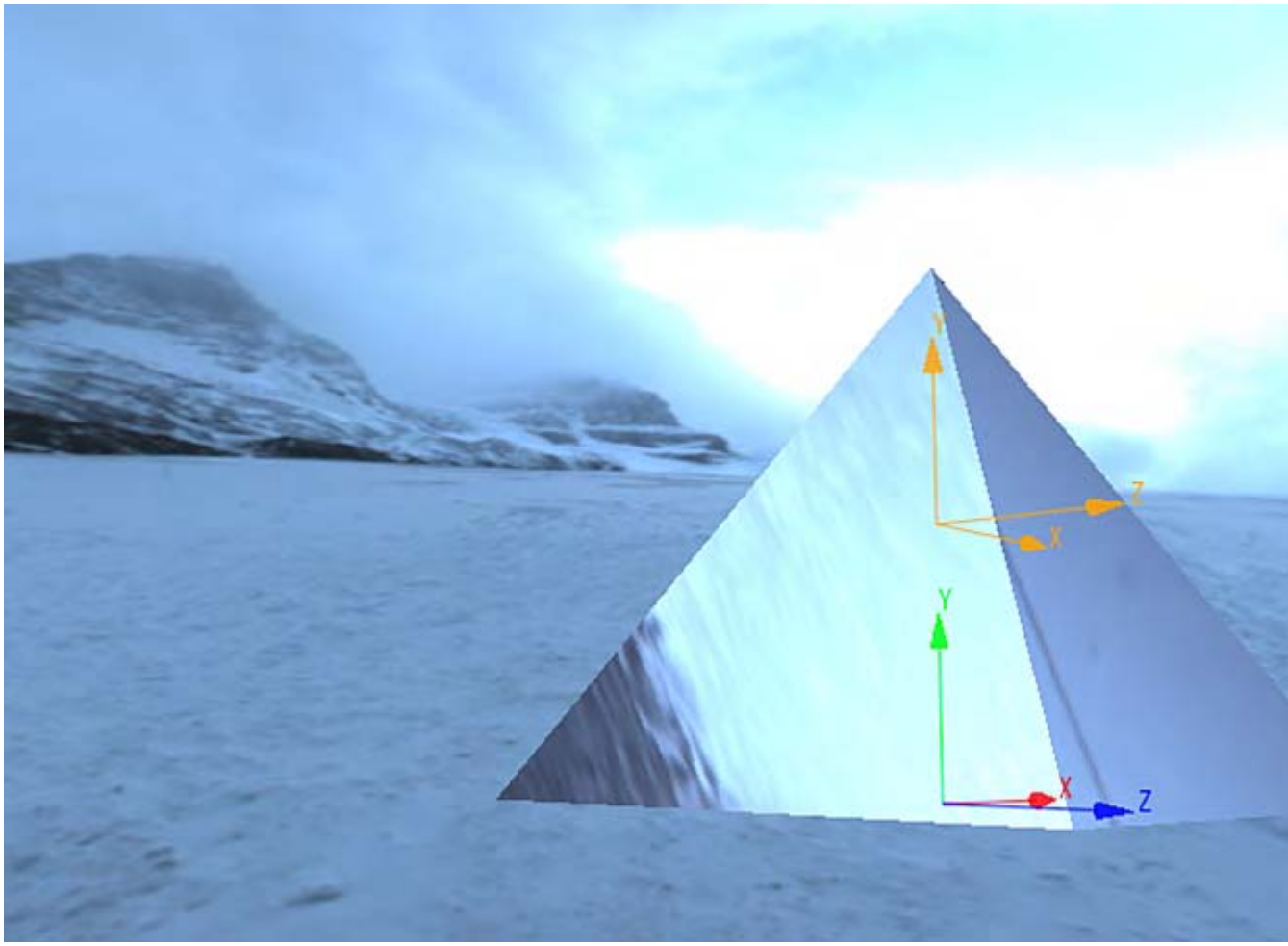
Texture Ratio field Displays the aspect ratio of all attached texture components. Editable.

Sampling X field Displays the size of the grid along the X axis to affect the number of samples taken to calculate the stamping effect. A higher value yields faster results, but may be less precise. Editable.

Sampling Y field Displays the size of the grid along the Y axis to affect the number of samples taken to calculate the stamping effect. A higher value yields faster results, but may be less precise. Editable.

Relighting: Image-Based Lighting (IBL)

Image-based lighting (IBL) is the process of illuminating scenes and objects (real or synthetic) using images of light from the real world. This is in contrast to using only direct light sources such as point lights or spotlights, which are more localized. A typical use of IBL is taking high-dynamic photos of a chrome ball placed in the original environment of a live-action shoot, then using the photos as an IBL map to simulate the lighting conditions of the shoot.



Light Probe Images courtesy of Paul Debevec, www.debevec.org

In Action, you can use IBL maps to:

- Colour blend images onto the backplate (the IBL map must be a shot of the environment that the backplate was taken from). This technique is useful fror green screen work.
- Light 3D geometry or text objects from every direction using the colours in the environment.

You can use IBL maps to make surfaces perfectly reflective or perfectly diffuse. You can also make an IBL map act as a virtual spherical background plate.

Action supports angular, cubic, and cylindrical IBL mapping types. You can use the Map Convert (page 1380) tool to convert images to one of these types.

Adding an IBL Map

IBL maps behave differently than other Light FX nodes, such as Lens Flares, since you don't attach it to a light in the scene. The IBL performs lighting based on media as an environment texture.

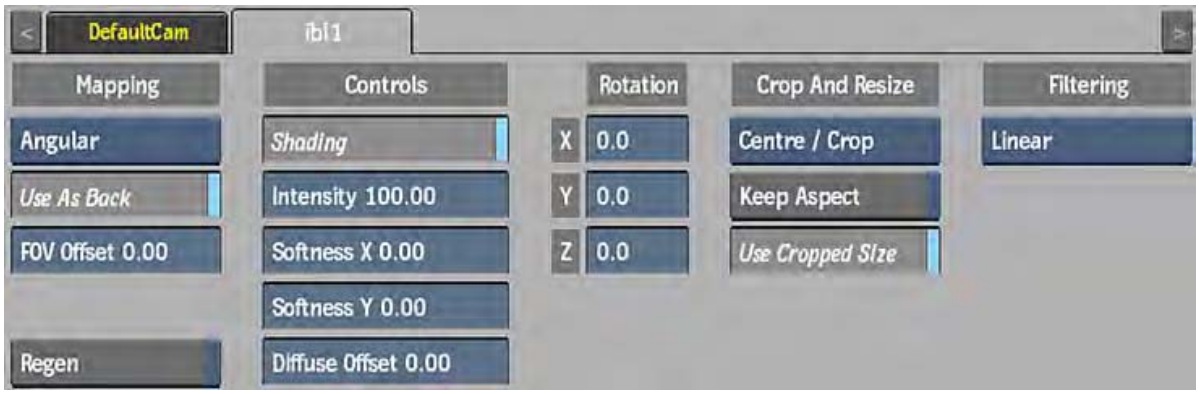
To add an IBL map to the scene:

1. Select the media to be uses as the IBL in the image list. Any type of media can be used, including, but not limited to, *.hdr* images.
2. Select the node in the scematic that you want to attach the IBL map to.
3. Do one of the following:
   - Drag the IBL node from the node bin and place it in the schematic.
   - Double-click the IBL node.

   To affect the entire scene, add an IBL map and make it a child of the Result camera (or any camera used for the comp output). To affect a specific object, make the IBL map a child of the object. You can use only one IBL to affect the entire scene, and only one IBL to affect an object.
4. To display the IBL menu, double-click the IBL object in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 415).

IBL Menu Settings

The settings in the IBL menu are as follows.



Mapping box Select the type of IBL texture mapping. Use the Map Convert (page 1380) tool to covert different mapping types.

IBL Type:	Description:	Example:
Angular	Usually a high-dynamic picture of a mirrored ball.	
Cubic	A series of images shown as an unfolded cube displaying six sides of the image.	
Cylindrical	An image mapped onto an un-folder cylinder (also known as longitude and latitude maps).	

Light Probe Images courtesy of Paul Debevec, *www.debevec.org*

Use As Back button Enable to use the selected IBL map as the background in the scene.

FOV Offset field Displays the field of view offset applied to an IBL map that is used as a background. Use to simulate a zoom in or out of the background. Editable.

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

Shading button Enable to light up the scene using added light sources. When Shading is disabled, no lighting effects appear in the scene; surfaces and 3D models appear flat.

Enable Shading for:

- Light sources
- Ambient or diffuse lighting for surfaces
- Specular highlights for surfaces and 3D models

This same button can be found in the Light and Action Setup menus.

Intensity field Displays the overall brightness of the IBL map. Editable.

Softness X field Displays the amount of X-axis blur applied to the IBL map (not available when Cubic is chosen as the mapping type). Editable.

Softness Y field Displays the amount of Y-axis blur applied to the IBL map (not available when Cubic is chosen as the mapping type). Editable.

Diffuse Offset field Displays the offset based on an attached diffuse map or the existing diffuse light in the image. Editable.

Rotation X field Displays the level of rotation of the IBL map along the X axis. Editable.

Rotation Y field Displays the level of rotation of the IBL map along the Y axis. Editable.

Rotation Z field Displays the level of rotation of the IBL map along the Z axis. Editable.

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

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

Use Cropped Size button Enable to replace the IBL map with the cropped size of the IBL media. Disable to use the cropped IBL media as is.

Filter box Select the type of filtering to apply to the IBL map.

About 3D Geometry

A powerful feature of Action is its ability to import 3D models created in other applications and combine them with existing clips. Compositing 3D models and characters with other media or a background can be done quickly and with a finite level of control.

3D geometric objects are manipulated like an image or a clip: you can animate their position and shape, apply textures and media, and light objects to produce a variety of effects.

You can import 3D polygon objects such as 3ds Max files, FBX files, Alembic files, Wavefront files, Inventor files, garbage masks, and Paint geometry. 3ds Max files contain object data, specifically, texture and materials. The FBX format acts as the intermediary between different file types. Files can be exported from another product to the FBX or ABC format and then imported into Flame Premium.

Paint geometry files are created by the Flame Premium Paint tool. If you want to work with polygon geometry in Action, import Paint geometry. Action ignores its animation and attributes, such as its colour, outline, and gradient. For example, in Paint, if you create a blue polygon, animate its scale, and save it as geometry, it is imported in Action as a white polygon with no animation.

NOTE Sample 3ds Max and FBX model files are located in the *usr/discreet/<product_home>/models/*(FBX or 3DS) libraries. All models are textured with an identical image. The library consists mostly of geometric primitives such as cubes, cylinders, and spheres.

Importing 3D Models

You can import a 3D model into one or several geometry nodes. You can also import multiple 3D models into a single animated geometry node.

To import a 3D model:

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

   The Import menu and file browser appear.

   NOTE When selecting Paint, GMask, or Photoshop, the subsequent controls described in this section do not apply. When selecting FBX, other controls appear. See Importing the FBX Format (page 536).
2. From the Import Type box, select the import format.
3. If the file you want to import has a file extension different from the one specified, type a file extension.
4. Enable or disable Geometry, Object, and Material options, as needed. See below for descriptions of the options.
5. Select the file to import from the file browser.
6. Click Load.

   The 3D model (Geom node) and axis is added to the scene.

   You can change the 3D model's colour, specular highlight, shine, and other material properties. See 3D Geometry Menu Settings (page 538).

To import multiple 3D models into an animated sequence:

1. Follow the same steps for adding a single 3D model, but from the Import file browser, select multiple 3D models by holding the `Shift` or `Ctrl` key.
2. Once the models are selected, click Load.

   The selected 3D models are loaded to the same line in the Media list.
3. Double-click the Geometry node in the schematic to access the Geometry menu, then click the Timing tab.

Import Settings

Geometry Import Settings

Depending on the type of import, some of the options differ.

Import Type box Select the 3D model type to import.

File Extension field Displays the default extension for the file type selected in the Import Type box.

Smooth button Enable to build normals for the 3D model. Enable if you are importing polygons that do not have normals.

Auto Fit In Scene button Enable to scale the imported model to fit into the current frame. When disabled, the imported model maintains the same size in which it was created.

Mesh Animations button Enable to import the Alembic scene animations and preserve complex geometry animations.

Cameras button Enable to import cameras from the Alembic format file.

Normal button Enable to import an Alembic model's normal information.

Separate Nodes button Enable to create individual nodes for all 3D models contained in a file. When disabled, the 3D model is added to the scene with its own axis.

Rotate Axis button Enable to rotate the 3D model by 90 degrees on the X-axis so that it is compatible with the target's coordinate system.

Create Media button Enable to load the textures of the geometry to the Media list. If a texture is used in multiple geometry maps, it is loaded only once in the Media list.

Timing Settings



Animation Mode box Select the animation mode to use for multiple 3D geometries imported into a single animated geometry node.

Select:	To:
Loop	Play in a continuous loop.
Once	Play once. The 3D geometry is no longer displayed.
Last Still	Play once, and hold the last frame.
Timing	Animate according to the timing in the animation channel.

Hold field Displays the number of continuous frames for Loop, Once, or Last Still Animation modes. Editable.

Slip field Displays the offset to the start point for Loop, Once, or Last Still Animation modes. Editable.



Timing Range option box Select an option to determine how Frame Timing values outside the timing range of the animation are handled. Available when Timing is selected in the Animation Mode box.

Select:	To:
Roll	Roll over the Frame Timing value.
Cut	(Geometry is not displayed.)
Round	Display the first or last geometry (with this option, you can select the first and the last geometry of your animation).

Frame Timing field Displays the value for the frame in the timing curve. Available when Timing is selected in the Animation Mode box. Editable.

About 3D Data in the FBX Format

Flame Premium supports the import of 3D data saved in the FBX format.

The FBX standard format provides a means for exchanging 3D data—3D polygonal models, cameras, lights, keyframe animation—for scene compositions between tools and packages developed by different manufacturers.

FBX support makes it possible to import Autodesk 3ds Max, Autodesk Maya, Autodesk MotionBuilder, Autodesk Mudbox, and Autodesk Softimage files. You can import models, scene compositions, lights, and camera data.

NOTE Flame Premium supports the import of most models, including polygonal, nurbs, and smooth bind skinning, while subdiv primitives are not supported. See the FBX *compatibility maps*.

About Mesh Animation

Mesh animation (geometry caching) records the position of every vertex in an animated 3D scene over time. This allows character animations, animated geometry deformations, or physical simulations to be "baked" and transported in the FBX file without an understanding of the animation technique used to create the effect. By exporting FBX scenes with geometry caching, and then importing into Action with Mesh Animations enabled, you can preserve complex geometry animations, such as nCloth animations.

All imported cached geometry can still be manipulated in Action like a normal geometry. Deform meshes, texture maps, lighting, and axis manipulations are all supported on the animated geometry. Animations can also be retimed within Action.

FBX Information Preserved at Import

Flame Premium supports the latest FBX SDK, in sync with Autodesk 3D applications. To ensure compatibility, all applications must use the same version of the FBX SDK.

The following FBX features are preserved in Action:

- Point lights, spotlights, area lights, ambient lights, and directional lights
- Shadow casting
- Object ID and material assignments
- Sub-materials, exposed explicitly in the Action schematic
- Bezier animation curves

Object Group and Material Nodes

FBX models are created with one or more faces, which leads to different representations in Action.

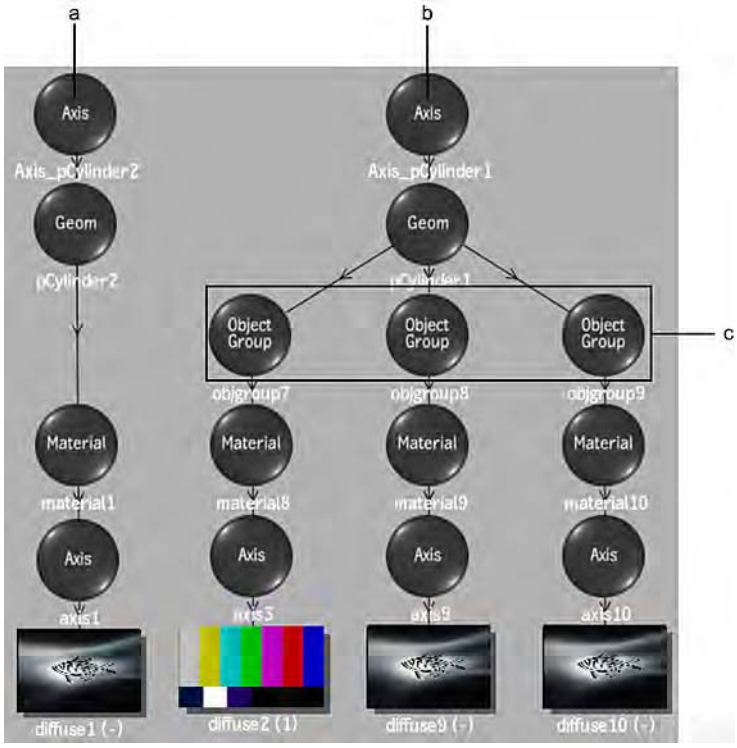
NOTE In this context, *face* actually means either a single face (such as one of six sides of a cube) or faces (such as a group consisting of 2 sides of a cube) that are grouped together to receive a single texture. How faces are grouped (if grouped at all) is decided when the model is created and cannot be modified in Flame Premium.

A model with a single face appears as a simple model, with a single Material node attached to the Geom node.

A model with multiple faces appears as a complex schematic where one Object Group node is created for each face. To each Object Group node is then attached the Material node. Having multiple Object Nodes allows you to modify, even replace, the textures applied to each shader.

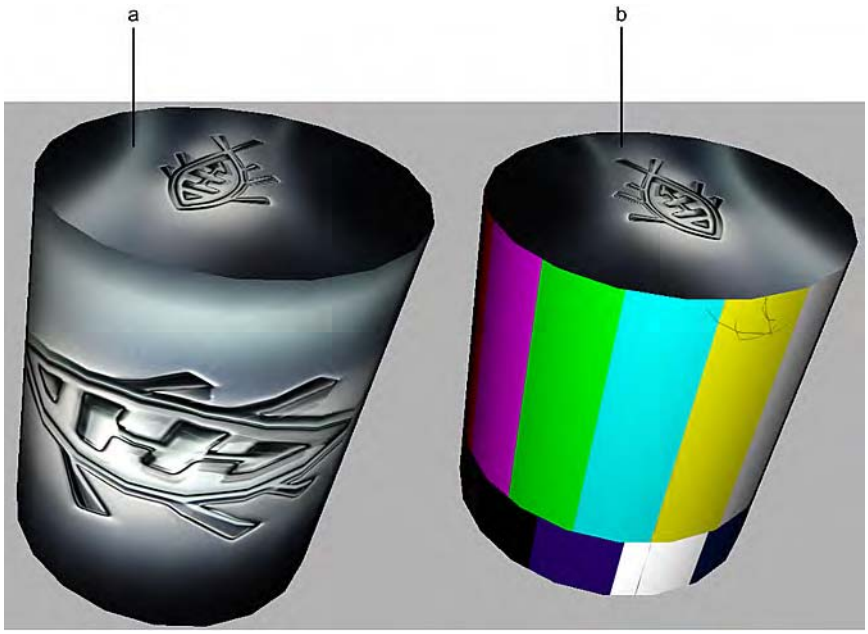
You cannot create new Object Group nodes, and their link to the Geom node is unbreakable; they have no editable attributes. Object Group nodes details how faces were applied to the FBX model, and allow you to texture differently the components of the FBX model.

In the following examples, one imported FBX model has a single face, while the other has three faces, one for each sides of the cylinder.



(a) FBX model with a single face (b) FBX model with multiple faces (c) Added Object Group nodes, one for each face

The schematic above creates the image below. Note how having different Object Group nodes allows you to link different textures.



(a) FBX model with a single face: one texture map for the whole model (b) FBX model with multiple faces: a different texture for each face

Importing the FBX Format

To import FBX format data in Action:

1. Do one of the following:
   - Drag the Import node from the node bar and place it in the schematic.
   - Drag the Import node from the node bar and place it where you want it in Result view.
   - Double-click the Import node. You do not need to be in Schematic view to add a node in this manner.
2. In the Import menu, select FBX as your format type.
3. Use the FBX import settings, as needed.
4. Navigate to the location where the FBX files are located.

   NOTE Make sure that you have write permissions to the folder where the FBX file is located, as Flame Premium decompresses the textures embedded in the FBX to an .fbm folder collocated with the FBX file.

5. Select a file from the file browser.

   The FBX data is imported into Action.

FBX Import Settings

Auto Fit In Scene button Enable to scale the imported model to fit into the current frame. When disabled, the imported model maintains the same size in which it was created, or you can use the FBX Units to Pixels field to manually set a scaling factor.

FBX Units to Pixels field Displays the scaling factor used on the imported FBX file to fit the scaling used in the application. One unit in the FBX file (default is cm) is converted to the number of pixels (default is 10) that you specify.

FBX Import Type box Select an FBX import type. Select Link to Original File to keep a live link to the original FBX file, but without control over the components of the scene. Select Create Local Copy to have control over the elements of the scene, but lose the link.

Keep FBX Frame Rate button Enable to use the frame rate of the FBX file as the frame rate in Action.

TIP It might be useful to enable Keep FBX Frame Rate when importing an FBX file before starting to build your animation, otherwise the timing of your animation may be affected.

Bake FBX Animation button Enable to add a keyframe at every frame of the imported FBX file. When disabled, Action translates the FBX keyframes to a comparable animation curve for comparable channels. Experiment with this option enabled and disabled to get your desired results.

Create Media button Enable to load the textures of the geometry to the Media list. If a texture is used in multiple geometry maps, it is loaded only once in the Media list.

Lights button Enable to import lights from the FBX format file.

Cameras button Enable to import cameras from the FBX format file. Free cameras are imported as target cameras (with a point of interest).

Mesh Animations button Enable to import the FBX scene animations and preserve complex geometry animations , such as nCloth animations.

Normals button Enable to import the model's normal information.

Working with FBX Scenes Linked to the Original Files

When importing an FBX file in an Action schematic, setting FBX Import Type to the option Link to Original File does not create a Geom node like Create Local Copy does, but rather creates an FBX Scene node.

You cannot perform the same operations on an FBX Scene node as on a Geom node.

- You can only parent Axis nodes.
- You cannot parent it to other nodes.
- You cannot select individual components; you always select the entire scene.
- There is only one node in the Priority Editor for the whole scene.

If someone updates the linked FBX file, click Refresh to update the FBX Scene node in Action.

The options available in an FBX Scene node are also different from the ones found in a Geom node.

FBX Scene Settings

FBX Scale field Displays the factor by which to scale the FBX scene. Does not scale the camera if it has already been extracted using the Extract Camera button. Editable.

Extract Camera button Click to extract the 3D camera from the FBX scene. It extracts and creates a new 3D camera node, scaled to the current FBX Scale . Once extracted, the camera is no longer linked to the FBX scale: changing the FBX scale does not affect the camera. If you need to edit then rescale the camera after changing the FBX Scale, delete the extracted camera and extract a new one.

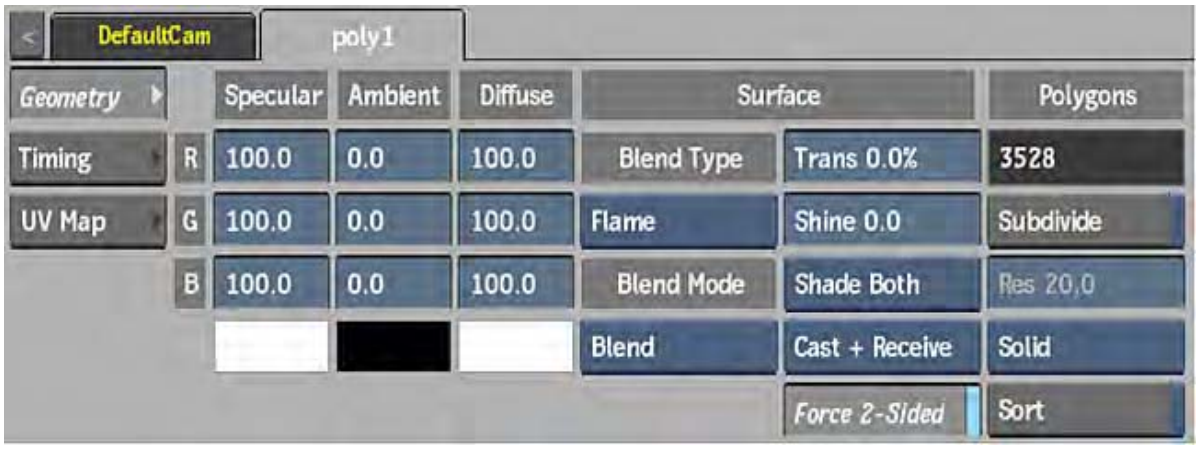
Sort button Enable so the 3D models contained in the FBX Scene are drawn according to their normals. Consider using this option if the scene contains semi-transparent 3D models to ensure they are correctly drawn.

FBX File Path field Displays the path to the FBX file. Click to replace the currently displayed FBX scene with another one.

Refresh button Click to update the FBX Scene node with the content of the FBX file. Use this if the FBX file has been updated since its import.

3D Geometry Menu Settings

You can change and animate parameters such as the colour, specular highlight, shine, and transparency of 3D models. You set these parameters using the Geometry menu. To access this menu, double-click a Geometry node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).



The Geometry controls are described as follows.

Lighting Settings

Red Specular field Displays the red specular highlight value. Editable.

Green Specular field Displays the green specular highlight value. Editable.

Blue Specular field Displays the blue specular highlight value. Editable

Specular colour pot Displays the colour of light reflected by the 3D model's surface. To enable the specular highlight, the Shine value must be larger than 0. Editable.

NOTE Specular lighting sets the colour of light reflected by the 3D model's surface. To enable the specular highlight, the Shine value must be larger than zero.

Red Ambient field Displays the red ambient colour value. Editable.

Green Ambient field Displays the green ambient colour value. Editable.

Blue Ambient field Displays the blue ambient colour value. Editable.

Ambient colour pot Displays the colour of the area of the 3D model that is not illuminated by a direct light source. Editable.

NOTE Ambient lighting sets colour to the area of the 3D model that is not illuminated by a direct light source. The edge of the ambient area mixes with the specular highlight colour and the diffuse colour.

Red Diffuse field Displays the red diffuse colour value. Editable.

Green Diffuse field Displays the green diffuse colour value. Editable.

Blue Diffuse field Displays the blue diffuse colour value. Editable.

Diffuse colour pot Displays the diffuse colour. Editable.

NOTE Diffuse lighting modifies the colour and illumination of the entire 3D model. Diffuse light mixes with the colour of the light sources used to illuminate the 3D model. The diffuse colour may also mix with the ambient colour and the colour of the specular highlight.

Surface Settings

Blend Type box Select whether to use Flame or Photoshop blend modes.

Blend Mode box Select how the 3D model and the scene are combined. The available list of modes depends on the selection in the Blend Type box. See Surface Blending Modes (page 467).

Transparency field Displays the transparency level of the 3D model. Editable.

Shine field Displays the intensity value of the specular highlight. When this value is zero, the specular highlight is disabled. Shine affects both size and intensity. Editable.

See Creating a Specular Highlight on a Model (page 543).

Sort Order box Select the drawing priority of the 3D model normals.

Select:	To draw the polygons:
Shade Both	That are both facing and opposite the camera.
Shade Front	Facing the camera last.
Shade Back	Opposite the camera last. This option is especially useful for semi-transparent models

Shadow Casting box Select how the selected geometry object will be affected by a Shadow Cast object in the scene. See Surface and Geometry Shadow Casters (page 508).

Force 2-Sided button Enable to have lights in the scene light both the inside and outside of the geometry (when shading is turned on).

Polygons Settings

Polygons field Displays the number of polygons in the 3D model. Non-editable.

Subdivide button Enable to create high-quality shading for polygon models.

Resolution field Displays the geometry resolution of the 3D model. Active when Subdivide is enabled. Editable.

Wireframe box Select a wireframe option for the 3D model. When you render the 3D model with Wireframe or Original Wire selected, it retains its light, shading, and texture attributes.

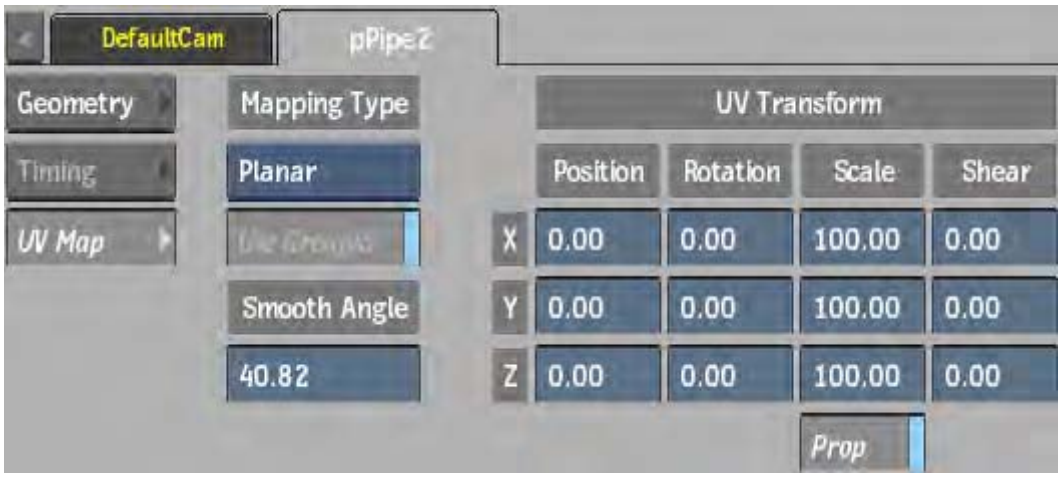
Select:	To:
Solid	Disable wireframe for the 3D model (filled polygons are drawn).
Wireframe	Display the model as a wireframe outline (triangular polygons only).

Select:	To:
Original Wire	Display the model as a wireframe outline (original mesh; any polygon type). May be useful for imported geometries.

Sort button Enable to determine how the 3D model is drawn according to its normals.

UV Map Settings

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



UV Mapping Type box Select the type of UV mapping to apply to the attached node.

When a Displace or Normal node is attached to a geometry, you may need a UV mapping type other than Default for the displace or normal pattern to have any effect on the geometry.

NOTE When a Diffuse node is attached to a geometry, you must select Wrap from the Mapping box in the Diffuse menu to be able to use the UV mapping settings. See Diffuse Mapping (page 615).

Use GroupId button Enable to respect GroupId information in an FBX file created in 3ds Max.

Smooth Angle button Enable to override existing normals in the geometry, then use the Smooth Angle field to change the value.

Smooth Angle field Displays the angle at which the edges of normals become hard. Changes to this field only affect the shading of the displacement, and not the shape. Editable.

Position X field Displays the position of the X axis. Editable.

Position Y field Displays the position of the Y axis. Editable.

Position Z field Displays the position of the Z axis. Editable.

Rotation X field Displays the rotation of the X axis. Editable.

Rotation Y field Displays the rotation of the Y axis. Editable.

Rotation Z field Displays the rotation of the Z axis. Editable.

Scale X field Displays the scale of the X axis. Editable.

Scale Y field Displays the scale of the Y axis. Editable.

Scale Z field Displays the scale of the Z axis. Editable.

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

Shear X field Displays the shear of the X axis. Editable.

Shear Y field Displays the shear of the Y axis. Editable.

Shear Z field Displays the shear of the Z axis. Editable.

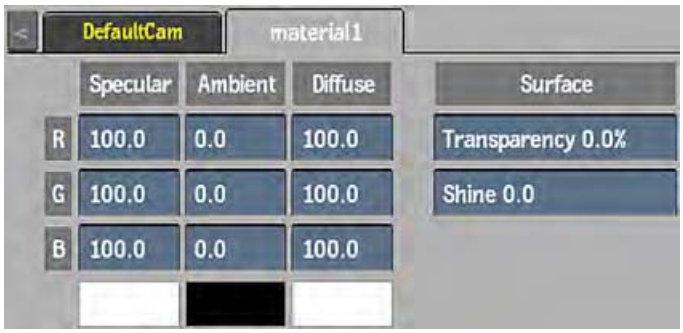
NOTE The UV Transform fields are only available if a Mapping Type other than Default is selected.

Material Menu Settings

If you import a 3ds Max model or an FBX file that has a texture applied to it, that texture map and its parent Axis node is attached to the geometry through a Material node.

You can use the materials imported with the model, or replace the texture with another texture map, or even a Substance texture.

Material Node Menu Settings



Lighting Settings

Red Specular field Displays the red specular highlight value. Editable.

Green Specular field Displays the green specular highlight value. Editable.

Blue Specular field Displays the blue specular highlight value. Editable

Specular colour pot Displays the colour of light reflected by the 3D model's surface. To enable the specular highlight, the Shine value must be larger than 0. Editable.

NOTE Specular lighting sets the colour of light reflected by the 3D model's surface. To enable the specular highlight, the Shine value must be larger than zero.

Red Ambient field Displays the red ambient colour value. Editable.

Green Ambient field Displays the green ambient colour value. Editable.

Blue Ambient field Displays the blue ambient colour value. Editable.

Ambient colour pot Displays the colour of the area of the 3D model that is not illuminated by a direct light source. Editable.

NOTE Ambient lighting sets colour to the area of the 3D model that is not illuminated by a direct light source. The edge of the ambient area mixes with the specular highlight colour and the diffuse colour.

Red Diffuse field Displays the red diffuse colour value. Editable.

Green Diffuse field Displays the green diffuse colour value. Editable.

Blue Diffuse field Displays the blue diffuse colour value. Editable.

Diffuse colour pot Displays the diffuse colour. Editable.

NOTE Diffuse lighting modifies the colour and illumination of the entire 3D model. Diffuse light mixes with the colour of the light sources used to illuminate the 3D model. The diffuse colour may also mix with the ambient colour and the colour of the specular highlight.

Surface Settings

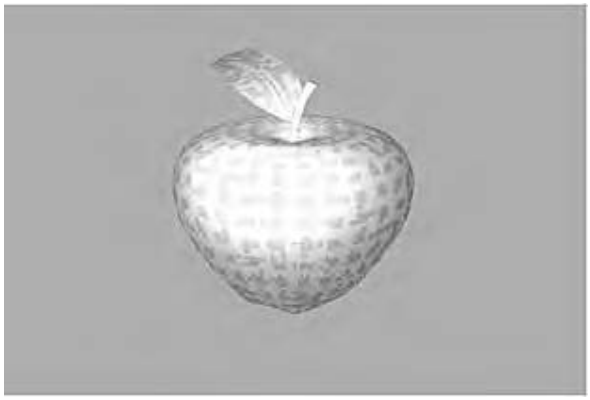
Transparency field Displays the transparency level of the 3D model. Editable.

Shine field Displays the intensity value of the specular highlight. When this value is zero, the specular highlight is disabled. Shine affects both size and intensity. Editable.

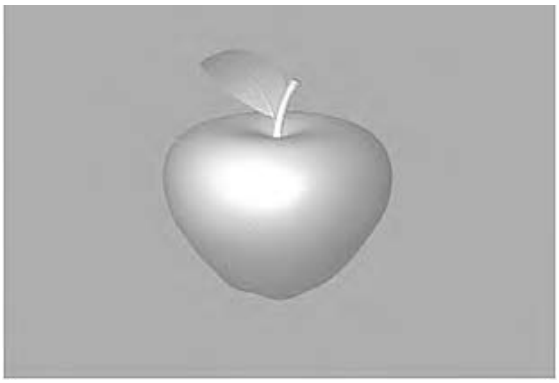
See Creating a Specular Highlight on a Model (page 543).

Adjusting Normals

When using transparency with 3D models, you may sometimes see the back polygons, giving the model a shattered or broken look. This happens when the drawing priority of the normals is not sorted properly. Enable Sort and select an option from the Sort Order box to sort the drawing priority back to front or front to back.



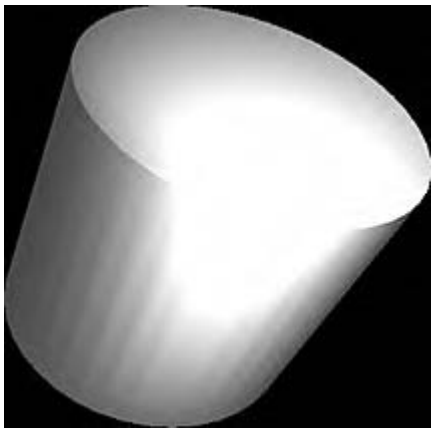
Drawing order of the 3D model's polygons is incorrect. Back polygons are drawn through when the model is transparent.



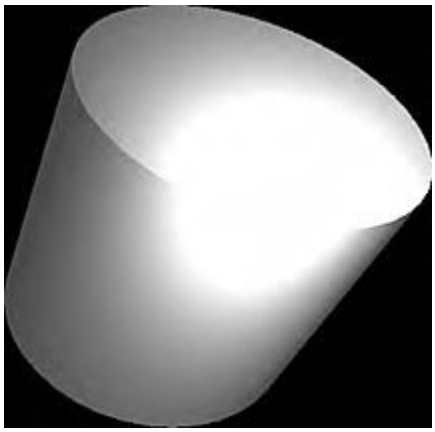
Select Front to sort the polygons front to back. The transparency is drawn correctly.

Subdividing a Model

The Subdivide feature is useful for creating precise highlights and spotlights. The polygons in the 3D model are subdivided at the time of render according to the value you specify, resulting in smoother rendered surfaces.



Spotlight on a polygon model rendered with Subdivide off.



Spotlight on a polygon model rendered with Subdivide set to 2.

To subdivide a model:

1. Select the model or geometry you want to subdivide.
2. In the Geometry menu, click Subdivide to enable the Subdivide field.
3. Edit the Subdivide value. The default value is 5.

   NOTE The smaller the value, the slower the rendering.

4. Click Process or Preview to see the subdivision effect.

You can animate the Subdivide channel in the Channel Editor; however, expect a longer rendering time when the Subdivide value changes over several keyframes.

Creating a Specular Highlight on a Model

Change the specular colour by entering values in the Specular red, green, and blue channel fields or using the colour picker. For example, if the specular colour is red and the light source is white, the specular highlight is also red. If the specular colour is yellow and the light source is red, the highlight is orange.

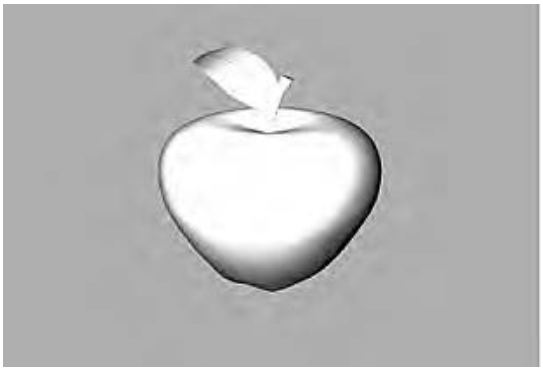
To use a specular highlight with a 3D model:

1. Add and position a light source in the scene.
2. In the scene, select the 3D model to which you want to add the highlight.
3. In the Action Setup menu, enable Shading.

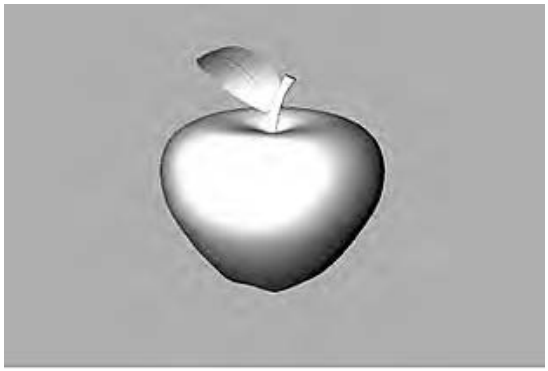


4 In the Geometry menu, set a value in the Shine field.

A high Shine value produces a dimmer highlight while a low Shine value produces an intense highlight. When the Shine field is set to zero, the 3D model does not have a highlight. The following example shows the same 3D model using two different Shine values.



Shine is set to 1. The specular highlight is intense.



Shine is set to 20. The specular highlight is dimmer.

Deforming Models and Surfaces

Use a deformation mesh to deform a 3D model or surface. To use 3D deformations, attach a Deform node to the 3D model or surface, and then use the Deform menu to modify the mesh. You can animate individual points on the deformation mesh, or move the entire mesh over the 3D model or surface to apply the deformation as the mesh passes over the object.

Adding a Deformation Mesh

When you add a Deform Mesh node, the deformation mesh appears over the 3D model or surface. You can view the source mesh, the destination mesh, or both to assist you as you deform the object. You can also turn both meshes off to view only the deformed object.

The deformation mesh consists of cells and lattices. You can divide the mesh into 1-100 lattices and each lattice can be divided by 1-3 cells. Increase the number of cells and lattices to deform specific areas of the object.

To add a deformation mesh:

1 In the schematic, select the axis for the image, 3D model, or 3D text.

2 Do one of the following:

- Drag the Deform Mesh node from the node bar and place it in the schematic.
- Drag the Deform Mesh node from the node bar and place it where you want it in Result view.

- Double-click the Deform Mesh node in the node bar. You do not need to be in Schematic view to add a node in this manner.

The deformation mesh is added to the selected object.



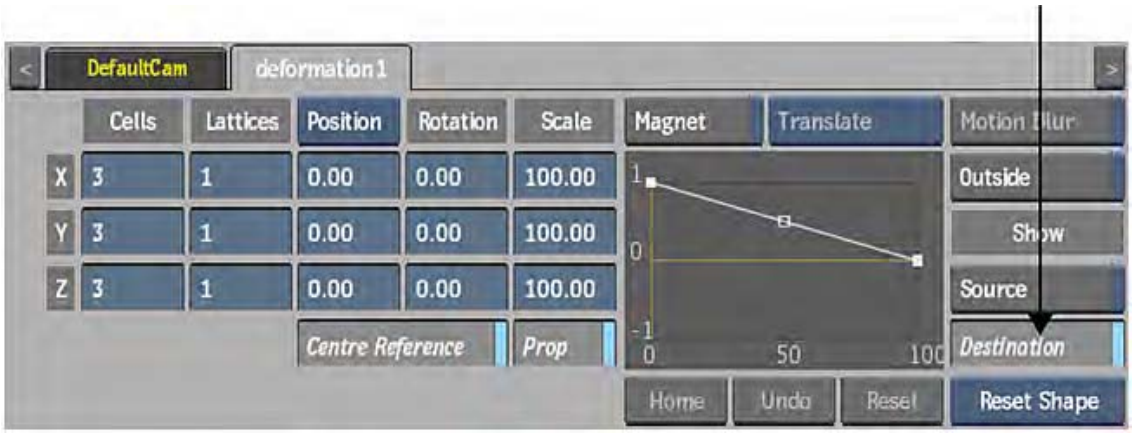
Image courtesy of Quietman

If you do not see the deformation mesh, follow the next steps.

3 Double-click the Deform Mesh node in the schematic.

The Deform Mesh menu appears.

4 In the Deform Mesh menu, enable the Show Destination button to view the deformation mesh in the image window.



Modifying a Deformation Mesh

By default, the deformation mesh has three X,Y, and Z cells and one X, Y, and Z lattice. You can change the number of cells (1-3) or lattices (0-100).

NOTE You can only change the number of cells or lattices before you modify the parameters of the mesh. If you modify a parameter, for example, translate a tangent, you cannot change the number of cells or lattices.

To change the number of cells or lattices:

1 In the image window, select the mesh.

2 In the Deform menu, change the number of cells in the Cells X, Y, and Z fields. By default they are set to 3, 3, 3.

3 In the Lattices X, Y, and Z field, change the number of vertices.

The cells or lattices are added to the deformation mesh.

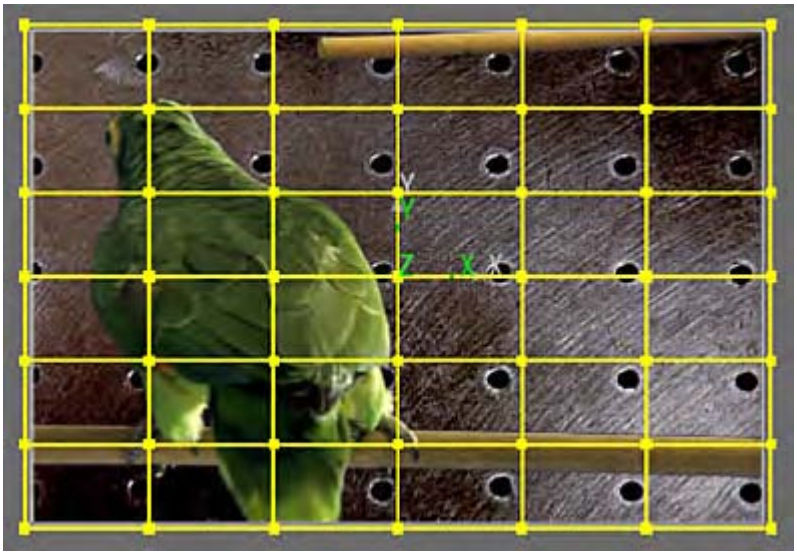


Image courtesy of Quietman

4 To view the original mesh, enable Show Source.

5 Use the Outside button to apply the deformation outside of the mesh (enabled) or constrain the deformation within the mesh (disabled).

To select points:

1 From the Edit Mode box, select Move or Select.

2 Select the point(s):

- To select a single point, click the point.
- To select multiple points, hold the `Shift` key and click the points you want to select.
- To select a range of points, hold the `Ctrl` key and draw a box around the points you want to select.
- To add a range of points to a selection, hold `Shift+Ctrl` and draw a box around the points you want to add to the selection.

When you move a point, all selected points also move.

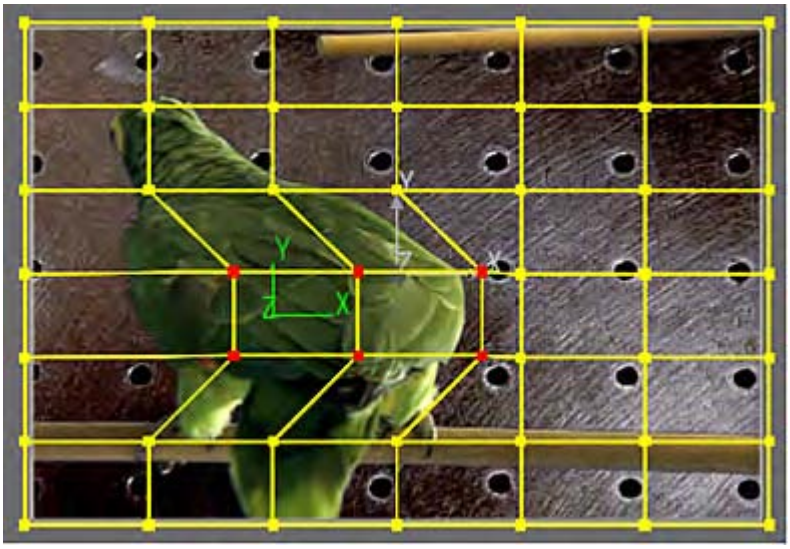


Image courtesy of Quietman

To transform a point on the mesh, drag it in the image window, or change the value in one of the X,Y or Z Translation fields.

Passing a Deformation Mesh Over an Object

Animate the deformation mesh so that it passes over the object. If the Outside button is disabled, the deformation is only applied to the parts of the object within the mesh. To achieve this effect, you must move the mesh over the object. You cannot move the object through the mesh because the object axis parents the mesh axis.

You can also use the channel editor to animate individual or multiple points for the deformation mesh.

To pass a deformation mesh over an object:

1. Add the deformation mesh to the object.
2. In the Deform menu, disable Outside.
3. Modify the points on the mesh to achieve the deformation you want.
4. In Schematic view, select the deformation axis.
5. Click Axis to view the Axis menu.
6. Move the deformation mesh so that it is on one side of the object.
7. Set a new keyframe with the deformation mesh on the other side of the object.

Transforming a Range of Points

Use the magnet to transform a range of deform mesh surface points. In the Deform Mesh menu, the magnet controls include the Magnet button, the Magnet Transformation box, and the Magnet Curve Editor. The magnet controls work the same way as the magnet controls in the Extended Bicubic (page 475) menu.

Deform Mesh Menu Settings

X Cells field Defines the number of cells along the X axis. Editable.

Y Cells field Defines the number of cells along the Y axis. Editable.

Z Cells field Defines the number of cells along the Z axis. Editable.

X Lattices field Defines the number of lattices along the X axis. Editable.

Y Lattices field Defines the number of lattices along the Y axis. Editable.

Z Lattices field Defines the number of lattices along the Z axis. Editable.

Position/Current mode box Select to modify the Position or Current axis controls.

X Position field Positions the selected points along the X axis.

Y Position field Positions the se1lected points along the Y axis.

Z Position field Positions the selected points along the Z axis.

X Current field Positions the selected vertex along the X axis.

Y Current field Positions the selected vertex along the Y axis.

Z Current field Positions the selected vertex along the Z axis.

X Rotation field Rotates the selected points along the X axis.

Y Rotation field Rotates the selected points along the Y axis.

Z Rotation field Rotates the selected points along the Z axis.

X Scale field Scales the selected points along the X axis.

Y Scale field Scales the selected points along the Y axis.

Z Scale field Scales the selected points along the Z axis.

Proportional Scale button Enable to scale the points proportionally.

Centre Reference button Enable to create a centre reference axis for selected points.

Magnet button Enable to transform a range of deform mesh surface points.

Magnet Transformation box Select a transformation type to use when Magnet is enabled.

Magnet Curve Editor Defines the weighted polarity from the centre to the edge of the magnet.

Magnet Curve Home button Resets the position of the magnet curve after panning.

Magnet Curve Undo button Undoes a change to the Magnet Curve Editor.

Magnet Curve Reset button Resets the Magnet Curve Editor.

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

Outside button Enable to apply the deformation outside of the mesh. Disable to constrain the deformation within the mesh.

Show Source button Enable to view the original mesh.

Show Destination button Enable to view the destination mesh.

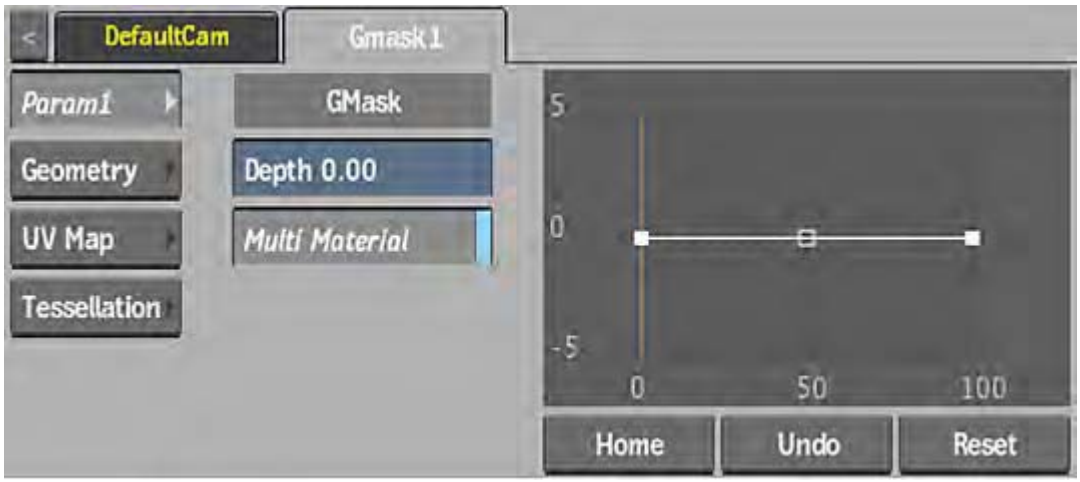
Reset box Select whether to resets the shape of the deformation mesh, or only the selected points.

Using Garbage Masks as 3D Geometry

Garbage mask files are created through the Keyer or by the GMask node in Batch. You can import a saved garbage mask setup into Action, and specify 3D properties, such as depth and geometry settings. If your garbage mask is animated, all shapes are imported into Action. Softness and offset settings in the imported garbage mask are not brought into Action. Any holes in the imported garbage mask are also holes in the 3D geometry. For help creating garbage masks, see Drawing a Mask (page 897).

TIP A library of preset garbage mask setups is available. To load a preset garbage mask, navigate to the */usr/discreet/<product_name>/gmask* directory when importing your 3D model.

Once you import a garbage mask as a 3D geometry into Action, use the GMask menu to extrude the garbage mask.



GMask Menu Settings

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

Multi Material button Enable to create an Object Group node for each of the front, back, and extrude of the 3D object. You can then attach a different texture map to apply to the different surfaces.

Bevel Curve Applies a bevel to the depth of the geometry when you manipulate the Bevel curve. You can move and add points to the curve, as well as adjust the tangent handles to produce different effects with the text string.

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

Undo button Undoes the last set of Bevel curve operations.

Reset button Resets the Bevel curve.

Click the Geometry (page 538) and UV Map (page 540) tabs to apply any of the other geometry settings to your garbage mask.

Click the Tessellation (page 575) tab to control the tessellation of the garbage mask.

About 3D Text

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

Using 3D Text Presets

A number of 3d text presets are included in Action, such as rotating or fading text to add to your scene. These presets can help you add complex text effects with just a few clicks.

To add a 3d text preset:

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

     The file browser opens.
2. From the Preset Type box, select 3D Text.



   The 3D Text Preset file browser appears, pointing to the default location of the presets: *usr/discreet/<product home>/3d_text_presets*.
3. Optional: Enable Scale to Action Resolution to load the preset in the current Action resolution.
4. Optional: Select which rendering settings to enable or disable in the preset (Z-Buffer, Shading, Polygon Resolution, and Colour Clamping).

   NOTE These settings are enabled by default, and by disabling any of them, you may not see the intended results in the preset.
5. Select the 3d text preset you want to load. Hold `Ctrl` and click to select multiple presets.

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

   The 3d text preset is then appended to your Action scene. In the 3D Text menu, you can change the default text string of the preset.

3D Text Presets Usage Tips

A number of 3D text presets are included in Action, such as fading or spinning 3D text effects to add to your scene. The presets are easily added to your scene from the Action node bar. For more information on the 3D Text node and its menu, see About 3D Text (page 550).

The 3D Text presets are designed for ease-of-use, so you should be able to easily identify the various schematic nodes and their corresponding menus. For example, the nodes in the schematic for each preset are renamed to give you an indication of the functionality of the node. Many of the presets include a *transition_ctrl* Axis node, that you can use to modify the timing of the text effect, as follows:

Rotation X=duration of the In effect; Y=start weight; Z=end weight. The Y and Z Rotation fields represent the shape of the "S" curve of the In effect, if applicable. The combined Y and Z values should equal 100.

Scale X=duration before the start of the In effect; Y=duration between the end of In effect and the start of the Out effect.

Shear X=duration of the Out effect; Y=start weight; Z=end weight. The Y and Z Shear fields represent the shape of the "S" curve of the Out effect, if applicable. The combined Y and Z values should equal 100.

Centre These fields include expressions to produce animation curves, and shouldn't be altered.

When loading a 3D Text preset, with a few exceptions, you can replace the **Type Your Own Text** string in the Text field of the 3D Text menu. Some of the presets have multiple text entries. In these cases, replace the text string of each Text Geom node with your preferred text string.

Presets Browser Settings

Preset Type box Select the category of presets to display in the browser.

Scale to Action Resolution button Enable to load the preset in the current Action resolution.

Z-Buffer button Enable to load the Z-buffer rendering settings of the preset.

Shading button Enable to load the shading rendering settings of the preset.

Polygon Resolution button Enable to load the rendering resolution settings of the preset.

Colour Clamping button Enable to load the resolution colour clamping settings of the preset.

3D Text Preset List

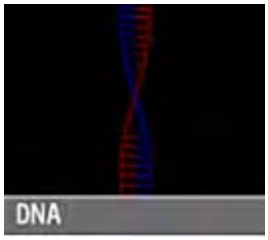
Use the following table to get a quick overview of the 3D text presets available in this release, along with particular comments, if applicable.

Proxy	Name/Description/Comments
	**3DRotate** Text on a rotating circular path. Edit timing using the Animation Track Editor.

Proxy	Name/Description/Comments
	**3DRotate_Shadow** Text rotating on a circular path with a fake shadow. Edit timing using the Animation Track Editor.
	**3DRotate_Transition** Transition between two text strings rotating on a circular path. Edit timing using the Animation Track Editor.
	**Behind_Camera_3D** Characters coming from behind the camera. Edit timing using *transition_ctrl* settings.
	**Bounce** Characters falling and bouncing. Edit timing using *transition_ctrl* settings. Set duration and amplitude of the bouncing.
	**Break_and_Fall** Characters breaking in two parts and falling down. Edit timing using the Animation Track Editor.

Proxy	Name/Description/Comments
	**Bubble_Pop** Characters forming a bubble which pops. Edit timing using *transition_ctrl* settings.
	**Chop_Sticks** Characters arriving one after the other with a scale and rotation effect. Edit timing using *transition_ctrl* settings.
	**Circle** Characters moving along a circular path. Edit timing using the Animation Track Editor.
	**Circle_Transition** Transition between two text strings, in which characters are moving along a circular path with a banking effect. Edit timing using *transition_ctrl* settings.
	**Climber** Characters climbing from the bottom to the top of the frame. Edit timing using *transition_ctrl* settings. Use offset.rotation.X to set the displacement duration and offset.rotation.Y to set the rotation duration.

Proxy	Name/Description/Comments
	**Clock_Transition** Transition between four text strings rotating like clock hands. Edit timing using *transition_ctrl* settings.
	**Colour_Flasher** Moving coloured flashing text. Edit timing using *transition_ctrl* settings. Colour_offset sets the time offset between red, green, and blue channels.
	**Counter** Incremental counter with numbers 000 to 999.
	**Diagonal** Characters dispersing diagonally in the frame. Edit timing using *transition_ctrl* settings.
	**Dispersion** Characters dispersing randomly. Use time_stretch.scaling.X to slow down or speed up the effect.

Proxy	Name/Description/Comments
	**Dispersion_Transition** Transition between two text strings using a dispersion movement. Edit timing using *transition_ctrl* settings.
	**DNA** Animated double helix structure using two text strings of the letter "T".
	**Dominos** Characters falling one after the other. Edit timing using *transition_ctrl* settings.
	**Fade_In_Flicker** Fade in of randomly flickering characters. Edit timing using *transition_ctrl* settings.
	**Fade_In_From_Centre** Fade in from the centre to the extremities of the text. Edit timing using *transition_ctrl* settings. Set characters.rotation.X to the number of characters in the Text field.

Proxy	Name/Description/Comments
	**Fade_In_From_Left** Fade in from the left to the right of the text. Edit timing using *transition_ctrl* settings.
	**Fade_In_Out_By_Character** Fade in and out one character after the other. Edit timing using *transition_ctrl* settings. Set characters.rotation.X to the number of characters in the Text field.
	**Fade_In_Random** Use time_stretch.scaling.X to slow down or speed up the effect.
	**Fade_In_To_Centre** Fade in from the extremities to the centre of the text. Edit timing using *transition_ctrl* settings.
	**Fade_Out_From_Centre** Fade out from the centre to the extremities of the text. Edit timing using *transition_ctrl* settings. Set characters.rotation.X to the number of characters in the Text field.

Proxy	Name/Description/Comments
	**Fade_Out_From_Right** Fade out from the right to the left of the text. Edit timing using *transition_ctrl* settings.
	**Fade_Out_Random** Use time_stretch.scaling.X to slow down or speed up the effect.
	**Fade_Out_To_Centre** Fade out from the extremities to the centre of the text. Edit timing using *transition_ctrl* settings.
	**Fast_Beam** Characters arriving from the right side of the frame, with scaling and flickering. Edit timing using *transition_ctrl* settings. Customize the flicker effect using *flicker* settings.
	**Flasher** Customize the flashing using *period_offset* settings.

Proxy	Name/Description/Comments
	**Flicker** Animate with period.rotation.X.
	**Flip_Transition** Transition between two text strings, with characters rotating on the Y axis. Edit timing using *transition_ctrl* settings.
	**Flutter** Characters arriving one after the other with a scale and rotation effect. Edit timing using *transition_ctrl* settings.
	**Flying_Transition** Transition between two text strings in which characters are flying out from the centre and coming back to it. Edit timing using *transition_ctrl* settings.
	**From_Back** Characters arriving from behind the camera. Use time_stretch.scaling.X to slow down or speed up the effect.

Proxy	Name/Description/Comments
	**From_Bottom** Characters arriving from the bottom of the frame, with rotation. Edit timing using *transition_ctrl* settings.
	**From_Camera_3D** Characters arriving from behind the camera and turning towards the camera. Edit timing using *transition_ctrl* settings.
	**Ghost** Text arriving toward the camera, reflecting, and fading. Edit timing using the Animation Track Editor.
	**Ghost_Cascade** Similar to Ghost, with a cascade applied. Edit timing using the Animation Track Editor.
	**Ghost_Rotation** Similar to Ghost with an animated ParticleDraw axis. Edit timing using the Animation Track Editor.

Proxy	Name/Description/Comments
	**Helix_Transition** Transition between two text strings in which characters are rotating on their X axis. Edit timing using *transition_ctrl* settings.
	**Insertion** Text is revealed frame after frame in insert mode. Edit timing using *transition_ctrl* settings. Use offset.rotation.X to set the starting value of the path.
	**Kern** Characters arriving and leaving the frame with kerning and rotation. Edit timing using *transition_ctrl* settings.
	**Mirror** Text on path with a mirror effect. Edit timing using the Animation Track Editor.
	**Negate** Text revealed using a negate blending effect. Edit timing using *transition_ctrl* settings.

Proxy	Name/Description/Comments
	**Ocean_Tide** Text moving slowly as if characters were floating on the sea. Use period and amplitude axes to customize the wavy effect.
	**Ocean_Tide_Transition** Transition between two text strings moving slowly as if characters were floating on the sea. Edit timing using *transition_ctrl* settings. Use period and amplitude axes to customize the wavy effect.
	**Path_Warp** Text on animated path producing a warping effect. Edit timing using the Animation Track Editor.
	**Pendulum** Characters moving as if attached to a pendulum Edit timing using *transition_ctrl* settings. Use period and amplitude axes to customize the oscillating effect.
	**Pulse_Wave** Characters moving vertically with a pulsating effect. Customize using period and amplitude settings.

Proxy	Name/Description/Comments
	**Quick_Rotate** Characters rotating on their X axis. Edit timing using *transition_ctrl* settings.
	**Quick_Twist** Characters scaled on their Y axis. Edit timing using *transition_ctrl* settings.
	**Random_Display** Random letters with a specific starting and ending text string. Edit timing using *transition_ctrl* settings. Type the start text in text_1 and the stop text in text_2.
	**Random_Hide** Characters disappearing randomly. Clipping Plane modified. Use time_stretch.scaling.X to slow down or speed up the effect.
	**Random_Letters** Letters displayed randomly.

Proxy	Name/Description/Comments
	**Random_Reveal** Characters appearing randomly. Clipping Plane modified. Use time_stretch.scaling.X to slow down or speed up the effect.
	**Replace_Transition** Transition between two text strings with rotation. Edit timing using *transition_ctrl* settings.
	**Reverse** Transition between two text strings in which characters reverse direction. Edit timing using the Animation Track Editor.
	**Rubber** Characters are stretched and released like rubber. Edit timing using *transition_ctrl* settings. Set period and amplitude of the bouncing.
	**Rubber_Transition** Transition between two text strings using a rubber effect. Edit timing using *transition_ctrl* settings. Set period and amplitude of the bouncing.

Proxy	Name/Description/Comments
	**Scaling_Transition** Transition between two text string using a scaling effect. Edit timing using *transition_ctrl* settings. Set characters.rotation.X to the number of characters in the Text field.
	**Shiver_Transition** Transition between two text strings with a shiver-like effect. Edit timing using *transition_ctrl* settings. Increase or decrease the turbulence using noise_amplitude.
	**Shuffle_In_Out** Characters arriving from the left, stopping in the centre, and leaving to the right. Edit timing using *transition_ctrl* settings.
	**Spin_Compress** Characters revealed with rotation, and leaving with a compression effect. Edit timing using the Animation Track Editor.
	**Spin_In_Out** Characters arriving and leaving with rotation. Edit timing using *transition_ctrl* settings.

Proxy	Name/Description/Comments
	**Spin_Transition** Transition between two spinning text strings. Edit timing using *transition_ctrl* settings.
	**Spinning_Letters** Characters thrown with a spinning effect. Edit timing using *transition_ctrl* settings. Experiment with transition_ctrl.scaling.Y and transition_ctrl.scaling.Z.
	**Spiral** Text on a spiral path. Edit timing using the Animation Track Editor.
	**Spiral_Transition** Transition between two text strings with a spiral effect. Edit timing using *transition_ctrl* settings.
	**Spring_Transition** Transition between two text strings with letters springing left and right. Edit timing using *transition_ctrl* settings. Customize the pulsing effect using period and amplitude settings

Proxy	Name/Description/Comments
	**Squash_Transition** Transition from a text string with letters being squashed to reveal the other text string. Edit timing using *transition_ctrl* settings.
	**Star** Text moving along a star path. Edit timing using the Animation Track Editor.
	**Stretch** Text arriving from the left of the frame with a stretching effect. Edit timing using *transition_ctrl* settings.
	**Subtract** Text revealed using a subtract blending effect. Edit timing using *transition_ctrl* settings.
	**Text_Falling** Characters moving along a line, then falling down. Edit timing using the Animation Track Editor.

Proxy	Name/Description/Comments
	**Titles** Three text strings on a path arriving from the upper left. Edit timing using the Animation Track Editor.
	**Titles_Stairs** Three text strings revealed with a spinning effect. Edit timing using *transition_ctrl* settings.
	**Track_In** Characters appear and disappear with scaling. Edit timing using *transition_ctrl* settings.
	**Tremor** Characters appear as if shaking. Increase or decrease the turbulence effect using noise_amplitude.
	**Twist** Characters arriving and leaving with a twisting effect. Edit timing using *transition_ctrl* settings.

Proxy	Name/Description/Comments
	**Typewriter** Text revealed frame after frame with a typewriter effect. Edit timing using *transition_ctrl* settings. Use transition_ctrl.rotation.X andtransition_ctrl.shearing.X to set the fade in duration of characters.
	**Typewriter_Cursor** Text is revealed frame after frame with a cursor preceding characters. Edit timing using *transition_ctrl* settings. Use transition_ctrl.rotation.X and transition_ctrl.shearing.X to set the fade in duration of characters. Set as many white squares as characters in the text node. Adapt cursor kerning if needed.
	**Vertical_In_Out** Characters arriving from the left side of the frame and moving vertically before going out to the right of the frame. Edit timing using *transition_ctrl* settings.
	**Vertical_Rotation_Transition** Transition between two text strings moving vertically. Edit timing using *transition_ctrl* settings.
	**Waveform** Waveform using Cursor font. Link text.character_anim.scaling.Y to an audio file.

Proxy	Name/Description/Comments
	**Whip_Effect** Characters whipped by a rotating effect. Edit timing using *transition_ctrl* settings.
	**Whip_Transition** Transition between two text string with characters whipped by a rotating effect. Edit timing using *transition_ctrl* settings.
	**Whirlwind_Transition** Transition between two text string with characters circulating by a rotating effect. Edit timing using *transition_ctrl* settings.
	**White_Background** Characters revealed on a white background. Edit timing using *transition_ctrl* settings. Change the background using white_bkgd_ax settings and white_bkgd kerning value.
	**Windy** Characters moving as if by wind. Customize with period and amplitude axes.

Proxy	Name/Description/Comments
	**Zig_Zag** Characters arriving and leaving following a zig-zag path. Edit timing using *transition_ctrl* settings. Change the path, if needed.

Adding a 3D Text Node

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

To add a 3D Text node to the scene:

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

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

   See Populating Menu Tabs of Selected Objects (page 415).

You can also add a 3D Text node using the Presets node. See Using 3D Text Presets (page 550).

Changing 3D Text Properties

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

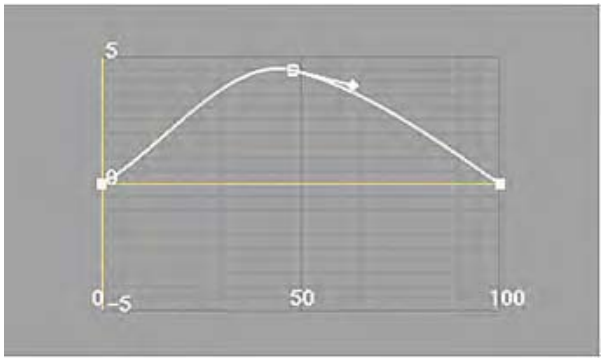
To change a text string:

1. Click the Text field.

   The on-screen keyboard appears, representing the character set for the selected font. Enable Up ASCII to access the rest of the character set. If the selected font has special symbol characters, enable Symbols to see them.
2. Type your text string or use the on-screen keyboard.
3. Click Exit Keyboard. The text string is displayed in the Text field and automatically updated in the scene.
4. Use the settings in the 3D Text tab to change the font, size, depth, and other text properties.

Creating Bevelled Text

Use the Bevel curve to add a bevelled edge to your 3D text. Use the options in the Edit Mode box to add, select, delete, or move keyframes on the Bevel curve. The Bevel curve behaves in much the same way as an animation curve in the Channel Editor. Experiment with different curves to create different effects.



Bevel curve



Resulting bevelled text

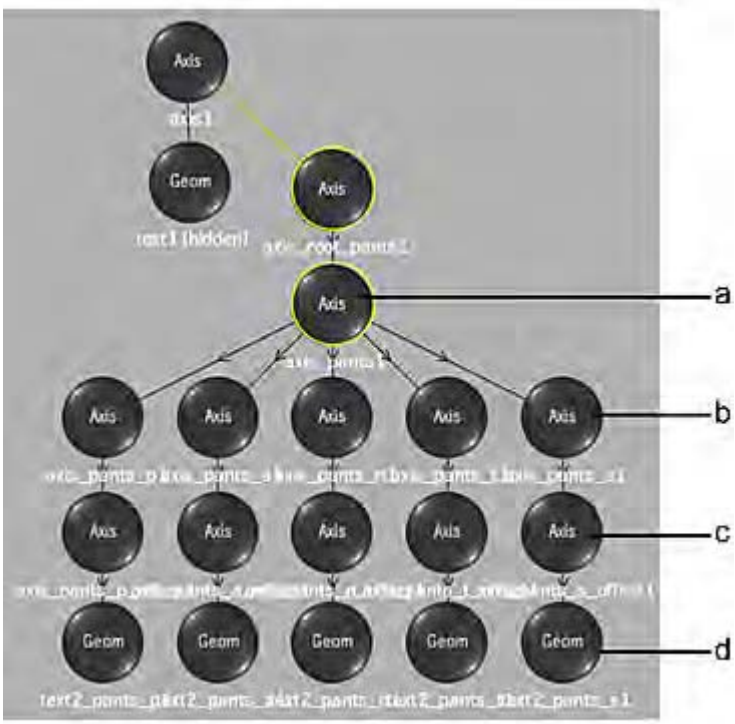
Separating Text

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

To separate text:

1. Select the text to separate.
2. From the Text tab, click Separate.

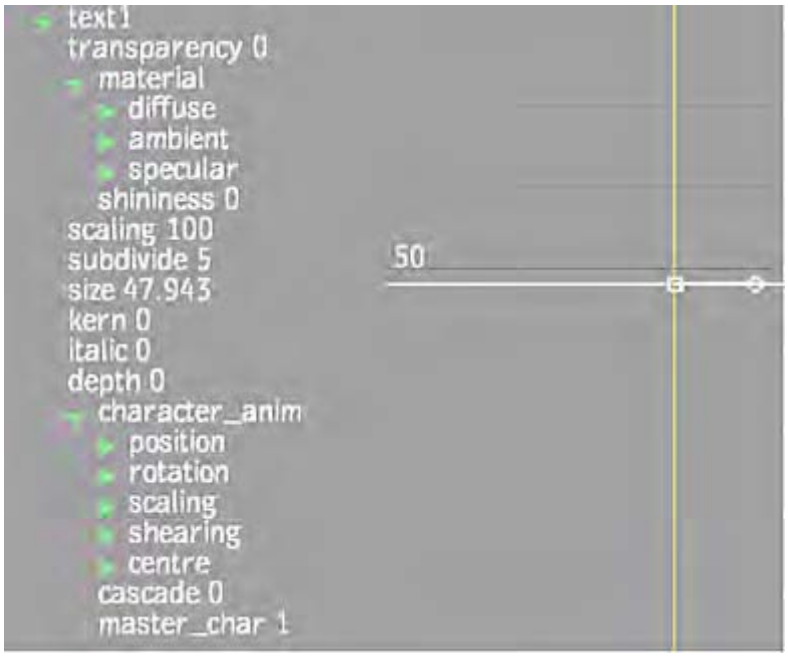
   Each letter of the text geometry is now an independent geometric object, and has its own axis and offset (to separate the letters). Each word is also given its own axis. The original 3D Text node is hidden in the schematic.



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

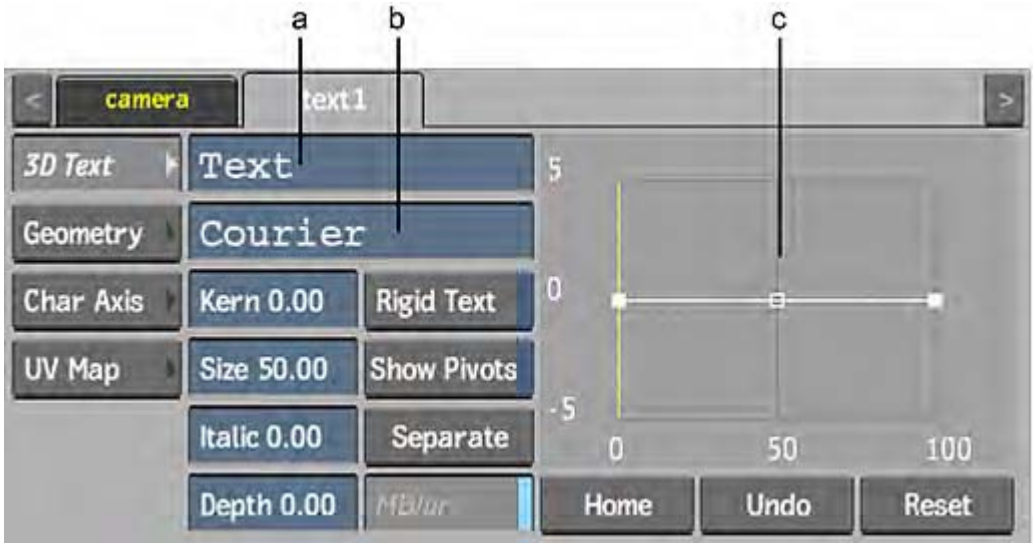
Animating 3D Text

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



3D Text Menu Settings

3D Text Tab



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

Text field Displays the current text string. Editable.

Font field Displays the current font. Click to open the font library to select a different font for the text.

You specify the default font in the Preferences menu. Also, you can install additional fonts for use with Flame Premium.

Kern field Displays the kerning value for the text string. Editable.

Size field Displays the font size for the text string. Editable.

Italic field Displays the level of italics for the characters in the text string. Editable.

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

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

Show Pivots button Enable to display the pivot points for each individual text character in the 3D Text string (displayed in the image window in red). When disabled, only the master character pivot point is displayed (in green).

This setting can also be found in the Character Axis tab.

Separate button Click to separate selected text so that each letter has its own axis. See Separating Text (page 571).

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

Bevel Curve Applies a bevel to the depth of the geometry when you manipulate the Bevel curve. You can move and add points to the curve, as well as adjust the tangent handles to produce different effects with the text string.

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

Undo button Undoes the last set of Bevel curve operations.

Reset button Resets the Bevel curve.

Multi Material button Enable to create an Object Group node for each of the front, back, and extrude of the 3D object. You can then attach a different texture map to apply to the different surfaces.

Character Axis Tab

You can change the axis properties of your 3D text string characters. This can be useful in offsetting your text from a 3D path.



(a) Cascade Alignment box (b) Vertical Pivot box (c) Horizontal Pivot box

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

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

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

Cascade Alignment box Select the flow of the cascade offset, with respect to the master character.

X Position field Displays the position of the offset along the X axis. Editable.

Y Position field Displays the position of the offset along the Y axis. Editable.

Z Position field Displays the position of the offset along the Z axis. Editable.

X Rotation field Displays the rotation of the offset along the X axis. Editable.

Y Rotation field Displays the rotation of the offset along the Y axis. Editable.

Z Rotation field Displays the rotation of the offset along the Z axis. Editable.

X Scale field Displays the scale of the offset along the X axis. Editable.

Y Scale field Displays the scale of the offset along the Y axis. Editable.

Z Scale field Displays the scale of the offset along the Z axis. Editable.

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

X Shear field Displays the shear of the offset along the X axis. Editable.

Y Shear field Displays the shear of the offset along the Y axis. Editable.

Z Shear field Displays the shear of the offset along the Z axis. Editable.

X Centre field Displays the centre of the offset along the X axis. Editable.

Y Centre field Displays the centre of the offset along the Y axis. Editable.

Z Centre field Displays the centre of the offset along the Z axis. Editable.

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

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

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

Geometry and UV Map Tabs

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

See 3D Geometry Menu Settings (page 538) for definitions of the specific Geometry and UV Map controls.

Changing Tessellation Properties

Tessellation is the process of tiling the curves' shapes with polygons. Flame Premium offers three different tessellation methods:

Tessellation Type box Select the tessellation type you want to apply to the geometry. More settings appear if you select Delaunay or Medial Axis.

- **Standard (GLU)** is the least taxing tessellation option, although it is also the least efficient.
- **Delaunay** generates a mesh composed entirely of triangular polygons. This method gives consistent and predictable results, and in particular, it will not give different results if the tessellated objects are rotated.
- **Medial Axis** creates concentric contour lines along the medial axes (averages between the input boundary curves), morphing from one boundary shape to the next.

  The Wireframe box is also provided in the Tessellation menu. It is the same setting as in the Geometry tab, but repeated here for ease-of-use.

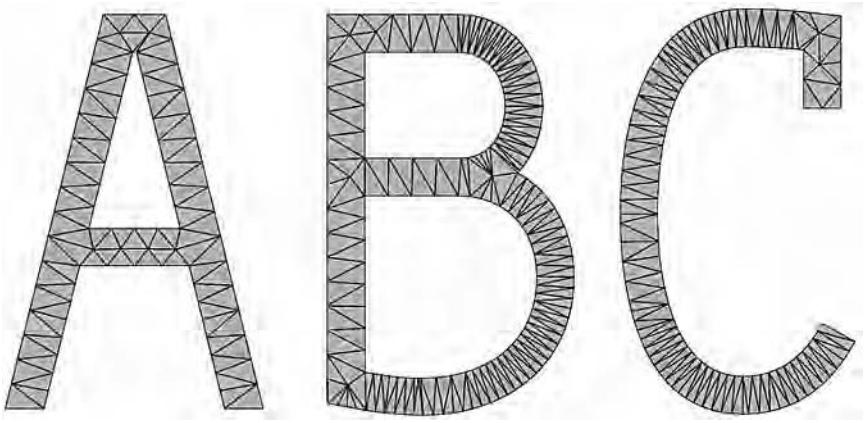
Each method has its own set of options, described in the sections that follow.

Standard (GLU)

The Standard (GLU) tessellation method is the legacy tessellation option; while being very light in its processing requirements, it is also the least efficient and precise. And compared to Delaunay and Medial Axis tessellation methods, it has no options to fine tune the resulting tessellation.

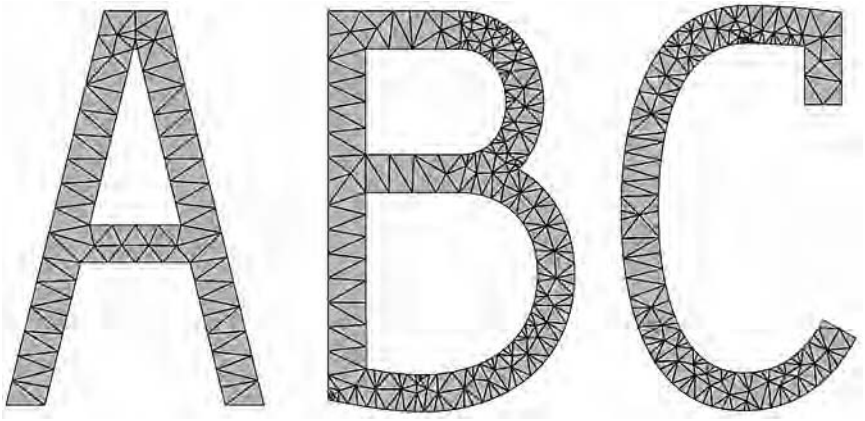
Delaunay

The Delaunay tessellation method (or more precisely, *constrained Delaunay tessellation*) generates a mesh composed entirely of triangular polygons. This method gives consistent and predictable results, and in particular, it does not give different results if the curves are rotated.



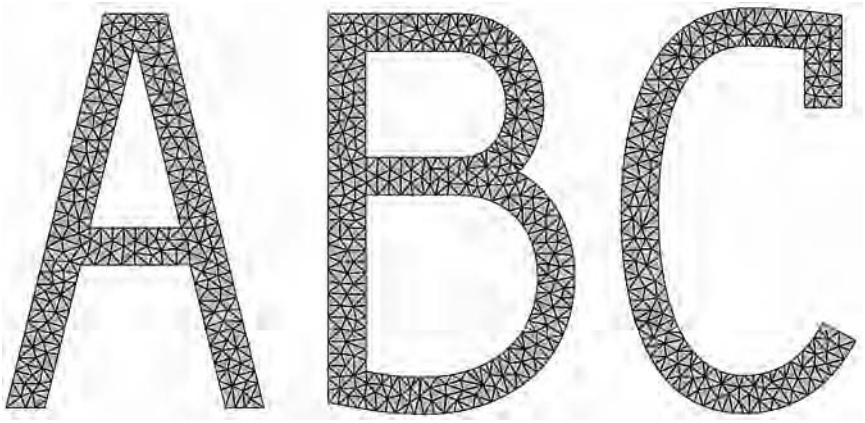
With this method, there are several options for fine-tuning the tessellation further.

Min Angle field Displays the smallest value of angle that the polygons can have. If a triangle contains an angle that is smaller than this value, it gets replaced by better-shaped ones. Eliminating small-angled triangles gives a more uniform shading and is more suited for deformations.



Min Angle = 20

Max Area field Displays the largest value of area that the polygons can have. If a triangle is larger than this value, it gets replaced by smaller ones. This allows the polygon mesh to be deformed more smoothly.



Max Area = 5

Max Vertices field Displays the total number of new vertices that can be added by the Minimum Angle and Maximum Area options. 0 by default.

Use this option as a precaution against accidentally setting the other options to values that would create huge amounts of geometry with long processing times.

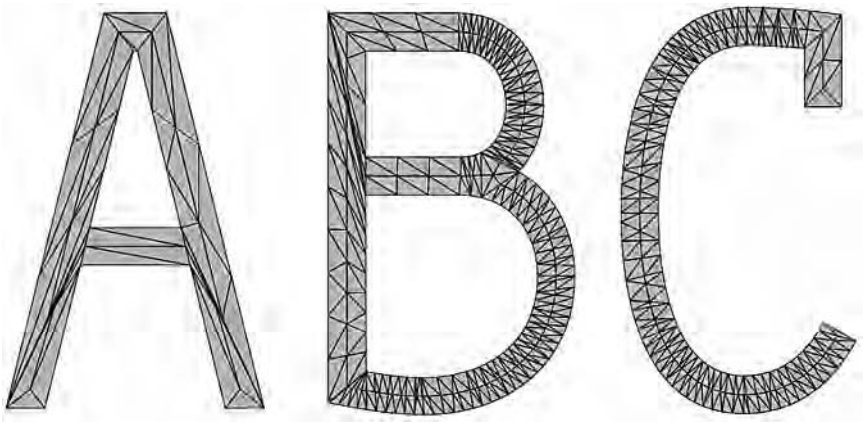
It is not recommended that you rely on this option to control the final number of vertices because it can force the tessellation to stop before the process is completed, thereby giving an unpredictable combination of polygon shapes and sizes.

Boundary Split box Select an option to control tessellation along boundaries.

Select:	To:
Free	Allow the boundary edges along the outer contour and inner holes to be split further during tessellation. This is particularly useful for text and other shapes that may contain straight edges that need to be deformed smoothly.
None (Contour Only)	Allow boundary edges along inner holes to be split, but not boundary edges along the outer contour. Note that this may affect the uniformity of the mesh if you enabled Min Angle or Max Area.
None (Contour and Holes)	Prevent any boundary to be split. Note that this may affect the uniformity of the mesh if you enabled Min Angle or Max Area.

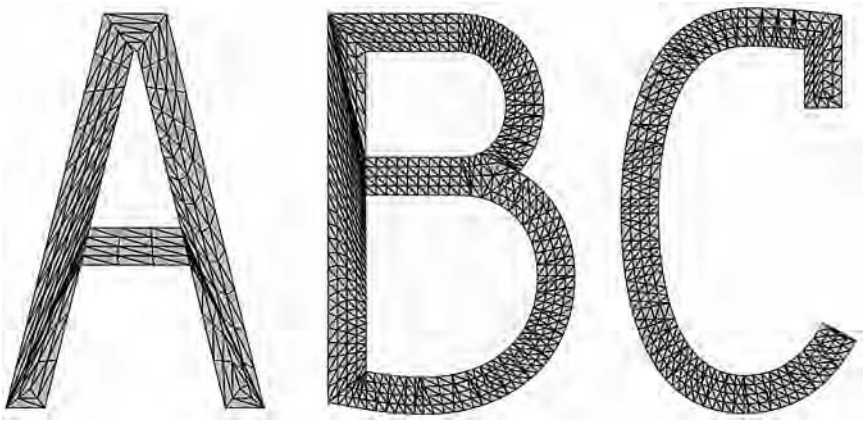
Medial Axis

The Medial Axis tessellation method creates concentric contour lines along the medial axes (averages between the input boundary curves), morphing from one boundary shape to the next. This method creates mainly quads with some triangles, so it is well-suited for subdivision surfaces.



With this method, there are several options for fine-tuning the tessellation further.

Loops field Displays the number of loops used in the tessellation, modified by the Adaptive toggle.



Loops = 2

Adaptive button Enable to set the Loops field as the average number of medial axes drawn and to keep the distance between them fairly constant. Disable to set the Loops field as the exact number of medial axes drawn per boundary (rounded to the nearest integer).

Backtrack Length field Displays the tessellation value at the extremities. Editable.

Set to:	To have:
0	The medial axis intersects boundaries at each point of concavity, which can often create many small triangles especially in sharp extremities.
Positive value	The medial axis does not extend completely to the boundary and the remaining area is tessellated with a fan shape.
Negative value	Sharper embossing effects

Split Edges to Enhance button Enable to add vertices to allow the contour lines to follow the medial axes more accurately. Turn this option off if there are no holes in the geometry; otherwise, there may be shading artifacts along internal curves.

Edge Tessellation box Select an option to control the shape of the polygons.

Select:	To:
None	Have long edges that are not split. This results in fewer polygons and lighter geometry, but the resulting long, thin polygons may not deform well.
Equal on Both Sides	Have a tessellation made of squarer polygons that deform better, at the cost of a heavier tessellation.

About Shading and Textures

Action uses shaders to compute the colour, lighting, shadows, and other attributes of each pixel or vertex of objects in the scene. Shaders use the processing pipeline of the GPU to accelerate object-specific rendering effects. You can use shaders to control the interaction between surfaces or models and the lights in the scene to contribute to the realism of a material simulated in a texture.

You can use any media to map textures to Action surfaces and geometries, thus adding detail such as depth and reflections to your 3D composites.

Substance Textures

Use the Substance Texture node to quickly create photorealistic procedural textures using a library of organic, fabric, and material presets. A loaded preset consists of a Material Node, a Substance node, as well as other texture maps and shaders, as needed.

TIP You can also create textures with your own input materials, such as photographs, using the Substance Materialize (page 585) node.

To add a Substance preset:

1. In the schematic, select the surface or 3D geometry to which you want to apply the Substance Texture.
2. Do one of the following:
   - Drag the Substance Texture node from the node bin and place it in the schematic.
   - Double-click the Substance Texture node.

     The file browser appears, pointing to the default location of the presets: *usr/discreet/<product home>/substance_presets/TEXTURE*

   TIP Switch to Proxies view to see a visual representation of the presets.
3. Navigate through the subfolders and select the Substance Texture preset you want to load.

   The preset, with applicable maps and shaders, is automatically loaded into Action.

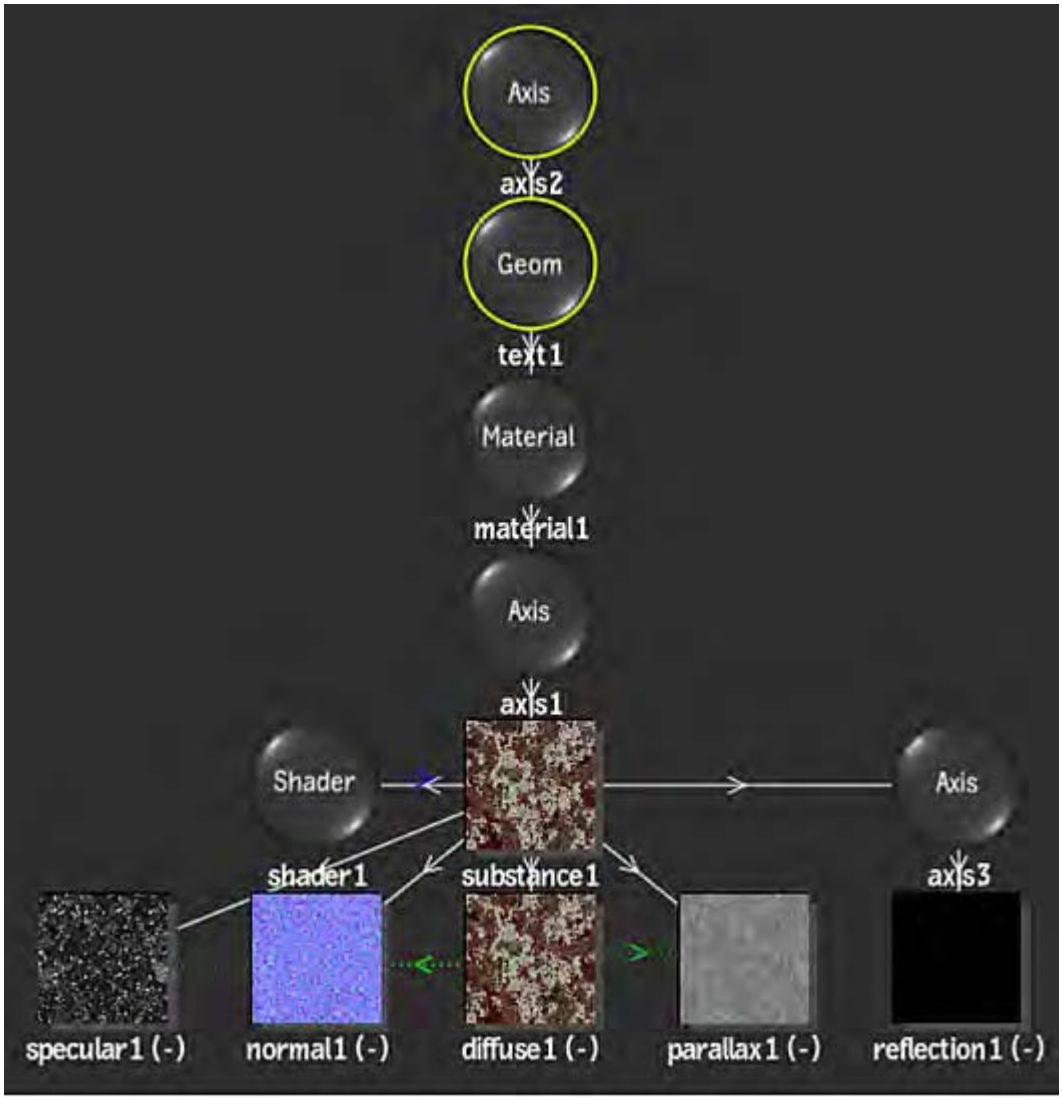
Using Third-Party Subtance Textures

While Flame Premium comes with a selection of Substance Texture presets, you can also use Substance Textures available in Autodesk Maya, Autodesk 3ds Max, or created with the Substance Designer from Allegorithmic. In these cases, a Substance Texture file uses the *.sbsar* file extension.

To use an *.sbsar* Substance Texture file, simply select Interop Format from the Substance Format box in the preset file browser, then navigate to and select your *.sbsar* file. The Substance Texture, with applicable maps and shaders, is automatically loaded into Action.

Navigating the Schematic with a Substance Texture

When you load a Substance Texture preset into the Action schematic, multiple nodes are added and connected with different types of links, creating a type of Substance group. You can work with the menus of the various nodes (such as the Parallax node) as you would if you had added the object manually.



Keep in mind the following when working with Substance Texture presets in the schematic and menus:

- The Substance node is parented by an Axis node, itself parented by a Material node, and is the parent of some or all of the following map nodes:
  - Specular node — see Specular Mapping (page 608)
  - Normal node — see Normal Mapping (page 604)
  - Diffuse node — see Diffuse Mapping (page 615)
  - Parallax node — see Parallax Mapping (page 600)
  - Displacement node (locked to Hardware Displacement) — see Displacement Mapping (page 595)
  - Emissive node — see Emissive Mapping (page 611)

  Media is automatically applied to these mapping nodes, and cannot be changed.
- The Substance node is also parented to a Reflection node, with its own axis. You can apply media to the Reflection node. See Reflection Mapping (page 619).
- A Shader node is parented from the Substance node. Lighting links (blue dotted lines) are applied from the Shader node to the Substance node and the originally selected surface or geometry. See Using the Shader Node (page 588).

- Some presets require you to add input media to feed into the Substance node and attached maps to create a plausible effect. For example, the *Broken_Glass* preset in the *Glass*category requires one input node. See Substance Textures with Inputs (page 581).
- All Map parameters are loaded with default values based on the loaded Preset. Reflection map parameters sometimes have an expression automatically set on its effect or softness values.
- The Diffuse node acts as an originating duplicate object to the present Specular, Normal, Emissive, and Parallax nodes (green dotted lines). Therefore, any common settings applied to one of these map nodes are applied to all map nodes. The common settings are found in the Texture tab of any of the map menus. See Mimicking, Copying, and Duplicating Objects (page 421).
- Since the Substance preset requires all of the loaded objects, and the parenting, lighting, and duplicate links to function correctly, you are unable to delete any of the objects separately, or break any of the links.
- Some presets load and use only the maps that are needed for that preset. For example, the *Defocused_Light* preset in the *Abstract* category includes only an Emissive map that blends in additive mode with the diffuse colour of the attached surface or geometry. In this case, the preset works best if the diffuse colour in the Geometry or Image menu is set to black.
- You cannot parent an object from any of the Substance objects. You can parent the Material node to another object to substitute for one of the Substance maps. For example, you can add a Diffuse map and connect it to the Material node, and then hide the Substance DIffuse map so that it is no longer applied to the Substance effect.
- If you select a surface or geometry before adding the Substance Texture preset, the connection to the Material node of the Substance node, and the lighting link from the Shader node are automatically applied. If you do not select a surface or geometry first, you have to make these connections manually for the Substance node to function correctly.

Substance Textures with Inputs

Some semi-transparent presets, such as those with water or glass, may require one or more media inputs to be added manually. This is done so that the Substance node and attached maps can correctly create a plausible effect. When you load a Substance Texture preset that requires media input, one or more input nodes are attached to the Substance node in the schematic.

To specify media as the input source:

1. Select the Input node in the schematic.
2. Select the appropriate media in the Media list.
3. Click Apply.

   NOTE If media is selected in the Media list prior to adding a Substance Texture preset that requires an input, the media is automatically added to the input. You can change the input by following the steps above. For presets with more than one input, only the first input is added automatically.

4. Double-click the Input node in the schematic to access the Input menu.

Input Menu Settings



Position X field Displays the X-axis offset applied to the clip. Editable.

Position Y field Displays the Y-axis offset applied to the clip. Editable.

Rotation field Displays the rotation offset applied to the clip. Editable.

Scale X field Displays the X-axis scaling offsets applied to the clip. Editable.

Scale Y field Displays the Y-axis scaling offsets applied to the clip. Editable.

Proportional button Enable to affect the scale fields proportionally.

Repeat Mode box Select how the input media is repeated (after Transform settings are applied).

Filter box Select the type of filtering to apply to the input media.

Destination Resolution box Select the square destination resolution of the input media.

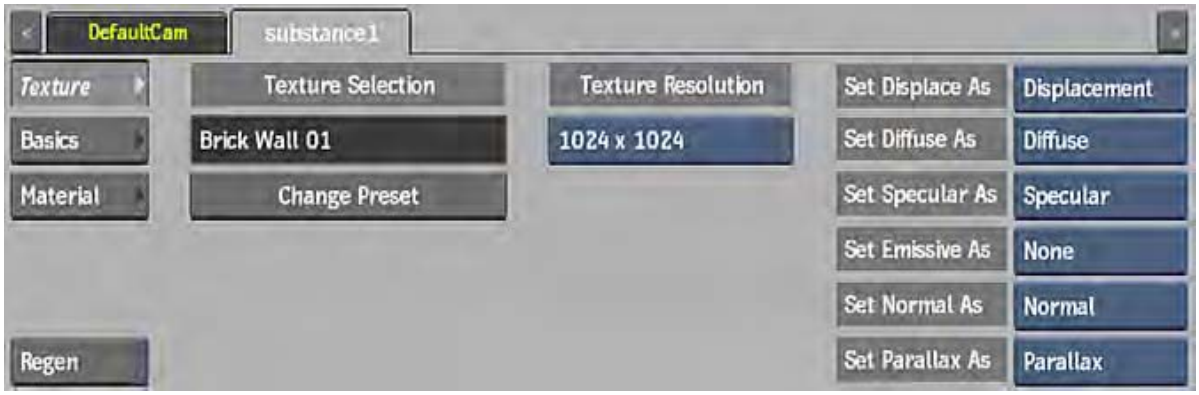
Perspective Correct box Enable to apply perspective transformations to the input. Switch to Object view (F8), and interactively align the perspective of what area you want to include in the input image.

Reset Box button Click to reset the Perspective box in the Object view.

Substance Menu Settings

Double-click the Substance node to access the Substance menu. The Substance menu is divided into three tabs.

The Texture tab contains settings related to the selection and resolution of the texture.



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

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

Texture Resolution box Select the resolution of the pattern.

Lock Width and Height button Not shown. Enable to lock the texture resolution width and height. Disable to be able to select different width and height values. Available when using third-party Substance Textures or the Substance Materialize node.

Texture Resolution Width box Not shown. Select the width resolution of the pattern. Available if Lock Width And Height is disabled.

Texture Resolution Height box Not shown. Select the height resolution of the pattern. Available if Lock Width And Height is disabled.

Set Displace As box Select a map to be used as the Displace pass. This is useful if you want to rewire one of the maps in the substance (such as the Bump map), that might be invisible, to be seen explicitly.

Set Diffuse As box Select a map to be used as the Diffuse pass. This is useful if you want to rewire one of the maps in the substance (such as the Bump map), that might be invisible, to be seen explicitly.

Set Specular As box Select a map to be used as the Specular pass. This is useful if you want to rewire one of the maps in the preset (such as the Bump map), that might be invisible, to be seen explicitly.

Set Emissive As box Select a map to be used as the Emissive pass. This is useful if you want to rewire one of the maps in the preset (such as the Bump map), that might be invisible, to be seen explicitly.

Set Normal As box Select a map to be used as the Normal pass. This is useful if you want to rewire one of the maps in the preset (such as the Bump map), that might be invisible, to be seen explicitly.

Set Parallax As box Select a map to be used as the Parallax pass. This is useful if you want to rewire one of the maps in the preset (such as the Bump map), that might be invisible, to be seen explicitly.

TIP Use the Output Rewire boxes to define how the images making up the Subtance are wired to each texture map. If a texture map is not used by the selected Substance, the Output Rewire box is set to None. You can also rewire a texture map to None to disable it and remove it from the Action Schematic (the map is not deleted, just disabled).

Regen button Enable to dynamically refresh the image as changes are made to the settings. If you notice a slowdown in interactivity, disable Regen.

NOTE The Regen button is available from all Substance menu tabs.

The Basics tab contains settings common to all Substance presets. Settings that are not applicable to a specific preset are greyed out.



Hue field Displays the colour range of the texture. Editable.

Luminosity field Displays the brightness level of the texture. Editable.

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

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

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

Depth field Displays the attenuation of depth of the texture. Editable.

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

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

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

Relief field Displays the frequency of surface detail between attached Parallax and Displacement maps. Editable.

Overall Speed field Displays the rate at which the animation plays. Editable.

Time Offset field Displays the start point of the animation. With a value of 0, the animation starts at frame 1. With a value of 100, the animation begins as if it has been generating for 99 frames. You cannot animate this field. Editable.

The settings in the Material tab vary depending on the preset chosen.



For example, the Material settings for the BrickWall01 preset allow you to age the bricks, and alter the number of the bricks on the X and Y axes.

Some presets include a Flow field in the Material settings. Animate the Flow field to create a motion behaviour specific to the preset.

Substance Materialize

The Substance Materialize node allows you to easily create realistic material textures based on your own image inputs. When you add a Substance Materialize node to the Action schematic, you'll notice the familiar Substance tree of maps and shaders. Simply specify your input image, as well as optional Height and Details maps, and use the menu settings to fine-tune the results.

Here is an example of the various inputs in a Substance Materialize:

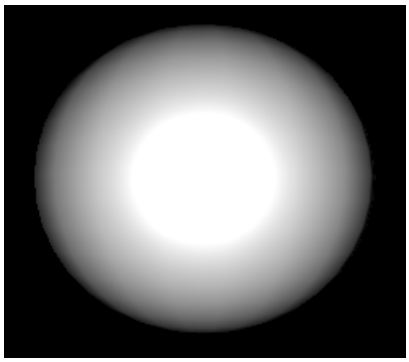
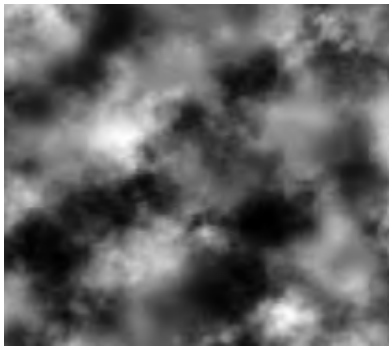
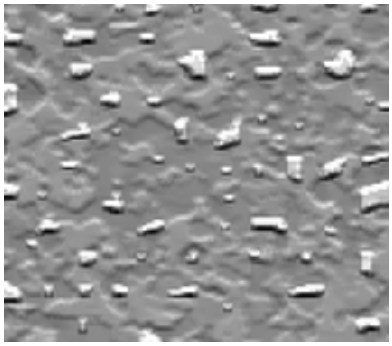
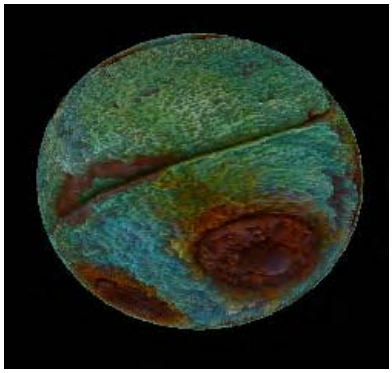
Image	Comments
	**Initial sphere geometry**
	**Input image** Single-frame .tif photo of rusty screws in a wall.
	**Height Map** An optional grayscale image file you can use to enhance elevation. Black represents the lowest elevation on the map, while white represents the highest.

Image	Comments
	**Details Map** An optional image file allowing you to specify fine details in your result.
	**Result** After applying inputs and various Substance settings.

Adding a Substance Materialize Node

To add a Substance Materialize node:

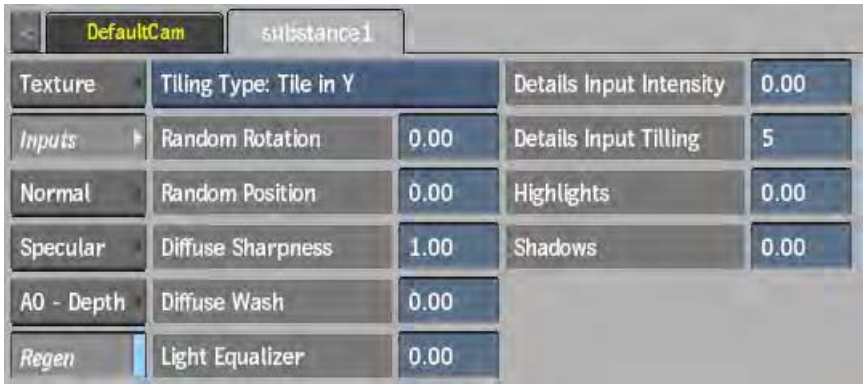
1. In the schematic, select the surface or 3D geometry to which you want to apply the Substance Materialize.
2. In the Media list, select the image that you want to use as the input. You can add this later, or even change the media.
3. Do one of the following:
   - Drag the Substance Materialize node from the node bin and place it in the schematic.
   - Double-click the Substance Materialize node.

   The Substance tree, with applicable maps, shaders, and inputs appears in the Action schematic.
4. To specify different media as the input source, select the input node in the schematic, select the media in the Media menu, then click Apply.
5. Optional: If you have Height and Details maps, you can also use them to enhance your Substance material. Simply apply them from the Media list in the same manner as with the Input image.
6. Double-click any of the nodes in the schematic to access its menu.

Materialize Settings

Once you have added a Substance Materialize node (with your Input image and optional Height and Details maps), you have settings in various menus that you can use to tweak your result.

The Substance menu is different than if you added a Substance Texture preset. You have tabs in the Substance menu with settings to control the texture, inputs, normal, specular, ambient occlusion, and depth. See the tooltips for help with each setting.



For each input, you also can use the Input menu (page 582) (as with Substance Textures with inputs), and the Object view (F8). The Object view lets you visualize changes that you make to the Input menu settings. In the Object view, you can also interactively align the perspective of the area you want to include in the input image.

Material Node

The Material node creates a central hub to control the specular, ambient, diffuse, transparency, and shininess of a geometry's children maps and substances.

Flame Premium automatically creates a Material node when you insert a Substance Texture or when you import 3ds Max or FBX models with textures.

Material Node Menu Settings



Lighting Settings

Red Specular field Displays the red specular highlight value. Editable.

Green Specular field Displays the green specular highlight value. Editable.

Blue Specular field Displays the blue specular highlight value. Editable

Specular colour pot Displays the colour of light reflected by the 3D model's surface. To enable the specular highlight, the Shine value must be larger than 0. Editable.

NOTE Specular lighting sets the colour of light reflected by the 3D model's surface. To enable the specular highlight, the Shine value must be larger than zero.

Red Ambient field Displays the red ambient colour value. Editable.

Green Ambient field Displays the green ambient colour value. Editable.

Blue Ambient field Displays the blue ambient colour value. Editable.

Ambient colour pot Displays the colour of the area of the 3D model that is not illuminated by a direct light source. Editable.

NOTE Ambient lighting sets colour to the area of the 3D model that is not illuminated by a direct light source. The edge of the ambient area mixes with the specular highlight colour and the diffuse colour.

Red Diffuse field Displays the red diffuse colour value. Editable.

Green Diffuse field Displays the green diffuse colour value. Editable.

Blue Diffuse field Displays the blue diffuse colour value. Editable.

Diffuse colour pot Displays the diffuse colour. Editable.

NOTE Diffuse lighting modifies the colour and illumination of the entire 3D model. Diffuse light mixes with the colour of the light sources used to illuminate the 3D model. The diffuse colour may also mix with the ambient colour and the colour of the specular highlight.

Surface Settings

Transparency field Displays the transparency level of the 3D model. Editable.

Shine field Displays the intensity value of the specular highlight. When this value is zero, the specular highlight is disabled. Shine affects both size and intensity. Editable.

See Creating a Specular Highlight on a Model (page 543).

Using the Shader Node

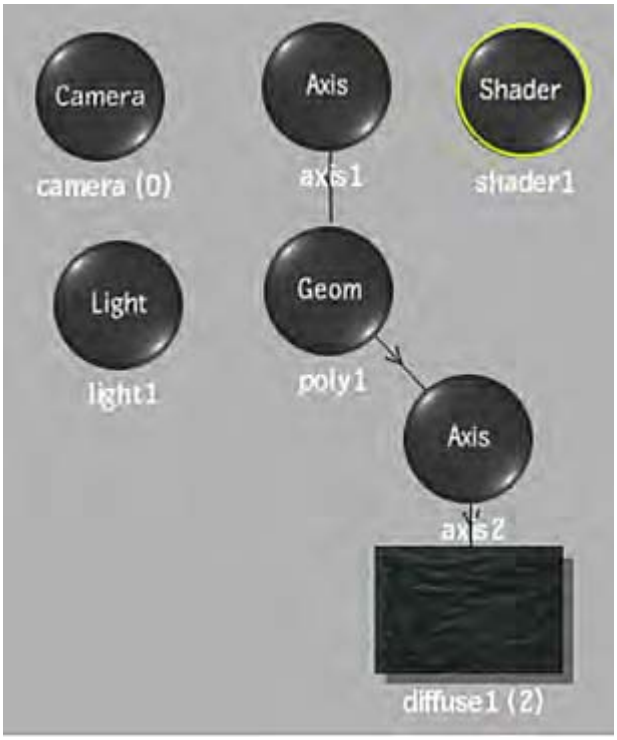
You can add a shader to your scene to apply to all objects in the scene. You also have the option of selectively applying a specific shading algorithm, or turning shading off completely. You can use multiple shaders in an Action scene, but only one shader can be applied to each object. Even if no shaders are present in the schematic, Action uses an implicit default shader that applies to the whole scene.

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

To add a shader to the scene:

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

A shader is added to the scene. If you select a surface or 3D geometry in the scene before adding the Shader node, a shader inclusion link is automatically applied. You can also apply inclusion and exclusion links manually. See Applying a Selective Shader (page 589).



Unlike many objects, a shader is not a confined object in the perspective space, and therefore is added without an axis.

2 To display the Shader menu, double-click the selected shader in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).

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



See Shader Types and Settings (page 592) for more information on the shader algorithms, and their settings.

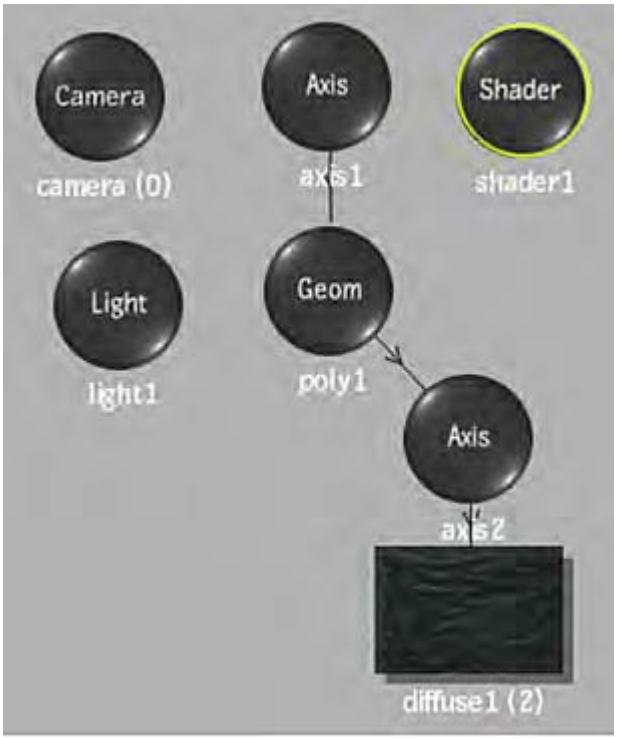
Applying a Selective Shader

If you select a surface or 3D geometry in the scene before adding a Shader node, a shader inclusion link is automatically applied; otherwise, the shader is applied to all objects. You can also apply inclusion and exclusion links manually. You may want a shader to only affect an individual or specific group of objects, or prevent a shader from affecting an individual or specific group of objects.

To apply selective shading:

1 Add a shader to the scene.

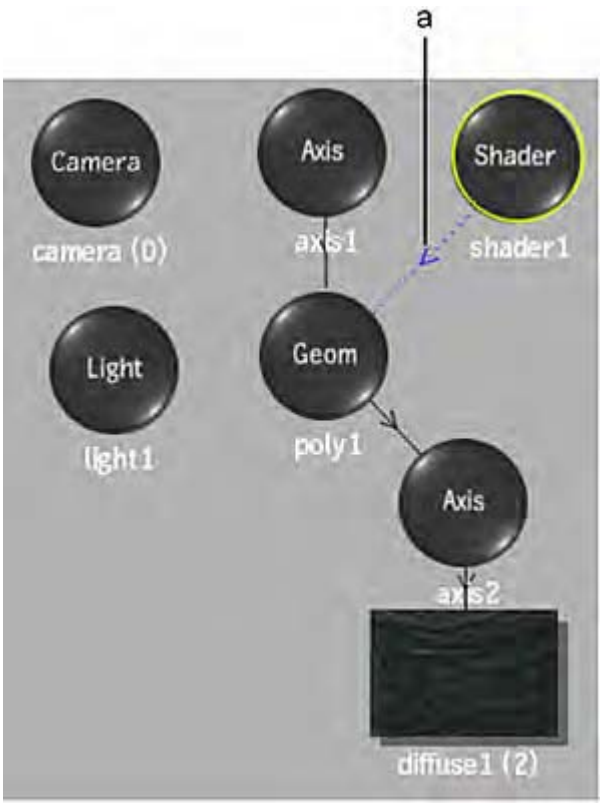
All objects are affected.



2 Select Lighting from the Edit Mode box.

3 To affect only a selected object, click the Shader node, and drag it to an object you want affected.

The selected object is connected to the shader by a blue dotted line with an arrow, and only the selected objects are affected.



(a) Shader inclusion link

4 To exclude an object, hold the `Alt` key while clicking and dragging from the shader to the object you do not want affected.

Excluded surfaces are connected to the shader by a red dotted line with an "X", and they are not affected by the shader.



(a) Shader exclusion link

NOTE To remove the inclusion or exclusion link, click and drag over the line that connects the Shader node to the object (while in Lighting mode).

Shader Types and Settings

Depending on the look you are trying to apply to an object or the scene, you can select from among different shader algorithms. The differences between the shader types are sometimes subtle, as they build upon the same algorithms, such as Fresnel or the Oren-Nayar diffuse model.

Some of the settings in the Shader menu change depending on the shader type chosen. See the following sections for the specific settings for each shader type.

Shader Type box Select a shading algorithm, or turn shading off.

Anisotropic Shader

Use the anisotropic shader to control the specular effect of the highlights.



X Roughness field Displays the shape of specularity of the shader along the X axis. Editable.

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

Fresnel Offset field Displays the total amount of specular light.

Fresnel Factor field Displays the amount of specular light at grazing angles.

TIP To cancel out the Fresnel effect altogether, set the Fresnel Offset to 0 and the Fresnel Factor to 100.

Cook-Torrance Shader

Use the Cook-Torrance shader for high specularity materials, such as metals or shiny plastics. This shader includes Fresnel controls for specularity.



Roughness field Displays the shape of specularity of the shader.

Fresnel Offset field Displays the total amount of specular light.

Fresnel Factor field Displays the amount of specular light at grazing angles.

TIP To cancel out the Fresnel effect altogether, set the Fresnel Offset to 0 and the Fresnel Factor to 100.

Fresnel Shader

The Fresnel shader contains only Fresnel controls for the specularity.



Roughness field Displays the shape of specularity of the shader.

Fresnel Offset field Displays the total amount of specular light.

Fresnel Factor field Displays the amount of specular light at grazing angles.

TIP To cancel out the Fresnel effect altogether, set the Fresnel Offset to 0 and the Fresnel Factor to 100.

Car Paint Shader

Use the Car Paint shader to blend between two colour tones, based on the viewing angle and the normal of the object. This shader includes Cook-Torrance shaders and Fresnel controls for the specularity.



Roughness field Displays the shape of specularity of the shader.

Fresnel Offset field Displays the total amount of specular light.

Fresnel Factor field Displays the amount of specular light at grazing angles.

TIP To cancel out the Fresnel effect altogether, set the Fresnel Offset to 0 and the Fresnel Factor to 100.

Paint Mix field Displays the viewing angle of the normal that occurs between parallel (mid colour) and perpendicular (edge colour). Editable.

Red Mid Colour field Displays the red mid colour value. Editable.

Green Mid Colour field Displays the green mid colour value. Editable.

Blue Mid Colour field Displays the blue mid colour value. Editable.

Mid colour pot Diplays the mid colour. Editable.

Red Edge Colour field Displays the red edge colour value. Editable.

Green Edge Colour field Displays the green edge colour value. Editable.

Blue Edge Colour field Displays the blue edge colour value. Editable.

Edge colour pot Displays the edge colour. Editable.

Displacement Mapping

Use displacement mapping to create a 3D model from a 2D surface. The values of a selected colour channel in the displacement source clip are used to create a displacement map. Displacement mapping uses the media's matte clip, so you can turn the matte on or off to get the desired effect.

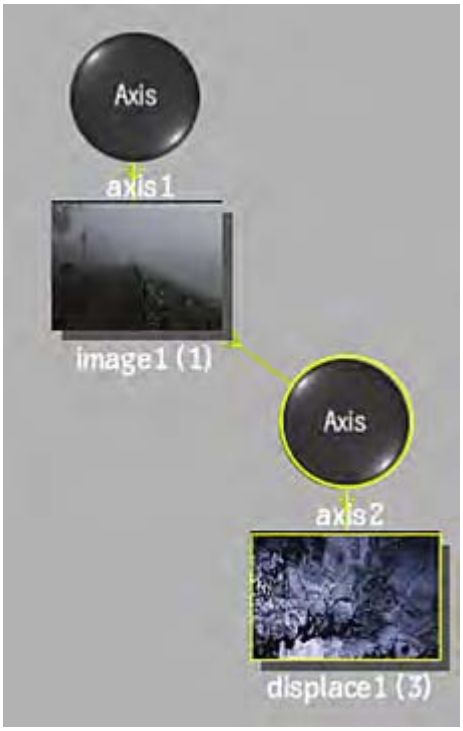
You have the choice of applying hardware or software displacement mapping. Hardware displacement mapping is GPU-accelerated and allows you to create a normal map if none exists. Software displacement mapping displaces the pixels of the surface along the positive or negative X, Y, and/or Z axes. Hardware Displacement is faster than software displacement, especially when using low resolution values in an image surface (high polygon count).

NOTE If the Displace node is attached to an imported 3D Geometry, you may need to select a UV Mapping mode other than Default in the Geometry menu for the displace pattern to have an effect on the geometry. See UV Map Settings (page 540).

To add a displacement map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the displacement.
2. Click Media.
3. In the Media menu, select the media you want to use for the displacement.
4. Do one of the following:
   - Drag the Displace Map node from the node bin and place it in the schematic.
   - Drag the Displace Map node from the node bin and place it where you want it in Result view.
   - Double-click the Displace Map node. You do not need to be in Schematic view to add a node in this manner.

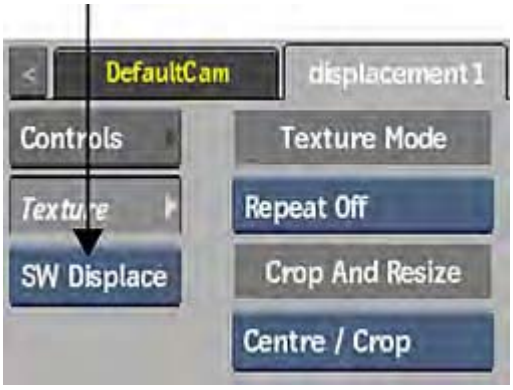
   The displace object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the displace node indicates the media used for the displacement.



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

5 Double-click the Displace node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).

The Displacement menu appears. You can choose between Hardware Displacement or Software Displacement using the Displacement Type box.



Software Displacement Menu Settings

Displacement Type box Select whether to use hardware or software displacement mapping.

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

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

Controls Tab



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

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

Softness rounds the edges of the displacement. The larger the softness, the smoother the displacement. Softness also affects rendering; the larger the softness, the longer it takes to render.

Displace Direction box Select the direction in which a displace occurs when a displacement map is attached to a surface. Geometries parented to a displacement map always use Normal Displace as the direction.

Select:	To Displace:
Flat Displace	In the X, Y, and Z directions.
Normal Displace	Bilinear and bicubic surfaces according to their normals.
Camera Displace	In the direction of the camera selected in the Displace Camera box.

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

Displace Camera box Active only when Camera Displace is selected in the Displace Direction box. Select which camera to take into account when using camera displacement.

Displace Camera field Displays the active displace camera number. Non-editable.

Displacement X field Displays the amount of displacement in pixel units along the X axis. Editable.

Displacement Y field Displays the amount of displacement in pixel units along the Y axis. Editable.

Displacement Z field Displays the amount of displacement in pixel units along the Z axis. Editable.

Texture Tab



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

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

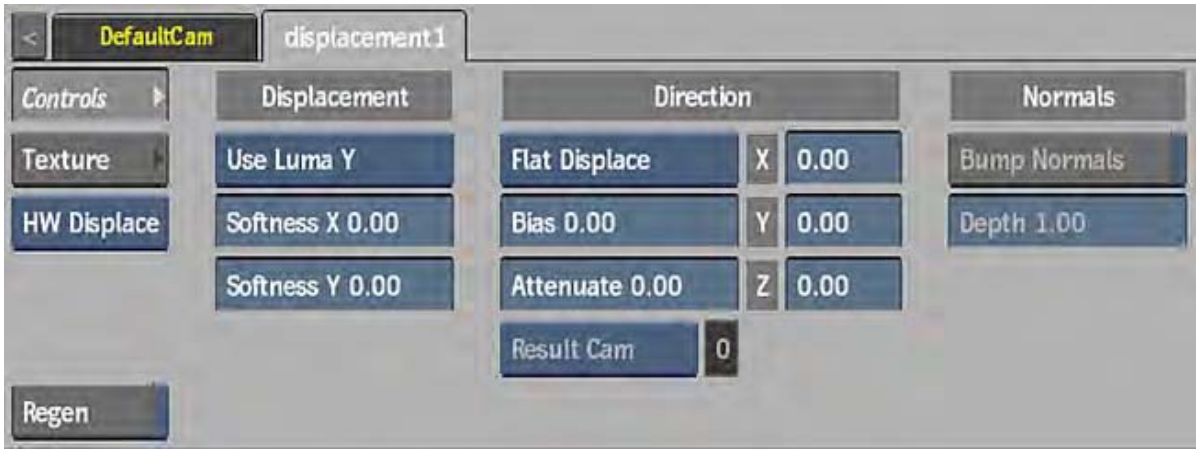
Hardware Displacement Menu Settings

Displacement Type box Select whether to use hardware or software displacement mapping.

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

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

Controls Tab



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

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

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

Displace Direction box Select the direction in which a displace occurs when a displacement map is attached to a surface. Geometries parented to a displacement map always use Normal Displace as the direction.

Select:	To Displace:
Flat Displace	In the X, Y, and Z directions.

Select:	To Displace:
Normal Displace	Bilinear and bicubic surfaces according to their normals.
Camera Displace	In the direction of the camera selected in the Displace Camera box.

Bias field Displays the distance between the real surface of the object and the perceptual ground (zero level) of the texture. Editable.

Attenuate field Displays the level of amplitude of the effect caused by the displacement map texture. Editable.

Displace Camera box Active only when Camera Displace is selected in the Displace Direction box. Select which camera to take into account when using camera displacement.

Displace Camera field Displays the active displace camera number. Non-editable.

Displacement X field Displays the amount of displacement in pixel units along the X axis. Editable.

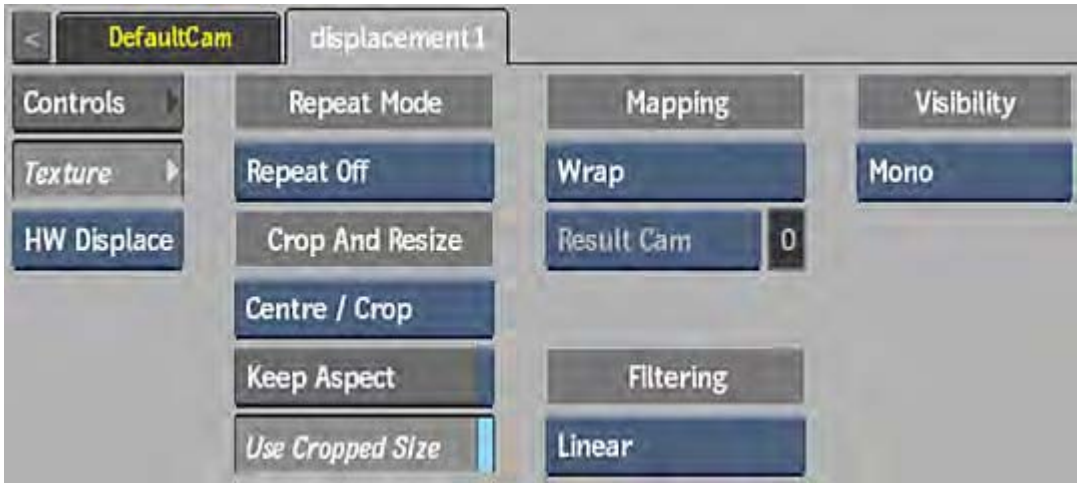
Displacement Y field Displays the amount of displacement in pixel units along the Y axis. Editable.

Displacement Z field Displays the amount of displacement in pixel units along the Z axis. Editable.

Bump Normals button Enable to allow the map to create a normal map for enhanced lighting effects. Used only if no normal map exists for the object using the displacement texture. The availability of this setting is dependant on your graphics card.

Depth field Displays the amount of Z scale applied to the generated normals. Higher values attenuate the effect toward the normals of the parent surface. Editable.

Texture Tab



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

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

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

Use Cropped Size button Enable to replace the map with the cropped size of the media. Disable to use the cropped media as is.

Mapping box Select the type of texture mapping.

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

Camera field Displays the active perspective camera number.

Filter box Select the type of filtering to apply to the map.

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

Parallax Mapping

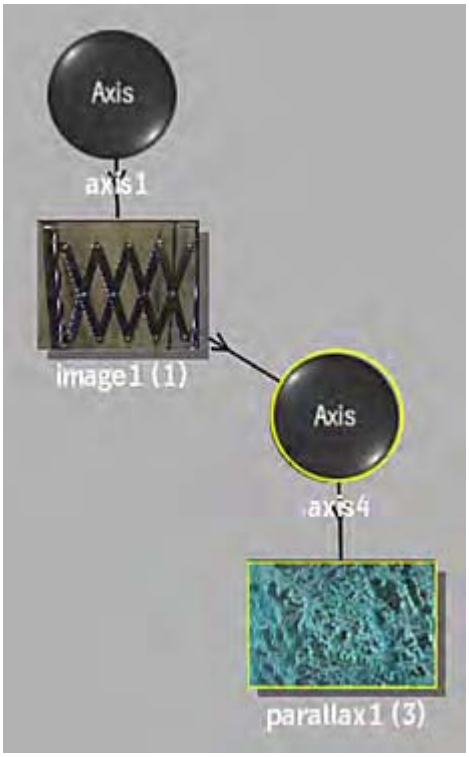
Parallax mapping can generate a visual result similar to that of a displacement map without actually displacing polygons. Parallax mapping uses the height map (similar to a displacement map) to determine how the object texture should be warped in order to simulate shading and occlusion effects, without actual polygonal displacement.

Because a parallax map does not actually change the polygons of the object, there are inherent limits to the illusion — looking at the object from the side will reveal the cheat. It is best used on surfaces and geometries where you do not see the angles or edges.

To add a parallax map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the parallax.
2. Click Media.
3. In the Media menu, select the media you want to use for the parallax.
4. Do one of the following:
   - Drag the Parallax Map node from the node bin and place it in the schematic.
   - Drag the Parallax Map node from the node bin and place it where you want it in Result view.
   - Double-click the Parallax Map node. You do not need to be in Schematic view to add a node in this manner.

   The parallax object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the Parallax node indicates the media used for the parallax.



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

5 Double-click the Parallax node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).

The Parallax menu appears.

Parallax Menu Settings

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

Controls Tab



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

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

Bias field Displays the distance between the real surface of the object and the perceptual ground (zero level) of the texture. A value of half of the Scale field value should give the best results.

Attenuate field Displays the level of amplitude of the effect caused by the parallax map texture. Editable.

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

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

Relief Mapping button Enable to augment the 3D surface detail by creating occlusions where changes occur in the parallax map. Rendering is slower when enabled, but the results may be better.

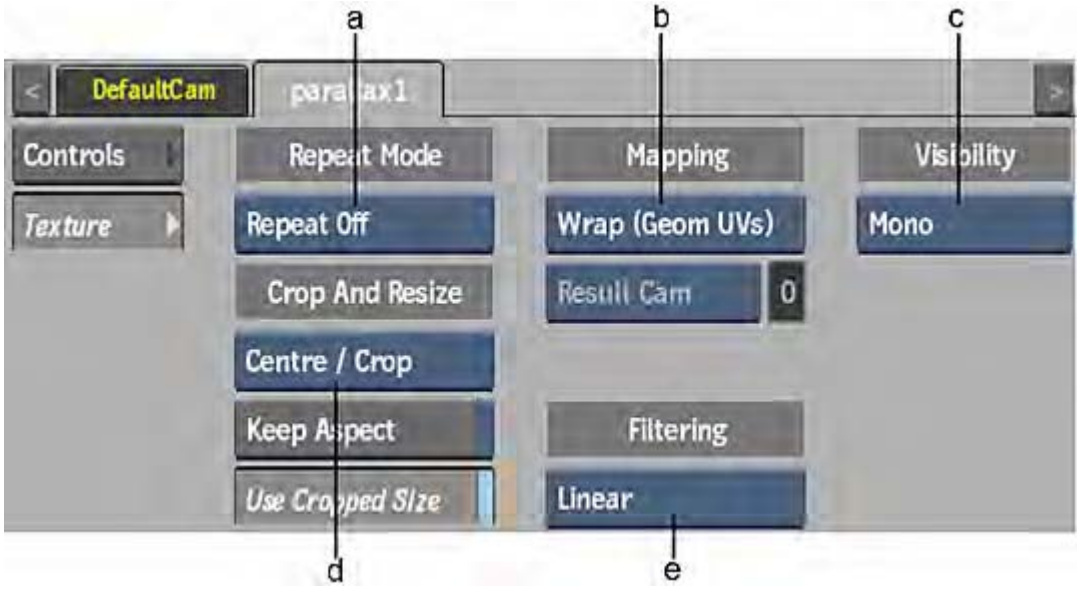
Linear Search field Displays the number of steps to search for the best depth in the relief map. Editable.

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

Bump Normals button Enable to allow the parallax map to create a normal map for enhanced lighting effects. Used only if no normal map exists for object using the parallax texture. Enabling also affects the Normal output in Action.

Depth field Displays the amount of Z scale applied to the generated normals. Higher values increase the effect of normals. Editable.

Texture Tab



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

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

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

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

Use Cropped Size button Enable to replace the parallax map with the cropped size of the parallax media. Disable to use the cropped parallax media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the texture map according to the object's texture coordinates. To use this option, you must import a model that has its own texture coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu. See UV Map Settings (page 540).

Plane Planar mapping applies the map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the texture values. Planar mapping positions the lower-left corner of the map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

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

Perspective Camera field Displays the active perspective camera number. Non-editable.

Filter box Select the type of filtering to apply to the parallax map.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

TIP You can set the default filtering type in the Action Setup menu Preferences tab.

Camera Type box Select the camera type visibility for the parallax map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

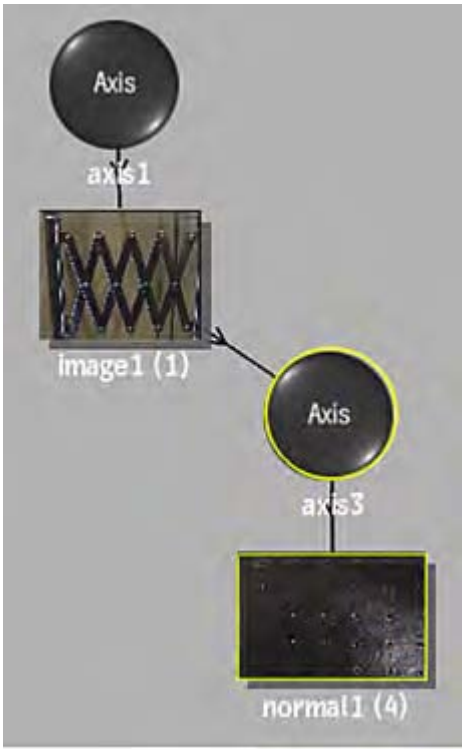
Normal Mapping

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

To add a normal map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the normal.
2. Click Media.
3. In the Media menu, select the media you want to use for the normal.
4. Do one of the following:
   - Drag the Normal Map node from the node bin and place it in the schematic.
   - Drag the Normal Map node from the node bin and place it where you want it in Result view.
   - Double-click the Normal Map node. You do not need to be in Schematic view to add a node in this manner.

   The normal object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the normal node indicates the media used for the normal.



   To specify different media as the normal source, select the media in the Media menu, then click Apply.
5. Double-click the Normal node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).

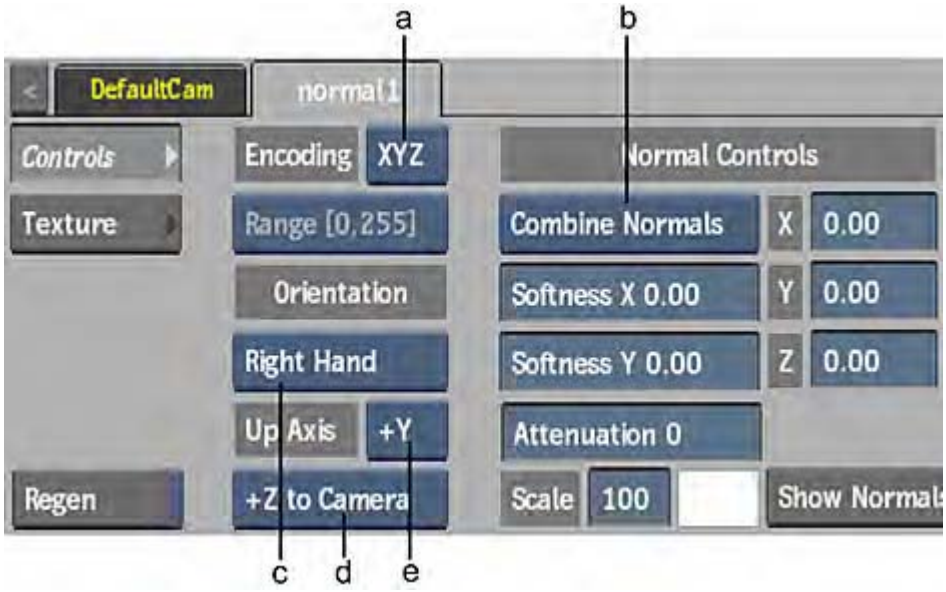
   The Normal menu appears.

Normal Menu Settings

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

Controls Tab

Use the Encoding and Orientation settings to help Action with information on how the normal map was encoded and rendered. Then use the Normal Controls to tweak the map in Action.



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

Encoding box Select the order that the normal map is encoded based on the interpretation of the RGB channels (XYZ or XZY).

Range box When working with floating point normal map media, select the range of the normal map media: [0, 1] or [-1, 1]. When working with 8-, 10-, or 12-bit images, the Range box displays the appropriate range for Action, but the box is greyed out.

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

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

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

Normals box Select whether to combine or replace the normal map with the surface normals.

Select:	To:
Combine Normals	Combine the normals map texture with the surface's normals.
Replace Normals	Apply only the normal map texture to the surface (ignoring the surface normal properties).

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

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

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

X Normal axis Displays the amount of offset in pixel units along the X axis.

Y Normal axis Displays the amount of offset in pixel units along the Y axis.

Z Normal axis Displays the amount of offset in pixel units along the Z axis.

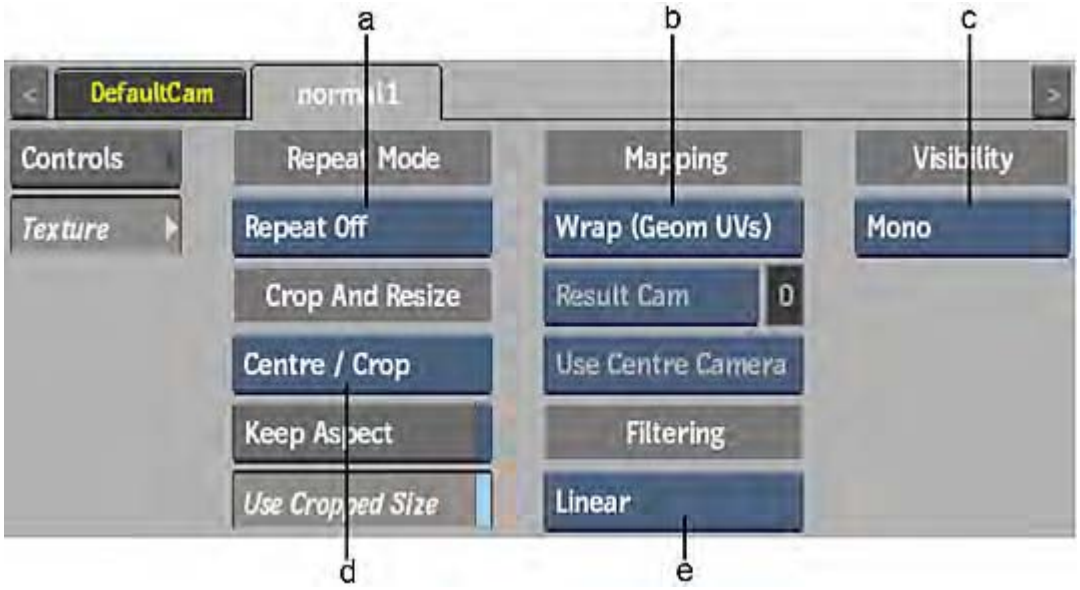
Show Normals button Enable to display normal vectors over the surface.

Normals colour pot Displays the normal vectors colour. Editable.

Scale field Displays the scale of the normal vectors. Editable.

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

Texture Tab



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

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

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

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

Use Cropped Size button Enable to replace the normal map with the cropped size of the normal media. Disable to use the cropped normal media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the texture map according to the object's texture coordinates. To use this option, you must import a model that has its own texture coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu.

Plane Planar mapping applies the map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the texture values. Planar mapping positions the lower-left corner of the map on the 3D model's axis. When you apply planar mapping, any

surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

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

Camera field Displays the active perspective or projection camera number. Non-editable.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the normal map.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

TIP You can set the default filtering type in the Action Setup menu Preferences tab.

Camera Type box Select the camera type visibility for the normal map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

NOTE If the Normal node is attached to a 3D Geometry node, you may need to select a UV Mapping mode other than Default in the Geometry menu for the normal pattern to have an effect on the geometry. See UV Map Settings (page 540).

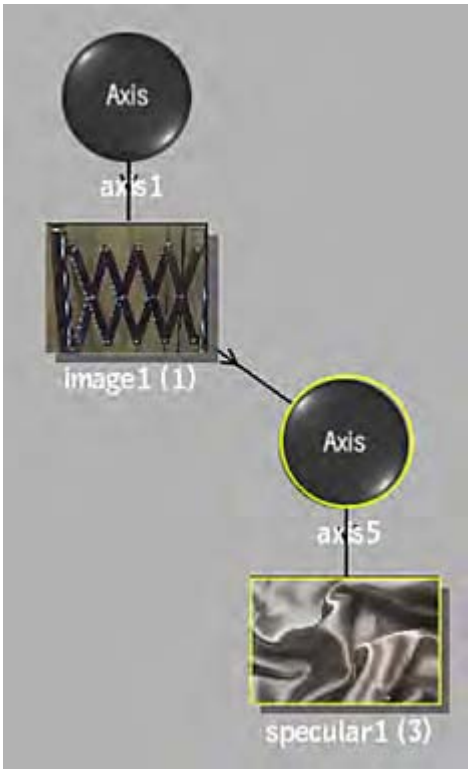
Specular Mapping

A specular map defines the shininess and highlight colour of a surface or geometry.

To add a specular map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the specular.
2. Click Media.
3. In the Media menu, select the media you want to use for the specular.
4. Do one of the following:
   - Drag the Specular Map node from the node bin and place it in the schematic.
   - Drag the Specular Map node from the node bin and place it where you want it in Result view.
   - Double-click the Specular Map node. You do not need to be in Schematic view to add a node in this manner.

   The specular object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the Specular node indicates the media used for the specular.



   To specify different media as the specular source, select the media in the Media menu, then click Apply.
5. Double-click the Specular node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).

   The Specular menu appears.

Specular Menu Settings

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

Controls Tab



Effect field Displays the amount of specular colour. Editable.

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

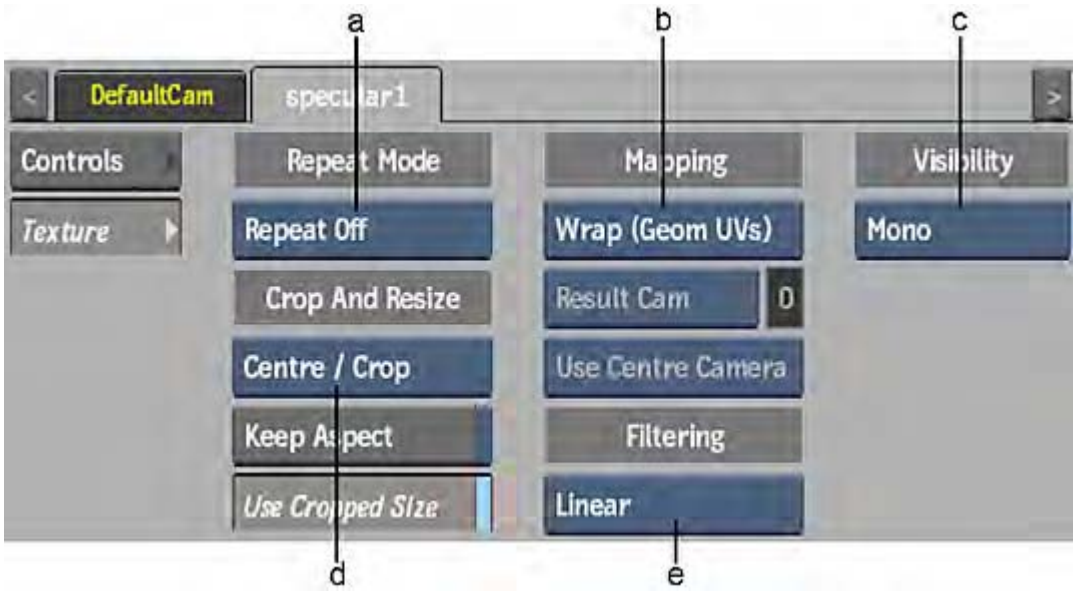
Softness Y field Displays the amount of Y-axis blur applied to the specular map. Editable.

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

NOTE To display proper results, the specular map also uses the value of the surface or geometry shine field. A shine value of 0.0 in the Surface or Geometry menu is interpreted as 1.0.

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

Texture Tab



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

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

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

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

Use Cropped Size button Enable to replace the specular map with the cropped size of the specular media. Disable to use the cropped specular media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the texture map according to the object's texture coordinates. To use this option, you must import a model that has its own texture coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu. See UV Map Settings (page 540).

Plane Planar mapping applies the map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the texture values. Planar mapping positions the lower-left corner of the map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

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

Camera field Displays the active perspective or projection camera number. Non-editable.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the specular map.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

TIP You can set the default filtering type in the Action Setup menu Preferences tab.

Camera Type box Select the camera type visibility for the specular map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

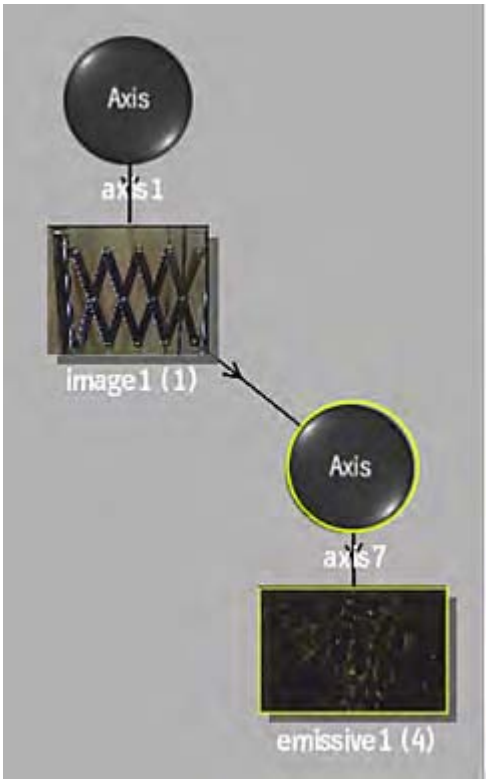
Emissive Mapping

An emissive map uses colours to simulate a glowing effect within the texture. The effect of an emissive map does not go beyond the limits of an object, and therefore cannot be used to generate glows around the rest of the scene or the surrounding objects.

To add an emissive map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the emissive.
2. Click Media.
3. In the Media menu, select the media you want to use for the emissive.
4. Do one of the following:
   - Drag the Emissive Map node from the node bin and place it in the schematic.
   - Drag the Emissive Map node from the node bin and place it where you want it in Result view.
   - Double-click the Emissive Map node. You do not need to be in Schematic view to add a node in this manner.

The emissive object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the Emissive node indicates the media used for the emission.



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

5 Double-click the Emissive node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).

The Emissive menu appears.

Emissive Menu Settings

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

Controls Tab



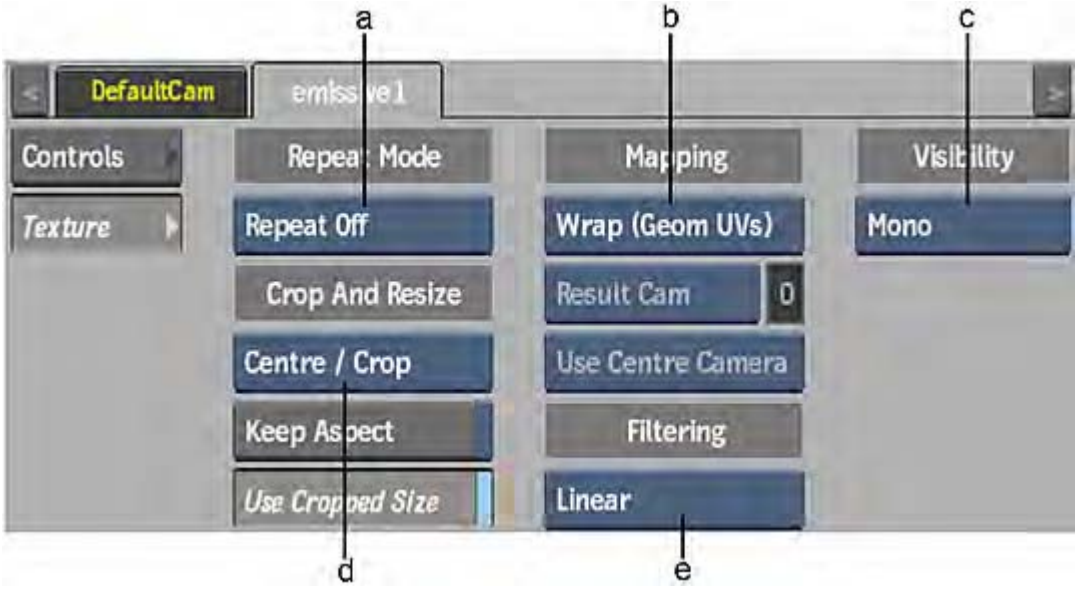
Effect field Displays the amount of emissive colour. Editable.

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

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

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

Texture Tab



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

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

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

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

Use Cropped Size button Enable to replace the emissive map with the cropped size of the emissive media. Disable to use the cropped emissive media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the texture map according to the object's texture coordinates. To use this option, you must import a model that has its own texture coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu. See UV Map Settings (page 540).

Plane Planar mapping applies the map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the texture values. Planar mapping positions the lower-left corner of the map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.

Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

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

Camera field Displays the active perspective or projection camera number. Non-editable.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the emissive map.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

TIP You can set the default filtering type in the Action Setup menu Preferences tab.

Camera Type box Select the camera type visibility for the emissive map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

Diffuse Mapping

Use a diffuse map to define the diffuse reflection and main colour of a surface, 3D model, or 3D text. Since the diffuse map and its axis are parented by the surface or geometry node, animating the parent's axis also animates the diffuse map, which has the effect of keeping the map properly in place on the model. A diffuse map uses the specular highlight, diffuse colour, and shine set by its parent.

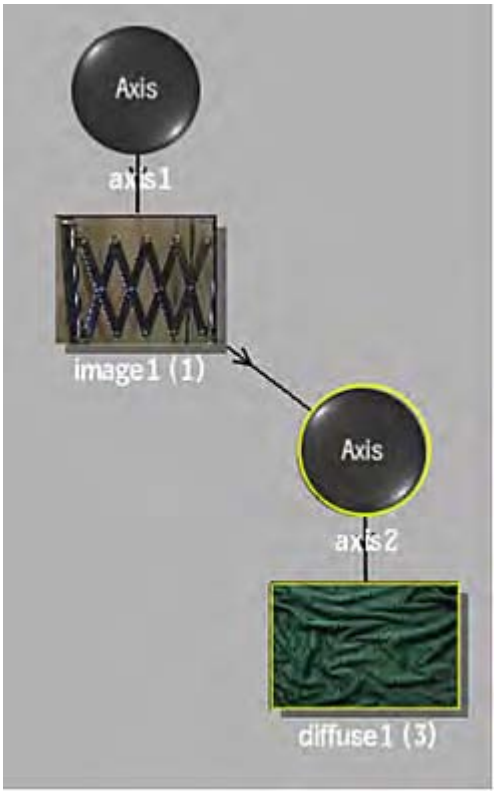
When adding a diffuse map to a shaded surface, the diffuse is used when generating the shadow. The diffuse is only used to apply the colour to the shadow, so effects such as surface displacement still reference the media associated with the surface. If a node has an applied diffuse map, it is the matte setting of the diffuse map that controls whether the object will be included in the various output mattes (scene matte, blend matte, for example).

To add a diffuse map:

1. In the schematic, select the surface, 3D model, or 3D text to which you want to apply the diffuse map.
2. Click Media.
3. In the Media menu, select the media you want to use for the diffuse map texture (you can change this texture later).

   A diffuse map uses the front and matte of the media. If you do not want to apply transparency to your diffuse map, turn its matte off.
4. Do one of the following:
   - Drag the Diffuse Map node from the node bin and place it in the schematic.
   - Drag the Diffuse Map node from the node bin and place it where you want it in Result view.
   - Double-click the Diffuse Map node. You do not need to be in Schematic view to add a node in this manner.

   The diffuse object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the Diffuse node indicates the media used for the diffuse.



To specify different media as the diffuse source, select the media in the Media menu, then click Apply. Alternately, you can load an external texture directly from the Diffuse menu.

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

6 Double-click the Diffuse node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).

The Diffuse menu appears.

Diffuse Menu Settings

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

Controls Tab



Effect field Displays the amount of diffuse colour. Editable.

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

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

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

Texture Tab



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

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

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

Use Cropped Size button Enable to replace the diffuse map with the cropped size of the diffuse media. Disable to use the cropped diffuse media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the diffuse map according to the object's diffuse coordinates. To use this option, you must import a model that has its own diffuse coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu. See UV Map Settings (page 540).



Plane Planar mapping applies the diffuse map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the diffuse values. Planar mapping positions the lower-left corner of the diffuse map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.



Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the diffuse map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

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

Camera field Displays the active perspective or projection camera number. Non-editable.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the diffuse map.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.

Select:	To apply:
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

TIP You can set the default filtering type in the Action Setup menu Preferences tab.

Camera Type box Select the camera type visibility for the map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

Texture Name field Displays the name of an externally loaded diffuse map texture. Non-editable.

Load Texture button Click to open the Import Media browser to select an external single-frame image file to be used as the diffuse map texture. This file is not added to the Media list.

NOTE When a diffuse map is connected to a flat surface, you can enable Auto Expand in the Surface menu to automatically resize the surface when the diffuse map's Axis settings are changed.

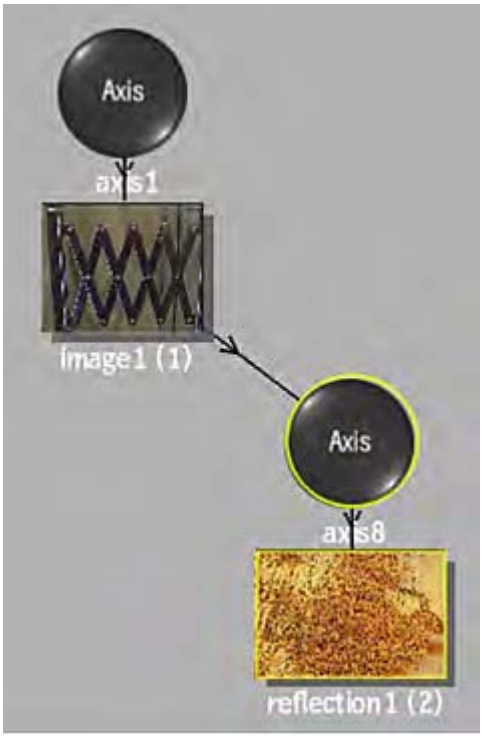
Reflection Mapping

Reflection mapping simulates a mirrored surface by using the specular reflection values in the map. You have the option of combining the result of a reflection map to a diffuse map.

To add a reflection map:

1. In the schematic, select the surface, 3D model, or 3D text to which you want to apply the diffuse map.
2. Click Media.
3. In the Media menu, select the media you want to use for the diffuse map.
4. Do one of the following:
   - Drag the Reflection Map node from the node bin and place it in the schematic.
   - Drag the Reflection Map node from the node bin and place it where you want it in Result view.
   - Double-click the Reflection Map node. You do not need to be in Schematic view to add a node in this manner.

   The reflection object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the Reflection node indicates the media used for the diffuse.



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

5 Double-click the Reflection node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).

The Reflection menu appears.

Reflection Menu Settings

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

Controls Tab



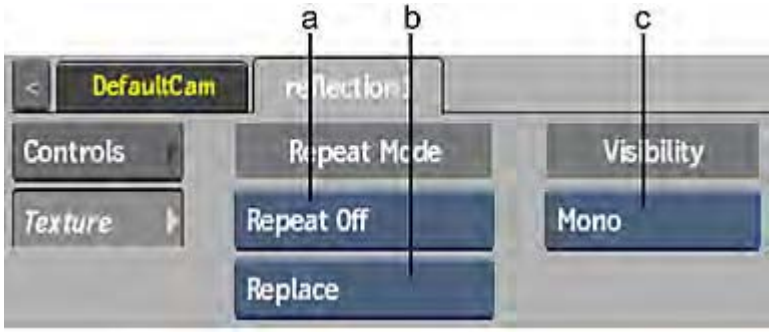
Effect field Displays the amount of diffuse colour. Editable.

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

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

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

Texture Tab



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

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

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

Camera Type box Select the camera type visibility for the map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

Position Mapping

A position map (sometimes called a world position pass), is an image where each pixel's R, G, B colour values represent the x, y, z coordinates of the corresponding vertex, in 3D world space. In Action, a position map works as a type of 3D displacement, that is, it displaces every pixel in that image to the actual location in space of the vertex it represents, allowing you to:

- Establish depth, position, and scaling in images, for compositing and creative purposes.
- Relight your scene.
- Create stereo objects.
- Creatively transfer the position of one object onto another object.

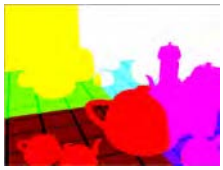
A typical workflow would be to import (from a 3D application) the beauty pass, the position pass (as a 16-bit floating point OpenEXR clip), the normal pass, and the FBX camera that rendered the scene. Then, apply the position and the normal map to the beauty pass image surface, and set the scale of the position map so that it aligns properly with the 3D camera frustum.

NOTE Make sure that your position pass rendering settings in the 3D application are sufficiently high so that the position map displays correctly in Flame Premium. For example, when rendering with Mental Ray in Maya, set the quality level to Production or higher.

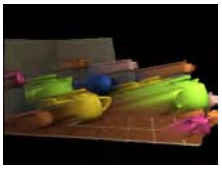
When looked from the original camera POV, your scene looks just like the rendered image. But since pixels are accurately located in space relative to the original 3D scene with the attached position map, you can correctly position 2D layers in the same perspective as objects of the 3D scene.



Beauty Pass



Position Map



Action scene with camera orbited

You can use position maps generated by 3D applications or Action itself (Position is available as an output type in the Output (page 435) menu).

To add a position map:

1. In the schematic, select the surface to which you want to apply the position map.
2. Click Media.
3. In the Media menu, select the media you want to use for the position map.
4. Do one of the following:
   - Drag the Position Map node from the node bin and place it in the schematic.
   - Drag the Position Map node from the node bin and place it where you want it in Result view.
   - Double-click the Position Map node. You do not need to be in Schematic view to add a node in this manner.

   The position object is added to the schematic with its own parent axis. The new axis is the child of the selected surface. In Schematic view, the number in brackets next to the name of the Position node indicates the media used for the position map.

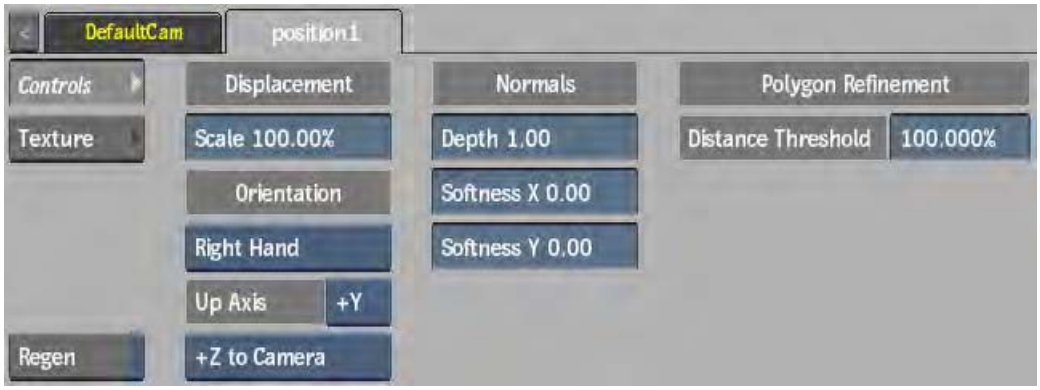
   To specify different media as the position map source, select the media in the Media menu, then click Apply.
5. Double-click the Position node in the schematic, or follow the tab population rules for the Object menu.

   The Position menu appears.

Position Menu Settings

Controls Tab

Use the Orientation settings to help Action with information on how the position map was rendered. The Normals settings affect the generated normals present in the position map.



Scale field Displays the scale of the position map in relation to Action space. Editable.

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

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

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

Depth field Displays the amount of Z scale applied to the generated normals. Higher values attenuate the effect toward the normals of the parent surface. Editable.

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

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

Distance Threshold field Displays the threshold at which the longest triangles producing disconnected meshes in the geometry are hidden. Editable.

Texture Tab



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

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

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

Use Cropped Size button Enable to replace the map with the cropped size of the media. Disable to use the cropped media as is.

Mapping box Select the type of texture mapping.

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

Camera field Displays the active perspective camera number.

Filter box Select the type of filtering to apply to the map.

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

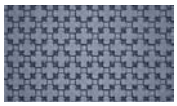
UV Mapping

A UV map is a pass which records the way the pixels of an image should be displaced within the image plane, so that the warped texture looks like the textured geometry from the point of view of the CG camera. The UV map acts as a 2D distortion field, or a pixel look up table, recording where every pixel of an input image texture should be, had it been applied on a geometry filmed by the CG camera. It is a very effective way of faking 3D mappings, without having a single polygon. It is most effective when combined with a Normal map, which can be used to simulate the shading of the actual geometry even though everything is perfectly flat.

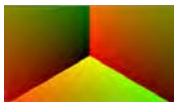
Applying a UV map to surface or geometry, warps its texture according to the defined UV. The U information is coded in the Red channel, while the V information is coded in the Green channel of the UV map image. For each pixel of the UV map, the red or green value corresponds to a given pixel on the unwrapped texture, and it is this pixel that gets displaced at the position of the same pixel in the attached surface or geometry.

Here are a few things to keep in mind when working with UV maps:

- To avoid pixelation in your imported UV passes, make sure that the UV pass is rendered as a 16-bit floating point image from your 3D application.
- It is a good idea to render UV passes without anti-aliasing to avoid warping on the edges of objects that the UV map is applied to. Since you can also output a UV Map from Flame Premium, make sure that anti-aliasing is not selected in the Render Options section of the Output menu.
- You can embed object IDs in the Blue channel when performing a UV render pass in Maya, allowing you to specify which objects are displaced within the UV map in Action. In Maya, when performing a UV Render Pass, enable *Embed Object ID in Blue Channel*. When disabled, blue channel information is ignored.
- For best results, in the Texture settings of the UV Map menu in Flame Premium, the Fit Method box should be set to Fill, and the Filter box to Nearest.



Flat texture



UV Map



Texture with UV Map applied

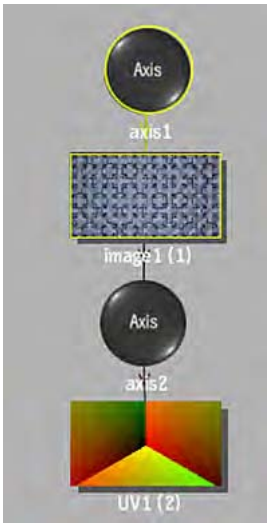
Adding a UV Map

To add a UV map:

1. In the schematic, select the surface or 3D geometry to which you want to apply the UV map.
2. Click Media.
3. In the Media menu, select the media you want to use for the UV map.
4. Do one of the following:
   - Drag the UV Map node from the node bin and place it in the schematic.
   - Drag the UV Map node from the node bin and place it where you want it in Result view.

- Double-click the UV Map node. You do not need to be in Schematic view to add a node in this manner.

The UV object is added to the schematic with its own parent axis. The new axis is the child of the selected surface or geometry. In Schematic view, the number in brackets next to the name of the UV node indicates the media used for the UV.



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

5 Double-click the UV node in the Schematic view, or follow the tab population rules for the Object menu.

The UV menu appears.

UV Menu Settings

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

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

Controls Tab



Effect field Displays the amount of displacement applied to the UV map. Editable.

Softness X field Displays the amount of softness along the X axis of the UV map. Editable.

Softness Y field Displays the amount of softness along the Y axis of the UV map. Editable.

Apply Type box Select which attached maps are affected by the UV map.

Use Object ID button Enable to use Object ID info embedded into the blue channel when using a UV map generated by Maya. When disabled, blue channel information is ignored.

The object ID allows you to control which objects within the UV map are displaced. In Maya, when performing a UV Render Pass, enable *Embed Object ID in Blue Channel*.

Object ID field Displays the specific blue value corresponding to an object ID. Editable.

Pick button Click to display the picker to select an object ID in the image through its blue value.

Texture Tab



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

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

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

Use Cropped Size button Enable to replace the UV map with the cropped size of the UV media. Disable to use the cropped UV media as is.

Mapping box Select the type of texture mapping.

Wrap (Geom UVs) Wrap mapping completely envelops the 3D model with the diffuse map according to the object's diffuse coordinates. To use this option, you must import a model that has its own diffuse coordinates. When using Wrap mode, you can also apply UV mapping settings from the Geometry menu. See UV Map Settings (page 540).



Plane Planar mapping applies the diffuse map without distorting the front plane of the 3D model, similar to a movie projector casting an image onto a screen. All 3D coordinates of the geometry are mapped to this plane to generate the diffuse values. Planar mapping positions the lower-left corner of the diffuse map on the 3D model's axis. When you apply planar mapping, any surfaces on the 3D model perpendicular to the front plane cause the pixels at the edge of the texture to project along the "sides" of the object.



Perspective Perspective mapping is similar to planar mapping, except that it performs a perspective transformation of the diffuse map based on the selected camera's field of view (FOV). When you select Perspective as the mapping type, the Perspective Camera box becomes active, allowing you to specify the active camera. The FOV of the camera has an impact on the resulting effect of any transform applied to the parent axis of the texture. On stereo cameras, the interaxial distance between left and right cameras also has an effect on the resulting perspective transform.

Projection The texture behaves as if it is projected by the selected camera. Projection mapping is useful as an alternative to projecting textures using the Projector node, especially when it is necessary to project while preserving a specific camera POV.

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

Camera field Displays the active perspective or projection camera number. Non-editable.

Stereo Camera Projection box Select whether to use the centre, left, or right camera from a stereo camera rig when projection mapping.

Filter box Select the type of filtering to apply to the UV map.

Select:	To apply:
Nearest	No filtering — the pixel of the texture closest to the screen pixel is displayed.
Linear	Basic bilinear filtering.
Anisotropic	Non-proportional filtering between X and Y (faster to process than EWA, but with a lesser quality).
Aniso+Linear	A combination of Anisotropic and Linear filtering.
EWA	A high-quality elliptical weighted average filter to produce enhanced rendering results (slower to process than other filters).
EWA+Linear	A combination of EWA and Linear filtering (offers the most advanced filter processing).

Camera Type box Select the camera type visibility for the UV map. For example, you can use this setting to apply a Left Eye and Right Eye camera type for two maps that are children of the same surface or geometry in a stereo scene.

Projecting Textures

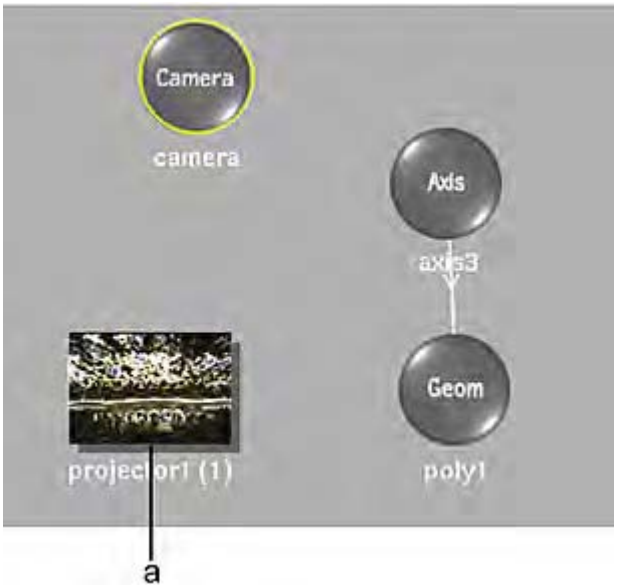
Like a slide projector, textures can be projected onto 3D models to create an effect where a texture is animated on an object in 3D space. The advantage of using a Projector node is that you can cast images on arbitrary surfaces. You can create a spotlight texture and project it onto a 3D model, so that you can see "into" it.

NOTE Some Action maps, such as Diffuse and Emissive maps, also allow you to use Projection mapping as a type of texture mapping. This method of projecting behaves as if it were projected by the selected camera.

To project a texture:

1. Click Media to access the Media menu.
2. In the Media list, select the media that you want to project.
3. Do one of the following:
   - Drag the Projector node from the node bin and place it in the schematic.
   - Drag the Projector node from the node bin and place it where you want it in Result view.
   - Double-click the Projector node. You do not need to be in Schematic view to add a node in this manner.

   The Projector node appears in the schematic and a projector icon appears in the scene.

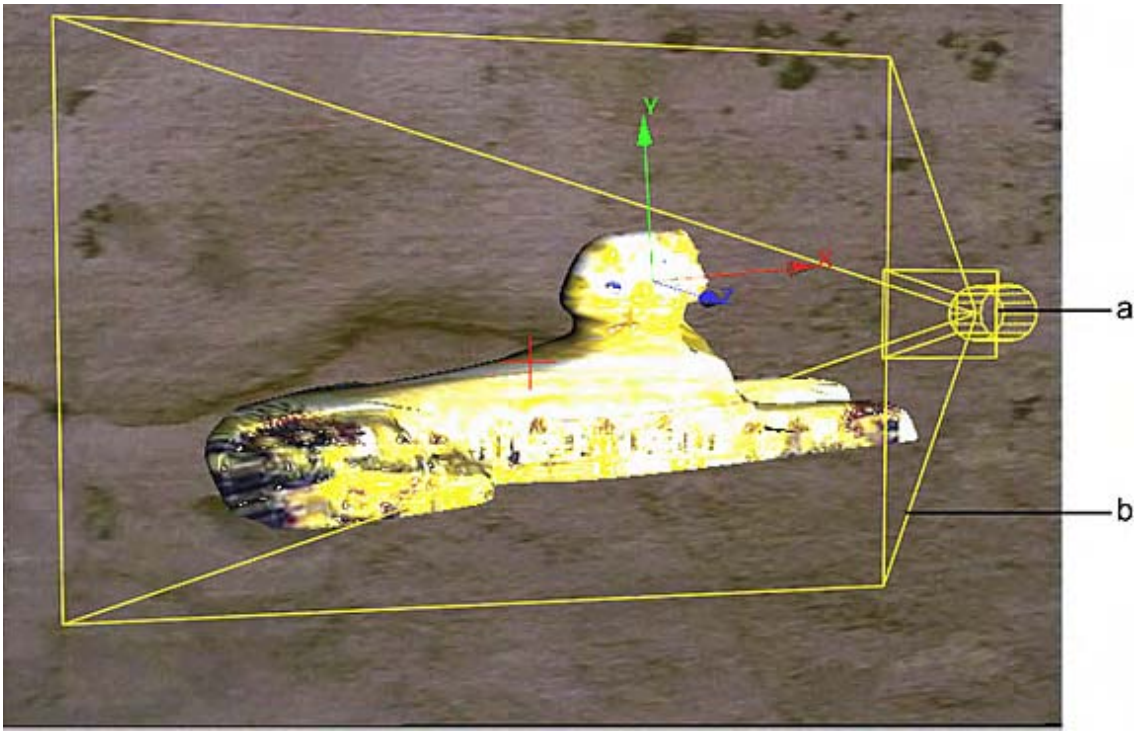


(a) Projector node

4. Double-click the Projector node to display the Projector menu, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).

   The Projector is selected.
5. Switch to Camera view to manipulate the projector in the scene using the View box.

   The following example illustrates projected media on a 3D model using Spotlight projection blend mode. See Blending Projections (page 631).



(a) Projector icon (b) Yellow outline shows the projector field of view

To add multiple projections:

1. Click Media to access the Media menu.
2. In the Media list, select the media you want to use as projectors by `Ctrl`-clicking the media.
3. Use the node bin to add a projector.

   Multiple Projector nodes appear in the schematic and projector icons appear in the scene.
4. Use the Order field in the Projector menu to set the order of projectors.

   The front projector is projector1. A new projector is always added to the back. Changing the order number of a projector affects the order of other projectors.

Projecting Textures Settings

Projector Menu Settings



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

Active button Enable to activate the projected texture effect in the scene. When disabled, the Projector node remains in the schematic and the projector icon appears smaller in the image area.

X Position field Positions the projector in 3D space along the X axis.

Y Position field Positions the projector in 3D space along the Y axis.

Z Position field Positions the projector in 3D space along the Z axis.

TIP You can also set the texture projector position by dragging the projector in the scene.

Motion Path button Enable to use motion path animation for the projector.

X Rotation field Rotates the projector in 3D space along the X axis.

Y Rotation field Rotates the projector in 3D space along the Y axis.

Z Rotation field Rotates the projector in 3D space along the Z axis.

Blend Mode button Select a mode to modify how you want to blend the projected image in the scene. See Blending Projections (page 631).

Transparency field Displays the transparency level of the projected texture effect. Editable.

Field Of View field Displays the projector's field of view value. Editable.

Aspect Ratio field Displays the ratio of height to width for the projected image. Editable.

Order field Displays the order of projectors. Editable.

Red Colour field Defines the red value of the selected projector. Editable.

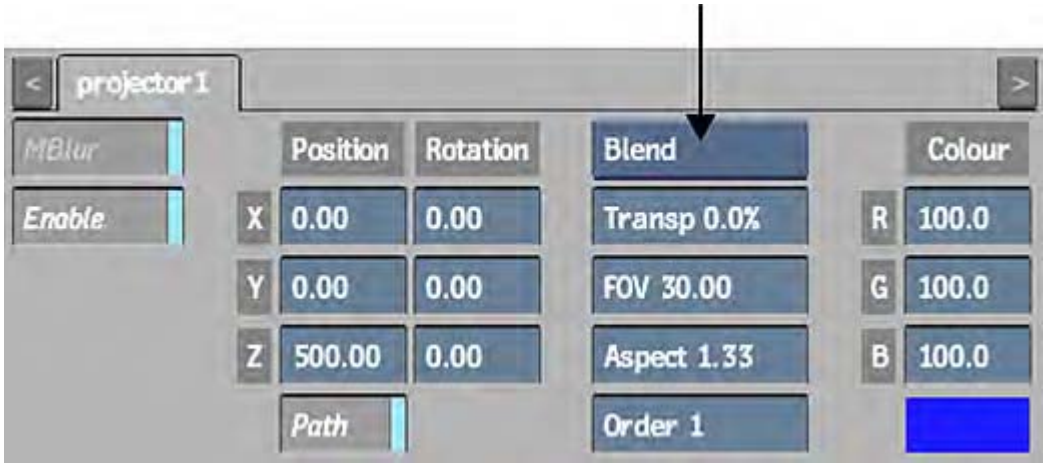
Green Colour field Defines the green value of the selected projector. Editable.

Blue Colour field Defines the blue value of the selected projector. Editable.

Projector colour pot Displays the colour of the projector. Editable.

Blending Projections

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



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

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

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

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

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

Hardware limitations may prevent transparency from working properly with Multiply.

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

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

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

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

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

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

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

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

Applying Selective Projections

You can use projections selectively; either inclusively or exclusively. Similarly to the Selective Lighting feature, you can connect a projection to an image, object, or its axis and make the projection affect only that connection (inclusive). Conversely, a projection that affects every object *except* the one it is connected to is exclusive.

To use selective projections, you must have a scene with a minimum of two objects in order for the selective projection to affect one and not the other.

To create an inclusive selective projection:

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

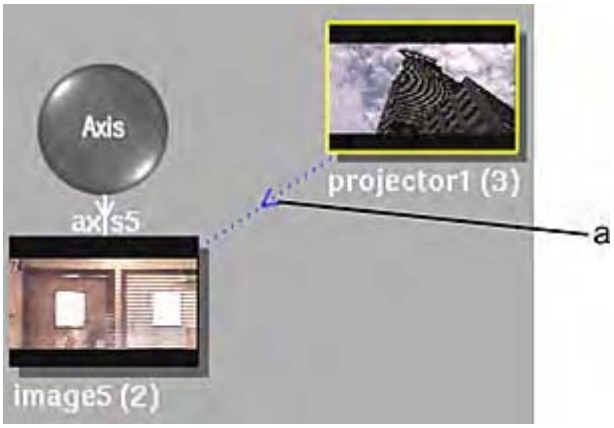
   The projection is added to the scene.
3. To make the projection selective, access the schematic by using the ~ key or by selecting Schematic from the View box.

   In the schematic, the projector is displayed much like an image. By default, it projects an image across the entire scene.
4. Select Lighting from the Edit Mode box to create lighting connections.



5. Do one of the following:
   - To project only a selected surface, drag a connection from the Projector node to one or several nodes in your scene. A selective projection connection is represented by a blue dotted line.

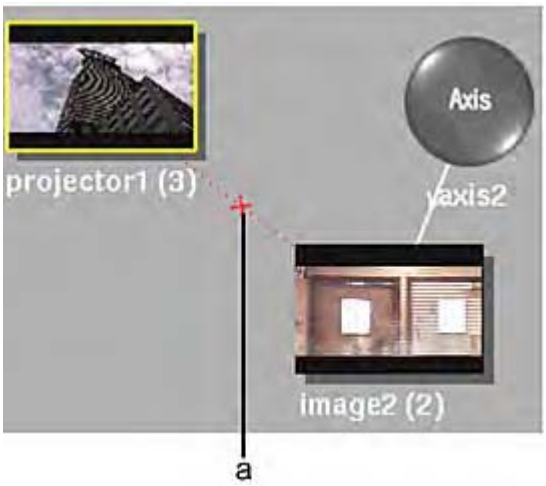
     Once a selective connection is created, the projector only affects the object to which it is connected.



(a) Inclusive projection connection

   - To exclude a surface, hold the `Alt` key as you drag a connection from the Projector node to a surface node. The exclusive connection is represented by a red dotted line.

     Once an exclusive connection is created, the projector ignores the node(s) to which it is connected.



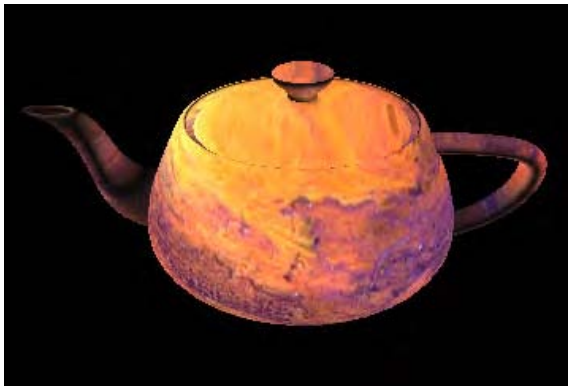
(a) Exclusive projection link

About Atomize

When you attach an Atomize node to a surface or geometry in Action, the object becomes a configurable point cloud. Pixels at each vertex of the surface or geometry become square or round points. The Atomize points always face the camera, and are displayed at the same size regardless of the distance from the camera. Atomize points inherit any properties of the parent object, as well as any attached maps (such as transparency, blend modes, and colours).

NOTE Atomize points are displayed in Action screen space, therefore they are affected by zooming in the image window.

Depending on the vertex resolution of the parent surface or geometry, the Atomize result can become quite detailed (more vertices equals more points), allowing you to creatively incorporate point clouds into your Action compositions.



Teapot Geometry with Diffuse Map



Teapot with Atomize applied

Adding an Atomize Node

To add an Atomize node:

1. In the schematic, select the surface or 3D geometry to which you want to apply the Atomize.
2. Do one of the following:
   - Drag the Atomize node from the node bin and place it in the schematic.
   - Drag the Atomize node from the node bin and place it where you want it in Result view.
   - Double-click the Position Map node. You do not need to be in Schematic view to add a node in this manner.

   The Atomize object is added to the schematic as a child of the selected surface or geometry.
3. Double-click the Atomize node in the schematic, or follow the tab population rules for the Object menu.

   The Atomize menu appears.

Atomize Menu Settings



Round Dot button Enable to display the points as circles; disable to display as squares.

Size field Displays the size of the points. Editable.

NOTE The size range is from 0 to 100, but the actual maximum value that will produce a change is determined by your graphics card. For example, you may notice that the size doesn't increase with a value higher than 64.

About 3D Paths

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

Adding a 3D Path Node

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

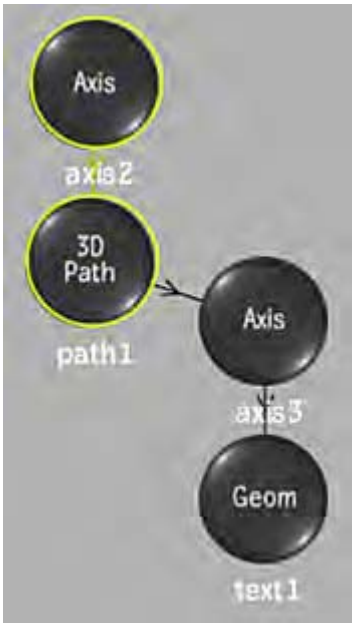
To add a 3D Path node to the scene:

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

A Path object (called path1, by default), with its parent axis, appears in the schematic.

2 Create mode is automatically selected in the Edit Mode box so you can create your spline. See Creating Splines (page 635).

3 Parent the 3D Path node to another object in your schematic, such as a Light node or a 3D Text node. The 3D path becomes part of the transformation hierarchy of the attached object.



3D Path node parented to a 3D Text node

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

See Populating Menu Tabs of Selected Objects (page 415).

Creating Splines

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

To create a spline:

1 Make sure that the Edit Mode box is in Create mode (this is the default when you first add a 3D Path node).

2 In the image window, click to add vertices.

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

3 To complete your spline, do one of the following:

- Click the first vertex to close the spline.
- Click Finish in the 3D Path tab to leave the spline open. If you decide later that you want to close the spline, enable Closed in the 3D Text tab.

4 Make sure that the Edit Mode box is in Move mode, so that you do not add more vertices by mistake.

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

Editing Splines

Use the settings in the 3D Path tab to work with the spline you created. You edit splines in the same way as you edit garbage masks, working with their vertices and tangents. See Manipulating Vertices and Tangents (page 903).



Finish button Click to finish an open spline.

Closed button Enable to close a spline from the last to first point.

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

Constant Shape button Enable to modify the spline's shape without setting keyframes. This forces all animatable parameters to be set for the whole clip rather than for only the current frame. It also removes any existing keyframes and applies the shape of the current frame to the rest of the clip.

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

X Vertex field Displays the position of the selected vertices on the X axis. Editable.

Y Vertex field Displays the position of the selected vertices on the Y axis. Editable.

Z Vertex field Displays the position of the selected vertices on the Z axis. Editable.

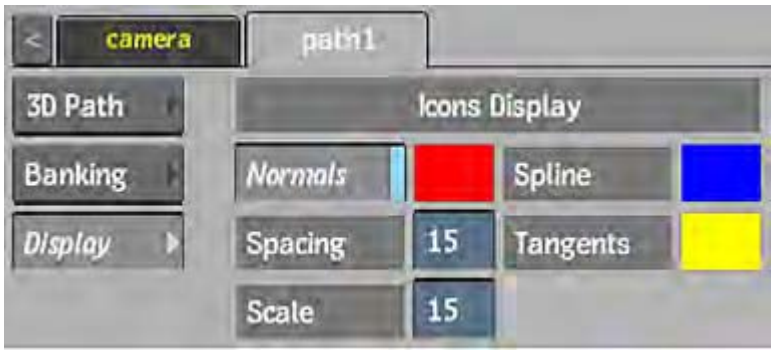
Auto Tangents button Enable to position a tangent for each vertex set, to create a smooth curve between the vertices. When enabled, it is possible to create a spline with both straight and curved segments.

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

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

Changing Spline Display Properties

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



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

Normals colour pot Displays the normals colour. Editable.

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

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

Spline colour pot Displays the colour for the display of splines. Editable.

Tangents colour pot Displays the colour for the display of tangents. Editable.

Positioning Objects on the Path

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



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

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

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

Aligning Objects on the Path

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



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



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

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

Select:	To:	Example:
Align Off	Not align the attached object to the path. In this case, the Orientation Axis box and banking controls are unavailable. This can be useful for vertical text effects.	
Align to Path	Align the attached object to the 3D path. You can then select which axis is aligned to the path in the Orientation Axis box, and use the banking controls.	
Look At	Point the attached object to a look-at object, such as a light. You can then select which axis is pointed to the look-at object in the Point Axis box, and use the banking controls. See Applying a Look-At Connection (page 638).	

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

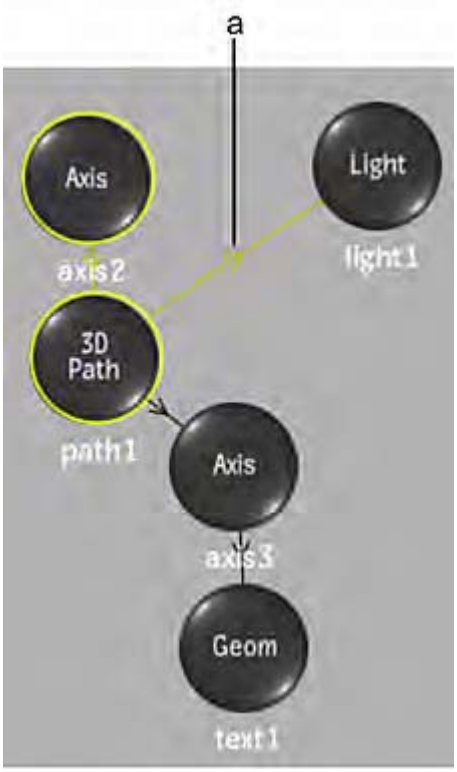
Point Axis box Select which axis is pointed to the attached look-at object. Available when Look At is selected in the Alignment option box.

Applying a Look-At Connection

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

To apply a look-at connection:

1. Do one of the following:
   - Select Look At in the Alignment option box.
   - Select Lookat in the Edit Mode box.
2. In the schematic, drag from the 3D Path node to an object with axis characteristics.
   The selected object is connected to the 3D Path node by an orange dotted line with an arrow.

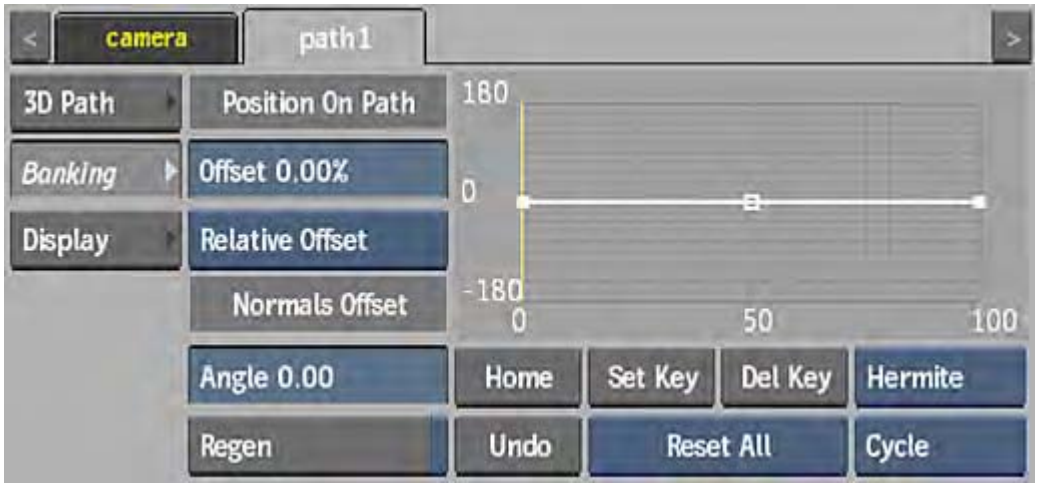


(a) Look-at connection

3. Select which axis looks at the attached object in the Point Axis box.
4. Optional: Use the banking curve to orient the attached objects.

Using the Banking Curve

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



The Position on Path settings are the same as those in the 3D Path tab. See Positioning Objects on the Path (page 637).

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

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

NOTE Changes made to the banking curve and Angle field are cumulative.

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

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

Undo button Undoes Banking curve operations.

Set Key button Sets the current values for the banking curve in the current frame (when Auto Key is disabled).

Del Key button Deletes the selected banking curve keyframes.

Reset Selection box Select whether to reset all of the banking settings (Reset All) or just the banking curve (Reset Key).

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

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

About the Camera

The scene is what you see through the camera lens. Typically, you work with the camera to frame and animate the view to achieve the effect you want. In Action, you have the choice of using the automatic camera or the manual camera, whose F-Stop, film size, and focal length you can set yourself. You can also animate specific camera properties.

Adding a Camera

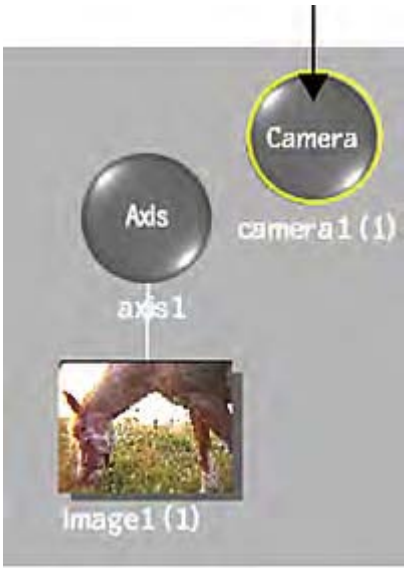
By default, a camera exists in the Action scene (you may need to pan in the schematic to see the camera node). You can add multiple cameras in order to change point of view or depth of field from one camera to

another. Add and animate multiple cameras when creating compositions. You can also switch from one camera to another at any point.

To add a camera:

1 Do one of the following:
- Drag the camera node from the node bin and place it in the schematic.
- Drag the camera node from the node bin to Result view, so you can see its effect on the scene before placing it exactly where you want.
- Double-click the camera node. The node appears next to the last added object. You do not need to be in Schematic view to add a node in this manner.

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



2 To display the Camera menu, double-click the selected camera in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 415).

The result camera always appears as the first tab on the right side of the Object menu. To allow you to easily access the camera without losing your place in the scene, this special camera tab (appearing in orange) does not follow the tab population rules. If a camera node is selected in the schematic, the special Camera tab does not appear, and the normal tab population rules apply.

Camera Menu Settings



X Eye field Displays the position of the camera eye on the X axis. Editable

Y Eye field Displays the position of the camera eye on the Y axis. Editable

Z Eye field Displays the position of the camera eye on the Z axis. Editable

Motion Path button Enable to animate the camera eye on a motion path. See Moving the Camera Eye and Point of Interest (page 645).

X Point of Interest field Displays the position of the point of interest on the X axis. Editable.

Y Point of Interest field Displays the position of the point of interest on the Y axis. Editable.

Z Point of Interest field Displays the position of the point of interest on the Z axis. Editable.

X Rotation field Displays the level of camera rotation along the X axis. Editable.

Y Rotation field Displays the level of camera rotation along the Y axis. Editable.

Z Rotation field Displays the level of camera rotation along the Z axis. Editable.

Camera Type box Select Free (to view the scene in the direction that you aim the camera), or Target (to aim the camera at a target object in the scene based on a point of interest).

Free cameras are easy to use because you do not have to manipulate the point of interest. You can simply animate the camera rotation or camera tilt as though it were on a tripod. Use the Distance field in conjunction with Free Camera.

Roll field Displays the amount of camera roll (available with Target Camera). Editable.

Use the Roll field in conjunction with Target Camera.

Field of View field Displays the camera field of view value, measured in degrees. Editable.

When PhysicalCam is disabled, use the field of view angle in the Y direction to adjust the width of the camera frustum. When PhysicalCam is enabled, use the focal length for the same purpose.

Distance field Displays the position of the camera's focus. This setting affects the depth of field (when enabled in the Action Setup menu). Editable.

Near field Displays the near view of the selected camera. Editable.

Far field Displays the far view of the selected camera. Editable.

See Moving the Clipping Planes (page 645).

Export Camera button Opens the Export Camera file browser to save a camera.

Exported data includes Eye X, Y, and Z; Poi X, Y, and Z; Fov; Roll, Target or Free Camera; motion path and explicit keyframe camera animation. See Importing and Exporting Cameras (page 646).

Import Camera button Opens the Import Camera file browser to import a camera. See Importing and Exporting Cameras (page 646).

Result Camera box Specify which camera is active. The active camera is the one that will be used when processing/rendering your scene.

Result Camera field Displays the active camera number. Non-editable.

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

Camera box Not shown. When in Camera view, select which camera is used in the image window.



Parenting Offset box Select an offset option for viewing an image when parenting a camera node. When parenting a camera node, the image offset gets reset to the camera origin, which is not always the desired viewing option. Origin sets the image to the camera origin; Target sets the image to the default viewplane distance relative to the camera; and Live Target sets the image to the current viewplane distance based on the FOV. The offset value is computed from the default camera field of view and the default image size, and does not change even if other camera parameters are changed. This value is displayed in the Parenting Offset field.

Fog box Select a fog type to cause objects to fade as their distance from the camera increases. See Applying Fog to the Scene (page 657).

Fog colour pot Displays the fog colour. Editable.

Fog Start field Displays the distance at which the fog should start. Editable.

Fog End field Displays the distance at which the fog should end. Editable.

Fog Range field Displays the range or distance from the camera for exponential type fog. Editable.

Physical Camera button Enable to activate the physical camera, which simulates a manual camera. Use the physical camera fields to adjust the field of view for the camera.

f-stop box Select a depth of field value. F-stop is the ratio between the focal length of the lens and the diameter of the aperture. For example, the F-stop value for a 100mm lens with a 50mm full aperture is 2 (or f/2).

Large f-stop values correspond to smaller apertures. The depth of field for your physical camera increases as you increase the f-stop value.

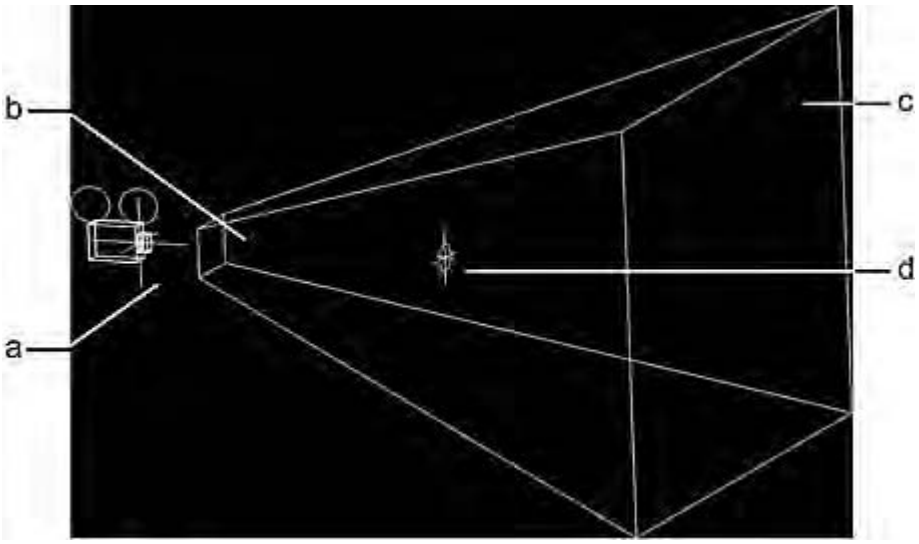
Film Size box Select a standard film size. As the film size increases, the focal length also increases and the focal point moves further away from the camera eye. Changing the film size also increases the focal length.

Focal Length field Displays the field of view. Changes to the Focal Length automatically update the field of view (based on the film size) and vice versa. Editable.

As the focal length increases, the field of view decreases. As the focal length decreases, the field of view increases.

About the Frustum

The volume of space viewed by the camera eye is called the frustum. The frustum is in effect a viewing pyramid. The camera eye is located at the apex of the pyramid, and the far clipping plane forms the base. The pyramid may be truncated by the near clipping plane.



(a) Camera eye (b) Near clipping plane (c) Far clipping plane (d) Camera interest point or look-at point

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

Viewing the Camera and Frustum

To see the camera and frustum:

1. From the View box, select Side.
2. In the image window controls, click  to zoom out from the scene.
3. Select Pan in the Edit Mode box and pan around the scene until you see the camera eye icon. Alternatively, use Orbit mode to pan around the scene in circular motion.
4. Go to the Camera menu and drag the Roll field until you see the four sides of the frustum.
5. To modify the frustum, do one of the following:
   - Change the position of the near clipping planes to alter the depth of the frustum. See Moving the Clipping Planes (page 645).
   - Change the position of either the camera eye or the camera's point of interest to alter the orientation of the frustum. See Moving the Camera Eye and Point of Interest (page 645).
   - If the Physical Camera button is disabled, enter a value in the FOV (field of view) field to adjust the width of the camera frustum.
   - If the Physical Cam button is enabled, enter a value in the Focal Length field to narrow or widen the frustum. You can also alter the depth of the frustum using only the near clipping plane.

Moving the Clipping Planes

The camera frustum is determined by six clipping planes: the left, right, top, bottom, near, and far clipping planes. The depth of the frustum is affected by the near and far clipping planes. The values for these channels are expressed in units relative to the position of the camera eye.

To move the clipping planes:

1. From the View box, select Side, Front or Top depending on how your camera is positioned. Ideally, you will want a view that profiles the camera so the near and far planes are visible.
2. In the Camera menu, enter a value in the Near field to edit the near clipping plane's position.

   The value in the Near field corresponds to the position of the near clipping plane. The default value is 1. Any object between the camera eye and the near clipping plane is outside the camera frustum and does not get processed in the final result.
3. Enter a value in the Far field to edit the far clipping plane's position.

   The value in the Far field corresponds to the position of the far clipping plane. The default value is 10000. Any object positioned behind the far clipping plane is outside the camera frustum and does not get processed in the final result.
4. Narrow or widen the frustum by modifying the camera's FOV (field of view). Increasing the FOV narrows the frustum and field of view. Decreasing the value widens the frustum and field of view.

Moving the Camera Eye and Point of Interest

Objects in the scene can be recorded from an arbitrary position as determined by the orientation of the camera eye in world space.

To change the position of the camera eye:

1. In the Camera menu, modify the Eye X, Y, and Z fields.

You can also animate the camera eye on a motion path. Enable the Motion Path button and drag the camera icon while viewing the scene in Top, Side, or Front view.

NOTE You can only animate the point of interest with a Target Camera.

The camera point of interest is the point in world space at which the camera eye is directed. The point of interest is always at the centre of the camera's frustum. Changing the position of the point of interest causes the orientation of the frustum to change. You can take advantage of the relationship between the point of interest and the frustum to make the camera follow a moving object. To do this, animate the point of interest while keeping the camera eye in a fixed position.

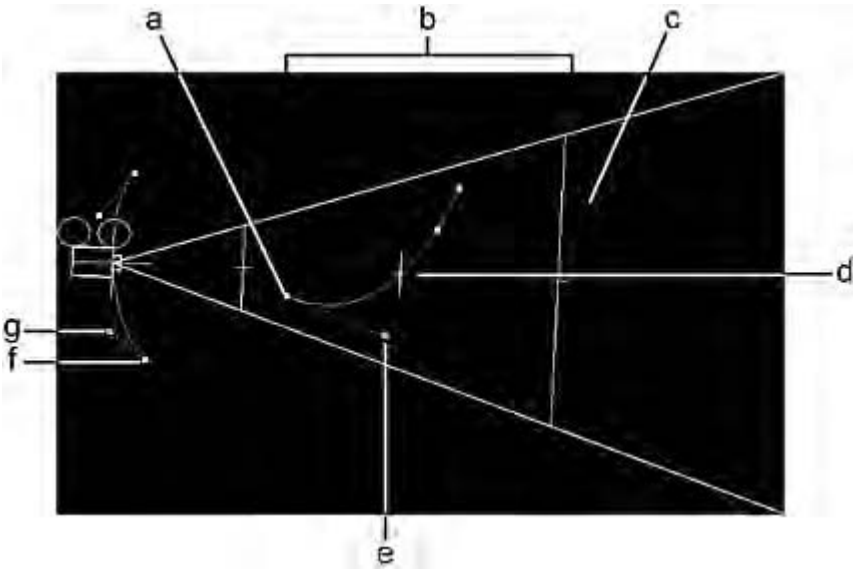
To change the point of interest:

1. In the Camera menu, do one of the following:
   - Modify the Interest X, Y, and Z fields.
   - Move the point of interest by dragging while viewing the scene in Top, Side, or Front view.

TIP You can animate the point of interest using a motion path by enabling the Motion Path button.

Adjusting the Depth of Field

Using depth of field, you can blur objects that are outside the camera's focal range. The further an object is from the focal range, the more it is blurred.



(a) Near sharpness plane (b) Focal range (c) Far sharpness plane (d) Motion path for point of interest (e) Camera point of interest (f) Motion path for camera eye (g) Camera eye

The Target camera uses depth of field and clearly indicates its focal range when in Top, Side, or Front view. With the Free camera, you can change the field of view by `Alt`-dragging the near sharpness plane. This changes the camera only; what you are viewing remains the same size.

To adjust the depth of field:

1. In the Action menu, click Setup.
2. In the Accumulation Settings section, enable Depth of Field. Objects are blurred on either side of the point of interest.
3. Change the amount of blur using the Depth of Field Softness field.

Importing and Exporting Cameras

You can either import a camera you created and edited in Action, or import one from another 3D application. Once a camera is imported, you can edit its parameters, change its animation, and then export it back to the application it came from.

FBX Cameras

Flame Premium supports the import and export of 3D data saved in the FBX 3D format. This format provides a means for exchanging 3D data for scene compositions—such as cameras—between tools and packages developed by different manufacturers.

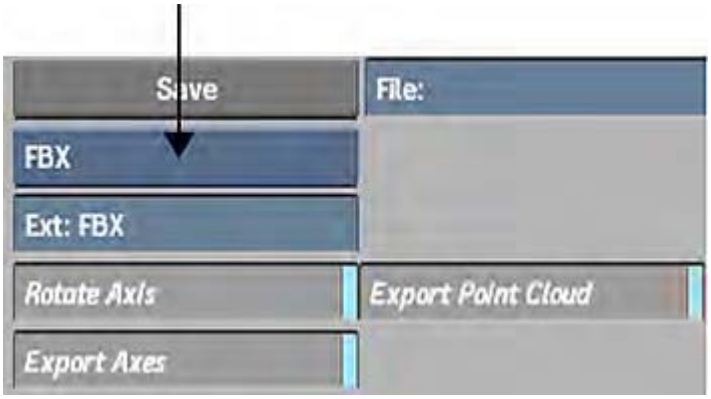
To export a camera to FBX format:

1. Select the camera you want to export.
2. In the Camera menu, click Export.



The Export Camera file browser appears.

3 From the Export Type box, select FBX.



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

Enable:	To:
Rotate Axis	Rotate the exported camera by -90° on the X-axis so that it is compatible with the co-ordinate system of the 3D application.
Export Axes	Export the animated axes present in the Action scene.
Export Point Cloud	Export the 3D point cloud created by the 3D Tracker.

5 Navigate to the location where you want to export the camera animation.

6 Enter a name for your exported camera in the file field.

7 Click Save.

3ds Max Cameras

An alternative to animating a camera's position is to import a camera animation from 3ds Max. 3ds Max camera animation can be saved as a . 3DS or .ase (ASCII Scene Export) file in your scene. While the .ase format contains only a camera's positional data, the .3DS format contains the camera's position, point of interest, roll and field of view values.

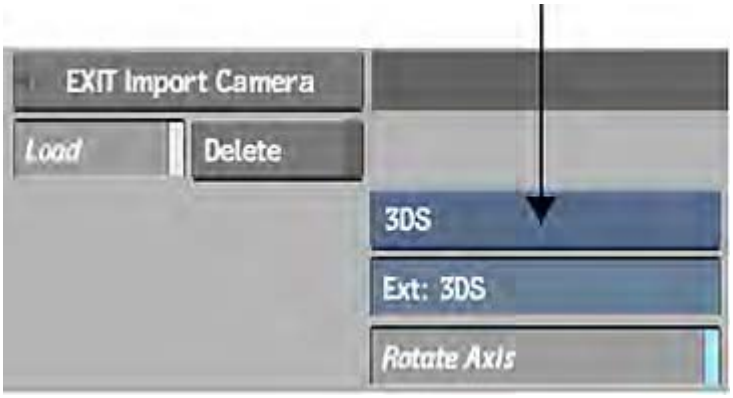
Once the camera is imported, you can edit any of the camera's values and, if necessary, export it back to 3ds Max in its native 3DS format using the Camera menu.



(a) Import Camera button (b) Export Camera button

To import a 3ds Max camera:

1. In the Camera menu, click Import.
   The Import Camera file browser appears.
2. From the Import Type box, select either the .3DS or .ase format.



3. If needed, enable Rotate Axis to rotate the imported camera by 90° on the X-axis so that it is compatible with Action's coordinate system.
4. Navigate to the location where a 3ds Max camera setup was exported, and select the file.
   You return to Action and the imported camera is applied to your scene.

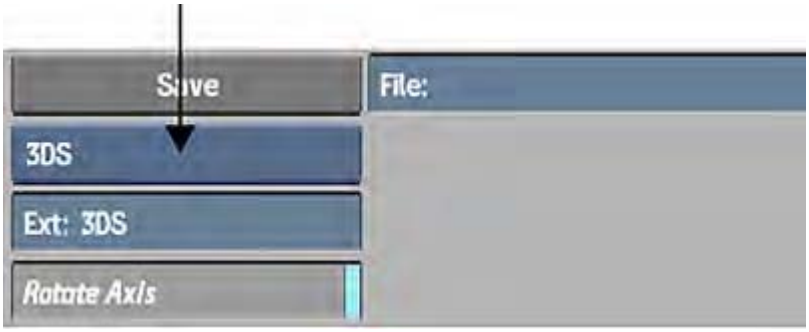
To export a camera to 3ds Max format:

1. Select the camera you want to export.
2. In the Camera menu, click Export.



The Export Camera file browser appears.

3 From the Export Type box, select 3DS.



4 If needed, enable Rotate Axis to rotate the exported camera by -90° on the X-axis so that it is compatible with the coordinate system of the 3D application.

5 Navigate to the location where you want to export the camera animation.

6 Enter a name for your exported camera in the file field.

7 Click Save.

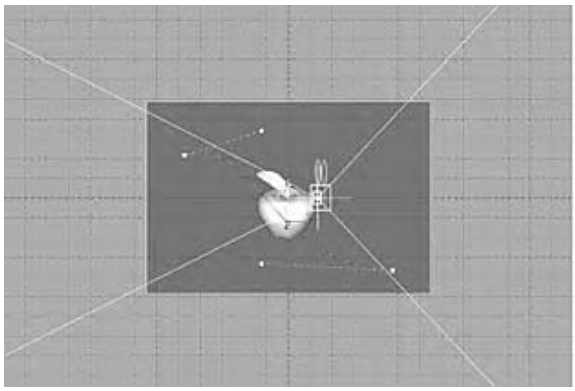
NOTE When exporting from Flame Premium, save the .3DS file in a directory that is readable by a 3ds Max system.

Camera and Orthographic Views

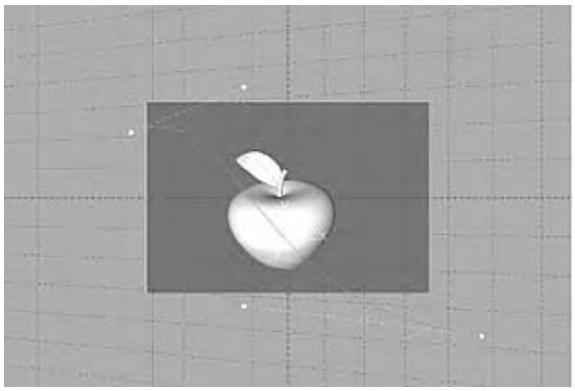
You can view the scene from various angles and display multiple views of these angles simultaneously. This is helpful in setting motion paths, light sources, camera angles, and animation keyframes more accurately.

You can view the scene from Camera view and three orthographic views. In Camera view, an object becomes smaller as it moves farther away from the camera. In orthographic view, an object remains the same size, regardless of its distance from the camera. Orthographic views are more helpful for aligning objects.

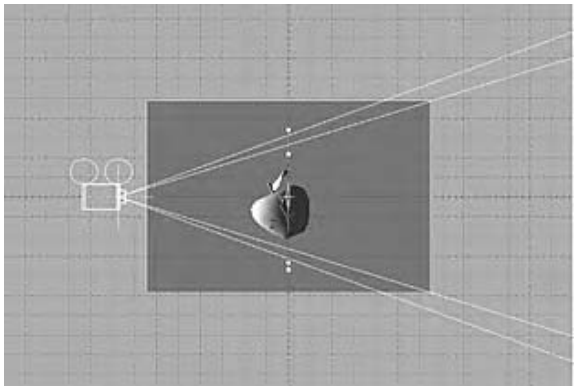
The following figures illustrate the different angles by which the scene can be viewed. The scene in this example contains a grey back clip and the 3D model of an apple.



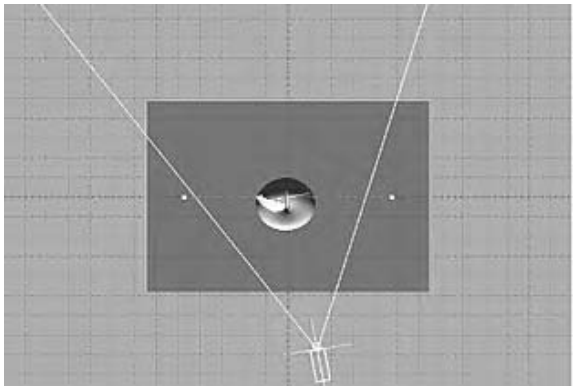
Front view



Camera view



Side view

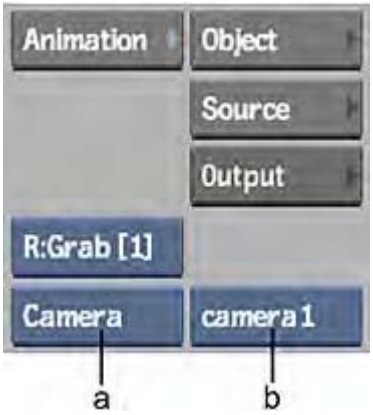


Top view

Setting the Camera and Orthographic Views

To set camera and orthographic views:

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



(a) View box (b) Camera box

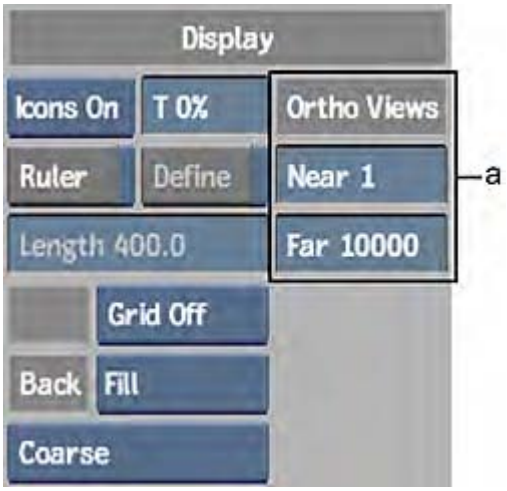
Select:	To:
Camera	View the scene in Camera view. This is the scene as viewed by the camera eye. In other words, your field of vision in world space is equivalent to the viewing frustum of the camera. The size of objects depends on their distance from the camera eye.
Top	View the scene as if you are positioned on the positive Y-axis. This is an orthographic view; there is no perspective deformation.
Side	View the scene from the side, as if you are positioned on the positive X-axis. This is an orthographic view; there is no perspective deformation.
Front	View the scene as if your line of vision is directed into the camera eye. This is an orthographic view; there is no perspective deformation.

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

3 Adjust the view with the Ortho Views controls, if needed.

When using an orthographic view, you may notice that parts of the object you are viewing are getting cut off. Adjust the near and far ortho views. You gain more space to view the object, but lose some viewing precision.

In the Display section of the Action Setup menu , adjust the Near and Far fields.



(a) Ortho Views in Setup menu

NOTE The Ortho Views parameters in the Action Setup menu are only for viewing objects, and cannot be animated or saved. The Near and Far fields in the Camera menu are used to set clipping planes. See Moving the Clipping Planes (page 645).

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

Modifying the Camera

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

To gesturally modify the camera:

1 Make a selection in the Edit Mode box.



Select:	To:
Track	Move the camera lens and look-at point.
Tilt	Tilt the camera up and down by moving the look-at point. Also changes the camera roll. This option only modifies Target cameras.
Roll	Rotate the camera on the Z-axis. This option only modifies Target cameras.
Orbit	Rotate the camera lens around the look-at point.
FOV	Move the camera field of view.
Dolly	Move the camera lens towards (zoom in) or away from (zoom out) the look-at point.

Select:	To:
Pan	Move the camera left and right by moving the look-at point. Also changes the camera roll, when not 0.00.

2 Drag the cursor in the image window.

The camera is modified. Related Camera menu controls are updated to reflect the changes.

Zooming In and Out

Use the Zoom option to move the camera eye toward or away from the point of interest. While viewing the scene in Camera view, zoom in or out from the point of interest to move the camera eye closer to or farther from the point of interest. In Top, Side, or Front view, you can enlarge or reduce the scene in the image window without affecting the camera. Zooming has no effect in Schematic view.

To zoom the camera:

1 From the Edit Mode box, select Zoom.

2 Place the cursor in the image window.

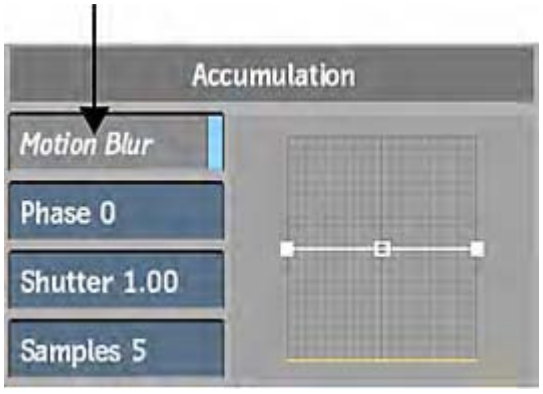
The cursor changes to a magnifying glass.

3 To zoom in, drag the cursor to the left. To zoom out, drag the cursor to the right.

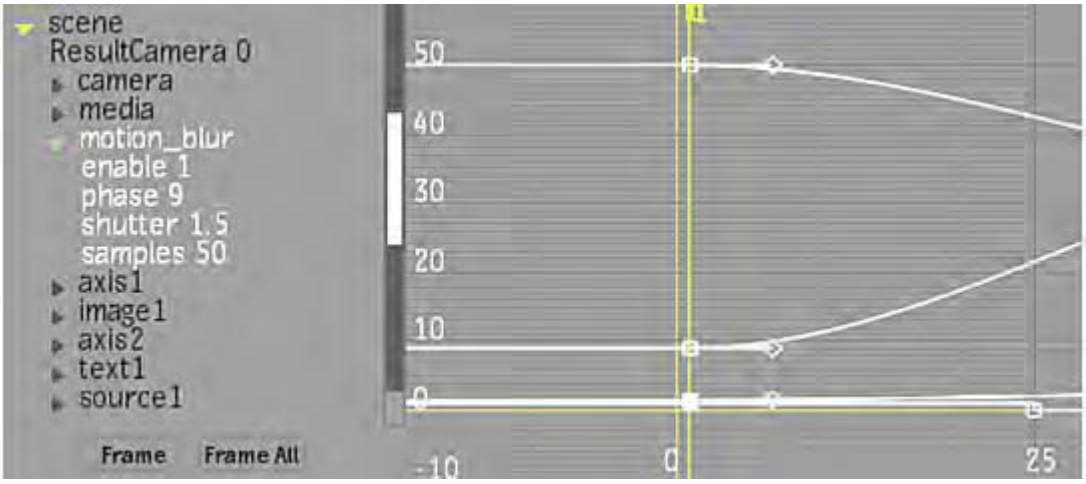
About Motion Blur

Use the Motion Blur tool to simulate the blur created by fast-moving objects. Motion Blur is used with both the normal and physical cameras.You can apply motion blur globally (to the entire scene) and then exclude objects in the scene from its effect.

To use motion blur, enable the Motion Blur button in the Action Setup menu and specify motion blur settings.



You can animate the Motion blur button, as well as the Phase, Shutter, and Samples fields. They can be found in the Channel Editor under the *motion_blur* folder.



See Accumulation Settings in Rendering Tab (page 423).

Blurring a Single Object

You can apply motion blur to an object rather than to the entire scene. To use the per object motion blur, you must apply Motion Blur globally, and then disable motion blur per object.

NOTE The motion blur curve in the Action Setup menu is global. It cannot be set per node. When motion blur is enabled, all surfaces have the same blur characteristics but not necessarily the same animation.

To blur a single object:

1. Enable Motion Blur in the Action Setup menu.

   This activates a global motion blur applied to everything within the scene.
2. Disable motion blur per object (for example, in the surface, axis or light menu) to exclude objects from the global motion blur.

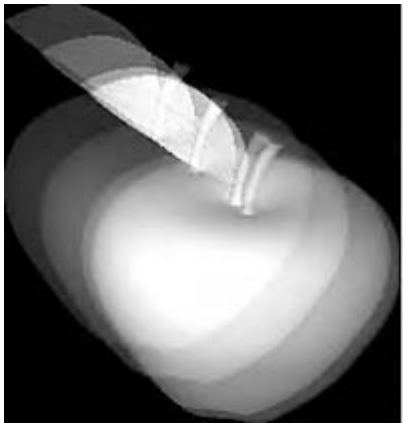
Creating Custom Motion Blurs

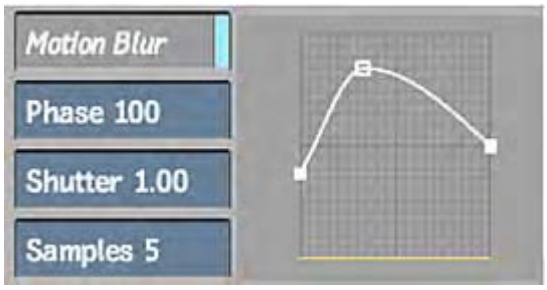
The motion blur curve controls the sample weight over the scope of the motion blur. The point on the left is the weight of the first sample and the point on the right is the weight of the last sample. By changing the curve, you can create custom motion blur effects such as a Gaussian blur.

As with the Channel Editor, you can add keyframes to the motion blur curve using Add mode, move keyframes with Move mode, and modify the curve's shape using tangent handles.

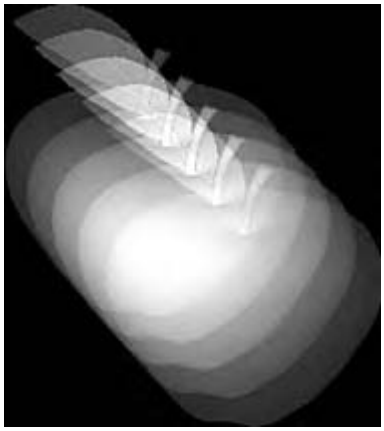


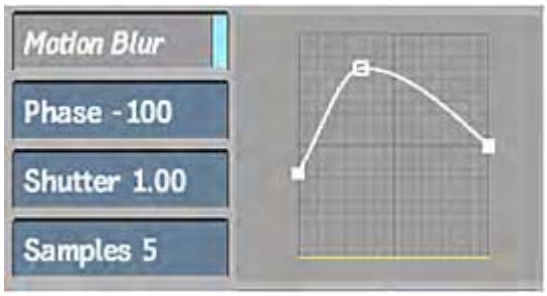
The original animation with Motion Blur disabled.





Motion Blur is enabled using a custom motion blur curve and phase set to 100.





Motion Blur is enabled using the same motion blur curve as above. Phase is set to -100.

TIP You can preview your motion blur effect by clicking Preview. This will display the rendered frame at the current frame in the timebar.

Simulating Motion Blur on a Still Object

You can add motion blur to an object that has no axis movement, by adding a second axis to simulate the motion.

To simulate motion blur on a still object:

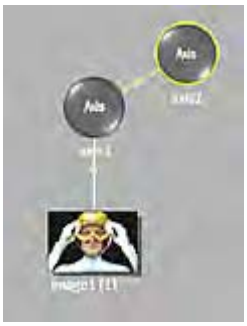
1. From the View box, select Schematic.
2. From the Node bin, drag an Axis node to the schematic.
3. Press `Shift` and drag the original Axis node over the new Axis node.



Still object with Axis



New axis added



New axis as child of original axis

Image courtesy of Das Werk

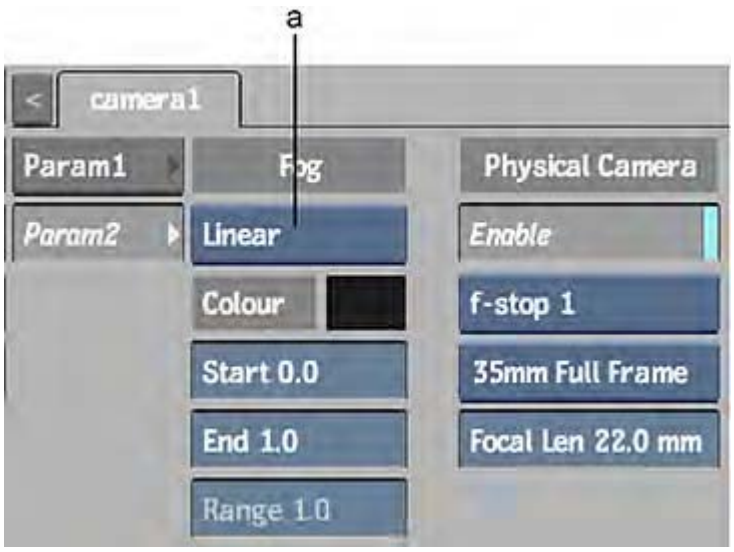
4. Enable Motion Blur and animate the new axis with the desired motion.

About Fog

Use fog in the scene to create visual effects such as mist, haze, and murky water. The fog effect causes objects to fade as their distance from the camera increases. The further an object is from the camera, the foggier the scene appears.

The fog effect is based on depth-cueing, which means the intensity of the fog varies along the Z-axis, and the specified colour gradually increases or decreases its intensity.

You control the density and colour of the fog using the fog controls in the Camera menu.



(a) Fog box

Try combining your fog effect with other effects such as transparency to create a more genuine look for non-solid substances such as water. You can also use expressions to improve the quality of fog in a scene so that it looks more natural. See Working with Expressions in the Channel Editor (page 1496).

Applying Fog to the Scene

To add fog to the scene:

1 In the Camera menu, select a fog type from the Fog box.

Type	Description
Linear	Specifies the distance at which fog should start and end.
Exponential	Fog = e -(density * z) where z represents the range or distance from the camera. The range should lie within the realm of the near and far clipping planes. The minimum value is 1.
Exponential2	Fog = e -(density * z)2. The exponential types provide more depth, more natural looking fog, and make the edges appear smoother.

2 For Linear fog, specify the distance at which the fog should start and end in the Start and End fields. For Exp or Exp2 fog, specify the range in the Range field.

3 To select the colour of the fog, click the colour pot to display the colour picker.

Typically, the fog colour should match the background colour of the image in the scene.

About the 3D Camera

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

Typically, you work with the 3D camera to frame and animate the view to achieve the effect that you want. You can also animate specific camera properties.

Use the 3D camera in Stereo mode to create three-dimensional renders with the illusion of a three-dimensional depth-of-field. When rendering a stereoscopic scene, Action takes into account all of the stereoscopic camera attributes. Action outputs two clips: one rendered for the left camera and one for the right camera. These clips can then be viewed in stereo mode, used in other stereo clips, output to VTR, or composited by another program.

Adding a 3D Camera

By default, a 3D camera exists in the Action scene when using stereo clips, or when Action has been set up to work in stereo (you may need to pan in the schematic to see the camera node). You can add multiple 3D cameras in order to change point-of-view or depth-of-field from one camera to another. You can add and animate multiple cameras when creating compositions. You can also switch from one camera to another at any point.

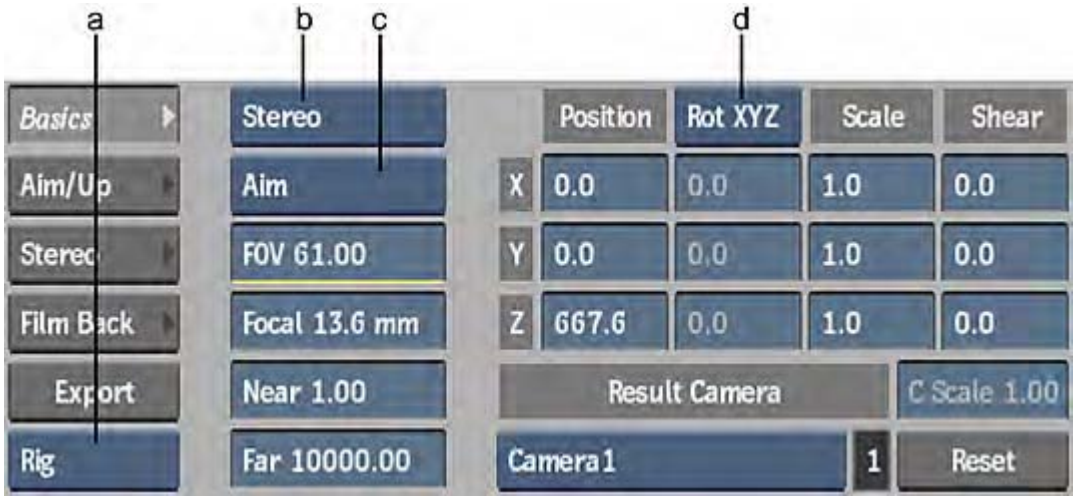
To add a stereo camera to a scene:

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

   A new 3D camera is added to the scene. An icon representing the camera is added to the schematic.
2. Select the 3D Camera node to make it the active camera in the scene.
3. In the Object menu that appears, set the Result Camera to a numbered camera representing a 3D camera.
4. In the Output menu, set the Mode to Stereo and set the Camera to Result Cam.

3D Camera Parameters

Basics Tab



(a) Stereo Camera View Type box (b) 3D Camera Type box (c) Camera Type box (d) Rotation Order box

Export FBX Camera button Opens the Export Camera file browser to save an FBX camera.

Stereo Camera View Type box Available when Stereo is selected in the 3D Camera Type box. Select Left, Right, or Rig (for Stereo Rig).

3D Camera Type box Select whether the 3D camera is stereo or mono.

Camera Type box Select whether the camera is Free, Aim, or Aim and Up.

Select:	For:
Free	Static scenes and for simple animations (up, down, side-to-side, in and out), such as panning out of a scene. A Free camera views the scene in the direction that you aim the camera. You can simply animate the camera rotation or camera tilt as though it were on a tripod. Use the Rotation fields in conjunction with a Free camera.
Aim	Slightly more complex animations (along a path, for example), such as a camera that follows the erratic path of a bird. The Aim camera ensures the camera is specifically aimed at a target object in the scene. Use the Roll and Aim fields in conjunction with the Aim camera.
Aim and Up	Complex animations, such as a camera that travels along a looping roller coaster. Use the Aim and Up camera to specify which end of the camera must face upward. Use the Roll, Aim, and Up fields in conjunction with the Aim and Up camera.

FOV field Displays the angular field of view value, measured in degrees. Use to adjust the width of the camera frustum. Editable.

Focal Length field Displays the focal length of the camera lens, measured in millimeters. Increasing zooms the camera in and increases the size of objects. Decreasing zooms the camera out and decreases the size of objects. Editable.

Near Clipping Plane field Displays the position of the near clipping plane, in pixels, which represents the distance from the camera to the closest point within which image details are processed. Editable.

Far Clipping Plane field Displays the position of the far clipping plane, in pixels, which represents the distance from the camera to the farthest point within which image details are processed. Editable.

See Moving the Near and Far Clipping Planes (page 666).

X Position field Displays the position of the camera, in pixels, on the horizontal (X) axis. Editable.

Y Position field Displays the position of the camera, in pixels, on the vertical (Y) axis. Editable.

Z Position field Displays the position of the camera, in pixels, on the perpendicular (Z) axis. Editable.

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

X Rotation field Displays the level of rotation of the camera on the horizontal (X) axis, in degrees. Active when Camera Type is set to Free. Editable.

Y Rotation field Displays the level of rotation of the camera on the vertical (Y) axis, in degrees. Active when Camera Type is set to Free. Editable.

Z Rotation field Displays the level of rotation of the camera on the perpendicular (Z) axis, in degrees. Active when Camera Type is set to Free. Editable.

X Scale field Displays the scale of the camera on the horizontal (X) axis, as a percentage. Editable.

Y Scale field Displays the scale of the camera on the vertical (Y) axis, as a percentage. Editable.

Z Scale field Displays the scale of the camera on the perpendicular (Z) axis, as a percentage. Editable.

X Shear field Displays the shearing of the camera (diagonal shift) on the horizontal (X) axis, as a percentage. Editable.

Y Shear field Displays the shearing of the camera (diagonal shift) on the vertical (Y) axis, as a percentage. Editable.

Z Shear field Displays the shearing of the camera (diagonal shift) on the perpendicular (Z) axis, as a percentage. Editable.

Result Camera box Specify which camera is active. The active camera is the one that will be used when processing/rendering your scene.

Camera Scale field Displays the size of the camera relative to the scene, independently for either Left or Right camera views. For example, if Camera Scale is set to 0.5, the camera's view covers an area half as large, but objects in the camera's view are twice as large. If Focal Length is set to 35, the effective focal length for the camera would be 70. Editable.

Reset button Resets the 3D Camera menu to its default settings.

FBX Unit Scaling field Not shown. This locked field displays the scale factor of the FBX camera used within the application. Use to help set the Pixels to FBX Units field when exporting an FBX camera. Non-editable.

Aim/Up Tab

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



3D Camera Type box Select whether the 3D camera is stereo or mono.

Camera Type box Select whether the camera is Free, Aim, or Aim and Up.

Select:	For:
Free	Static scenes and for simple animations (up, down, side-to-side, in and out), such as panning out of a scene. A Free camera views the scene in the direction that you aim the camera. You can simply animate the camera rotation or camera tilt as though it were on a tripod. Use the Rotation fields in conjunction with a Free camera.
Aim	Slightly more complex animations (along a path, for example), such as a camera that follows the erratic path of a bird. The Aim camera ensures the camera is specifically aimed at a target object in the scene. Use the Roll and Aim fields in conjunction with the Aim camera.

Select:	For:
Aim and Up	Complex animations, such as a camera that travels along a looping roller coaster. Use the Aim and Up camera to specify which end of the camera must face upward. Use the Roll, Aim, and Up fields in conjunction with the Aim and Up camera.

Parenting Offset box Select an offset option for viewing an image when parenting a camera node. Origin sets the image to the camera origin; Target sets the image to the default viewplane distance relative to the camera; and Live Target sets the image to the current viewplane distance based on the FOV.

Parenting Offset field Displays the offset value, as computed from the default camera field of view and the default image size. This value does not change even if other camera parameters are changed. Non-editable.

X Aim field Displays the position of the aiming target of the camera on the horizontal (X) axis, in pixels. Editable.

Y Aim field Displays the position of the aiming target of the camera on the vertical (Y) axis, in pixels. Editable.

Z Aim field Displays the position of the aiming target of the camera on the perpendicular (Z) axis, in pixels. Editable.

X Up field Displays the up direction on the horizontal (X) axis, in pixels. Editable.

Y Up field Displays the up direction on the vertical (Y) axis, in pixels. Editable.

Z Up field Displays the up direction on the perpendicular (Z) axis, in pixels. Editable.

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

Stereo Tab



Stereo Mode box Select the method for computing the zero parallax plane.

Select:	To:
Converged	Compute the zero parallax plane by toeing-in the cameras. You can compare this effect to our focusing on an object by rotating our pupils inwards. However, a dangerous side effect may occur where you get a keystone effect on the pairs of render images, causing visual confusion in other elements in the scene. In a rendered image, our focus tends to cascade over the entire image and we are not focusing on a single object, which is not

Select:	To:
	true in real life. You should only use Converged when an object is at the center of the screen with no scene elements at the render borders on either the left or right camera frustum.
Off-axis	Compute the convergence plane by shifting the frustum using camera film back. This is the safer way to compute stereo image pairs and avoids keystone artifacts. Off-axis is the default setting.
Parallel	Create a parallel camera setup where there is effectively no convergence plane. This is useful for landscape settings where objects exist at infinite focus.

Interaxial Separation field Displays the distance between the left and right cameras, in pixels. Editable.

Zero Parallax field Displays the distance on the camera view axis, in pixels, where the zero parallax plane occurs (the point where objects appear off screen). Objects in front of the zero parallax plane have negative parallax. Objects behind the zero parallax plane have positive parallax. Editable.

NOTE In general, your object should be behind the zero parallax plane. In other words, the camera distance should be greater than the zero parallax plane value. The zero parallax value, the camera separation, and focal length are all used to determine the shift that must be applied to film back on the respective left and right cameras. The zero parallax distance is enabled only when Stereo Mode is set to Off-Axis or Toe-In.

Zero Parallax Plane button Enable to display the zero parallax plane.

Zero Parallax colour pot Displays the colour used for the zero parallax plane. Editable.

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

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

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

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

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

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

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

Film Back Tab



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

Film Gate box Select a preset film frame format type. This action automatically sets the corresponding Camera Aperture, Film Aspect Ratio, and Lens Squeeze Ratio. To set these attributes individually, set Film Gate to User. The default setting is User.

Camera Aperture X field Displays the width of the camera Film Gate setting, measured in inches. This setting has a direct effect on the camera's angle of view and automatically updates Film Aspect Ratio. Editable.

Camera Aperture Y field Display the height of the camera Film Gate setting, measured in inches. This setting has a direct effect on the camera's angle of view and automatically updates Film Aspect Ratio. Editable.

Film Aspect Ratio field Displays the ratio of the camera aperture width versus height. Modifying this field automatically updates the Camera Aperture fields. Editable.

Lens Squeeze Ratio field Displays the amount horizontal compression that is applied to the image. Used with some cameras (for example, anamorphic cameras), which compress the image horizontally to record a wider aspect ratio image onto a square area on film. Editable.

Film Roll Value field Displays the amount of rotation applied to the film back. The rotation occurs around the specified pivot point. Used to compute a film roll matrix, which is a component of the post-projection matrix. Editable.

Film Roll Rotation Order box Select how the roll is applied with respect to the pivot value.

Enable:	**To:**
Rotate-Translate	First rotate the film back, then translate it by the pivot point value.
Translate-Rotate	First translate the film back, then rotate it by the film roll value.

Pre Scale field Displays the artificial 2D camera zoom that is applied before the film roll. Used in 2D effects. Editable.

Post Scale field Displays the artificial 2D camera zoom that is applied after the film roll. Used in 2D effects. Editable.

Film Fit Resolution Gate box Select the size of the resolution gate relative to the film gate (Film fit). If the resolution gate and the film gate have the same aspect ratio, then the Film Fit setting has no effect.

Select:	To:
Fill	Fit the resolution gate within the film gate.
Horizontal	Fit the resolution gate horizontally within the film gate.
Vertical	Fit the resolution gate vertically within the film gate.
Overscan	Fit the film gate within the resolution gate.

Film Fit Offset field Displays the offsets of the resolution gate relative to the film gate either vertically (if Film Fit Resolution Gate is Horizontal) or horizontally (if Film Fit Resolution Gate is Vertical). Film Fit Offset has no effect if Film Fit Resolution Gate is Fill or Overscan. Editable.

Film X Offset field Displays the horizontal offset, in pixels, of the resolution gate and the film gate relative to the scene. Changing the Film X Offset produces a two-dimensional track. Editable.

Film Y Offset field Displays the vertical offset, in pixels, of the resolution gate and the film gate relative to the scene. Changing the Film Y Offset produces a two-dimensional track. Editable.

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

Film X Translate field Displays the artificial 2D horizontal camera pan. Used in 2D effects. Editable.

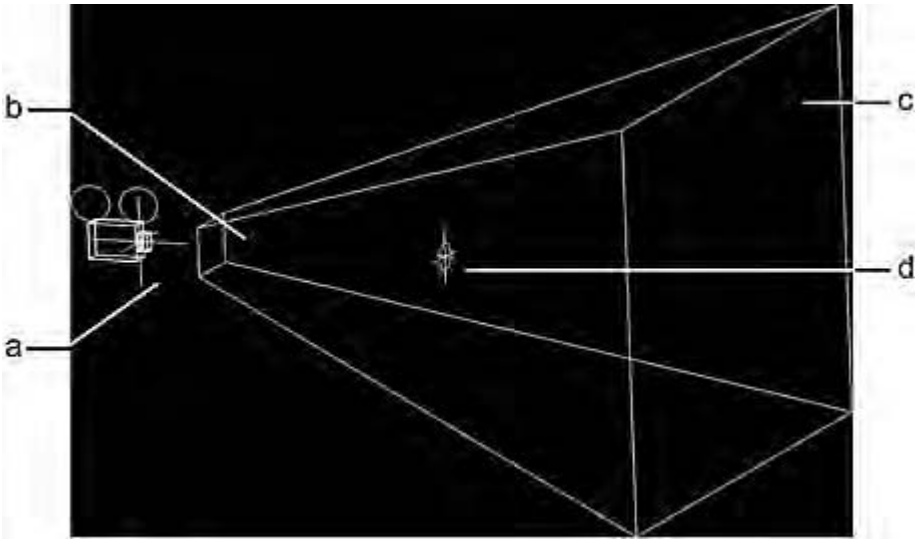
Film Y Translate field Displays the artificial 2D vertical camera pan. Used in 2D effects. Editable.

Film Roll X Pivot field Displays the horizontal pivot point from the center of the film back, which is used during the rotating of the film back. Used to compute the film roll matrix, which is a component of the post projection matrix. Editable.

Film Roll Y Pivot field Displays the vertical pivot point from the center of the film back, which is used during the rotating of the film back. Used to compute the film roll matrix, which is a component of the post projection matrix. Editable.

Working with the Frustum

The volume of space viewed by the camera is called the frustum. The frustum is, in effect, a viewing pyramid. The camera is located at the apex of the pyramid, and the far clipping plane forms the base. The pyramid may be truncated by the near clipping plane. The point of interest, or aim, is the target at the center of the camera's view.

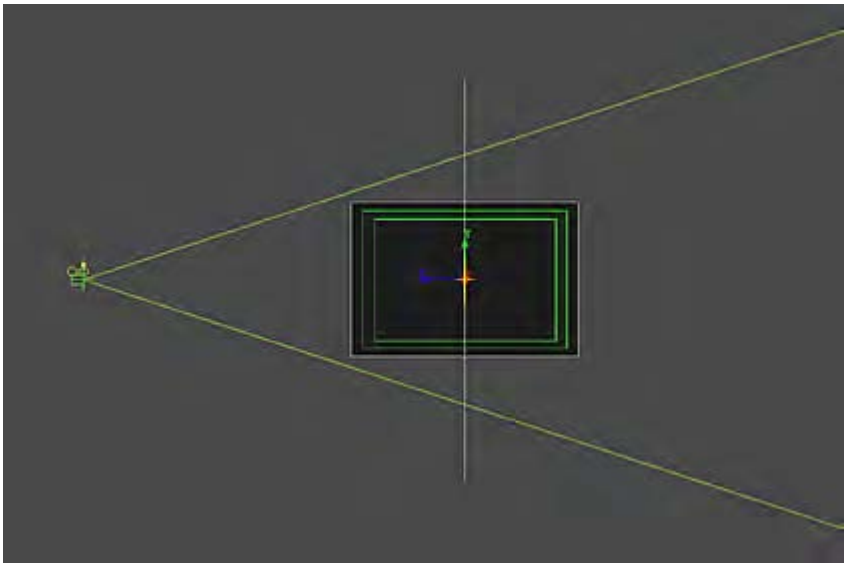


(a) Camera (b) Near clipping plane (c) Far clipping plane (d) Aiming target

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

To see the camera and frustum:

1. From the View box, select Side.
2. In the image window controls, click  to zoom out from the scene.
3. Enable the Pan button, and pan around the scene until you see the camera icon.

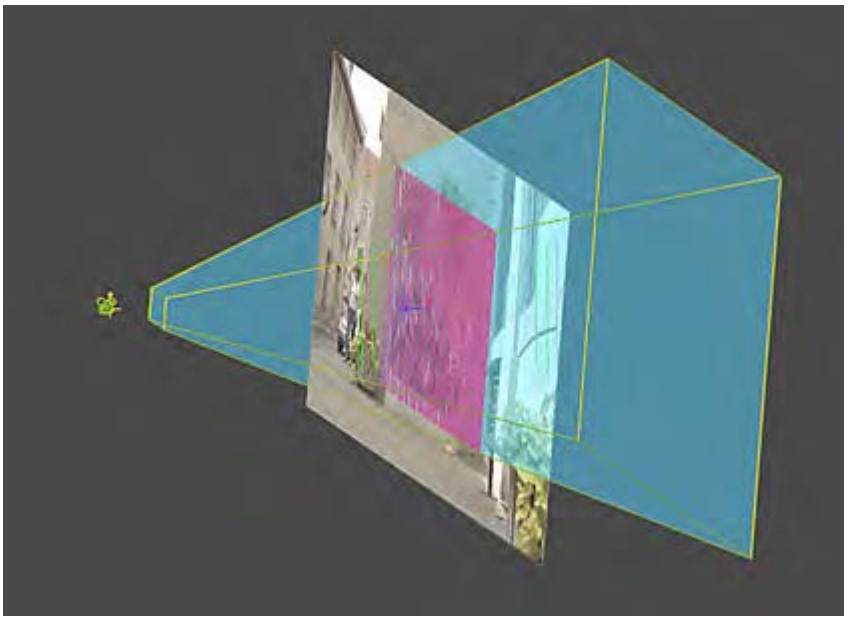


4. From the Camera menu, click the Aim/Up tab.
5. Ensure that Camera Type is set to Aim, or Aim and Up.
6. Drag the Roll field until you see the four sides of the frustum.
7. On the Stereo tab, enable Zero Parallax Plane and Safe Stereo Viewing.

   The safe viewable volume of the frustum, and the zero parallax plane area appear and are denoted with a transparent colour. The aiming target is centered at zero parallax.

8 Change the position of the near and far clipping planes to alter the depth of the frustum. See Moving the Near and Far Clipping Planes (page 666).

9 Change the position of the camera to alter the orientation of the frustum. See Moving the 3D Camera (page 667).



10 Change the position of the camera's aim to alter the orientation of the frustum. See Adjusting the Aim of the 3D Camera (page 667).

11 Adjust the angle of view to adjust the size of the objects as viewed by the camera. See Adjusting the Field of View (page 668).

Near and Far Clipping Planes

The camera frustum is determined by six clipping planes: the left, right, top, bottom, near, and far clipping planes. The depth of the frustum is affected by the near and far. The values for these channels are expressed in pixels relative to the position of the camera.

Set the Near and Far clipping planes to the lowest and highest respective values that produce the desired result. If the distance between the near and far clipping planes is much larger than is required to contain all the objects in the scene, the image quality of some objects may be poor.

TIP Objects that you want to render are usually within a certain range from the camera. Setting the near and far clipping planes just slightly beyond the limits of the objects in the scene can help improve image quality.

The ratio of far:near clipping planes determines the depth precision. Try to keep that ratio as small as possible for better results. Since most of the depth precision is concentrated around the near clip plane, try to avoid a lot of detail on distant objects.

Moving the Near and Far Clipping Planes

To move the near and far clipping planes:

1 From the View box, select Side or Top, depending on how your camera is positioned.

Either of these views profiles the camera so that the near and far clipping planes are clearly visible.

2 From the Camera menu, on the Basics tab, enter a value in the Near field to edit the position of the near clipping plane.

The value in the Near field corresponds to the position of the near clipping plane, in pixels, from the front of the camera. The farther away that the near clipping plane is placed, the higher the value. The default value is 1. Any object between the camera and the near clipping plane is outside the camera frustum and does not get processed in the final result.

3 Enter a value in the Far field to edit the position of the far clipping plane.

The value in the Far field corresponds to the position of the far clipping plane, in pixels, from the front of the camera. The lower the value, the closer the far clipping plane is placed. The default value is 10000. Any object positioned behind the far clipping plane is outside the camera frustum and does not get processed in the final result.

Moving the 3D Camera

Objects in the scene can be recorded from an arbitrary position as determined by the orientation of the camera in world space. The position of the camera can be moved left or right, up or down, or closer or farther from the central point of interest. The values for camera position are expressed in pixels relative to the aiming target.

To change the position of the 3D camera:

1 From the View box, select Front, Side, or Top, depending on how your camera is positioned, to provide a clear view of the camera motion.

2 From the Camera menu, on the Basics tab, modify the Position X, Y, and Z fields.

You can also animate the camera using keyframes and the Channel Editor.

Adjusting the Aim of the 3D Camera

The aiming target is the point in world space at which the camera is directed. The aiming target is always at the centre of the camera's frustum. Changing the position of the camera's aim causes the orientation of the frustum to change. You can take advantage of the relationship between the aim and the frustum to make the camera follow a moving object. To do this, animate the aim while keeping the camera in a fixed position. You can adjust the aim only with an Aim, or Aim and Up camera.

To adjust the aim of the 3D camera:

1 From the View box, select Front, Side, or Top, depending on how your camera is positioned, to provide a clear view of the camera motion.

2 Do one of the following:

- From the Camera menu, on the Aim/Up tab, modify the Aim X, Y, and Z fields.
- Move the aiming target by dragging.

You can also animate the aim using keyframes and the Channel Editor.

Adjusting the Field of View

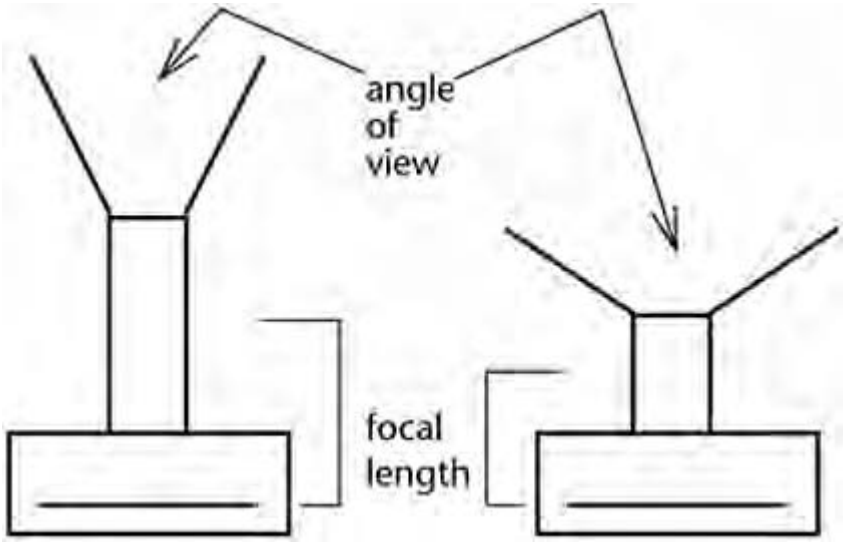
For every shot, you can decide how big an object appears in the frame, for example, whether a shot includes an entire character or just its head and shoulders. This is directly controlled by adjusting the field of view.

The field of view is the amount of a scene (measurable in terms of area) that can be viewed by the camera. The field of view can be altered by changing the angle of view or the distance between the camera and the subject. See Moving the 3D Camera (page 667).

About the Angle of View

The angle of view, or more accurately, the angular field of view, is the amount of a scene (measurable in terms of an angle in degrees, originating at the camera position) that can be viewed by the camera. The angle of view is not affected by changes in the distance between camera and subject. It can be modified directly, in the FOV field, or indirectly, via the Focal Length field.

The angle of view and the focal length are inversely proportional. As you extend the camera's focal length, the field of view gets narrower. As you shorten the focal length, the field of view gets larger.



Playing with the relationship between distance and angle of view affects the focus and perspective of objects in the scene that can be used to creative advantage.

For example, a pair of objects that are placed a distance apart from each other, but in line with the camera, can appear differently when changing these two parameters.

If these objects are viewed by a camera at long range, but using a lens with a high focal length (narrow angle of view), they will appear large in the frame and in equal focus, and will seem to be located on the same plane.

These same objects, when viewed at close range, but using a lens with a low focal length (wide angle of view), will appear to also fill the frame, but the size difference between them will be exaggerated (the foreground object will appear much bigger than the background object) and there will be a very noticeable focus difference.

Focal Length

The focal length of a lens is the distance from the center of the lens to the film plane. The shorter the focal length, the closer the focal plane is to the back of the lens. Focal length is usually expressed in millimeters.

The object's size in the frame is directly proportional to the focal length. If you double the focal length (keeping the distance from the camera to the object constant), the subject appears twice as large in the frame. The size of the object in the frame is inversely proportional to the object's distance from the camera. If you double the distance, you reduce the size of the object by half in the frame.

Adjusting the Angle of View

To adjust the angle of view:

1. From the Camera menu, ensure that the Basics tab is active.
2. If you are using degrees for angle of view, enter the value directly in the FOV field.

   Increasing the FOV value widens the frustum and decreases the Focal Length. Decreasing the FOV value narrows the frustum and increases the Focal Length.
3. If you are using different lens sizes, enter a value in the Focal Length field.

   Increasing the Focal Length narrows the frustum and decreases the angle of view. Decreasing the Focal Length widens the frustum and increases the angle of view.

Importing and Exporting 3D Cameras

Flame Premium supports the import of Alembic 3D data, and the import and export of 3D data saved in the FBX 3D format. These formats provide a means for exchanging 3D data for scene compositions—such as cameras—between tools and packages developed by different manufacturers.

You can either import a camera you created and edited in Action, or import one from another 3D application. Once a camera is imported, you can edit its parameters, change its animation, and then export it back to the application it came from.

NOTE Some parameters in Maya are not supported in FBX, such as Shear, FilmPostScale, and CamScale. Also, many parameters that are animatable in Maya and Flame Premium are not supported as animation curves in FBX. The 3D Camera does not support depth of field.

Export a 3D camera from Action to FBX format, which can be used later by any other FBX-compatible application.

To export a 3D camera from Action to FBX format:

1. Select the camera that you want to export.
2. In the 3D Camera menu, click Export.



The Export Camera file browser appears.

3. Set the FBX Scale and Export Options, as needed.
4. Navigate to the location where you want to export the camera animation.
5. Enter a name for your exported camera in the file field.
6. Click Save.

Export FBX Cameras Settings

Pixels to FBX Units field Displays the scaling factor to apply to the exported FBX file to be used in the 3D application. Use the Units box to select the unit of measurement. Editable.

Units box Select a unit of measurement to apply to the exported FBX file.

Rotate Axis button Enable to rotate the 3D model by 90 degrees on the X-axis so that it is compatible with the target's coordinate system.

Export Axes button Enable to export the animated axes present in the Action scene.

Export Point Cloud button Enable to export the 3D point cloud created by the Analyzer.

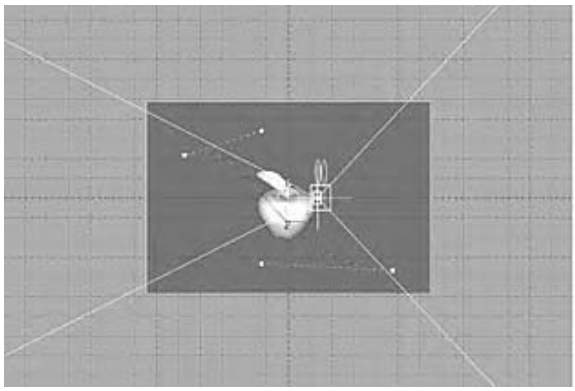
Bake Animation button Enable to add a keyframe at every frame of the exported FBX camera file.

Camera, Working, and Orthographic Views

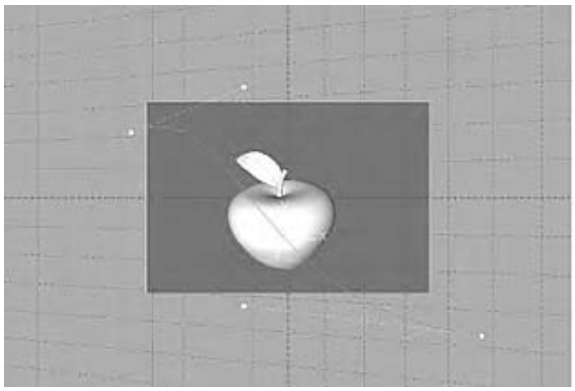
You can view the scene from various angles and display multiple views of these angles simultaneously. This is helpful in setting light sources, camera angles, stereo parameters, and animation keyframes more accurately.

You can view the scene from Camera view, the Working view, and three orthographic views. In Camera view, an object becomes smaller as it moves farther away from the camera. Working view is very similar to Camera view, except that no camera settings are affected. Working view is useful for trying out different settings and positions without actually making changes that can affect your cameras. In Orthographic view, an object remains the same size, regardless of its distance from the camera. Orthographic views are more helpful for aligning objects.

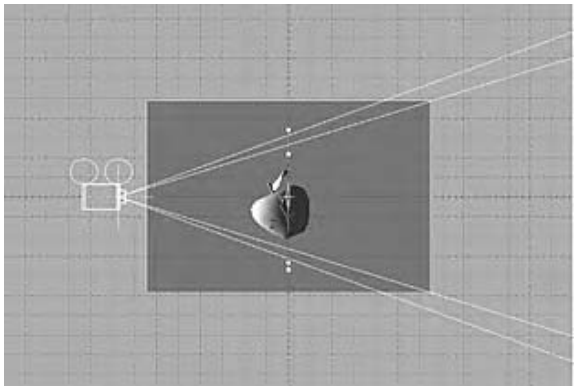
The following figures illustrate the different angles by which the scene can be viewed. The scene in this example contains a grey back clip and the 3D model of an apple.



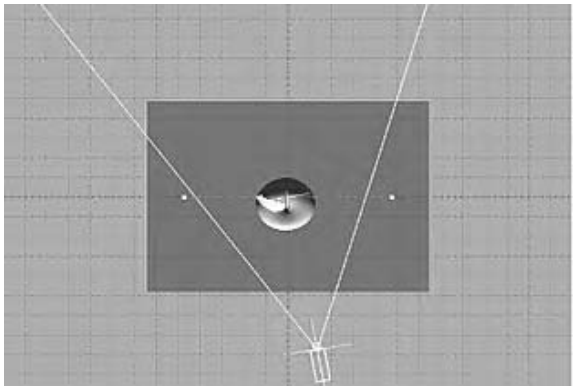
Front view



Camera view



Side view

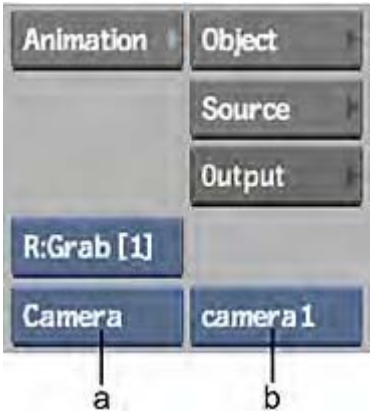


Top view

Setting Camera, Working, and Orthographic Views

To set camera, working, and orthographic views:

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



(a) View box (b) Camera box

Select:	To:
Camera	View the scene in Camera view. This is the scene as viewed by the camera eye. In other words, your field of vision in world space is equivalent to the viewing frustum of the camera. The size of objects depends on their distance from the camera eye.
Top	View the scene as if you are positioned on the positive Y-axis. This is an orthographic view; there is no perspective deformation.
Side	View the scene from the side, as if you are positioned on the positive X-axis. This is an orthographic view; there is no perspective deformation.
Front	View the scene as if your line of vision is directed into the camera eye. This is an orthographic view; there is no perspective deformation.
Working	View the scene just as in Camera view. Try out different positions and adjustments without affecting the camera settings.

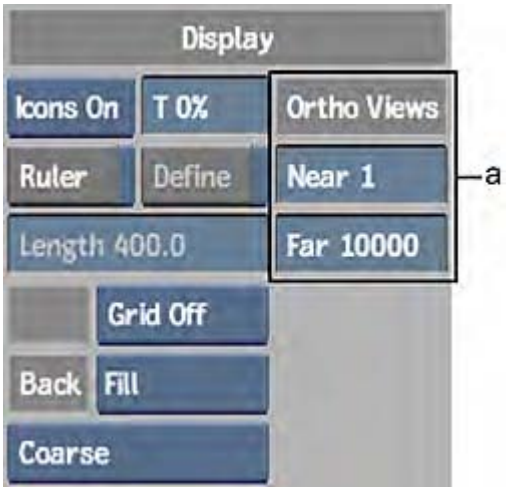
2 If you have multiple cameras in the scene, use the Camera box to define which camera is used for the Camera view in the image window. The camera selected in this box is not necessarily the camera used to process the scene.

3 Use the Working view if you want to view the scene using different adjustments, without saving any of the settings to a camera setup. For this reason, no parameters are animatable.

4 Adjust the view with the Ortho Views controls, if needed.

When using an orthographic view, you may notice that parts of the object you are viewing are getting cut off. Adjust the near and far ortho views. You gain more space to view the object, but lose some viewing precision.

In the Display section of the Action Setup menu, adjust the Near and Far fields.



(a) Ortho Views in Setup menu

NOTE The Ortho Views parameters in the Action Setup menu are only for viewing objects, and cannot be animated or saved. The Near and Far fields in the Camera menu are used to set clipping planes. See Moving the Near and Far Clipping Planes (page 666).

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

Modifying the Camera

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

To gesturally modify the camera:

1 Make a selection in the Edit Mode box.



Select:	To:
Track	Move the camera lens and look-at point.
Tilt	Tilt the camera up and down by moving the look-at point. Also changes the camera roll. This option only modifies Target cameras.
Roll	Rotate the camera on the Z-axis. This option only modifies Target cameras.
Orbit	Rotate the camera lens around the look-at point.
FOV	Move the camera field of view.

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

2 Drag the cursor in the image window.

The camera is modified. Related Camera menu controls are updated to reflect the changes.

Zooming In and Out

Use the Zoom option to move the camera eye toward or away from the point of interest. While viewing the scene in Camera view, zoom in or out from the point of interest to move the camera eye closer to or farther from the point of interest. In Top, Side, or Front view, you can enlarge or reduce the scene in the image window without affecting the camera. Zooming has no effect in Schematic view.

To zoom the camera:

1 From the Edit Mode box, select Zoom.

2 Place the cursor in the image window.

The cursor changes to a magnifying glass.

3 To zoom in, drag the cursor to the left. To zoom out, drag the cursor to the right.

Optimizing Attributes of the 3D Camera

Here are some general guidelines for tweaking the stereo attributes of the 3D camera:

- Many parameters are relative to each other, and can be scaled and changed upon import using the Auto Fit in Scene or FBX Unit to Pixels settings.
- Tweak the Interaxial Separation to move the cameras closer to or farther away from one another.

  NOTE You should re-adjust your Interaxial Separation if you change your output device, since the settings for one display method may differ from another.

- Increase the Zero Parallax to move objects further away from the camera. The 3D effect becomes less pronounced in this case. Decrease your Zero Parallax to move objects closer to the camera. You can see more depth if you do this.

  The stereoscopic effect is the most realistic when the Zero Parallax Plane is in between the two objects.
- You may need to re-adjust your camera attributes if you change the resolution of your output device.
- You can also increase the Far Clip Plane to increase the depth of the camera.
- In Anaglyph viewing mode, the red/cyan colors for objects are swapped depending on whether they are behind or in front of the parallax plane. For objects behind the zero parallax plane, they appear in cyan/red. For objects in front of the zero parallax plane, they appear in red/cyan.

About the Analyzer

Use Action's Analyzer to compute the path of live-action camera and object motion in 3D space. Using the calculated position and motion of the virtual camera, you can match image sequences perfectly, placing

any element in the scene. The perspective of the element you place in the scene changes with the perspective of the background as the camera moves. The virtual camera motion is intended to be identical to the motion of the actual camera that shot the scene.

Use the following workflow as a quick start guide to the Analyzer. Follow the links for more detailed information.

Step 1

Select media (mono or stereo) to analyze and add an Analyzer node.

Optional steps:

- Perform a lens correction.
- Adjust the four corners of the perspective grid to set the focal length.
- Add manual trackers to obtain a more predictable and consistent result.

NOTE Adding manual trackers can be initiated post-process.



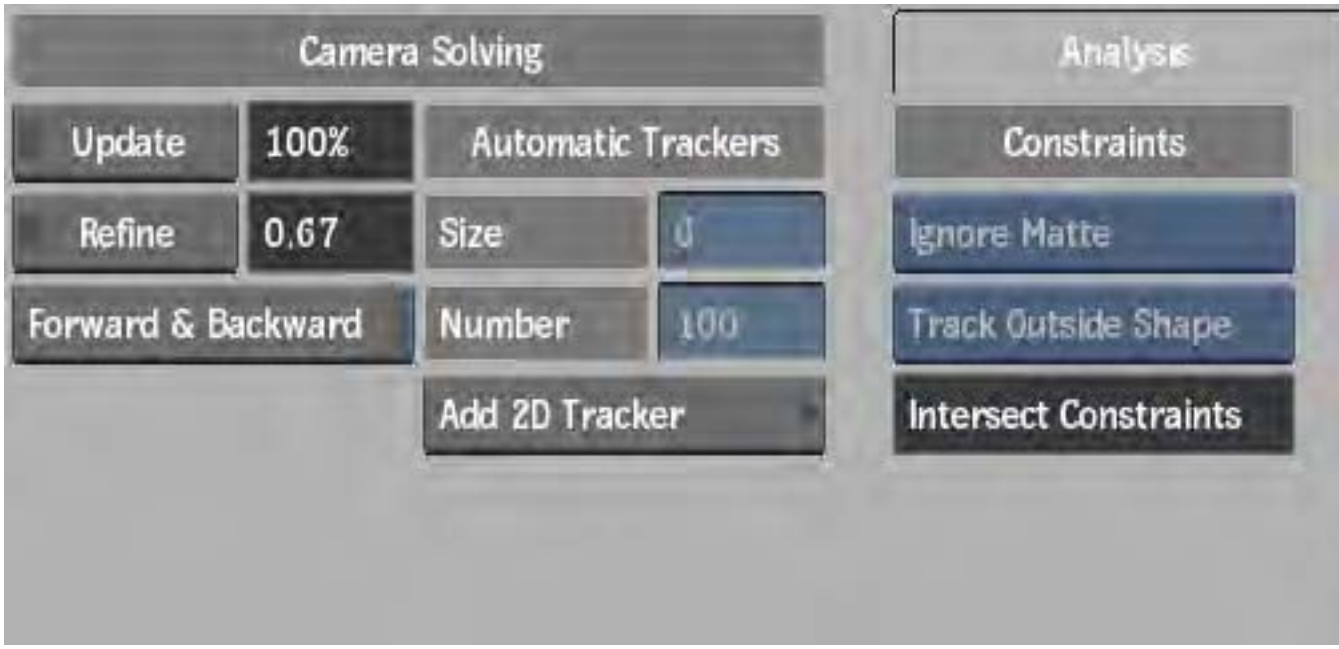
Tracking using the Analyzer Node (page 678)

Step 2

Perform Camera Tracking in the Analyzer menu.

Optional steps:

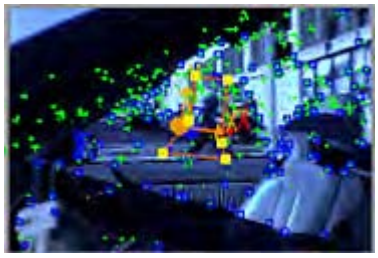
- Add mask constraints to moving areas or areas not wanted in the analysis.
- Add properties of the camera that shot the footage to be analyzed.



Camera Tracking (page 680)

Step 3

Fine-Tune and recalibrate or refine the camera tracking analysis. This step is optional depending on the results of your initial analysis.



Fine Tuning the Analysis (page 689)

Step 4

Create a Point Cloud of selected points after the analysis.



Converting the Camera Analyzer Results (page 692)

Step 5

Perform Object Tracking. If needed, after camera tracking, you can track moving objects in the scene — such as inside the masks that were not tracked in the camera tracking analysis.



Object Tracking (page 693)

Preparing to Analyze

A good analysis result is often footage dependant, therefore you may need to perform object tracking after the initial camera tracking.

Here are some things to keep in mind that can improve your analysis:

- The Analyzer works best when tracking an image sequence that has a moving camera or environment. Masks should be used to isolate the background and the objects in the scene that have different motions. Masking objects cause them to be ignored in the tracking process. You should also mask any logos or watermarks in your image sequence.
- The analysis is based on point-like and corner-like image features. Balls, spheres, lines, and smooth surfaces are not considered.
- Crop out any black areas around your image (such as letterbox areas). When cropping, it is very important to keep the optical centre of the image in the centre, that is, the crop should be symmetrical in both dimensions.
- 3D tracking analyses each frame extensively and with high resolution clips, the process can be lengthy. Unlike 2D tracking, however, you are not required to analyse full-resolution clips. In many cases, analyzing proxies will produce acceptable 3D tracking results quicker than the time that would be required for the full-resolution clips.
- If you have information about the camera that shot the scene you want to track, such as the focal length of the lens and the film back size, it is recommended you specify these details to improve the results of the tracking analysis.

Obtain better tracking results by adding manual trackers

Add manual trackers to produce a more consistent track result. Each tracker consists of an inner reference box, which establishes the reference point for the tracking, and an outer tracker box, which follows the movement of the reference point. You can explicitly choose specific trackable features in the image (see How the Stabilizer Works) (page 860).

While trackers can be added both pre- or post-processing, it is recommended to do so pre-process if there is going to be a difficult sequence to track. Add trackers post-process to improve the results of the automatic processing. If added post-process, click the Update button.

When tracking, the more track points you have in each frame, the smoother and more accurate the tracking results. As you place the trackers on the image, consider the following guidelines:

- Scatter the trackers by placing them on markings, corners, and shadows in the widest area possible. For example, avoid placing all trackers on the floor; place them on walls and other objects in the scene, as well as on the floor.
- Create a sense of depth by positioning trackers on points that lie in different planes, as well as on points located in the foreground and the background of the sequence.
- Avoid tracking points such as highlights or a point where the foreground and a background object meet, as they do not represent physical 3D points.
- Balance the number of trackers within each frame of the sequence so that as you move through the clip, some points leave the frame and other points appear in the frame. However, maintain a balance so that too many points do not leave or enter the frame at the same time.
- Avoid positioning the trackers in uniform areas or on linear edges where the track points may slide along the edge.

To add manual trackers:

1. In the Analyzer menu, click the Add 2D Tracker button.

   The Stabilizer tool opens and automatically displays the Analyzer media
2. Click Add Tracker.

   A tracker will appear on the middle of the view.
3. Place the tracker over a trackable feature.

   Add as many as you need to improve the tracking.

   Manually added trackers will be assigned a number, incrementing from the first tracker. Manual trackers can then be selected by either:

   - Selecting them directly on the screen or,
   - Selecting the corresponding Tracker button.
4. Click Analyze.
5. Click Return when done to return to the Analyzer menu.

   The trackers will now be visible in the Analyzer view in a different color from the automatically generated trackers.

NOTE Manually added trackers have persistence. They cannot be filtered, and they will not be deleted in a reset. However a tracker can be manually deleted by first selecting it, and in Fine Tuning, clicking Delete.

Adding an Analyzer Node

When you add a Stereo or Mono Analyzer in Action, a number of nodes are added to the schematic.

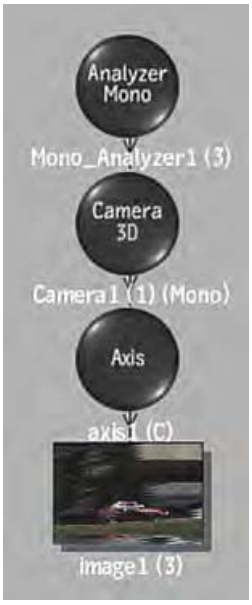
To add an analyzer to a scene:

1. From the Media list, select the media that you want to analyze. If you want to analyze a stereo clip, select the left clip, then press **Ctrl** and select the right clip.
2. Do one of the following:
   - Drag the Analyzer Mono or Analyzer Stereo node from the node bin and place it in the schematic.

- Double-click the Analyzer Mono or Analyzer Stereo node. You do not need to be in Schematic view to add a node in this manner.

The image window automatically switches to an analyzer view to help you view the interactive result of your analysis.

In the schematic view, you can see that an Analyzer node is added to the scene. The number in brackets next to the name of the Analyzer node indicates the media used for the analyzer. The Analyzer node is parented to a 3D Camera node, which is synced to the analysis of the Analyzer node, as well as an Image (or Stereo Object) node with an axis.

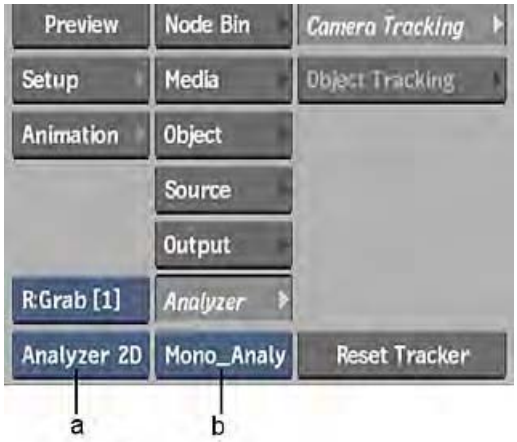


3 To display the Analyzer menu, double-click the Analyzer node in the schematic, or click the Analyzer button.

In the Analyzer menu, you can access Camera and Object tracking settings.

Viewing Analyzers

When you add a mono or stereo analyzer node to your scene, the image window automatically switches to an analyzer view to help you view the interactive result of your analysis. There are other views available from the View box for working with analyzers.



(a) View box (b) Analyzer View box

View box Select an option to set the view in the image window.

Select:	To Display:
Analyzer 2D	2D tracks after analysis. By default, these are displayed as blue squares in the image.
Analyzer 3D	3D points after a mono analysis. By default, these are displayed as green crosses in the image. The F7 keyboard shortcut is a toggle keyboard shortcut; each time you press it, the view changes from Analyzer 2D to Analyzer 3D.
Analyzer 3D Left or Analyzer 3D Right	3D points from the left or right eye perspective after a stereo analysis. The F7 keyboard shortcut is a toggle keyboard shortcut; each time you press it, the view changes from Analyzer 2D to Analyzer 3D Left to Analyzer 3D Right.
Analyzer Wk)	The Analyzer Working view. The Analyzer Working view provides you with the ability to view your analyzed scene using a working camera, which you can modify to view the scene without modifying the tracked virtual camera. This can be helpful to orbit the scene in order to view the depth of 3D points, for example.

Analyzer View box Select which analyzer to display in the image window.

If your scene has multiple analyzers, use the Analyzer View box to select which analyzer to display in the image window. This setting is available when the image window is in Analyzer 2D or Analyzer 3D view.

Camera Tracking

You can perform automatic 3D tracking based on the camera properties of the device used to acquire the image sequence you want to track. You can specify these properties, or let the analyzer automatically detect the best solutions for the analysis. Even if you want to perform object tracking on specific moving areas of the scene, you also start with an analysis using camera tracking to create a properly tracked camera.

NOTE To improve the results of the track, consider adding 2D manual trackers either pre- or post-process (see Obtain better tracking results by adding manual trackers (page 678)).

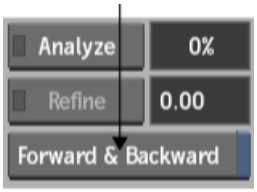
To create a 3D camera track:

1 Add an Analyzer Mono or Analyzer Stereo node, as explained in Adding an Analyzer Node (page 678).

2 In the Camera Tracking menu, set Analysis options. For example, decide if you need to use a matte or GMask in the analysis. In an image sequence of a busy street, you can create a matte or mask of moving elements (such as cars and people) to isolate this area from the analysis. See Analysis Settings (page 686) for details of each Lens Correction, Constraints, and Camera setting.



3 Enable Forward & Backward to track the image sequence backward after the forward tracking has completed. This option takes longer, but you may get better results.



4 If needed, adjust the size and number of trackers.

Smaller trackers can speed up the calculation, while larger trackers make the analysis more robust with regard to image noise and variations. A general rule is to increase the size of the trackers when tracking high-resolution footage (2K or larger) that contains more noise.



5 Click Analyze.



You can see a progress indicator beside the Analyze button. You can interrupt the analysis and resume it by clicking Analyze again. Most of the time, tracking occurs in the background, allowing you to continue working while tracking. In Batch, you can use connected input and matte clips as the media to be tracked. In this case, tracking becomes a foreground process.

After tracking has completed and you press Confirm, the Analyze button changes to Update, and you can see the 2D tracks (the blue squares in the following example) and 3D points (green crosses) in your image, provided that you are in Analyzer 3D view.



Image courtesy of Behavior Communications Inc.

The 3D camera connected to the analyzer node synchronizes to the results of your 3D tracking, and any further changes you make to the 3D track are reflected in this camera.

6 If you are satisfied with the results of the tracking analysis, proceed to Converting the Camera Analyzer Results (page 692). If you want to tweak your track results, see Fine Tuning the Analysis (page 689).

NOTE As a general rule, a generated Refine field value between 1 and 1.5 is considered a good quality track.

Set the focal length by adjusting the perspective grid

The tracking analysis algorithm can make a best estimate of the focal length for the camera without any user input (see Analysis Settings (page 686)). In order to refine the algorithm's analysis you can use a perspective-based grid to define the focal length. The focal length determined through the perspective grid is then used as part of the analysis. If you do not use this perspective grid or manually enter a value for the focal length, the algorithm guesses at an estimate of the focal length.

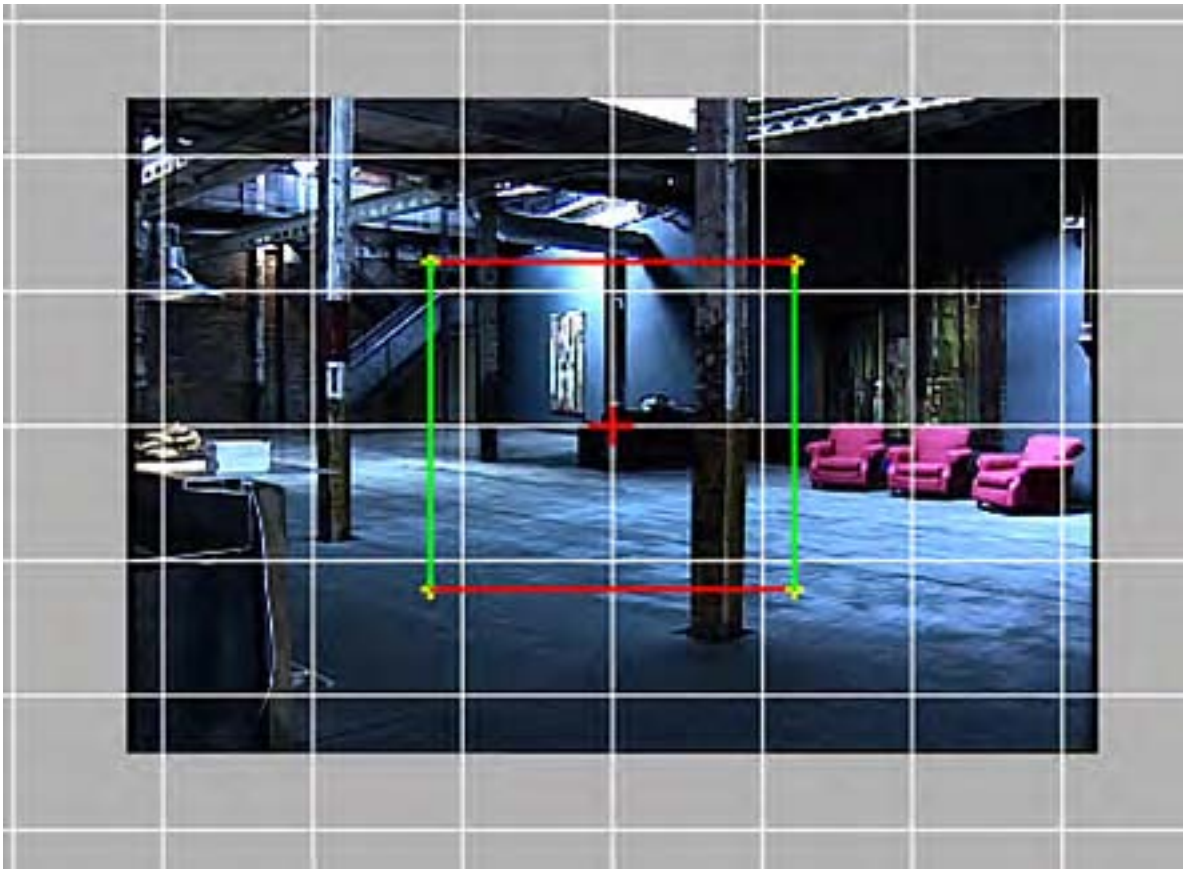
To set the focal length:

1 With the Analyzer object selected in the schematic, select the Analyzer 3D view from the View box.

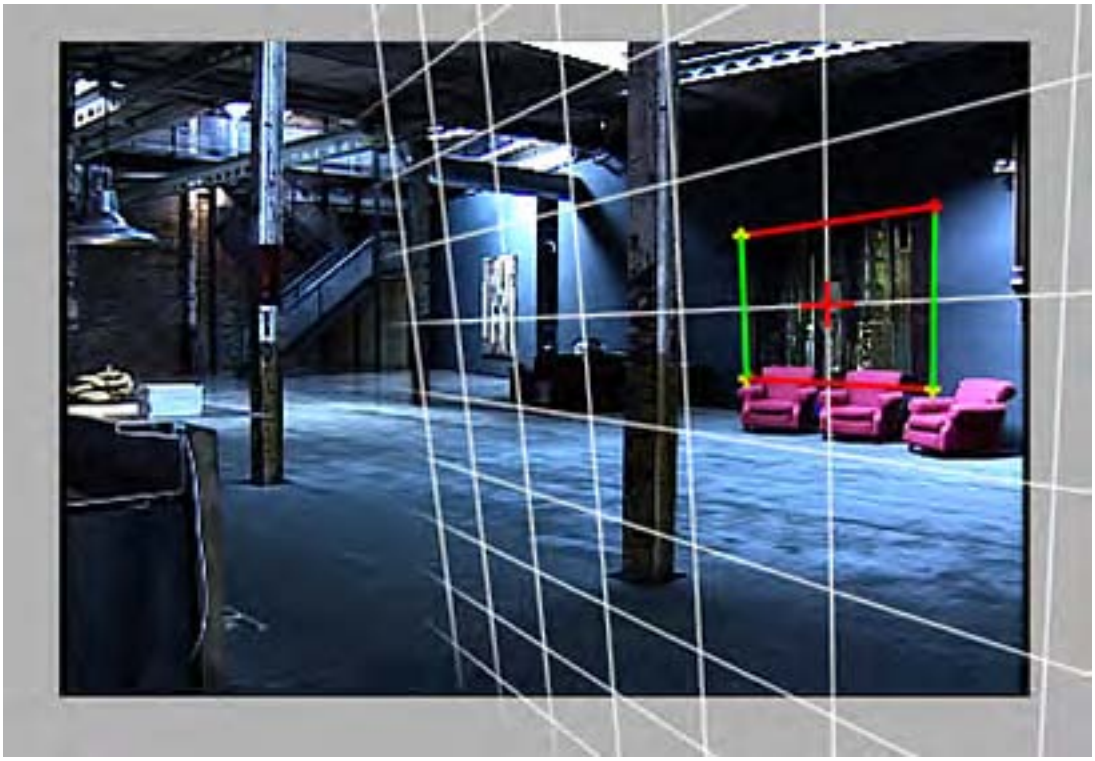
2 From the Analysis tab in the Analyzer, select Perspective Based from the Focal Length box.



A perspective grid appears in the view with the selected media.

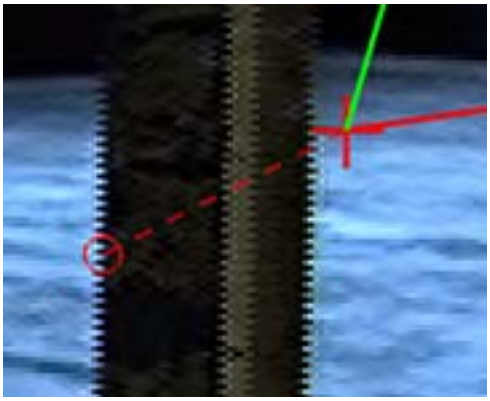


3 While in Analyzer 3D view, drag each of the four corners of the rectangle to the desired location to form a rectangular grid. The rectangle automatically aligns to the new perspective. The Range Error, which displays the error estimate for the focal length, should be small relative to the focal length (should be less than 10% of the focal length).



NOTE When dragging one of the corner points of the rectangle, if your desired location cannot be computed (for example, if a point goes past another point on the plane), the location is remembered with a dotted line and red circle. Once you move other corners, your original location may now become viable, and the rectangle and grid align properly.

(See Perspective Grid (page 492) for specific details on placing the corner points.)



4 If you want to add manual trackers pre-process do so now.

5 Click the Analyze button to start the tracking.

6 When the tracking analysis is complete, click Confirm.

Camera Tracking Settings

Camera Tracking Settings

Use the settings in the Camera Tracking menu to perform an automatic 3D track.

Camera Tracking button Displays the Camera Tracking menu, where you can track based on the settings of the camera.

Track button Tracks the image (when complete, Track button changes to Update).

Analyze Progress field Displays the progress of analyzing or updating. Non-editable.

Analyze button Click to delete all previous 3D points, and start tracking based on new information.

Refine button Click to use the current results of the track analysis as a starting point, and refine from this point. Click Refine again to stop the process.

Pixel Error Value field Displays a representation of the distance of 2D tracks from repositioned 3D points. Non-editable.

Forward & Backward button Enable to track the image sequence backward after the forward tracking has completed.

Size field Displays the size of the trackers. Editable.

Number field Displays the number of 2D tracks created by the analysis. Editable.

Reset Tracker button Resets all tracker data in the Analyzer menu.

Tracker Display Settings



Tracker Display button Enable to display 2D tracks in the image.

Tracker Display colour pot Displays the colour of the trackers. Editable.

Point Cloud Display button Enable to display 3D tracker points in your image.

Points Display colour pot Displays the colour of the points. Editable.

Transparency field Displays the transparency level of the trackers and points in the image. Editable.

Point Size field Displays the size of the 3D points in the image. Editable.

Grid box Select the type of grid to display in the scene. Use to position objects in the scene more accurately.

Select:	To:
Grid Off	Disable the grid.
Grid XY	Use a grid constructed on the X and Y planes.
Grid XZ	Use a grid constructed on the X and Z planes. The XZ grid is visible only when the camera is moved from its default position.
Grid YZ	Use a grid constructed on the Y and Z planes. The YZ grid is visible only when the camera is moved from its default position.

NOTE The Action grid is independent from the global grid in the Grids and Guides menu.

You can change the display of the perspective grid with the following settings:

Show Perspective button Enable to display the grid corners in the Analyzer view.

Grid button Enable to display the perspective grid in the Analyzer view.

Grid Colour pot Displays the colour of the grid lines. Click to change the colour.

Zoom field Displays the zoom factor of the magnifier. Editable up to 5x.

Analysis Settings

Use the Analysis settings before clicking Track. The Analysis tab of the Analyzer menu is divided into a number of sections.

Constraints Settings

Use the constraint settings pass information to the analyzer about any mattes or masks in the scene.



Matte Media box Select whether a matte is used for tracking.

Select:	To:
Ignore Matte	Not use a matte.

Select:	To:
Track Inside Matte	Use a matte to delimit the tracking results. White areas of the matte are considered for calculating the solution, and black areas are ignored.
Track Outside Matte	Use an inverted matte to delimit the tracking results. Black areas of the matte are considered for calculating the solution, and white areas are ignored.

GMask Option box Select whether to track inside or outside of an attached GMask shape.

While in an Analyzer view, you can add a GMask node and instantly draw the constraining mask. If added while in Analyzer view, the GMask and Axis nodes are automatically connected to the image as part of the Analyzer tree. If you want to use an existing GMask node, make sure that you parent the Surface (mono analyzer) or Stereo Object (stereo analyzer) to the Axis of the GMask in the schematic.

If the area to be constrained is moving, you can perform shape tracking from within the GMask menu. See Tracking a Mask (page 713).

Intersect Constraints field Displays that the intersection of the two constraints above are used as the tracking region. Non-editable.

Camera Settings

Before analysing the 3D motion, define your camera properties. The camera represents the device used to acquire the sequence of images you are tracking. For each camera parameter, you can choose to let the auto 3D tracker calculate the value automatically, you can specify the value yourself, or you can set a perspective grid to estimate the focal length (see Adding an Analyzer Node) (page 678). This information can help the automatic 3D camera tracker calculate better results.



NOTE You can set the camera properties before initial tracking, or after initial tracking when calibrating or refining the track.

Camera Type box Select the type of camera motion in the tracking shot.

Select:	To:
Free 3D Motion	Track a clip shot with a free-moving camera.
Pan and Tilt	Track a rotating clip shot with a camera on a tripod.
Auto Detect Motion	Automatically detect the camera type and track accordingly (default value).

Focal Length box Select Auto to let the 3D tracker calculate the camera values automatically. Select Manual to specify the values yourself. Select Perspective Based to use the perspective rectangle in the image to automatically determine the focal length of the camera.

Length field Displays the focal length of the camera, in millimeters. Editable.

Range Error field Displays the error of the estimate for the focal length. This should be a small number relative to the focal length. Non-editable.

Zoom button Enable to calculate the zoom value of the reconstructed camera for each frame.

Film Ratio field Displays the film aspect ratio of the camera. Non-editable.

Film Back box Select the film back size of the camera that shot the scene.

Width field Displays the width of the film back. Editable.

Film Back Units box Select the unit of measurement (inches or millimeters) for the film back width.

Lens Correction Settings

You can apply lens correction before or after your initial analysis. If you make any changes after analysis, you'll need to update or refine the track. While not necessarily required, performing a lens correction may give better tracking results.



Lens Correction Mode box Select Manual and set the Magnitude and Adjust fields if you want to apply lens correction during analysis.

Magnitude field Displays the magnitude of lens correction. Editable.

Adjust field Displays the level of secondary adjustment of lens correction. Editable.

Stereo Analysis Settings

The Stereo Analysis settings display information about the 3D camera after a stereo analysis is performed. These are locked fields, and are meant as informational only.



Stereo Mode field Displays the stereo mode of the 3D camera after a stereo analysis. Non-editable.

Interaxial Separation field Displays the interaxial distance between the left and right cameras of the 3D camera after a stereo analysis. Non-editable.

Zero Parallax field Displays the distance on the camera view axis (where the zero parallax place occurs) of the 3D camera after a stereo analysis. Non-editable.

Fine Tuning the Analysis

If the initial camera track analysis does not give desired results, you can use some or all of the Fine Tuning options to update and refine your track analysis. These procedures are not necessarily required, but depending on your image and the initial tracking, may give better tracking results.

Defining Real World Measurements

You can use the Real World Measures settings to select two points in your analyzed scene, and enter a measurement that represents the distance between the two points. Establishing real world units is helpful when exporting the point cloud through the 3D camera for use in a 3D application.



To define real world units in your analyzed scene:

1. In the Real World Measures section of the Fine Tuning tab, enable Define.
2. Select a point in the image, then press `Alt` and select a second point.

   A red line is drawn between the two points.
3. Disable Define.
4. In the Length field, enter a measurement of the length between the two points, in the unit of your choice (centimeters, inches, etc.). The Action scene scales to the distance you enter.

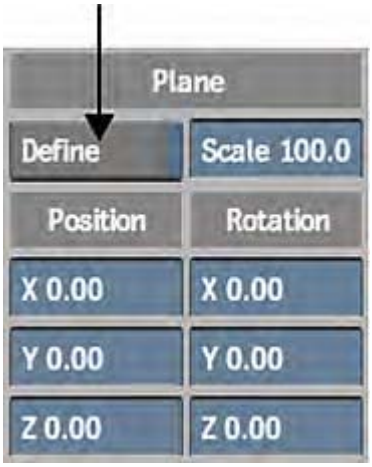
   If you export the point cloud through the 3D camera, you'll be able to specify the unit of measurement in the Export Camera menu (page 669).

Setting the Orientation

Although it is not mandatory to define a ground plane in your image, it helps orient the reconstruction of the cameras.

To define the camera track ground plane:

1. In the Plane section of the Fine Tuning tab, enable Define.



NOTE If you selected Pan and Tilt as the camera type (or if the Auto Detect switched to Pan and Tilt) the Plane Define button is not available. The rest of the settings in this group are still available, but only for orientation purposes.

2 Select a minimum of three points in the image that represent the plane of the X/Z axes, such as the ground, a table, or any flat surface.

The selected points appear as red squares with white crosses.

TIP You may want to disable the tracker temporarily and point display options (or raise the transparency level) to help you find and select the plane points.

3 Use the Position and Rotation controls to define the orientation of the ground plane.

4 Use the Scale field to specify the scale of the scene.

5 Disable Define.

Filtering trackers to fine tune the track analysis

Use the Filter settings to delete lower quality trackers.

To fine-tune the track analysis:

1 Use the Quality slider to adjust the number of good trackers kept. The higher the quality setting, more low quality trackers are selected, such as trackers that drift off their initial reference point. Click Delete to delete the selected trackers.



Trackers of lower quality may hinder the accuracy of the camera tracking.

NOTE After you have made a change that requires the 3D tracking analysis to be refined or updated, notice that the LED next to the Refine and Update buttons turns yellow. This signifies that a Refine or Update is required, but you do not have to perform it until you have completed your tracker selections.

2. Adjust the Short slider to select short duration trackers, that is, trackers that only track a feature for a few frames. Click Delete to delete the selected trackers, leaving the longer duration trackers intact.
3. You can manually select and delete trackers from the image that you feel are not tracking properly. Do one of the following:
   - To select an individual tracker, click the tracker, and then click Delete.
   - To select multiple trackers in the same area, `Ctrl`-drag a selection box over a series of trackers, and then click Delete.
   - With Delete mode selected in the Edit Mode box, select trackers in the image.

Refining or Updating the Track

Once you are satisfied with your fine tuning changes, you can refine or update your Analyzer.



To refine or update the 3D track:

1. Depending on the changes you have made, you can choose to refine or update the 3D track. Do one or both of the following:
   - Click Update.

     NOTE The update operation deletes all previous 3D points and starts over based on the new information. Depending on your footage, and how many trackers you added, deleted, or linked, multiple updates may yield different results.

   - Click Refine.

     The track analysis uses the current results as a starting point, and refines from this point.

Click Refine again to stop the process once an acceptable pixel error value is reached. The pixel error value is a representation of the distance of the 2D tracks from the computed 3D points.

TIP The refine process is footage-dependant, so your acceptable pixel error value may change depending on what is tracking. Since the refine process continues until you stop it, as a general rule, if the pixel error value does not change for a length of time (for example, 30 seconds), you can stop the refine process. The lower the pixel error value, the more accurate the reconstructed track is.

Fine Tuning Settings

Real World Measures Settings

Define button Enable to define a target length between two points in the image. Select one point, then press Shift and select the second point. When defining is complete, disable Define. Establishing real world units is helpful when exporting the point cloud through the 3D camera for use in a 3D application.

Length field Displays the measurement of the length between the two points, in the unit of your choice (centimeters, inches, etc.). Editable.

Plane Settings

Plane Define button Enable to define the ground plane in the image (when defining is complete, disable Define).

Plane Scale field Displays the scale of the ground plane. Editable.

X Position field Displays the position of the ground plane on the X axis. Editable.

Y Position field Displays the position of the ground plane on the Y axis. Editable.

Z Position field Displays the position of the ground plane on the Z axis. Editable.

X Rotation field Displays the rotation of the ground plane on the X axis. Editable.

Y Rotation field Displays the rotation of the ground plane on the Y axis. Editable.

Z Rotation field Displays the rotation of the ground plane on the Z axis. Editable.

Filter Settings

Quality field Displays the number of good trackers that are kept, as a percentage. Editable.

Short field Displays the percentage of short duration trackers (trackers that only track a feature for a few frames). Editable.

Delete button Deletes the selected trackers.

Converting the Camera Analyzer Results

When you are satisfied with the results of the 3D camera tracking analysis, you can convert the selected reconstructed points to a point cloud object or actual axes in your scene. The point cloud object is useful because you can easily snap objects to it.



To create a point cloud object or axes from the 3D camera tracking results:

1. In the Analyzer Mono node, select the points in the image that you want converted to a point cloud 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. Enable Fixed Camera if you want to generate a fixed camera and a moving 3D point cloud.

   Typically, 3D tracking results in a moving camera and fixed points, but you also can set the camera to be fixed and the points to move.

3 Do one of the following:

- Click Create Point Cloud

  Selected point are converted to a point cloud object with a parent axis. Double-click the newly created point cloud object to access its menu, where you can change display settings and enable snapping. See Using the Point Cloud Object (page 697).

- Click Create Axis

  Selected points are converted to axes with a parent axis. The axes synchronize to the results of your 3D camera tracking, and any further changes you make to the 3D track are reflected in these axes.

  NOTE If you want to apply settings manually to a synchronized axis, change its name so that it does not update automatically when 3D Track settings are changed. If you reset the 3D Track settings, the synchronized axes become regular Action axes.

4 Use the created point cloud or axes to view the reconstructed scene geometry or camera motion.

You can attach objects such as surfaces, 3D text, and 3D models to the new point cloud or axes to help position them in 3D space.

Output Settings

Fixed Camera button Enable to generate a fixed camera and a moving 3D point cloud.

Create Axis button Click to convert selected points to axes.

Create Point Cloud button Click to convert selected points to a point cloud object. Double-click the new point cloud node in the schematic to open the Point Cloud menu.

Object Tracking

If the image sequence you are tracking has multiple moving objects, you can perform object tracking to track these objects relative to the same camera. Since you perform camera tracking first, you can use the camera data generated from the camera tracking analysis. For example, you can perform a camera tracking analysis on the complete scene, then use masks or mattes to perform multiple object tracking passes focusing on various moving objects in the scene. Each result can be converted to separate point clouds or axes, but all results conform to the 3D camera synced to the original camera tracking.

NOTE To improve the results of the track, consider adding 2D manual trackers either pre- or post-process (see Obtain better tracking results by adding manual trackers (page 678)).

To create an 3D track analysis based on object properties:

1 Create a camera track analysis, as explained in Camera Tracking (page 680).

2 Click Object Tracking to switch to the Object Tracking menu.



3 From the Object Motion box, make a selection about the object you are tracking in relation to the camera.



Select:	To:
Free 3D Motion	Track an object moving independently from the camera.
Pan and Tilt	Track an object rotating around the camera, or far away from the camera.
Auto Detect Motion	Automatically detect the motion type of the object and track accordingly. For small objects, Auto Detect may not be able to establish the proper motion. In this case, select Free 3D Motion or Orbit Around Cam.

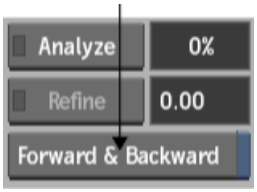
4 Set any needed constraints:



- Use the Matte Media box to select whether to use a matte for object tracking. If you select to track inside the matte, white areas of the matte are considered for calculating the solution, and black areas are ignored (vice-versa for tracking outside the matte).
- Use the GMask Option box to select whether to track inside or outside of an attached GMask node. This option is only available if a GMask node is attached to Analyzer tree. If you used a GMask constraint while camera tracking, the opposite option is automatically selected for object tracking.

The tracking analysis uses an intersection of the constraints, so you may choose to hide or disconnect gmasks if you want to perform a separate object track for each one.

5 In the Object Tracking section, enable Forward and Backward to track the image sequence backward after the forward tracking has completed. This option takes longer, but you may get better results.



6 Set the size of the automatic trackers in the size field.



7 Click Analyze.



Most of the time, tracking occurs in the background, allowing you to continue working while tracking. In Batch, you can use connected input and matte clips as the media to be tracked. If this case, tracking becomes a foreground process.

You can see a progress indicator beside the Track button. You can interrupt the analysis and resume it by clicking Track again. After tracking has completed and you press Confirm, the Track button changes to Update, and you can see the 2D tracks and 3D points in your image.

Filtering trackers to fine tune the track analysis

Use the Filter settings to delete lower quality trackers.

To fine-tune the track analysis:

1. Use the Quality slider to adjust the number of good trackers kept. The higher the quality setting, more low quality trackers are selected, such as trackers that drift off their initial reference point. Click Delete to delete the selected trackers.



Trackers of lower quality may hinder the accuracy of the camera tracking.

NOTE After you have made a change that requires the 3D tracking analysis to be refined or updated, notice that the LED next to the Refine and Update buttons turns yellow. This signifies that a Refine or Update is required, but you do not have to perform it until you have completed your tracker selections.

2. Adjust the Short slider to select short duration trackers, that is, trackers that only track a feature for a few frames. Click Delete to delete the selected trackers, leaving the longer duration trackers intact.
3. You can manually select and delete trackers from the image that you feel are not tracking properly. Do one of the following:
   - To select an individual tracker, click the tracker, and then click Delete.
   - To select multiple trackers in the same area, `Ctrl`-drag a selection box over a series of trackers, and then click Delete.
   - With Delete mode selected in the Edit Mode box, select trackers in the image.

Setting the Scale

After tracking has occurred, you can set the scale of the tracked object. Since you are tracking a specific object as part of an image, setting the relative scale of the object in relation to the image helps you to position objects in the reconstructed scene when you convert the 3D points into a point cloud or axes.



Refining or Updating the Track

Once you are satisfied with your fine tuning changes, you can refine or update your Analyzer.



To refine or update the 3D track:

1. Depending on the changes you have made, you can choose to refine or update the 3D track. Do one or both of the following:
   - Click Update.

     NOTE The update operation deletes all previous 3D points and starts over based on the new information. Depending on your footage, and how many trackers you added, deleted, or linked, multiple updates may yield different results.

   - Click Refine.

     The track analysis uses the current results as a starting point, and refines from this point.

Click Refine again to stop the process once an acceptable pixel error value is reached. The pixel error value is a representation of the distance of the 2D tracks from the computed 3D points.

TIP The refine process is footage-dependant, so your acceptable pixel error value may change depending on what is tracking. Since the refine process continues until you stop it, as a general rule, if the pixel error value does not change for a length of time (for example, 30 seconds), you can stop the refine process. The lower the pixel error value, the more accurate the reconstructed track is.

Converting Object Tracking Results

When you are satisfied with the results of the 3D object tracking analysis, you can convert the selected reconstructed points to a point cloud object or actual axes in your scene. The point cloud object is useful because you can easily snap objects to the point cloud. An image that does not deform is the best candidate for the point cloud.



To create a point cloud object or axes from the 3D object tracking results:

1. Select the points in the image that you want converted to a point cloud 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 Do one of the following:

- Click Create Axis

  Selected points are converted to axes with a parent axis. The axes synchronize to the results of your 3D camera tracking, and any further changes you make to the 3D track are reflected in these axes.

  NOTE If you want to apply settings manually to a synchronized axis, change its name so that it does not update automatically when 3D Track settings are changed. If you reset the 3D Track settings, the synchronized axes become regular Action axes.

- Click Create Point Cloud

  Selected point are converted to a point cloud object with a parent axis. Double-click the newly created point cloud object to access its menu, where you can change display settings and enable snapping. See xxx.

3 Use the created point cloud or axes to view the reconstructed object motion.

You can attach objects such as surfaces, 3D text, and 3D models to the new point cloud or axes to help position them in 3D space. See Using the Point Cloud Object (page 697).

Object Tracking Settings

Use the settings in the Object Tracking menu to perform an object tracking analysis.

Object Tracking button Displays the Object Tracking menu. where you can track moving objects in the scene.

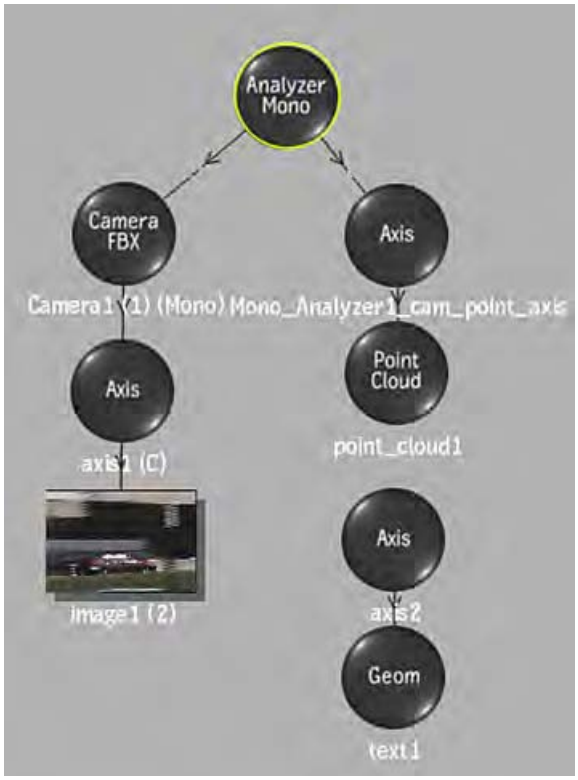
Object Motion box Select an option about the object's motion in relation to the camera.

Object Scale field Displays the scale of the tracked object. Editable.

Using the Point Cloud Object

Once you have created one or more point cloud objects from your camera or object tracking analysis, you can use the Point Cloud menu to help you snap objects to the points in 3D space and have them perfectly follow the movement in the scene.

In the schematic, you can see that the axis of a point cloud object is automatically attached to the Analyzer node. You can then add objects (such as a 3D text geometry), and use the Point Cloud menu settings to help you position the object into your scene.



NOTE If the point cloud object is moving (that is, you performed object tracking), you'll need to parent the point cloud object to the object you want to position, so that it can properly follow the movement of the point cloud.

To access the Point Cloud menu, double-click the point cloud object in the schematic.

Point Cloud Menu Settings



Point Cloud Display box Select how the point cloud is displayed in the image. You can choose between spheres, boxes, and crosses.

Point Cloud Colour pot Displays the colour of the point cloud points. Editable.

Point Size field Displays the size of the point cloud points. Editable.

Transparency field Displays the transparency level of the point cloud points. Editable.

Delete Selected button Click to delete selected points from the point cloud.

Snap To Point button Enable to snap an object automatically to a point in the point cloud object. Use Shift and drag the object towards the point cloud.

Snap Tolerance field Displays the minimum distance between the object and the point cloud for snapping to occur. Editable.

Transform box Select how an attached object behaves based on the movement of the point cloud.

Select:	To:
No Transform	Apply no transformation on the attached object.
Plane Transform	Align an object based on the average plane of the maximum number of points.
Centroid Transform	Align an object to the centre of the point cloud.

About Source Nodes

Use source nodes in Action to create more advanced techniques such as transforming or replacing a front or matte clip or applying motion blur. Source nodes can be used to separate front and matte clips in media and then apply separate transformations to each clip. For example, if you apply media to a surface and you want to create an effect where the matte moves into the scene, add a source that isolates the matte clip. A matte source lets you animate the matte's position separately from the front. You can also use source nodes to apply several matte clips to a single front clip.

NOTE When applying colour corrector, blur, and crop effects with source nodes, the effects are applied to the result of the source of the media. In contrast, keyer effects are applied only to the input clip of the media.

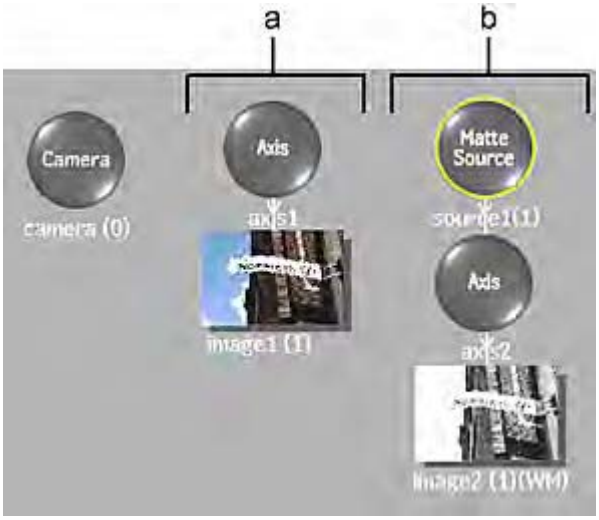
Creating a Matte or Front Source

When you create a matte or front source, a new branch appears in the schematic showing the matte or front clip is separated from its media, ultimately replacing the matte or front with itself. This method of using source nodes lets you animate the media's matte or front separately.

To create a matte or front source:

1. In Action, click Media.
2. In the Media list, select the matte or front media that you want to replace.
3. Do one of the following:
   - Drag the Source Matte or Source Front node from the node bin and place it in the schematic.
   - Double-click the Source Matte or Source Front node. You do not need to be in Schematic view to add a node in this manner.
   - Drag the Source Matte or Source Front node from the node bin directly to the image window. The view switches to Source view automatically.

   A source node is created in Schematic view that parents an axis and a surface.



(a) Normal branch (b) Source branch

To view the contents of the source node in the image window, see Viewing a Source Node (page 702).

NOTE You can change a source from a front to a matte source, or vice versa, from the Source Type box in the Source menu.

4 Select the axis or surface parented by the source and create the animation.

The changes applied to the axis or surface connected to a source are applied to the media's matte or front.

For example, on a matte source, if you change the axis's scale to 80%, the media's matte is scaled 80% when used with the media's front. In addition, if you replace the surface by a bilinear or bicubic, you can create complex animations where the matte behaves like a page turn leading into the scene.

Accessing the Source Menu

Once you added source nodes to your scene, you can access the Source menu to apply various settings.

To access the Source menu:

1 Do one of the following:

- Select a Source Matte or Source Front node in the schematic, and then click the Source menu button.

  NOTE If no source node is selected when first accessing the Source menu, or if there are no source nodes present in the scene, all of the options in the menu are disabled.

- Double-click a Matte or Front Source node in the schematic.

Parenting and Redrawing Source Nodes

A source cannot be parented by other objects in the schematic. If you attempt to parent a source, the connection is refused. You can, however, parent objects inside a source node and they appear in your scene based on the Source Type settings in the Object Image menu.

NOTE Be careful when you unparent or delete a source. Surfaces or other objects that were parented by the source are added to the scene.

See Connecting Action Nodes (page 420).

Replacing the Front or Matte Clip

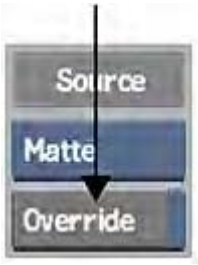
When working with source nodes, you can replace the media's front and matte clips with the front, matte, or a combination of the front and matte clips from other media. These changes can be made in the Source menu, or from the Sources tab in the Object Image menu of the image attached to the source node.

To replace the media's front or matte:

1. Double-click the source node in the schematic.
2. In the Source menu, change the source node from Matte to Front or vice versa.



3. Enable Override.



4. Select the type of source from the Source Type box.



   You can also change the source type in the Object Image menu for an image parented by a source node. The source type set in the Object Image menu is independent of the source type set in the Source menu for the parent source node or nodes.

5. If you selected Custom, select the clip to use as the front and matte from the image parented by the source. You can select Front, Matte, or White for each.

For example, if you select F: Matte and M: Matte, only the matte of the source's child image is rendered and used. A summary of your choice can be seen under the image node in the schematic. In this case you will see (MM).

NOTE You can combine multiple parent sources each with their own override setting.

Source Settings

Source box Select a Front or Matte source.

Override button Enable to override a source node selection on the parent source.

Source Type box Select a source type. When selecting Custom, you can also designate a Front and Matte type in the boxes below.

Select:	To:
Front only	Set the source to use only the front of the selected surface; the matte will be replaced with white (displayed as FW under the selected surface in the schematic).
Matte only	Set the source to use only the matte of the selected surface; the front will be replaced with white (displayed as WM under the selected surface in the schematic).
Custom	Set the source to have all the options of replacing the front and the matte of the selected surface by White, original Front or original Matte.

Front Type box Select the clip to use as the front from the image parented by the source.

Matte Type box Select the clip to use as the matte from the image parented by the source.

Source View box Select which source to view in the image window. See Viewing a Source Node (page 702).

Viewing a Source Node

There are different ways to view a source node while working in Action. For example, a 2-up view with Schematic and Source views allows you to view the complete scene and the interactive result of your source node.

The Src Working view provides you with the ability to view your source scene using a working camera, through which you can experiment without saving anything while preserving your camera setups. Each source node has its own working camera apart from the regular camera. See Setting the Camera and Orthographic Views (page 650).

You can also select Source Result, Source Front, and Source Matte views from the View box.

To view a source node:

1 From the View box, select a Source view.



(a) View box (b) Source View box

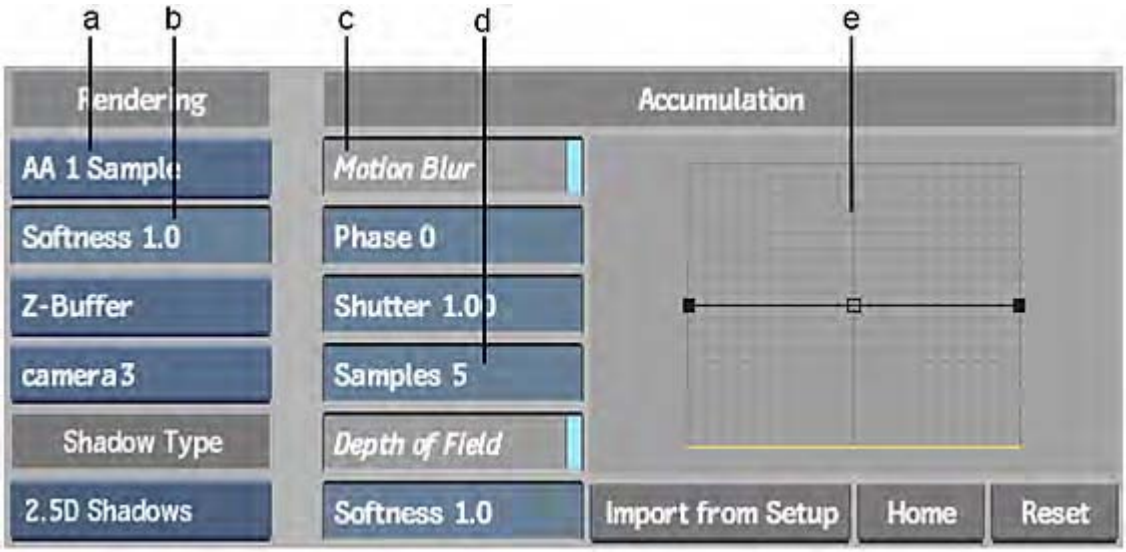
2 Press `F6` again to cycle through the Source Result, Source Front, and Source Matte views.

3 From the Source View box, select which source node you want to view.

Blurring with Source Nodes

Motion blur and anti-aliasing can be set for either the front or matte clip of the media. Motion blur simulates the blur created by fast-moving objects by blurring the motion of the front or matte media.

Similar settings for anti-aliasing and motion blur can be found in the Action Setup menu. You have the option of using the same settings from the Setup menu for source nodes, or you can create specific source node settings.



(a) Anti-Aliasing box (b) Anti-Aliasing Softness field (c) Motion Blur button (d) Samples field (e) Motion Blur Curve

Rendering Settings

Anti-Aliasing Sample box Select an anti-aliasing sampling level.

Softness field Displays the softness of the anti-aliasing sample. Editable.

Z-Buffer Mode box Select an option to determine whether the distance from the camera eye is considered.

Select:	To:
Z-Buffer	Arrange sources according to their distance from the camera eye.
Z-Buffer Off	Not consider the distance from the camera eye when arranging sources.
Shadow Mix	To render each shadow in the correct Z order with its corresponding source.

Camera Selector box Select the child camera under a parent source node.

Shadow Type Settings

Source Shadow Type box Select the type of shadow cast for the source node. This setting is available in the Shadow Cast menu only if the shadow cast node is a child of a source node, and is repeated in the Source menu.

Accumulation Settings

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

Phase field Displays the frame that motion blur is based on (before or after the current frame). Editable.

Shutter field Displays the duration of motion blur at each frame. Editable.

Samples field Displays the quality level of motion blur and the depth of field produced by the number of samples taken at each frame. Editable.

Increasing the number of samples causes the processing time to increase linearly. The number of motion blur samples is multiplied by the number of anti-aliasing samples. To reduce the total number of passes made for each frame, reduce the level of anti-aliasing when Motion Blur is enabled.

TIP You can animate the Motion Blur button, as well as the Phase, Shutter, and Samples fields in the Channel Editor under the *source* > *motion_blur* folder.

Depth of Field button Enable to use the camera's depth of field.

Softness field Displays the softness of the depth of field. A low value yields a sharp falloff between focused and unfocused regions. Editable

Motion Blur Curve Displays the sample weight over the scope of the motion blur.

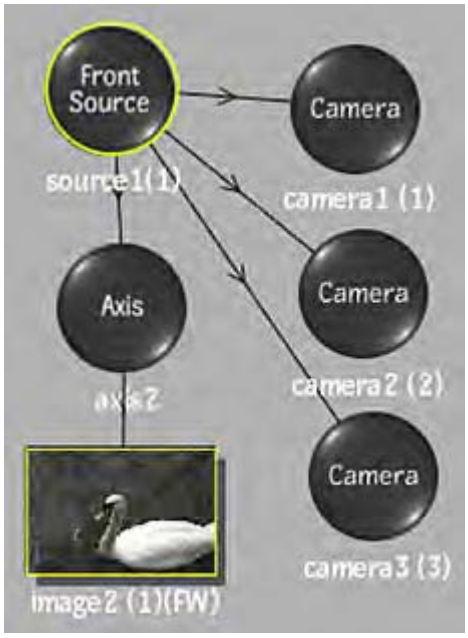
Import from Setup button Click to use the anti-aliasing and motion blur values from the Setup menu.

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

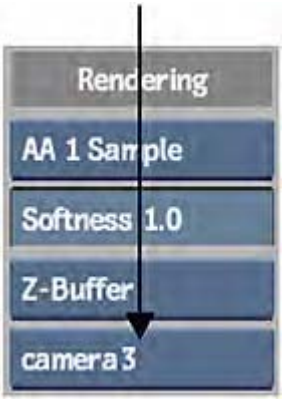
Reset button Resets the anti-aliasing and motion blur settings to their default values.

Adding Cameras

You can add multiple cameras under a parent source node to change the point of view (by orbiting to a different orientation, for example). Camera 0 is always the default camera in a scene (you may need to pan in the schematic to see this camera).



If you have multiple child cameras under a selected parent source node, you can switch from one camera to another from the Camera Selector box.



About Masks

A GMask object is an animatable spline-based object that you draw directly in an Action scene that allows you to apply transparency effects to the scene. A mask can be applied to all media by default, or limited to selected media. Depending on what you want to accomplish, you can create a transparency that reveals the background image, or hide a portion of the matte source or a front source.You can move the mask in the 3D environment using the Axis object.

Adding a Mask

When you add a GMask node to the schematic, it is added with an Axis node.

To add a GMask node:

1. Do one of the following:
   - Drag the GMask node from the node bin and place it in the Schematic view.
   - Drag the GMask node from the node bin and place it in Result view.
   - Double-click the GMask node.

   A GMask object (called gmask1, by default), with its parent axis, appears in the schematic. Once you have added the object, you can create the mask. See Creating a Mask (page 707).

   NOTE To create a mask using a GMask Plane node, GMask Ellipse node or GMask Rectangle node, see GMask Plane (page 709) and Gmask Preset Shapes (page 709).

Adding a GMask node to the schematic will apply the mask to the entire scene. You link the GMask object to specific media to apply a mask selectively. You can also apply a link to exclude a mask.

To apply selective masks:

1. Add a GMask object to the scene. All media is affected.
2. Select GMask from the Edit Mode box.
3. Do one of the following:
   - To apply a mask to an image, click the GMask node, and drag it to an image you want masked.

     The selected image is connected to the mask by a blue line with an arrow, and only the selected surfaces are masked.

- To exclude a mask from an image, hold the `Alt` key while clicking and dragging from the GMask node to the image you do not want masked.

  Excluded images are connected to the mask by an aqua dotted line with an "X", and they are not masked.

Changing Basic Mask Properties

You can customize a mask's properties by changing settings in the node's Mask tab.

Transparency field Displays the percentage of transparency of the mask. Editable.

Intensity field Displays the blend between the outgoing and incoming image inside the mask. Editable.

Smoothing field Displays the percentage of gradient blur softness. Editable.

Camera box Specify which camera to take into account when working with a GMask. Select Auto Frame when the GMask is co-planar with the image. When the GMask is under a Perspective Grid node, or is not under the image, select a specific camera to view the scene from the same perspective in Result (F4) or Object (F8) view.

Camera field Displays the active camera number. Non-editable.

Closed button Enable to connect a spline from the last point in the mask to the first control point (enabled automatically if you close the mask gesturally).

Invert button Enable to apply settings to the area outside of the mask.

Lock button Enable to lock the relative position of vertices.

Animation box Select Shape Animation to group all keyframes into a single channel called shape; select Vertex Animation to locate all keyframes on their respective channels.

Blend Mode box Displays the blending mode of the mask when it overlaps with another mask. Locked when a master blend option is selected in the Output menu.

Correlation box Select how overlapping gradients are blending. This box is displayed when an Autodesk Maya Composite blending mode is selected in the Blend Mode box.

Post Processing box Select whether to use the mask as a mask or occluder when encountering post processing effects. If you attach a Gmask to a specific post-processing node, for example a blooming effect, and choose Use as Occluder, only the effect will be blocked by the occluder.

Motion Blur button Enable to apply a blur effect to the selected mask. (Motion blur must be enabled in the Setup menu.)

Visibility box Select to apply the mask to the Right, Left, or Mono camera in a stereo scene.

Creating a Mask

After you have added a GMask object to the scene, you can now create the mask itself.

NOTE To create a mask using a GMask Plane node, GMask Ellipse node or GMask Rectangle node, see GMask Plane (page 709) and Gmask Preset Shapes (page 709).

To create a mask:

1. In the Edit Modes box, select Create (`N`).

2 Do any of the following to draw the mask:
   - Click to add vertices.
   - Shift-click to hide vertices and tangents as you add them.
   - Shift-drag to draw the mask freehand. Vertices and tangents are hidden as you add them. After closing the mask, you can use the Lasso Fit field (in the Vertices tab of the GMask menu) to increase or decrease the number of vertices that define the freehand segments of the mask.
   - Click and drag out the tangents to adjust the curvature of the spline at the current vertex.
3 Complete the mask by closing the mask shape or leaving it as an open spline.

Editing the Mask

You can edit the mask in three ways: by gestural action, by using the Edit Box, or by modifying the settings in in the GMask tabs. Editing operations, such as adding and removing vertices, can be effected whether or not the mask is completed. You can also edit mask properties, move vertices and tangents, add and edit the gradient, track the mask, change display properties before mask completion, and make shape transformations by enabling the Edit Box button.

To add points to a mask:

1 In the Edit Modes box, select Add (A).
2 Do one of the following:
   - Click on a spline between two existing vertices to add a new vertex.
   - Click on the first or last point of an open mask and click where you want to add a new point to continue drawing the mask.

To delete points from the mask:

1 In the Edit Modes box, select Delete (D).
2 Click on a vertex to delete it.
   If the vertex is in a mask that is a closed shape, the splines that connected it to adjacent vertices will be replaced by a single spline.

Closing the Mask

A mask can be closed gesturally or in the Mask tab.

1 Create the mask.
2 Do one of the following:
   - Create a spline to connect the last point in the mask to the first point.
     The mask is completed once it is a closed shape. The Closed button is enabled.
   - In the Mask tab, enable the Closed button.
     A spline connects the last point in the mask to the first point.

NOTE It is still possible to change the size of the mask by enabling the Edit Box.

Finishing the Mask

You can finish a mask without closing it to make a shape.

To finish a mask:

1. Create the mask. Do not close the shape.
2. Click the Finish button.

The mask is complete. The Finish button changes to the Edit Box button. The Edit mode changes from Create to Move.

A mask can still be adjusted after it is finished.

To adjust the mask:

1. Enable the Edit Box button.
2. Click and drag any of the points along the border of the Edit Box to adjust the mask.

You can also adjust the mask using the following keyboard shortcuts:

- Press `Alt` to drag the Edit Box proportionately.
- Use `Shift` to drag the opposite sides of the Edit Box proportionately.
- Use `Shift + Alt` to drag the Edit Box proportionately from the centre.

GMask Plane

You can add a Gmask Plane node to the schematic in order to accomplish specific tasks such as creating wipe effects or having better controls over reflection effects.

To add a GMask Plane node:

1. Do one of the following:
   - Drag the GMask Plane node from the node bin and place it in the schematic.
   - Drag the GMask Plane node from the node bin and place it in Result view.
   - Double-click the GMask Plane node.

A GMask object (called gmask1, by default), with its parent axis, appears in the schematic, and a single straight spline that is opaque on one side and transparent on the other, appears in the image. You can use the manipulator handles to adjust the Gmask Plane, and add as many gradient points to the spline as you want to.

GMask Preset Shapes

You can add a GMask preset node to the schematic. Two preset nodes are available: Ellipse and Rectangle.

To add either a GMask Ellipse or Rectangle node:

1. Do one of the following:
   - Drag the GMask preset node from the node bin and place it in the Schematic view.
   - Drag the GMask preset node from the node bin and place it in Result view.
   - Double-click the GMask preset node.

A GMask object (called gmask1, by default), with its parent axis, appears in the Schematic view. Once the GMask preset node has been created, it will have the same functionality and settings as a normal GMask

node and can therefor be manipulated in the same ways. When a GMask has been finished, the user can still make shape transformations to the mask by clicking the Edit Box button.

NOTE The Finish button becomes the Edit Box button once the GMask is finished.

Using the Mask in a Stereoscopic Workflow

When creating a mask for a stereo object, you can apply it to both eyes simultaneously. You can change the camera visibility options to apply a mask to only the Left Eye or Right Eye. Use the Visibility box to set the camera visibility options.

To set up stereo objects in Action, see Starting a Stereoscopic Session (page 485).

Manipulating Vertices and Tangents

You can manipulate vertices and tangents during and after mask creation. Edit vertices and tangents gesturally when the edit mode is set to Move or Break. Alternatively, edit them using the Vertices tab.

Vertices selected as a group can be transformed around a localized pivot. The angle and length of tangent handles can also be manipulated as a group.

Selecting Vertices and Tangents

You can select vertices and tangent handles when Move is selected in the Edit Mode box.

- To select an individual vertex or tangent handle, click the vertex or tangent handle.
- To select multiple vertices, `Ctrl`-drag a selection box over a series of vertices.
- To select multiple vertices using the pen, press the pen button and drag a selection box over a series of vertices.
- To add more vertices to an existing selection, press `Shift+Ctrl` and drag a selection box over the additional vertices.
- To deselect all vertices, click anywhere outside the mask.

Moving Vertices and Tangents

You can move vertices and tangents using the Move option of the Edit Mode box:

- To move a vertex or tangent in any direction, select Move. Click the vertex or tangent and drag.
- To move selected vertices in any direction, select Move. Click one of the selected vertices and drag.
- To ensure better continuity on a garbage mask, automatically adjust tangents as you move vertices or scale the mask. Enable Auto Adjust and drag a vertex. To toggle temporarily enable or disable Auto Adjust, press and hold `G` and then drag a vertex. Auto Tangents must be enabled to use the Auto Adjust feature.
- To keep tangent handles an equal length, `Ctrl+Alt`-drag one vertex handle to automatically change the length of the other tangent handle at the same time.

You can use the Vertices tab to move vertex and tangent positions.

The Selection fields display the current position of the vertex. If multiple vertices are selected, the position displayed is of the vertex that is closest to the initial vertex created in the mask.

Depending on the Transformation and Pivot Type options, you can move sets of selected vertices by editing the Translate, Scale, Shear and Rotation fields. You can also move tangent handles independently.

Transforming a Selection of Vertices

You can select multiple vertices in a mask and transform them as a group using the parameters. With the exception of translating vertices, transformation results are dependent on the position of the pivot point, displayed as a green or red circle in the image window. All transformations can be animated.

To transform a localized selection of vertices:

1. Select the node of the mask you want to edit.
2. `Ctrl`-drag over the vertices that you want to transform as a group.
3. Click the Vertices button.
4. Select an option from the Transformation box.

   Fields associated with the transformation type will be enabled.
5. Select an option from the Pivot Type box. (If Translate is selected as the transformation type, the box is disabled.)

   NOTE If the pivot point is selected, the position can also be moved by editing the X and Y Selection fields.
6. Do either of the following:
   - The active fields can be used to transform the selection.
   - Move the selection gesturally in the image window.

Vertices and Tangents Settings

Vertices Settings

Transformation box Select the type of modification that is permitted on the selected vertices. In the image window, the pivot icon indicates the transformation type.

Pivot Type box Select to place the pivot in the center of the mask (Centroid) or vertex selection set (Selection), or set the pivot gesturally (Manual).

X Selection field Displays the position on the X axis of the selected vertex. Editable.

Y Selection field Displays the position on the Y axis of the selected vertex. Editable.

X Translate field Displays the position on the X axis of the selected vertex. Editable.

Y Translate field Displays the position on the Y axis of the selected vertex. Editable.

X Scale field Displays the scale value on the X axis of the selected vertices. Editable.

Y Scale field Displays the scale value on the Y axis of the selected vertices. Editable.

Proportional button Enable to change the scale fields proportionally.

X Shear field Displays the shear value on the X axis of the selected vertices. Editable.

Y Shear field Displays the shear value on the Y axis of the selected vertices. Editable.

Proportional button Enable to change the shear fields proportionally.

Z Rotation field Displays the rotation value on the Z axis of the selected vertices. Editable.

Tangents Settings

In Angle field Displays the angle of the incoming tangent handle. Editable.

In Distance field Displays the length of the incoming tangent handle. Editable.

Out Angle field Displays the angle of the outgoing tangent handle. Editable.

Out Distance field Displays the length of the outgoing tangent handle. Editable.

Auto Tangent button Enable to create tangents automatically. When enabled, the Tracker will affect tangents.

Auto Adjust button Enable to modify tangents on either side of the selected vertex automatically. Press G to temporarily enable or disable during a gestural operation.

Equal Length button Enable to set tangents on either side of vertices to be the same length (based on the proximity to the next vertex).

Lasso Fit button Set the tolerance for the number of points in the segments of the mask if it was drawn freehand.

Breaking, Removing and Adding Tangents

You can separate two tangent handles (break the tangent) and move them separately using the Break option from the Edit Mode box:

- To break and move a tangent handle, select Break (B) and click the tangent handle. The tangent is displayed as a dashed line, indicating it is "broken."
- To reconnect two broken tangent handles, click the vertex or select Auto (U) and click either of the two tangent handles. The tangent is displayed as a solid line.
- To change the position of an individual tangent handle after releasing the cursor, select Move (M).

In the Vertices tab, use the fields in the Tangents section to change the length and angle for the incoming and outgoing tangent handles. If the tangents do not combine to form a straight line, the tangents will be broken and displayed as a broken line.

You can also use the Break option to remove tangents from vertices by clicking the vertex while in Break mode. When you remove a tangent, the interpolation of the spline between the two vertices is modified.

The shape of the border line differs depending on whether adjacent vertices are broken:

- If adjacent vertices are unbroken, the border line curves as it approaches the vertices.
- If adjacent vertices are broken, the border line is straight as it approaches the vertices.

You can create a garbage mask composed entirely of straight edges by removing the tangents from all the vertices:

- To remove the tangent of a vertex, select Break (B) and click the vertex.
- To add a tangent back to a vertex, select Auto (U) and click the vertex. Alternatively, using the Break option, click the vertex and drag the tangent out again.

Adding a Gradient to a Mask

The gradient allows you to create a region of softness around the mask edge. It applies a gradient according to the distance of the borders from the mask spline. You can create a uniform gradient or customize it at different parts of the mask. The gradient can have two softness borders, one inside and one outside the mask edge.

To add a gradient to a mask:

1. Change the Edit mode to Add (A).
2. Hold down Shift and drag away from the spline.
3. Click on the mask spline.
4. (Optional) Do one of the following:
   - Shift-click on the spline again and drag in the same direction as the existing gradient to add a point in the gradient at a different position and distance from the mask.
   - Shift-click on the spline and drag away from it in the other direction to create a second gradient.

The position and distance of a gradient point from the mask spline can be controlled gesturally in Move mode (M) or in the Gradient tab.

Gradient Settings

Position field Displays the relative position of the gradient control point in relation to the vertices. For example, if the control point is between the third and fourth vertices, its value is 3.5. Editable.

Offset field Displays the distance from the gradient spline to the mask spline. Editable.

Gradient Curve Displays the transparency distribution of the gradient. The y axis represents the percentage of transparency and the x axis represents the percentage of distance from the vertex to the spline. This curve can be animated. You can edit, add and delete points from the curve.

Set Key button Sets the current values for the gradient curve in the current frame(when Autokey is disabled).

Delete Key button Deletes the selected gradient curve keyframes.

Reset Selection box Select whether to reset all of the gradient settings (Reset All) or just the gradient curve (Reset Key).

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

Undo button Undoes the gradient curve operations.

Tracking a Mask

You can track a mask in your scene from the GMask object's Tracking tab. You can track the mask vertices using the Tracker (page 878), or track shapes using the GMask object's own shape tracker.

To apply shape tracking to a mask:

1. Select the mask you want to track.
2. Open the Tracking tab.
3. Under Shape Tracking, set the Use Media field to assign the front and matte clip.
4. Under Tracking, adjust the fields as needed.
5. Select an option from the Track Direction box to determine if the frames before or after the current frame will be used for analysis.

   TIP When tracking a shape that cuts off during analysis (for example, a person walking into or out of the frame), begin shape tracking when the whole shape is apparent, and track in the direction of the frames at which the shape begins to disappear.

6. Click the Shape Track button to perform the shape tracking analysis.

7 To perform additional analysis in a different direction from the first frame of the analysis, toggle the Forward and Backward box and click Shape Track again.

Tracking Settings

Vertices Tracking Settings

Tracker menu button Opens the Tracker menu where you apply tracking data to selected vertices.

Tracking View box When the current node is attached to a Source node, select to track the media in the Source Front, Source Matte or Source Result view.

Camera box Specify which camera to take into account when working with a GMask. Select Auto Frame when the GMask is co-planar with the image. When the GMask is under a Perspective Grid node, or is not under the image, select a specific camera to view the scene from the same perspective in Result (F4) or Object (F8) view.

Camera field Displays the active camera number. Non-editable.

Shape Tracking Settings

Shape Track button Click to generate tracking data. Tracking information is stored in the Axis node to which the mask is attached.

Shape Track Progress field Displays the progress of the tracking operation. Non-editable.

Use Media field Displays the layer number (as defined in the Media list) of the media that will be tracked. Editable.

Track Axis Hierarchy button Enable to overwrite and store tracking information in the topmost Axis node in the schematic hierarchy.

Transform Type box Select the type of shape tracking to perform.

Track Direction box Select to track the image sequence forward or backward from the current frame.

Trackers Settings

Size field Displays the area in pixels of each tracker. Editable.

Maximum Number field Displays the number of trackers to use in the analysis. Editable.

Motion Tolerance field Set a value to determine the tolerance level of acceptable changes in the pixel environment. If an external object passes in front of the tracked object, a low tolerance value ensures that the motion of an external object does not get tracked through the analysis. Editable.

Filtering field Displays the level of smoothing to apply to the shape tracking animation curve. Editable.

Mask Display Properties

You can change the display properties of your mask in the Display tab.

Spline colour pot Displays the colour of mask splines. Editable.

Vertices colour pot When the mask is active, displays the colour of the unselected vertices. Editable.

Gradient colour pot When the gradient is active, displays the colour of the gradient spline. Editable.

Tangents Display option box Select to show all, selected, or no tangents on any mask that is selected.

Rendering the Mask

In the Output menu, you can set up preferences to render the mask. See Processing Multiple Outputs (page 435).

To make the mask visible in the output, enable Use GMask in the Render Options panel.

The mask output in the render is dependent on the settings of the Type box In the Output Options panel. Do any of the following:

- Set the Type box to Comp to reveal the background in the masked area in the render.
- Set the Type box to Matte or Media Matte to render a matte output of the scene or the selected object in the scene that reveals the colour, intensity and smoothing in the masked area.

About Action's Particle System

Use Action's 3D particle system to create a variety of effects such as snow, rain, fog, and tornadoes, as well as fireworks, sparks, and explosions.

Particles are 3D objects that originate from a surface, light, or 3D object. Particles can be spheres, cones, squares, points, lines, or any image, 3D text, or 3D object. Particles are generated in the direction of the normals of a surface, light, or 3D object—a normal is a line perpendicular to the surface of an object.

In Action, the main components for generating particles are generators, manipulators, and bouncers. You can also use the particle system to explode a 3D object or an image.

NOTE Rendering particles is faster if Z-Buffer On is selected in the Rendering section of the Action Setup menu.

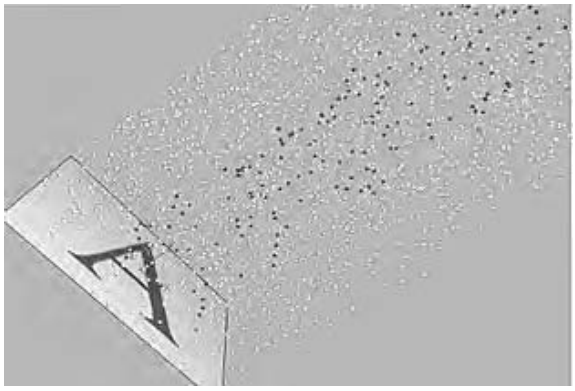
Generators

To create a particle stream, start by adding a particle generator that creates a stream of particles when attached to a light source, 3D object, or surface. You can use three different generators: light source, surface, and 3D object.

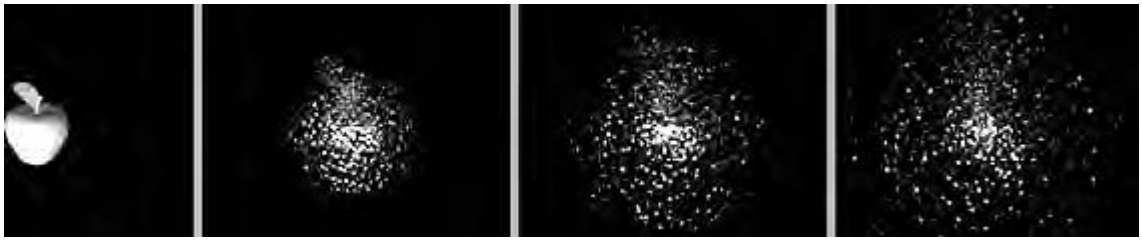
Particle generators have two settings:

- The Generate setting creates a stream of particles using the shape of a surface, a 3D object, or the spread of a light source. See Generating Particles (page 749).



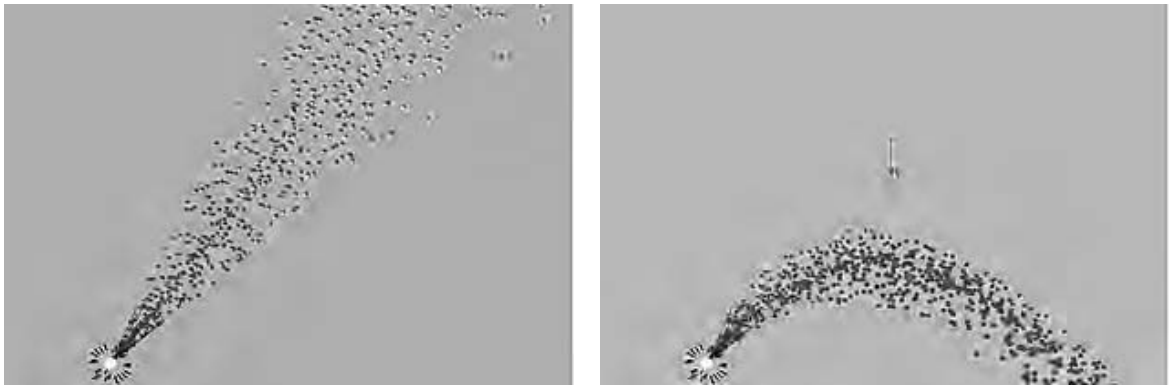


- The Explode setting breaks up a surface or 3D object. See Exploding Objects and Surfaces (page 762).



Manipulators

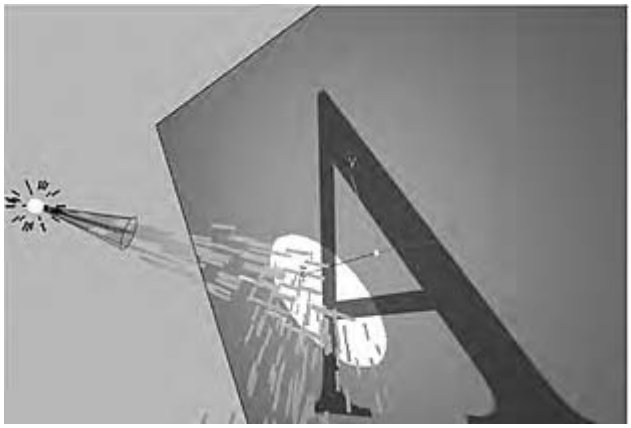
Particle manipulators are used to influence either the position or the speed of particles. Manipulators can simulate the effects of a vortex, gravitational pull, or the pull toward a point, line, or plane. Manipulators are parented by the particle stream that they influence.



Action offers eight manipulators. Instructions on how to use each manipulator are provided in Manipulating Particles (page 765).

Bouncers

Particle bouncers are used to influence the behaviour of particles when they come into contact with a surface. You can parent an axis or light source to a particle bouncer. See Bouncing Particles (page 783).



NOTE Action setups created in a pre-9.0 version of Flame Premium that use a Variation value for Number, LifeTime, Speed, and TrailSize can be restored in the current version, but will render slightly differently.

Particles Workflow

The following table shows a possible workflow when working with particles in Action.

Step:	Refer to:
1. Add a particle generator.	Generating Particles (page 749).
2. Add a particle manipulator.	Manipulating Particles (page 765).
3. Set the particle path.	Forming a Particle Path (page 774).
4. Add bouncers.	Bouncing Particles (page 783).
5. Modify the particle stream.	Setting Stream Properties (page 752).
6. Apply texture to particles.	Applying Textures to Particles (page 761).
7. Add gravity.	Simulating Gravity (page 769).
8. Render particles.	Setting Particle Rendering Properties (page 755).

Using Particle Presets

A number of particle presets are included in Action, such as a fog effect to add to your scene. These presets can also help you learn how the particle system works. The presets are divided into a number of categories and are easily added to your scene from the Action node bin.

To add a particle preset:

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

     The file browser opens.
2. From the Preset Type box, select Particles.



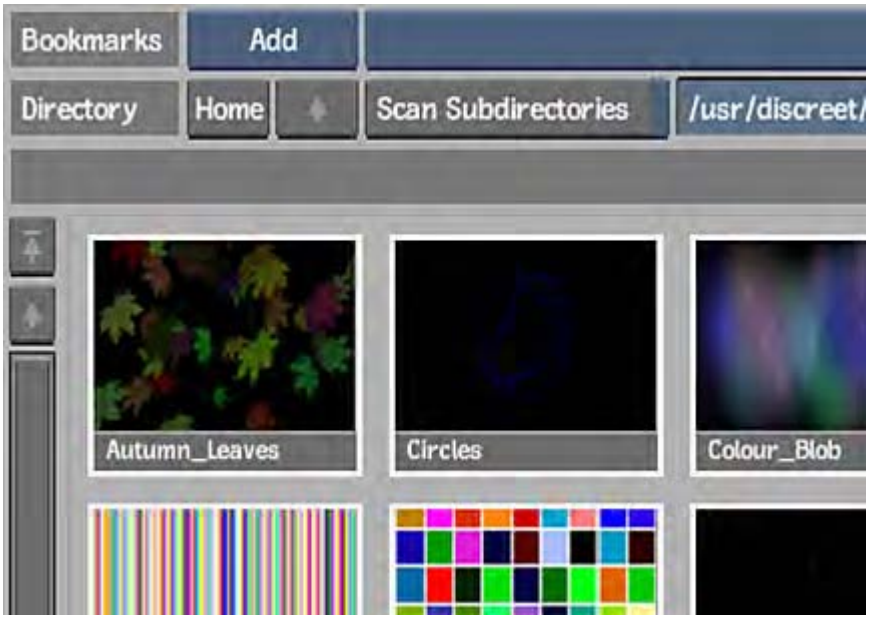
   The Particle Preset file browser appears, pointing to the default location of the presets: *usr/discreet/<product home>/particle_presets*.
3. Optional: Enable Scale to Action Resolution to load the preset in the current Action resolution.

4 Optional: Select which rendering settings to enable or disable in the preset (Z-Buffer, Shading, Polygon Resolution, and Colour Clamping).

NOTE These settings are enabled by default, and by disabling any of them, you may not see the intended results in the preset.

5 Navigate through the subfolders to select the particle preset you want to load. Hold `Ctrl` and click to select multiple presets.

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



6 Click Load.

The particle preset is then appended to your Action scene.

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.

Presets Browser Settings

Preset Type box Select the category of presets to display in the browser.

Scale to Action Resolution button Enable to load the preset in the current Action resolution.

Z-Buffer button Enable to load the Z-buffer rendering settings of the preset.

Shading button Enable to load the shading rendering settings of the preset.

Polygon Resolution button Enable to load the rendering resolution settings of the preset.

Colour Clamping button Enable to load the resolution colour clamping settings of the preset.

Particle Preset Reference

A number of particle presets are included in Action, such as pyrotechnic or liquid effects to add to your scene. These presets can also help you learn how the particle system works. The presets are divided into a number of categories and are easily added to your scene from the Action node bar.

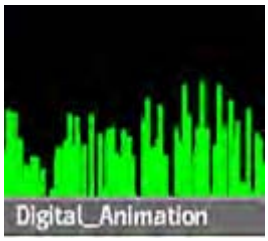
Use the following tables to get a quick overview of the particle presets, along with best use comments, if applicable. For more information on particles in general, see About Action's Particle System (page 715).

Abstract/Backdrops

Use these presets as backdrops or to create abstract art particle effects in your scene.

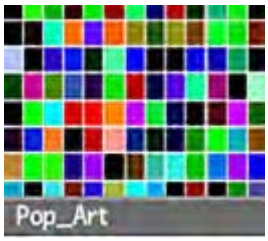
Proxy	Name/Description/Comments
	**Animated_Coloured_Background** Use Number and Lifetime to slow down or speed up the animation. Try with Z Rotation and Spin settings.
	**Art_Coloured_Lines** Speed sets the amount of stripes. Try with Colour variance in motif_generator menu.
	**Autumn_Leaves** Use Number, Size, and Speed settings. Try with the Rotation and Spin settings.
	**Burning_Text** Particle Generator and cascade effect to burn the text and make it disappear.

Proxy	Name/Description/Comments
	**Circles** Use Number, Lifetime, and Size settings. Try with the Spin setting.
	**Colour_Blob** Animated coloured gradients. Use Number, Lifetime, and Size settings. Try with Blending modes and Lighting.
	**Colour_Cloud** Animated. Use Number and Size settings. Try with Z Rotation and Lighting.
	**Colour_Vortex** Animated. Play with circ_gradient_ax (especially Y Scaling). Try with Z Spin setting. For a nice spiral effect, try Timesteps = 2 and Trail values up to 50.
	**Coloured_Arcs** Arcs rotating towards the camera. Try text_ax (posX=140, RotZ=200, ScaleX=30).

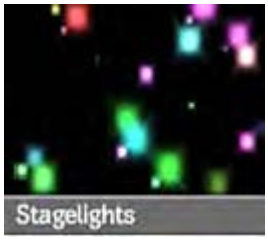
Proxy	Name/Description/Comments
	**Concentric_Circles** Animated. Use circles_emitter Number and Lifetime fields to set the amount of circles.
	**Concentric_Circles_2** Use Number and Size. Speed up or slow down animation with Lifetime.
	**Crazy_Serpentine** Random circular patterns. Try with the size manipulator (Magnitude). Modify the Power animation in vortex manipulator.
	**Crazy_Stars** Pulsing coloured stars. Use Number and Size settings.
	**Digital_Animation** Animated rectangles. Use Number, Lifetime, Size, and Speed settings. Change colour using light_colour.

Proxy	Name/Description/Comments
	**Dots_Spiral** Animate Size setting. Try with Magnitude in size manipulator. Change colour using the Geometry settings of dots_generator.
	**Dots_Vortex** Use the Size setting. Try with Magnitude in the size manipulator and with Power in the vortex manipulator (try Power = 5). Move the camera and see the effect of gravity.
	**Fairy_Text** 3D text made of coloured stars. Try star_text_ax (Scale=22) and generator (Number=95, Number_V=25, Lifetime=75, Size=12, Size_V=20, Speed_V=0,1).
	**Flashing_Squares** Influence the colour using the light's colour settings.
	**Glow_Scanline** Vertical and horizontal moving glows. Use Number, Lifetime, and Size to characterize horizontal & vertical glows. Use Magnitude in the V_size and H_size manipulators to set the scaling's amplitude. Use Magnitude in the V_horizontal_osc manipulator to set the horizontal motion's amplitude. Change colour with the light.

Proxy	Name/Description/Comments
	**Glowing_Streaks** Coloured streaks coming from behind the camera to the centre of the frame. Try Number=110, Number_V=35, Lifetime=1, and Speed_V=70. Orbit the camera.
	**Glowline** Moving horizontal glows. Use Number, Lifetime, and Size settings to characterize the glow. Try with Magnitude in the size manipulator to scale the glows.
	**Hypno** Influence the colour using the light's colour settings.
	**Hypnotic** Try with Rotation and Spin settings. Orbit the camera.
	**Kaleidoscope** Change the colour using the light's colour settings.

Proxy	Name/Description/Comments
	**Light_Canvas** Grid of animated glows. Influence the colour using the light's colour settings. Characterize the vertical and horizontal glows using the circgrad_V_ax and circgrad_H_ax axes.
	**Light_Dots** Speed up or slow down the effect using the Power setting in the size Animator.
	**Moving_Glows** Circular illuminated glow. Change colour by colour correcting the white media. Use Number and Size to characterize the ring. Use the pos_scale axis to set the position and rotation of the ring in the scene.
	**Path_Light** Animated glow on a path.
	**Pop_Art** Try with a close-up shot of a face as REPLACE_ME media. Zoom in to have fewer coloured squares in the frame, or zoom out to have more.

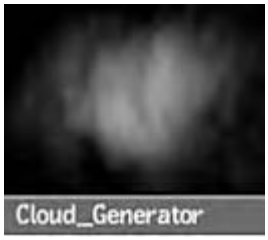
Proxy	Name/Description/Comments
	**Psychedelic_Texture** Animated. Use Magnitude in the size manipulator to set pattern size. Try with the Trailsize field in the Generator menu.
	**Random_Texture** Animated. Use Number, Lifetime, Size, and Speed settings to characterize the texture. Try with the Spin settings to set the animation.
	**Raying_Sphere** Black sphere emitting coloured streaks. Use Number and Speed settings. Try with the Tailsize field.
	**Sky** Moving cloudy sky. Use Number and Colour_V settings to characterize the sky. Use Position settings in the sky_motion manipulator to set how the clouds are moving.
	**Space_Blob** Coloured glows with a burning effect. Try the PD_ax settings, and influence colour using the Light's colour settings.

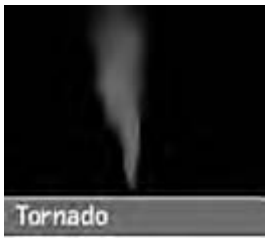
Proxy	Name/Description/Comments
	**Spectrum_Lights** Animated vertical glows. Use Number, Lifetime, and Size to characterize the lights. Try with Magnitude in the size manipulator to control the scaling amplitude of the lights.
	**Stagelights** Use Number, Lifetime, and Size. Try with Magnitude in the size manipulator to control the scaling amplitude of the stagelights.
	**Star_Spiral** Use Number, Lifetime, and Size. Try with Magnitude in the size manipulator to control the scaling amplitude of the stars.
	**Text_Atom** Animated 3D Text birth.
	**Text_Distort** Use Number, Lifetime, and Size. Try with Rotation_V, Spin, and Spin_V settings. Try also to change the light's colour and the glow_generator's Material settings.

Proxy	Name/Description/Comments
	**Text_Dots** Use Number, Lifetime, and Size. Try with Rotation_V, Spin, and Spin_V settings. Try also to change the light's colour and the glow_generator's Material settings.
	**Text_Glow** Use the Size setting. Try with Magnitude in the size manipulator.
	**Text_Particules** Animated cascade effect with the 3D Text Geom node.
	**Texture** Use the Size setting. Try with the Rotation_V setting. Play with the texture_ax settings.

Atmospheric

Use these presets to create weather-like effects, such as clouds, rain, or snow.

Proxy	Name/Description/Comments
	**Cloud_Generator** A one-frame cloud. Use Number and Size settings. Try with the texture's Axis settings and the geometry's Magnitude and Scaling settings.
	**Fog** Use Number, Lifetime, Size, and Speed to characterize the fog.
	**Heavy_Fog** Use Number, Lifetime, Size, and Speed to characterize the fog.
	**Rain** Use Number and Speed to trigger the amount of rain. Colour correct the white media's Front, and set rain_generator Transparency to improve blending.
	**Rainfall** Use Number and Speed to trigger the amount of rain. Colour correct the white media's Front, and set rain_generator Transparency to improve blending. Use gravity's Rotation_Y to make it windier.

Proxy	Name/Description/Comments
	**Snow** Use Number, Size, and Speed for characterizing the snow.
	**Snow_Graphical** Use Number, Size, and Speed for characterizing the snow. Play with lighting.
	**Snow_Windy** Use Number, Size, and Speed for characterizing the snow. Try with wind's Magnitude and axis animation to mimic the wind.
	**Surfing_Clouds** Flying into the clouds. Use gravity's Rotation_X expression to stay above clouds, or dive into them.
	**Tornado** Use Magnitude in the size manipulator to scale the tornado. Use Magnitude and Power in the twist manipulator to alter the shape of the tornado.

Proxy	Name/Description/Comments
	**Tornado_ZatZero** Same as Tornado, but with emitter Z=0.

Liquids

Use these presets to create fluid effects, such as bubbles.

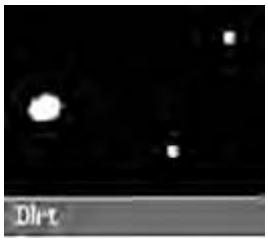
Proxy	Name/Description/Comments
	**Bubbles_Graphical** Use Number, Size, and Speed settings to characterize the bubbles. Process with Motion Blur enabled for better rendering.
	**Bubbles_Rising** Bubbles coming toward camera. Use Number, Size, and Speed settings to characterize the bubbles. Use Mass_V to randomize the bubble motion.
	**Inside_Water_Rays** Sun rays under water. Use water_generator for the water and rays_generator for the rays coming from the sun above the surface.

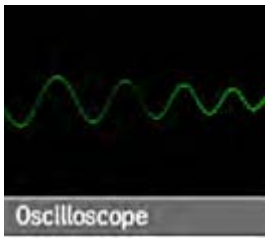
Proxy	Name/Description/Comments
	**Sea** Water surface. Try the Spin_V settings to animate the sea. Customize by adding more lights to the scene.
	**Water_Bubbles** Use Number, Size, and Speed to characterize the bubbles. Try the Spin & Spin_V settings to control the flow.
	**Waterfall** Use Number, Size, and Speed to characterize the water. Try with size and gravity manipulators.

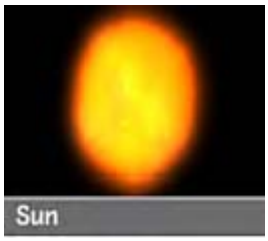
Miscellaneous

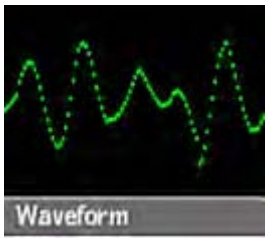
Use the miscellaneous presets to add dirt or scratches to your scene, as well as many other effects.

Proxy	Name/Description/Comments
	**Balloons** Animated. Use Number, Size, and Speed settings to characterize the balloons. Try the Spin & Spin_V settings to control the flow.

Proxy	Name/Description/Comments
	**Blue_Thing** Moving glow following a path. Use Number, Lifetime, and Size settings to characterize the glow. Try with Rotation and Spin for motion. Change the path.
	**Broken_Window** Pieces from a broken window flying towards the camera.
	**Dirt** Use Number and Size settings. Try with variance.
	**Dust** Modify dust size using PD_ax Scale and generator Number settings.
	**Dust_Ray** Dust revealed by rays of light.

Proxy	Name/Description/Comments
	**Hair** Use Number and Size settings. Try with variance. hair_generator4 is linked to hair_generator1, and hair_generator3 is linked to hair_generator2.
	**Microscope_View** Modify lights. Try neuronal_tex_ax settings.
	**Oscilloscope** Change the colour by using beam_generator's Diffuse setting. Use axis to control the beam's amplitude, frequency, and phase.
	**Paint_Effect** Use Number, Lifetime, and Size settings to customize the paint effect. Try changing the texture media (white square, soft cloud, etc.).
	**Scratch** Use axis to control the scratch's position, the amplitude of the horizontal motion of the scratch, and the frequency of the scratch. Hide some generators for fewer scratches, or duplicate generators if you want more scratches.

Proxy	Name/Description/Comments
	**Shotgun** Slow-motion shotgun effect.
	**Simple_Sun** Processes quicker than Sun.
	**Sun** Astronomical view of the sun. Slower to process than Simple_Sun.
	**Vu_Meter** Animated. Use beamgen Speed, PD_axis Scale_Z, and beam_emitter positioning to modify the number and width of stripes.
	**Vu_Meter_AudioLink** Similar to Vu_Meter, but you can link an audio clip to produce the animation.

Proxy	Name/Description/Comments
	**Waveform** Animated audio signal monitor effect. Use axis to control the beam.

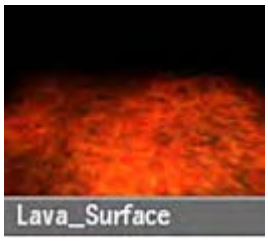
Pyrotechnic

Use the pyrotechnic presets to create fire and flame effects.

Proxy	Name/Description/Comments
	**Birthday_Sparkles** Customize the sparkles with Number, Lifetime, and Speed.
	**Bouncing_Sparkles** Sparkles bouncing on a plane. Use white_ax settings to place the bouncer plane in the scene.
	**Camp_Fire** Adapt the source of fire with emitter_ax. Characterize the flame with Number, Lifetime, and Size.

Proxy	Name/Description/Comments
	**Camp_Fire_Smoke** Adapt the source of fire with emitter_ax. Characterize the flame with Number, Lifetime, and Size. Adjust the manipulators.
	**Candle_Flame** Characterize the flame with Number, Lifetime, and Size.
	**Explo_Zero_Gravity** Explosion in a zero gravity environment.
	**Explosion** Exploding towards the camera.
	**Fire_Ring** Characterize the fire with Number, Lifetime, and Size. Adjust the camera.

Proxy	Name/Description/Comments
	**Fire_Turbulence** Fire agitated by the wind.
	**Fireworks** Animated.
	**Fireworks_BigBlue** Animated.
	**Fireworks_BigRed** Animated.
	**Fireworks_Cartoon** Animated.Process with Motion Blur enabled.

Proxy	Name/Description/Comments
	**Fireworks_on_Path** Adjust the path.
	**Gas_Flame** Characterize the flame with Number, Lifetime, and Size.
	**Lava** Lava flowing from the top of a volcano. Change the flow using the path manipulator.
	**Lava_Surface** Speed up or slow down the lava using the transform manipulator position settings.
	**Match_Flame** Characterize the flame with Number, Lifetime, and Size.

Proxy	Name/Description/Comments
	**Meteor** Meteor burning in space. Adjust the camera.
	**Moving_Sparkles** Animate sparkles on a path. Change the path.
	**Rocket_Propulsion** Rocket engine output.
	**Text_Plasma** Customize with Number, Lifetime, and Speed.

Smoke

These presets allow you to create various smoke effects.

Proxy	Name/Description/Comments
	**Carbo_Snow** Snow coming toward camera. Use Size to characterize the snow. Use the Magnitude setting of the gravity manipulator to set the snow's motion.
	**Dark_Smoke** Use on a light background. Use Number, Lifetime, Size, and Speed to characterize the smoke. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.
	**Fire_Smoke** Characterize the smoke with Number, Lifetime, Size, and Speed settings. Use the Magnitude setting of the gravity manipulator to set the smoke's motion. Modify the fire colour using the fire_light_1 and fire_light_2 colour settings.
	**Fire_Smoke_ZatZero** Similar to Fire_Smoke, but with emitter Z=0.
	**Gun_Smoke_MB** Smoke produced by a shotgun. Process with Motion Blur enabled.

Proxy	Name/Description/Comments
	**Heavy_Smoke** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.
	**Heavy_Smoke_ZatZero** Similar to Heavy_Smoke, but with emitter Z=0.
	**Jet_Engine_Smoke** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.
	**Nozzle** Characterize the smoke with Number, Lifetime, Size, and Speed settings. Use vortex settings to set the smoke's motion. Change the bouncer. Not scalable.
	**Pyroclastic_Cloud** Change the path. Characterize the smoke with Number, Lifetime, Size, and Speed settings. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.

Proxy	Name/Description/Comments
	**Smoke** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.
	**Smoke_Light** Characterize the smoke with Number, Lifetime, Size, and Speed settings.
	**Smoke_Lookat** Smoke pointing to a target. Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.
	**Smoke_Track** Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude setting of the gravity manipulator to set the smoke's motion.
	**Smoke_ZatZero** Similar to Smoke, but with emitter Z=0.

Proxy	Name/Description/Comments
	**Train_Smoke** Smoke produced by a steam engine in motion. Use Number, Lifetime, Size, and Speed settings to characterize the smoke. Use the Magnitude setting of the train_speed manipulator to control the train's speed and its effect on the smoke.
	**Train_Smoke_ZatZero** Similar to Train_Smoke, but with emitter Z=0.
	**Vapour** Vapour rising from the ground. Use Number, Lifetime, Size, and Speed settings to characterize the vapour. Try the Rotation_V and Spin_V settings to control the vapour animation
	**Wheel_Burning_MB** Animated. Process with Motion Blur enabled.

Space

Use these presets to create astronomy effects.

Proxy	Name/Description/Comments
	**Aurora** Aurora borealis effect. Use Number and Size to characterize the aurora. Use Lifetime to speed up or slow down the animation. Change colours by modifying the left_light & right_light settings.
	**Blue_Star** Slow moving star. Use Number, Lifetime, Size, and TrailSize settings.
	**Blue_Star_2** Slow moving star. Similar to Blue_Star, but with a bigger hotspot in the middle. Use Number, Lifetime, Size, and TrailSize settings.
	**Blurry_Sparks** Use Number, Lifetime, Size, and Speed settings to set how the sparks are emitted. Use the transparency, gravity, and size manipulators to specify the behaviour of the falling sparks.
	**Cosmic_Light** Try with the PD_ax settings.

Proxy	Name/Description/Comments
	**Cosmic_Marguerite_Blue** Pulsating blue light. Try with galaxy_tex_ax settings (posX=0, posY=0, scaleX=14, scaleY=125).
	**Cosmic_Marguerite_Green** Pulsating and rotating green light. Orbit the camera.
	**Eye_Strike** Moving glows. Try with the circ_gradient1_ax and circ_gradient2_ax settings.
	**Glowing_Star** Try with the circgrad_tex_ax settings. Orbit the camera.
	**Light_Mvt_1** Try with the PD_ax settings. Process with Motion Blur enabled.

Proxy	Name/Description/Comments
	**Light_Mvt_2** Orbit the camera.
	**Light_Mvt_3** Orbit the camera (Orbit 90°).
	**Light_Mvt_4** Change the Rotation and Spin settings.
	**Light_Speed** Use Number, Speed, and TrailSize settings. Use the speed manipulator's power to set the Z acceleration.
	**Moving_Star_Field** Flying into space. Use Number and Speed settings.

Proxy	Name/Description/Comments
	**Pulsar** Pulsating glow. Use Number, Lifetime, Size, Speed, and TrailSize settings.
	**Red_Pulse** Pulsating red star. Customize the star with Number and Speed settings. Set the star's intensity with transparency magnitude.
	**Red_Thing** Glowing red light. Change the colour with the Light Colour settings.
	**Space_Dust** Reset vortex (Rot_Z) and try with vortex (Rot_Y). Try vortex (Rot_Y=-90). Change the camera settings.
	**Space_Odyssey** Change the space_tex_ax settings (animated) and emitter_ax1 Rot_X setting.

Proxy	Name/Description/Comments
	**Star** Glowing animated star. Use Number, Size, and Speed settings to characterize the star. Change colour using the light's Colour settings.
	**Star_Field** Flickering stars in the sky. Modify Number at frame 0 to control the amount of stars in the sky.
	**Star_Glow** Glowing aggregate of stars. Customize the generator with the Rotation and Spin settings.
	**Stellar** Orbit the camera and change the stellar_tex_ax settings.
	**Stellar_Fusion** Rotating celestial bodies. Orbit the camera.

Generating Particles

You can create a particle stream that behaves in different ways depending on the type of generator you use. When creating a particle stream, you can use three different generators: light source, surface, and 3D object.

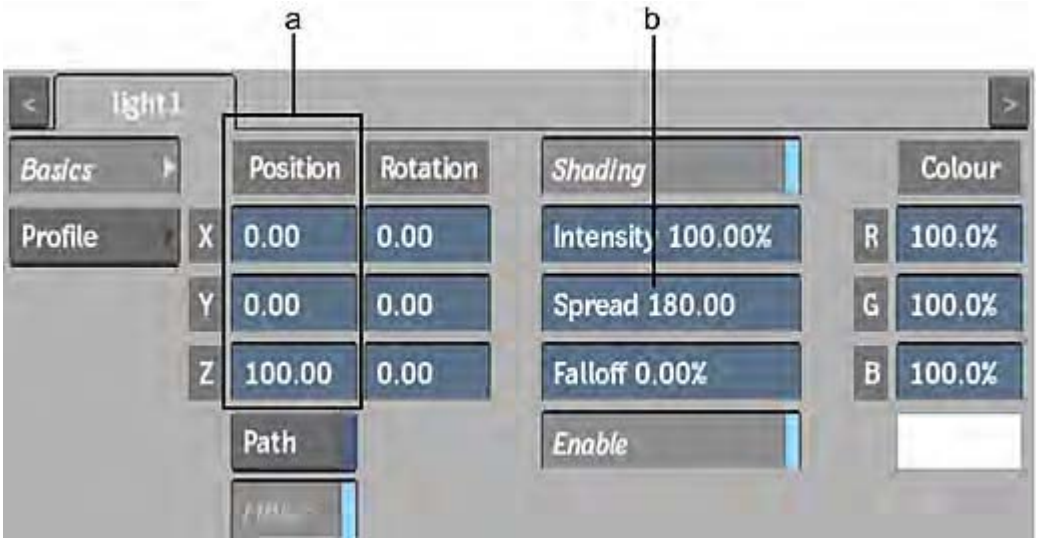
Use as a Generator:	To:
light source	Create a stream of particles starting from a single point.
surface	Generate particles based on the area, shape, and colour of a surface.
3D object	Generate particles from the centre of each polygon.

Using a Light Source

When a particle generator is attached to a light source, it uses the position and rotation of the light source to move and rotate the particle generator.

To add a particle stream using a light source:

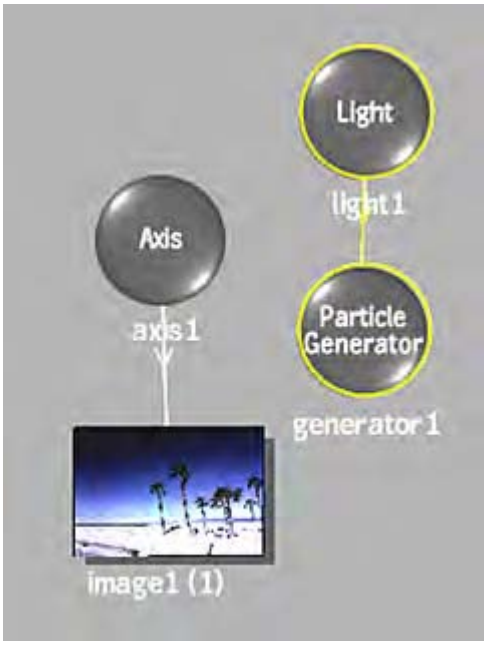
1. Add a Light node to your scene.
2. Double-click the new light in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).
3. Adjust the spread and position of the light source in the Light menu. Use the Spread field to set the spread of the particle stream. The Intensity, Falloff, and Colour fields are not used by the particle generator.



(a) Position fields (b) Spread field

4. With the light node selected in the schematic, do one of the following:
   - Drag the particle generator node from the node bin and place it in the schematic.
   - Double-click the particle generator node. You do not need to be in Schematic view to add a node in this manner.

   A particle generator object is added and automatically connected to the selected light source.



5 Set the particle generator's properties. See Customizing the Particle Stream (page 752).

6 If you do not want the light source to illuminate the scene, deactivate the light by clicking the Enable button in the Light menu. When Enable is deactivated, the light source is used as a particle generator only.

With a light source, you can animate the position of the particle stream by moving the light, changing the spread, and rotating the light. Because the generator is a light source, you can animate the position of the particle generator using a motion path.

Using a Surface

Use a surface as a particle generator to generate particles based on the area, shape, and colour of a surface. You can use a surface's matte to restrict where particles are generated. When the matte is off, the particle stream is generated from the entire surface. When the matte is on, particles are only generated where the matte is white or grey.

You can animate the shape of the surface using its tangent handles. The starting area of the particle stream matches the surface's shape animation. See Changing the Shape of a Surface (page 465).

To add a particle stream using a surface:

1 Select or add a surface that you want to use as a particle generator.

2 Adjust the surface's shape, position, and other attributes. See Modifying Surfaces (page 458).

3 With the surface node selected in the schematic, do one of the following:

- Drag the particle generator node from the node bin and place it in the schematic.
- Double-click the particle generator node. You do not need to be in Schematic view to add a node in this manner.

A particle generator object is added and automatically connected to the selected surface.



4 Set the particle generator's properties. See Customizing the Particle Stream (page 752).

5 If you do not want the surface to appear in the scene, you can hide the surface in Schematic view. A particle stream is generated from the hidden surface.

6 To make the generated particles change as the surface's clip changes, enable Indirect in the Particle Generator menu.

Using a 3D Object

You can use a 3D object as a particle generator. Particles are generated from the centre of each polygon.

To add a particle stream using a 3D object:

1 Select or add the 3D object that you want to use as a particle generator.

2 With the 3D object node selected in the schematic, do one of the following:

- Drag the particle generator node from the node bin and place it in the schematic.
- Double-click the particle generator node. You do not need to be in Schematic view to add a node in this manner.

A particle generator object is added and automatically connected to the selected 3D object.



3 Set the particle generator's properties. See Customizing the Particle Stream (page 752).

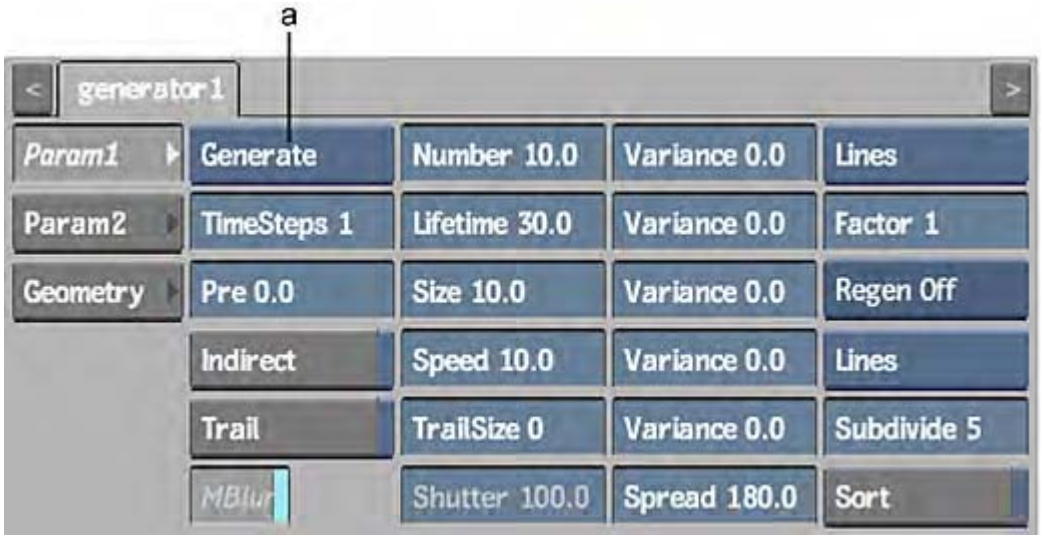
4 If you do not want the 3D object to appear in the scene, hide the model in Schematic view. A particle stream is generated from the hidden model.

Customizing the Particle Stream

Whether you use a light source, 3D object, or surface, the particle stream uses the same properties, allowing you to create custom particles and apply textures to particles. To access these properties, double-click the particle generator in the schematic, or follow the tab population rules for the Object menu. See Populating Menu Tabs of Selected Objects (page 415).

Setting Stream Properties

The following properties in the Particle Generator Param1 menu affect the particle stream.



(a) Stream Type box

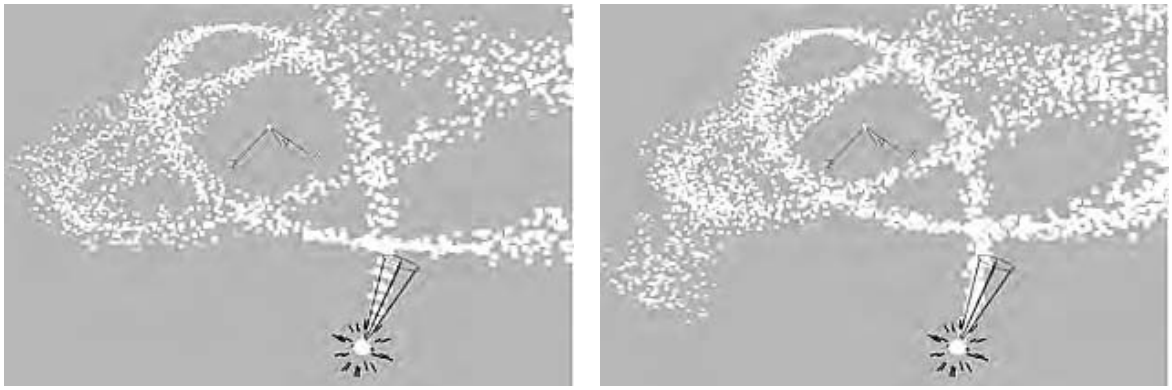
Stream Type box Select whether the particle stream will Generate or Explode.

TimeSteps field Displays the time simulation of the Generator per frame. By default, Timestep is set to 1, which means that time simulation is once per frame. Editable.

When Timestep is set to 2, the position of each particle is calculated once between frames. Any change in speed or position caused by manipulators or bouncers is applied between frames. When Timestep is set to 3, the position of each particle is calculated twice between frames. The greater the timestep value, the longer it takes to process.

Typically, you use Timestep with a particle manipulator or bouncer on a particle stream. In the following example, the same particle settings and manipulator (AccPoint) are used. Only the timestep values are different. The particle stream on the left uses a timestep of 1 while the stream on the right uses a timestep of 6.

In the following figure, notice the particle stream on the right is more accurate.



Pre field Displays the start point of the particle stream. With a value of 0.0, the particle stream starts creating particles at frame 1. With a value of 100, the clip begins as if the particle stream has been generating for 99 frames. Editable.

Indirect button Enable to allow the part of the surface used by the particle to change with each frame. For example, if the media applied to the surface changes from red to blue over 10 frames, each particle changes from red to blue as the surface changes. Disable to lock the part of the surface used by the particle to the frame where the particle was generated. For example, if the media applied to the surface changes from red

to blue over 10 frames and a particle is generated at the first frame, when the surface was red, the particle stays red.

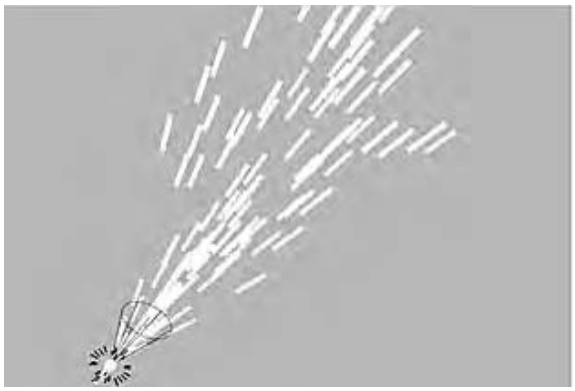
Trail button Enable to draw each particle with a trail. Set the length of the trail in the TrailSize field.

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

Shutter field Displays the length of the tail for lines, cones, and quads. Shutter is expressed as a percentage. At 100%, the tail's length is the same as the head (Size field). At 400%, the tail's length is four times the size of the head. Editable.



Shutter at 100%



Shutter at 400%

Spread field Displays the spread for the second particle generator when two particle generators are parented. When the first particle stream dies, the second particle stream begins. Editable.

See Combining Particle Streams (page 786). This also sets the spread of a particle stream that bounces off surfaces. See Bouncing Particles (page 783).

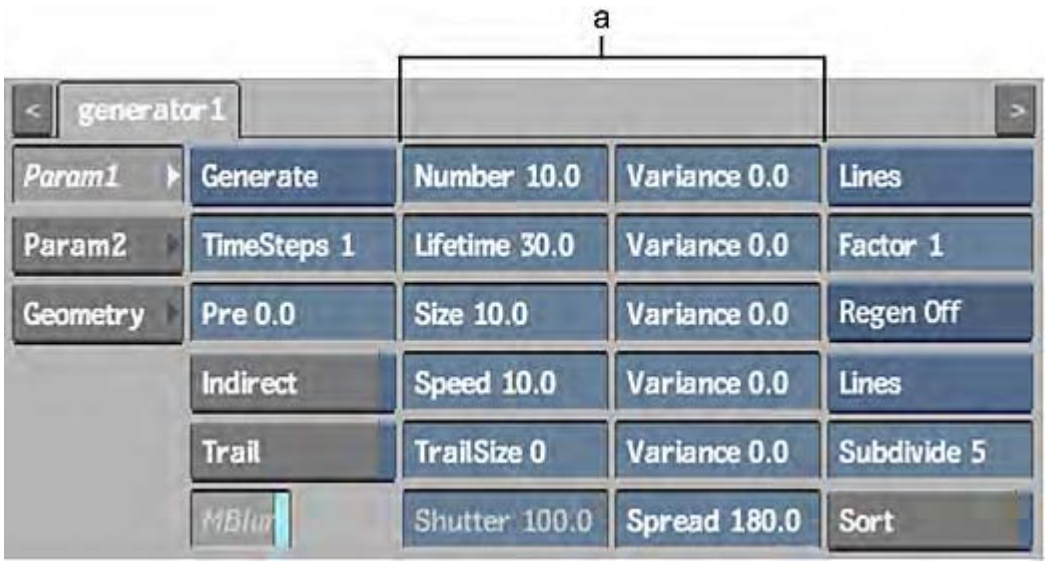
Setting Particle Properties

In the middle of the Generator Param1 menu, a table lists the different particle generation properties.

Each parameter has a variance value used to specify how much the parameter varies each time particles are generated. Exactly what gets varied depends on the parameter.

For example, if you set the Number parameter to 10 and its Variance parameter to 5, the number of particles generated at each frame is randomly selected between 5 and 15.

The unit of measurement used by each Variance field matches the parameter it is varying. For example, if the Number parameter is expressed in particles per frame, so is its Variance field. The Speed parameter is in pixels per frame and so is its Variance field.



(a) Particle generation properties

Number field Displays the number of particles generated per frame. Editable.

If you specify a number less than 1 (from 0.1 to 0.9), particles are generated randomly. For example, if you set the number to 0.1, a particle is generated every 10 frames. The exact frame within those 10 frames when the particle is generated is randomly selected. If you set the number of particles to 0.3, a particle is randomly generated approximately every three frames.

Number Variance field Displays how much the number varies each time particles are generated. Editable.

Lifetime field Displays the number of frames each particle lasts. For example, if Lifetime is 20 frames, a particle generated at frame 15 disappears at frame 35. Editable.

Lifetime Variance field Displays how much the lifetime varies each time particles are generated. Editable.

Size field Displays the size of each particle, in pixels. This parameter only affects cones, spheres, quads, squares, and objects. Editable.

Size Variance field Displays how much the size varies each time particles are generated. Editable.

Speed field Displays the speed of each particle, in pixels per frame. Editable.

Speed Variance field Displays how much the speed varies each time particles are generated. Editable.

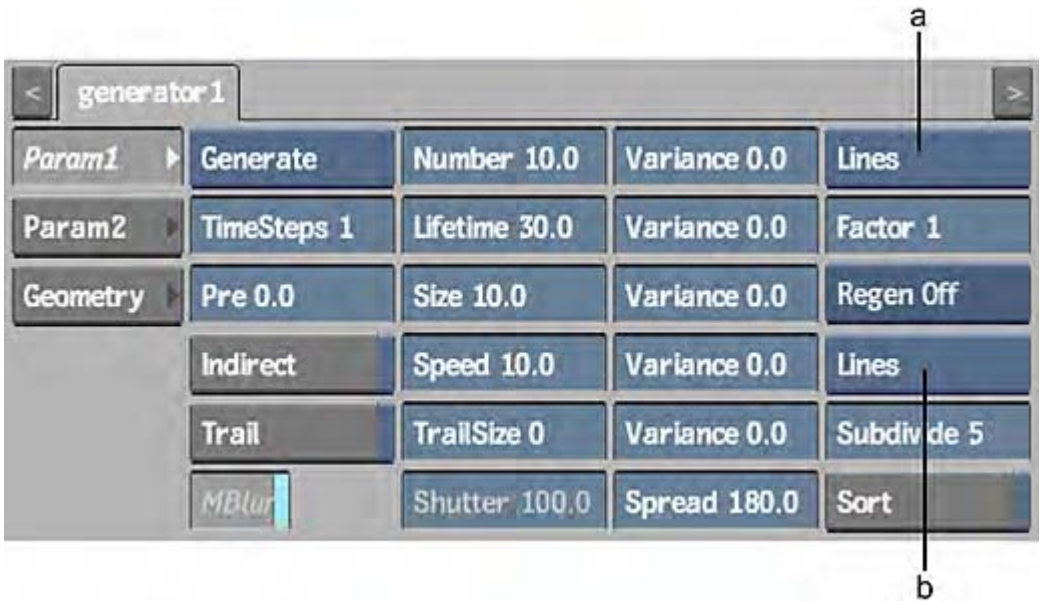
TrailSize field Displays the length of each particle's trail, in particles. A particle trail only appears when the Trail button is enabled. Editable.

TrailSize Variance field Displays how much the trail size varies each time particles are generated. Editable.

Setting Particle Rendering Properties

The following Generator properties affect the rendering and previewing of the particle stream.

NOTE Rendering particles is faster if Z-Buffer On is selected in the Rendering Section of the Action Setup menu.



(a) Interactive Type box (b) Render Type box

Interactive Type box Select the type of particle to display in the scene. For example, as you move between frames, the particle stream is redrawn. If you are using a custom 3D object for each particle, redrawing the scene may take a while. By changing the interactive type to a point or line, your interaction with Action is much quicker.

Factor field Displays the number of particles shown in the scene while working in Action (use to speed up work - does not affect the rendered result). Editable.

For example, if you generate 1000 particles with manipulators and bouncers, interacting with Action may become slow because calculations have to be made for each particle. If you specify a factor of 10, the number of particles is reduced to 10%, or 100 particles. Using Action is then faster, yet when you click Preview or Process, 1000 particles are created.

Regenerate box Select how you want to update the scene when you change particle generator parameters. Regen All updates the particle and the object generating the particle, but may cause performance slowdowns. Regen Anim only updates the particle in the scene.

Select:	To:
Regen All	Automatically update the particle and the object generating the particle (such as a surface) when you change any particle generator parameter. This option allows you to see the particle with all settings, such as position and colour correction, but may cause performance slowdowns, since the generating object and particle stream are recalculated each time a parameter is changed.
Regen Anim	Automatically update the particle in the scene when you change any particle generator parameter. The object generating the particle is not automatically updated in this case, so you may not see your exact desired results. If Regen All is causing slowdowns, you can try this option to get a faster update.
Regen Off	Not update the scene automatically, unless the frame is changed.

Render Type box Select the type of particle used when rendering the scene.

Select:	To:
Node	Use your own 3D object, text, or a surface for each particle. See Using Custom Particles (page 759).
Cones	Use a cone for each particle. You can adjust the size of the cone's tail.
Spheres	Use a 3D object of a sphere for each particle.
Polygons	Explode 3D objects and surfaces. See Exploding Objects and Surfaces (page 762).
Quads	Use a square with a tail for each particle. You can adjust the size of the square's tail.
Squares	Use a square for each particle.
Lines	Use a line for each particle.
Points	Use a single pixel for each particle. Points are not affected by size or trail.

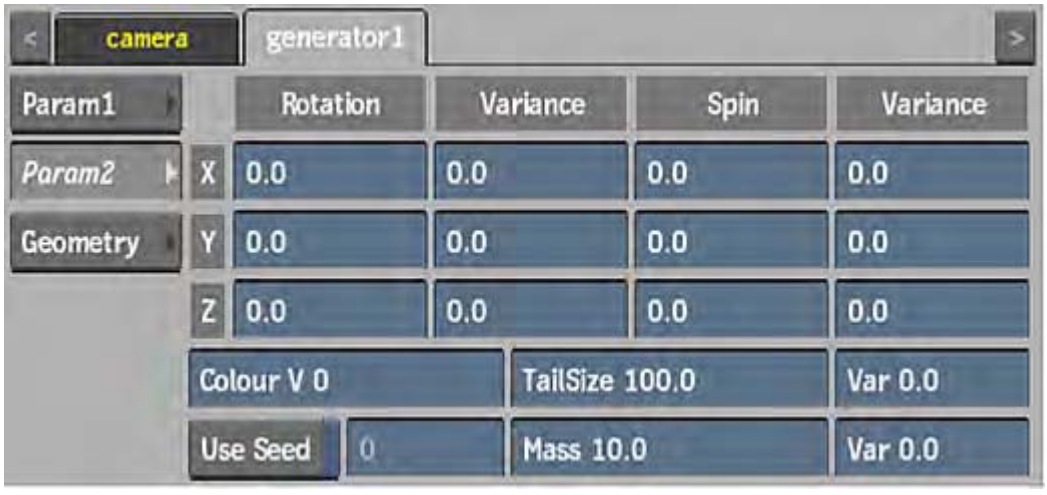
Subdivide field Displays the number of polygons used to render cones and spheres. The larger the number of subdivisions, the smoother the cones or spheres are drawn, but rendering time is increased. Editable.

If you are using an imported 3D object or text for a particle, use the Subdivide field and button in the Geometry menu to set the number of polygons. The Geometry menu's Subdivide field works differently than the Subdivide field in the Generator menu. See Subdividing a Model (page 543).

Sort button Enable to determine how the particles are drawn, according to the Z-order. If a Particle Generator node is selected, the state of the Sort button is the same in the Param1 and Geometry menus.

Setting Additional Generator Properties

Additional particle generator controls can be found in the Param2 tab.



X Rotation field Displays the X orientation for particles that have just been born (spawned). Editable.

Y Rotation field Displays the Y orientation for particles that have just been born (spawned). Editable.

Z Rotation field Displays the Z orientation for particles that have just been born (spawned). Editable.

X Rotation Variance field Displays the variation of the X rotation of a particle once spawned from the particle generator. Editable.

Y Rotation Variance field Displays the variation of the Y rotation of a particle once spawned from the particle generator. Editable.

Z Rotation Variance field Displays the variation of the Z rotation of a particle once spawned from the particle generator. Editable.

X Spin field Displays a value to act as a speed setting to an X rotation animated over time. Editable.

Y Spin field Displays a value to act as a speed setting to a Y rotation animated over time. Editable.

Z Spin field Displays a value to act as a speed setting to a Z rotation animated over time. Editable.

X Spin Variance field Displays a variation to the X Spin channel once particles are spawned from the particle generator. Editable.

Y Spin Variance field Displays a variation to the Y Spin channel once particles are spawned from the particle generator. Editable.

Z Spin Variance field Displays a variation to the Z Spin channel once particles are spawned from the particle generator. Editable.

Colour Var field Displays the variation of the colour of each particle based on the colour set in the Diffuse colour bar (Particle Geometry menu). Editable.

Use Seed button Enable to override the particle's seed (useful if you want to change the position of each particle in the particle stream).

Seed field Displays the random seed value for the particle stream properties. Editable.

TailSize field Displays the width of a particle's tail, expressed as a percentage. Works only with quad and cone particle types. Editable.

TailSize Variance field Displays the variation of the TailSize value, as a percentage. Editable.

Mass field Displays the mass of each particle (used with Damping when manipulating particles). Editable.

See Setting Manipulator Properties (page 765).

Mass Variance field Displays the variation of the mass, as a percentage. Editable.

Setting Geometry Properties

You can use the Particle Geometry menu controls to add colour, specular highlight, shine, and transparency to all particles. To access this menu, click the Geometry tab in the Generator menu.

Use the available particle geometry controls to affect the particle stream, whether it is a surface or a geometry. The controls behave in the same manner as when using with the Geometry menu for a 3D model. See 3D Geometry Menu Settings (page 538).

Setting Channel Editor Properties

The Channel Editor includes the particle properties included in the Particle menus for which you can set values and create particle animation effects. You set these properties in conjunction with those located in the Particle Generator, Particle Manipulator, and Particle Bouncer menus.

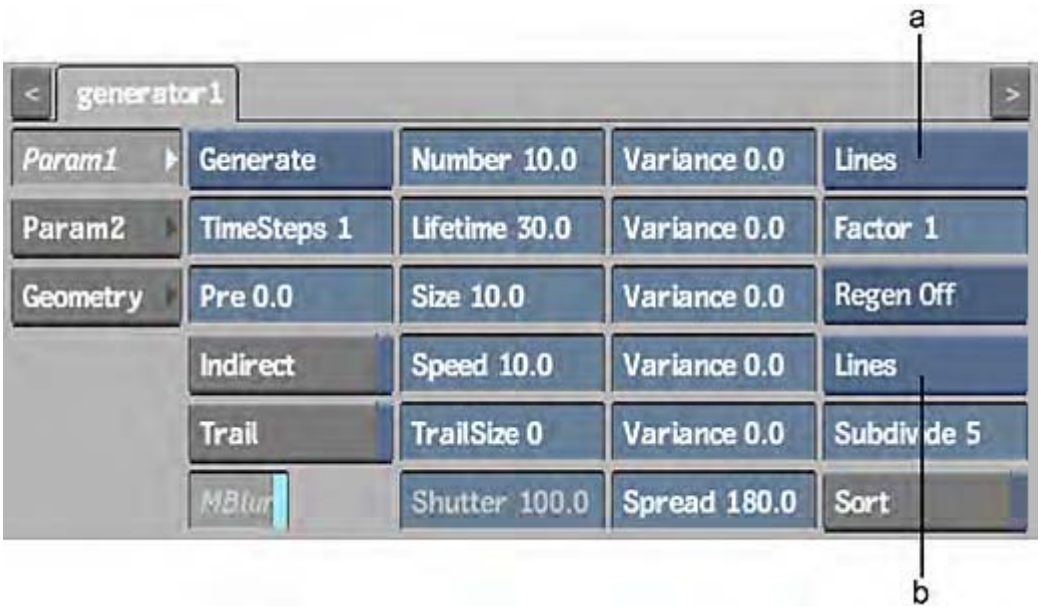
You must first add a particle generator, a particle manipulator, or a particle bouncer to the scene to view and set values for their corresponding properties in the channel hierarchy of the Channel Editor.

Using Custom Particles

You can create particle streams using your own 3D objects or text. You can also use the shape of any surface for each particle. For example, create custom particles using a light source, 3D object, or surface as a particle generator. See Generating Particles (page 749).

To use a 3D object or text as a custom particle:

1. From the node bin, add a particle generator to your scene.
2. In the schematic, double-click the Particle Generator node to access the Generator menu.



(a) Interactive Type box (b) Render Type box

3. From the Render Type box, select Node as the particle type and change the Interactive Type box to Node to view the 3D object or text as you make changes to your particle stream.
4. Import the 3D object or create the text. See Importing 3D Models (page 532) or About 3D Text (page 550).

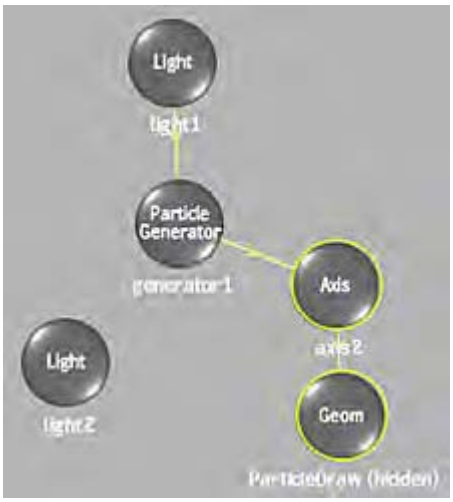
   TIP The 3D object or text used for the custom particle can also include a deformation mesh. See Deforming Models and Surfaces (page 544).

5. In Schematic view, rename your 3D object or text “ParticleDraw”. This name is case sensitive so make sure it is typed exactly as shown.
6. Parent the particle generator to the 3D object or text.

   If you want to add transformations, such as rotating all the particles, parent the particle generator to the axis of the 3D object or text as shown in the next step.

7 Hide the 3D object or text.

The particle generator uses the 3D object or text named ParticleDraw. The following figure shows the schematic and the result of using the letter A as a custom particle and a light source as the particle generator.





To use a surface as a custom particle:

1 Add a particle generator.

You can use a light source, 3D object, or surface as a particle generator. See Generating Particles (page 749).

2 Select Node as the particle type in the Render Type box. To view the surface as you make changes to your particle stream, change the Interactive Type box to Node as well. See Setting Particle Rendering Properties (page 755).

3 Add a surface to the scene. It can be an image, a bilinear, a bicubic, or an extended bicubic surface. See Adding Surfaces (page 458).

NOTE You do not have to apply media to the surface because the particle generator only uses the surface's shape and not its media. To have an image appear on each particle, you have to apply a texture to the particle generator. See Applying Textures to Particles (page 761).

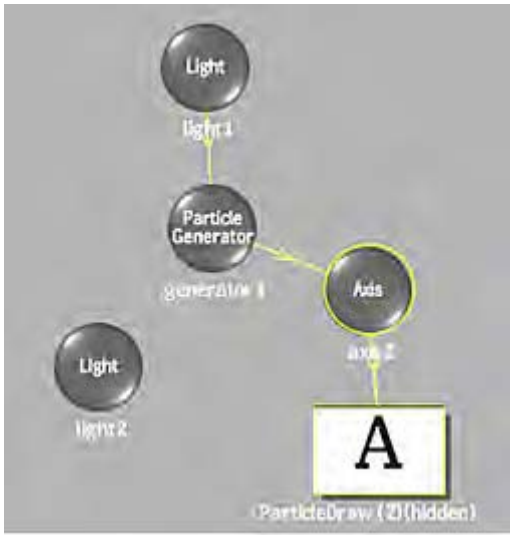
4 Rename your surface “ParticleDraw”. This name is case sensitive so make sure it is typed exactly as shown.

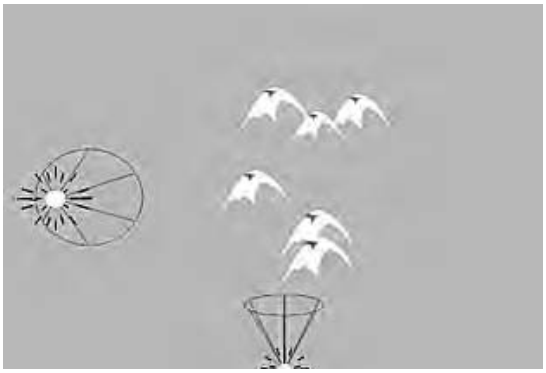
5 Parent the particle generator to the surface.

If you want to add transformations, such as rotating all the particles, parent the particle generator to the axis of the surface as shown in the next step.

6 Hide the surface.

The following figure shows the schematic and the result of using a bicubic surface as a custom particle and a light source as the particle generator. The shape of the bicubic surface is animated and its axis is used to rotate the particles.





NOTE You can animate the shape of a bilinear, bicubic, or extended bicubic surface, and have the particles reflect the animation.

Applying Textures to Particles

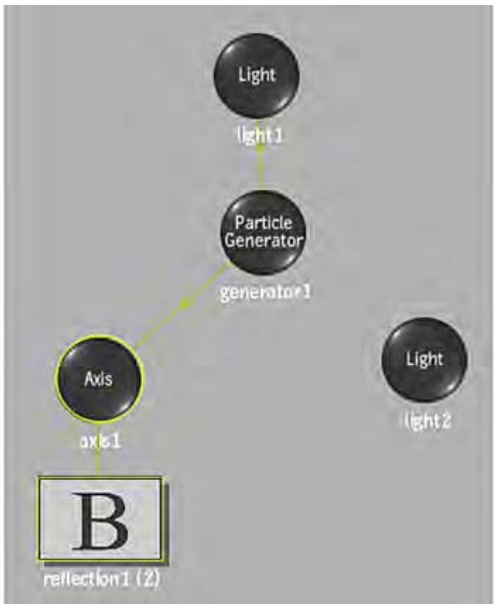
You can apply textures to particles using the same method as applying textures to 3D objects and text. You can also set its texture mapping using the Texture menu.

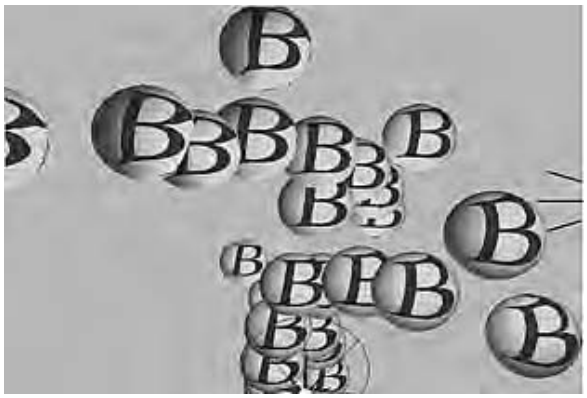
To apply a texture to a particle stream of spheres:

1. Create a particle generator.
2. From the Render Type box, select Spheres as the particle type. To view the 3D object or text as you make changes to your particle stream, change the Interactive Type box to Spheres as well. See Setting Particle Rendering Properties (page 755).
3. In the schematic, select the Particle Generator.
4. From the Node bin, double-click the a texture map node.

   A texture is added using the currently selected media and is parented by the particle generator.
5. Apply the appropriate media to the texture object and set texture menu settings.

For example, the following shows the schematic and the result of a particle stream using spheres and a reflection map, making each particle resemble a glass ball.





Exploding Objects and Surfaces

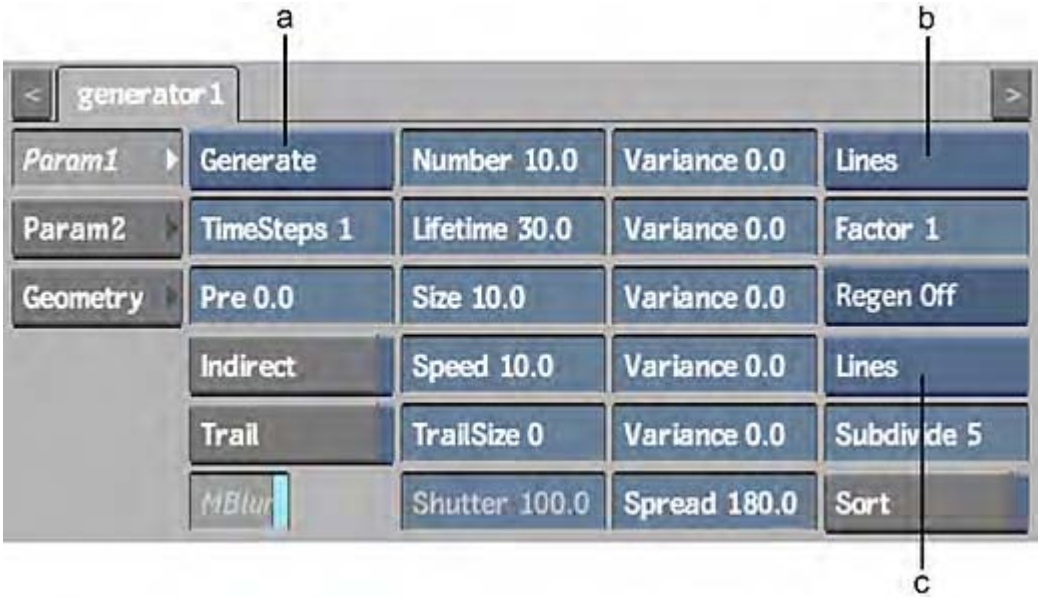
Whether you use a 3D object or a surface, exploding works much the same way. The shape of the 3D object or the surface is recreated by the particle generator, then the shape is exploded into polygons. For this to work correctly, the original 3D object or surface must be hidden.

Exploding Objects

The properties used to explode an object cannot be animated. When something explodes, it happens at a single point in time. This means that all the settings in the Particle Generator menu are used at once, at the beginning of the explosion.

To explode a 3D object:

1. Import a 3D object or create 3D text.
2. In the schematic, select the 3D object.
3. Add a particle generator. See Generating Particles (page 749).
4. From the Particle Generator menu, select Explode from the Stream Type box.



(a) Stream Type box (b) Interactive Type box (c) Render Type box

5 From the Render Type box, select Polygon as the particle type.

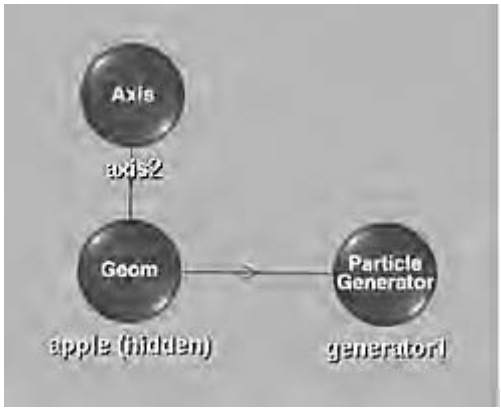
To view the explosion as you make changes to your particle stream, change the Interactive Type box to Polygon as well. See Setting Particle Rendering Properties (page 755).

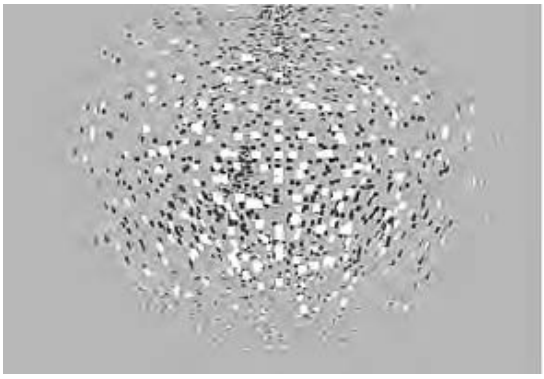
6 In the Number field, set the number of polygons to be removed from the 3D object with each pass. For example, if you specify 10 as the number of polygons per frame, then at each frame, 10 polygons are removed from the 3D object.

7 Set the other properties in the PartGen menu. See Customizing the Particle Stream (page 752).

8 Hide the 3D object or 3D text.

The following figure shows the schematic and the result of exploding a 3D object of an apple. The number of polygons is set to 200. The result shows frame 20.





Exploding Surfaces

To make the media applied to the surface appear during the explosion, you must also apply a texture to the particle generator.

To explode a surface:

1. Select or add a surface and adjust its shape, position, rotation, and other attributes. See Adding Surfaces (page 458).
2. Select the surface in Schematic view and add a particle generator. See Generating Particles (page 749).

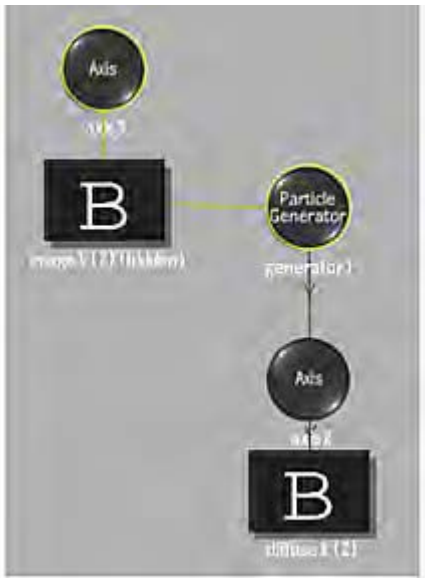
   A particle generator is automatically connected to the surface.
3. From the Particle Generator menu, select Explode as the Stream Type.
4. Select Polygon as the particle type from the Render Type box. To view the explosion as you make changes to your particle stream, change the Interactive Type box to Polygon as well. See Setting Particle Rendering Properties (page 755).
5. Set the other properties in the Particle Generator menu. See Customizing the Particle Stream (page 752).
6. In the Rendering section of the Action Setup menu, adjust the default resolution.

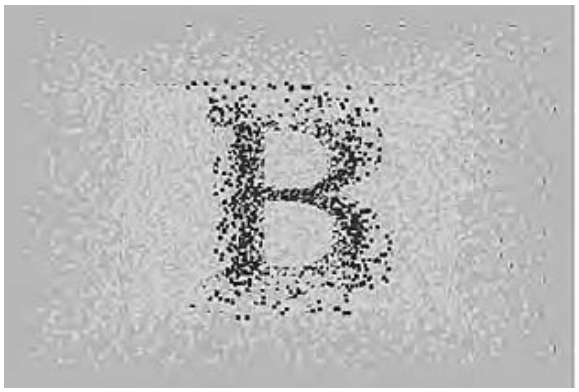


   Resolution is used to set the size of the polygons exploded from the surface. The higher the resolution, the larger the polygon.
7. Hide the surface.
8. Select the particle generator in Schematic view.
9. From the Node bin, double-click the Diffuse Map node.

   A diffuse object appears in the schematic and is parented by the particle generator. See Diffuse Mapping (page 615).
10. Double-click the diffuse object and, in the Diffuse menu, select Wrap from the Mapping box.
11. Apply the appropriate media to the Diffuse element and set any other Diffuse menu settings.

The following figure shows the schematic and the result of exploding a surface with diffuse mapping set to Wrap.





Manipulating Particles

You use manipulators to direct the particle stream and create a number of different effects, such as a swirling vortex, a swarm of objects orbiting a point, or a waterfall. A particle stream can have an unlimited number of manipulators. Manipulators only influence the particle stream that parents them.

When manipulating particles, you use the Particle Manipulator menu to set a manipulator's type, magnitude, power, and damping properties, as well as position, rotation, and scale.

Particle manipulators can be animated using motion paths. You can use the same operations on a particle manipulator that you use on an axis.

Creating a Particle Manipulator

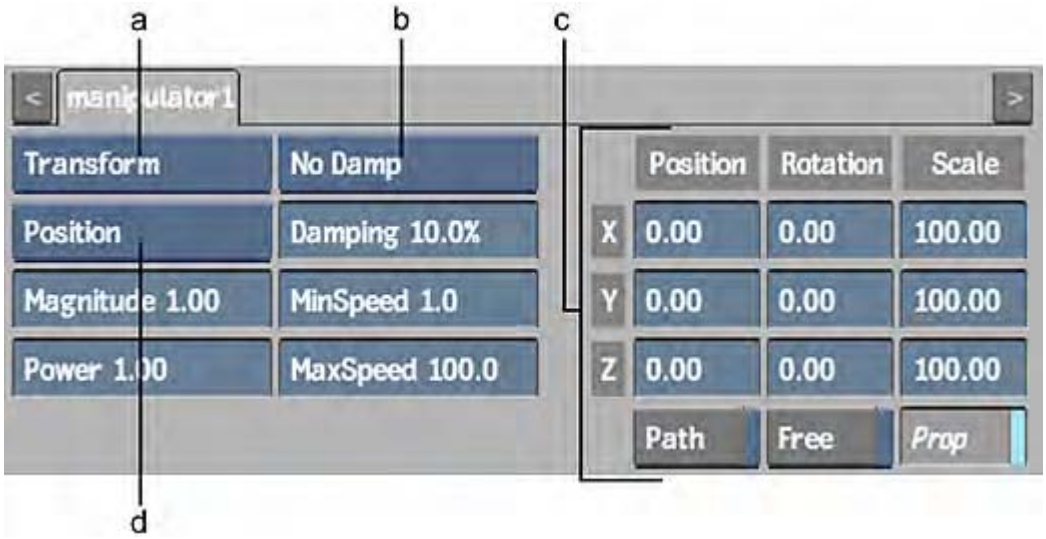
To create a particle manipulator:

1. In the schematic, select the particle generator that you want the manipulator to influence.
2. From the Node bin, double-click the Particle Animator node.

   A particle animator element is added to the scene and parented by the selected Particle Generator element.

Setting Manipulator Properties

You use the Particle Manipulator menu to select the type of manipulator, its speed and position, and the power of its falloff. To access the Particle Manipulator menu, double-click the particle animator node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).



(a) Manipulator Type box (b) Damping box (c) Axis controls (d) Influence box

The particle manipulator controls are described as follows.

Manipulator Type box Select the type of manipulator.

Select:	To:
Gravity	Simulate the effect of gravity. See Simulating Gravity (page 769).
Transform	Apply transformations from the Axis menu to the position or speed of each particle. See Applying Transformations (page 768).
Vortex	Mimic the effect of a vortex. See Creating a Vortex Effect (page 770).
Acceleration Point	Pull particles toward a point. See Using an Acceleration Point Manipulator (page 771).
Acceleration Line	Pull particles toward a line. See Using an Acceleration Line Manipulator (page 772).
Acceleration Plane	Pull particles toward a plane. See Using an Acceleration Plane Manipulator (page 773).
Path	Make the particles follow a path. See Forming a Particle Path (page 774).
Function	Enter a mathematical expression to use as a particle manipulator. See Using a Function as a Manipulator (page 775).

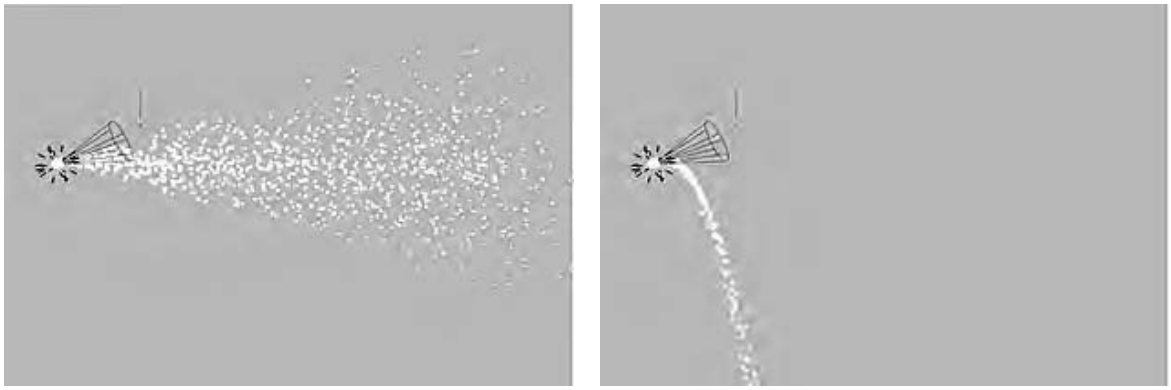
Influence box Select whether the manipulator influences each particle's position or speed.

Set the initial speed of each particle using the Speed field in the Surface Particle Generator menu.

By selecting Speed in the Influence box located on the Particle Manipulator menu, each particle's speed is changed with each pass by the selected manipulator.

When you influence by position, the position is applied only once. For example, the following figure uses a gravity manipulator to illustrate the difference between position and speed. The gravity manipulator on

the left uses the default gravity manipulator with a magnitude of 4 and with position as the influence. The gravity manipulator on the right uses exactly the same settings except the influence is set to speed.



Magnitude field Displays the unit of measurement specific to each manipulator. Editable.

Magnitude is used by each manipulator for everything from extra scale when using Transform, to a pixel per frame acceleration factor when using AccPoint, AccLine, or AccPlane.

Power field Displays the amount of falloff from the centre of a manipulator. Editable.

If you specify 0 as the power, there is no falloff; the manipulator's influence is universal. This means that a particle is affected no matter where it is located in the scene. A negative value makes particles that are farther away affected by the manipulator.

Damping box Select the type of friction to apply (damping affects the speed of particles by applying friction based on each particle's mass, size, or both).

Select:	**To:**
No Damp	Turn off damping. Particles have no friction.
Damp Mass	Activate damping based on mass. The greater a particle's mass, the slower its speed in relation to the manipulator. You set the mass using the Mass channel in the Channel Editor.
Damp Size	Activate damping based on particle size. The larger the particle, the slower it moves in relation to the manipulator. This corresponds with the Size field in the Surface Particle Generator menu.
Damp Both	Activate damping based on mass and particle size.

Damping Percentage field Displays the percent value of how much Damping affects the position or speed of the particles. Editable.

Minimum Speed field Displays the minimum speed range for particles affected by Damping. Editable.

Maximum Speed field Displays the maximum speed range for particles affected by Damping. Editable.

NOTE You can use the Min and Max Speed fields to force a minimum and maximum speed on particles without damping. To do this, select Damp Mass, Damp Size, or Damp Both. Change the Damping Percentage to 0% and change the Min and Max Speed fields accordingly. Use this feature to stop particles from being over accelerated. For example, when you use a vortex manipulator, the closer particles pass to the centre of the vortex, the more they are accelerated. Occasionally, a particle may be over accelerated and shot too far, too fast. By forcing a maximum speed, particles are not accelerated past a certain value.

Load button Not Shown; available when Function is selected as the Manipulator type. Loads a mathematical expressions to be used as a particle manipulator.

Save button Not Shown; available when Function is selected as the Manipulator type. Saves a mathematical expressions to be used as a particle manipulator.

Expression field Not Shown; available when Function is selected as the Manipulator type. Displays a mathematical expression to be used as a particle manipulator. Editable.

See Using a Function as a Manipulator (page 775).

Axis controls Use these controls to animate the position, rotation, and scale of the manipulator.

Control	Description
Position	Use the Position X, Y, and Z fields to displace each particle along the X, Y, and Z axes.
Rotation	Use the Rotation X, Y, and Z fields to rotate each particle. Each particle is not the centre of its rotation. Particles rotate around the manipulator's axis.
Scale	Use the Scale X, Y, and Z fields to increase or decrease the speed or position of each particle along the X, Y, and Z axes. For example, if speed is the selected influence, a Scale X of 100% has no effect while a Scale X of 105% increases the speed along the X axis by 5% with each pass. Enable Prop to proportionally scale the axes.

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

Free button Enable to ignore transformations from parent axes.

Applying Transformations

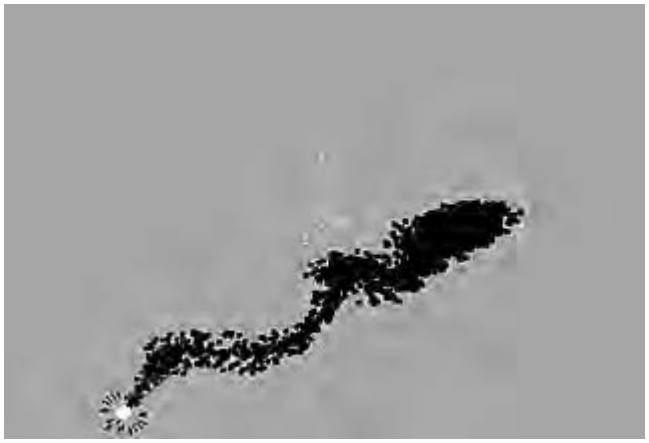
Use a transform manipulator to apply the accumulated transformations from the Axis controls to the position or speed of each particle. Unlike other manipulators, the transform manipulator is not used to position the manipulator in the scene. Transform manipulators are always placed at the centre of the particle stream they influence.

The following figure provides an example of creating a particle stream influenced by a transform manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.





In the Particle Manipulator menu, use Magnitude as an extra scaling factor whose sensitivity depends on the transformations in the Axis menu. For example, the following is a copy of the above particle stream with Magnitude set to 0.99 instead of 1.0; the particle generator and transform settings are the same.



Simulating Gravity

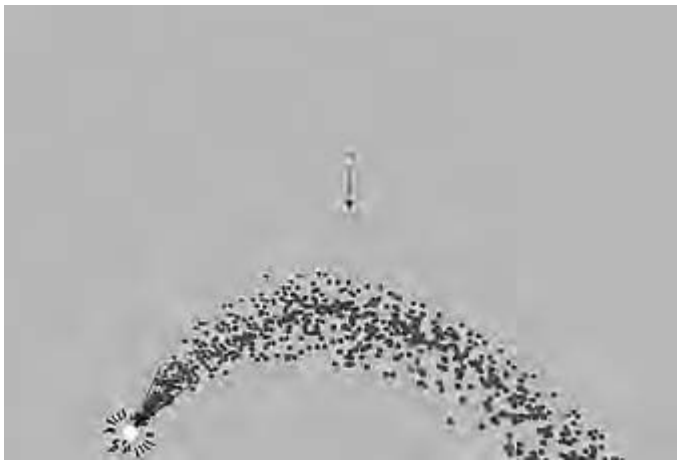
Use a gravity manipulator to simulate the effects of gravity. This manipulator works on speed or position. The Power field has no effect.

Magnitude specifies the gravitational pull in pixels per frame. A higher magnitude produces a higher gravity, which exerts a greater influence on each particle.

The icon for a gravity manipulator uses an arrow to indicate the direction of the influence. You can change the position and rotation of the arrow using the Axis menu. You can also change the scale of the gravity manipulator as another way of increasing its influence.

The following figure provides an example of creating a simple particle stream influenced by the gravity manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.





Creating a Vortex Effect

Use a vortex manipulator to influence the motion and rotation of particles. This manipulator works on speed or position. Power is the falloff from the centre of the vortex and magnitude is used for the scale of the vortex's rotation.

You can also use the Axis controls to apply additional transformations to the vortex. Scale changes the size of the vortex, and rotation affects the spin of the vortex.

The following figure provides an example of creating a simple particle stream influenced by the vortex manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.





NOTE The icon used to show a vortex does not indicate the limit of the vortex. The icon is only used to represent the vortex in the scene and is not meant as an accurate depiction of the size, strength, or limit of the vortex's influence.

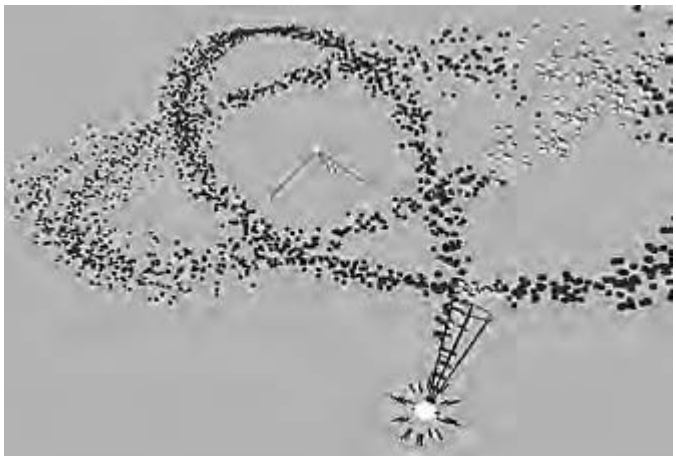
Using an Acceleration Point Manipulator

The Acceleration Point manipulator mimics the gravitational pull between bodies. Unlike the gravity manipulator, which pushes particles in a specific direction, Acceleration Point continually pulls particles toward itself on all three axes (X, Y, Z).

Magnitude is the gravitational pull in pixels per frame. A higher magnitude produces a higher gravity, which results in a greater influence. Power is the distance from the centre, or the radius, of the pull.

The following figure provides an example of creating a simple particle stream influenced by the Acceleration Point manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.





NOTE The Axis controls are used only to position the Acceleration Point in the scene. Scale and rotation have no effect.

Using an Acceleration Line Manipulator

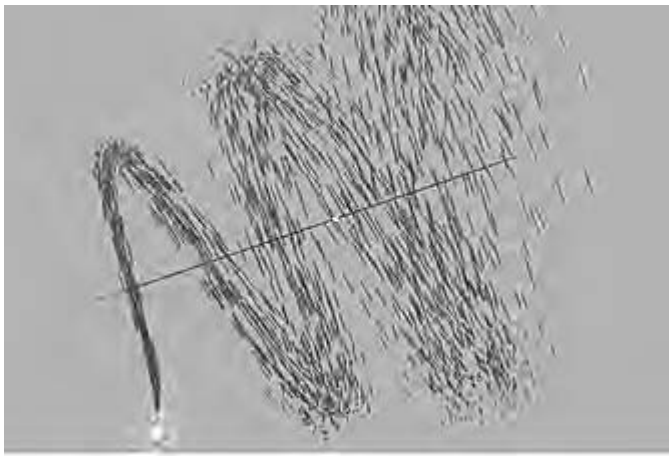
Like Acceleration Point, the Acceleration Line manipulator continually pulls particles toward itself. Acceleration Line, however, manipulates particles on two axes instead of all three.

You can set the two axes by rotating the manipulator using the Rotation fields. The Axis controls are also used to position the Acceleration Line manipulator in the scene. Scale may change the appearance of the Acceleration Line icon, but has no effect on particles.

Magnitude is the gravitational pull in pixels per frame. A higher magnitude produces a higher gravity, which results in a greater influence. Power is the falloff from the manipulator, or the radius of the pull.

The following figure provides an example of creating a simple particle stream influenced by the Acceleration Line manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.





Using an Acceleration Plane Manipulator

Like Acceleration Point and Acceleration Line, the Acceleration Plane manipulator continually pulls particles toward itself. With Acceleration Plane, however, particles are manipulated on only one axis set by rotating the plane using the Rotation fields.

Use the Axis controls to position the Acceleration Plane manipulator in the scene. The Scale and fields are not used.

Magnitude is the gravitational pull in pixels per frame. A higher magnitude produces a higher gravity, which results in a greater influence. Power is the falloff from the manipulator, or the radius, of the pull.

The following figure provides an example of creating a simple particle stream influenced by the Acceleration Plane manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.





NOTE Because Acceleration Plane influences particles on only one axis, the effect may not be noticeable until you change views or orbit the camera.

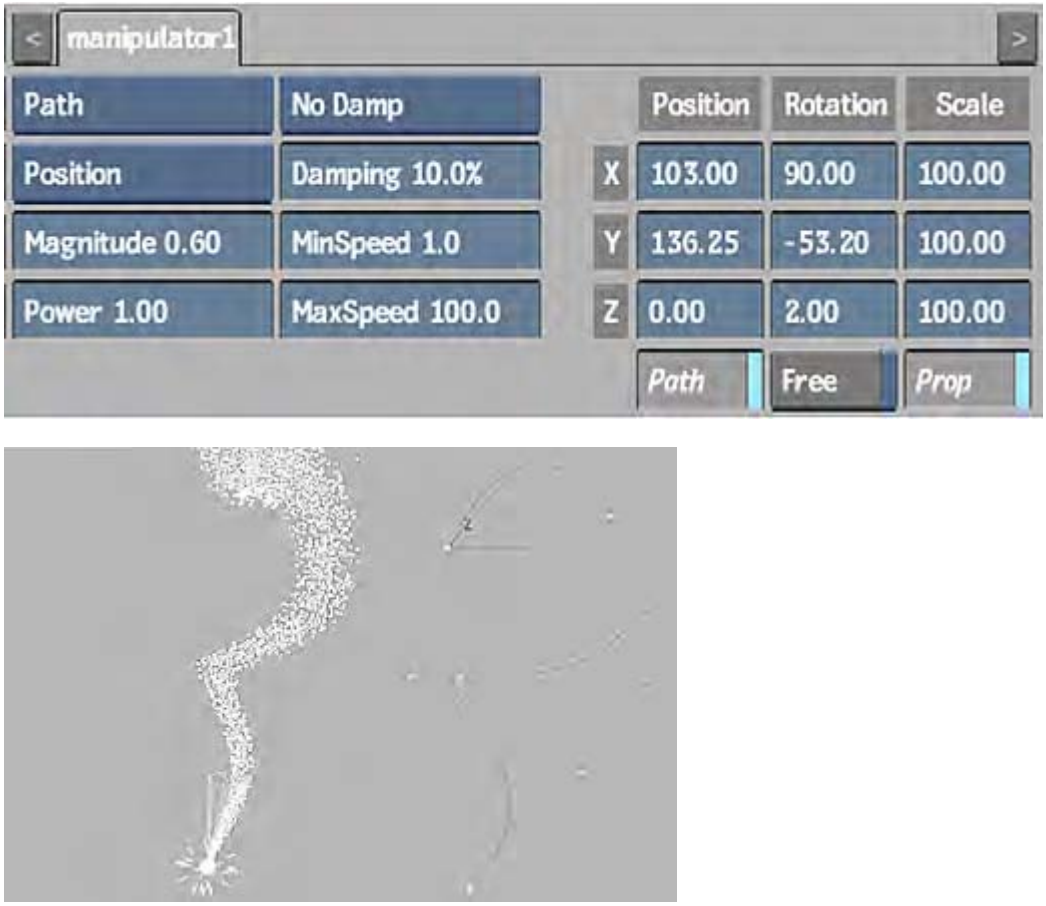
Forming a Particle Path

Use the path manipulator to make particles follow the motion of a path.

The motion of a path is applied to the speed or position of each particle. Note that particles do not directly follow the path. You can make the particles seem to follow the path by lowering the particle speed and magnitude, which is used as a scaling factor. Power is not used.

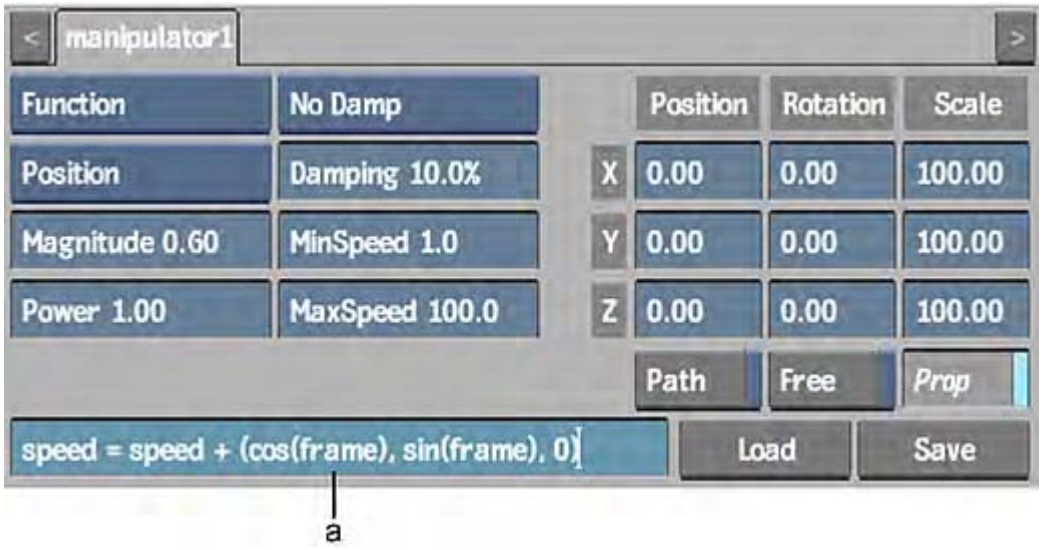
Use the Axis controls to animate the path. You can also use the Rotation and Scale fields to add extra transformations.

The following figure provides an example of creating a simple particle stream influenced by the path manipulator. The settings in the Particle Manipulator menu and the resulting effect are shown.



Using a Function as a Manipulator

Select Function to specify your own mathematical expressions to be used as a particle manipulator. You can use the channels from the Particle Manipulator and Particle Generator menus in your expressions, as well as arithmetic operations, mathematical conventions, functions, and constants.



(a) Expression field

The arithmetic operators, conventions, constants, and functions that you can use in your expressions are listed in the following sections. Before writing your own expressions, you should understand the following:

- A vector is a 3D coordinate written using the convention (x, y, z) where x, y, and z are separate values.

  For example, pos represents the position of each particle. If you want to increase the Y position of each particle with each pass, use the expression pos = pos + (0,1,0).

- Make sure assigned values and vector values are within an acceptable range. For example, transparency (opacity of each particle) is a value between 0 and 1. The expression transparency = size does not work unless size (particle size from the PartGen menu) falls between 0.0 and 1.0.

  To make transparency dependent on size, size must be divided by an appropriate value. For example, if size is between 1 and 10, use the expression transparency = size / 10.

- Some functions return scalars and other functions return vectors. Make sure that when you use a function, it returns the right value and that this value is within an acceptable range.

  For example, rgb is a vector of values between 0.0 and 1.0. The expression rgb = (0, 0, noise3(pos)) gives an error because noise3 returns a vector. The expression rgb = (0, 0, frame) does not give an error, but frame is the frame number and it is never less than 1. This means the blue channel is always set to full blue (1).

- You can specify more than one function by separating each with a semicolon.

  For example, to place the two expressions in the Expression field, you would type:

  speed = speed + pos; pos = pos + (0,size,0)

Expression field operators are listed in the following table.

Operator	Description
=	Equals
+	Addition
-	Subtraction

Operator	Description
*	Multiplication
/	Division
%	Percentage
(x,y,z)	Vector where *x, y, z* may also be the results of functions
==	Equivalence
!=	Not equal to
<	Less than
<=	Less than or equal to
>	Greater than
>=	Greater than or equal to

Particle and manipulator variables that you can use in the Expression field are listed in the following table.

Particle Symbol	Description
pos	Position, a vector (x,y,z).
speed	Speed, a vector (x,y,z).
rgba	Red, green, blue, and alpha colour channels for each particle, expressed as a 4D vector (r,g,b,a). Each component is a value between 0 and 1.
rgb	Red, green, and blue colour channel, expressed as a vector (r,g,b). Each component is a value between 0 and 1.
red	Red channel, a value between 0 and 1.
green	Green channel, a value between 0 and 1.
blue	Blue channel, a value between 0 and 1.
transparency	Transparency of each particle (surface or geometry), a value between 0 and 1.
lifetime	Lifetime of each particle, in frames.
lifetimeI	A value between 1.0 and 0.0 where 1.0 is when a particle is first generated and 0.0 is when it ends.

Particle Symbol	Description
mass	Mass of each particle.
size	Size of each particle, in pixels.
tailSize	Width of the particle's tail, a value between 0 and 1.

The following variables are read only. You cannot change them in the Expression field, but you can use them in your calculations.

Manipulator Symbol	Description
frame	Current frame
magnitude	Value from the Magnitude field
power	Value from the Power field
damping	Value from the Damping field
minSpeed	Value from the MinSpeed field
maxSpeed	Value from the MaxSpeed field

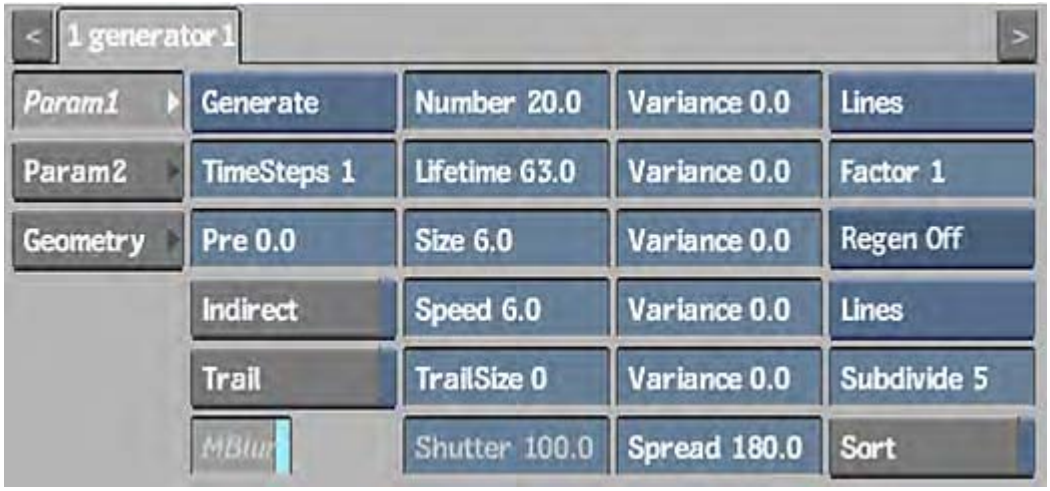
Single argument arithmetic functions are listed in the following table.

One Argument	Description
sin(*a*)	Sine of *a*
cos(*a*)	Cosine of *a*
tan(*a*)	Tangent of *a*
asin(*a*)	Arcsine of *a*
acos(*a*)	Arccosine of *a*
atan(*a*)	Arctangent of *a*
exp(*x*)	Exponential function of *x*
expm1(*x*)	Equivalent to exp(*x*)-1
log(*x*)	Natural logarithm of *x*
log10(*x*)	Base 10 logarithm of *x*

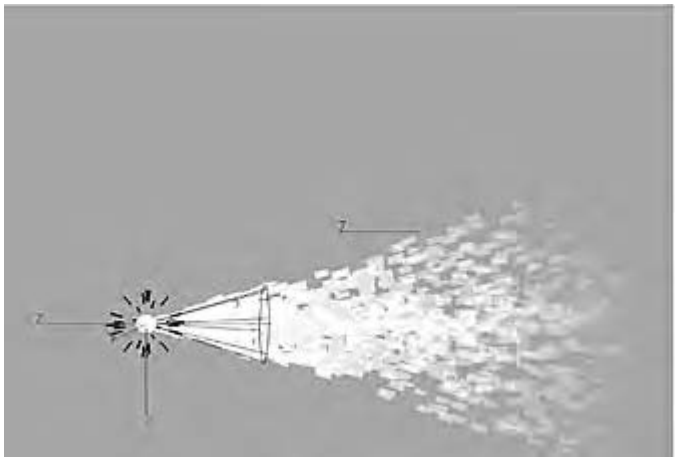
One Argument	Description
log1p(*x*)	Equivalent to log (1+ *x*)
sqrt(*x*)	Square root of *x*
abs(*x*)	Absolute value of *x*
trunc(*x*)	Integer value of *x*
floor(*x*)	Smallest integer greater than or equal to *x*
ceil(*x*)	Largest integer greater than or equal to *x*
round(*x*)	*x* rounded to the nearest integer
radians(*a*)	*a* converted to radians
degrees(*r*)	*r* converted to degrees
sign(*x*)	Returns +1 or -1 depending on the sign of *x*
length(*p*)	Euclidean length of point *p*
noise(*v*)	Noise of vector *v*, returns a float
fnoise(*v*)	Fractal noise vector *v*, returns a float
noise3(*v*)	Noise of vector *v*, returns a vector

Making Particles Transparent

The following example illustrates how to make particles become transparent as they reach the end of their lifetime using the expression transparency = lifetimeI. The settings in the Particle Manipulator menu, Particle Generator menu, and the resulting effect are shown.







Making Particles Spin

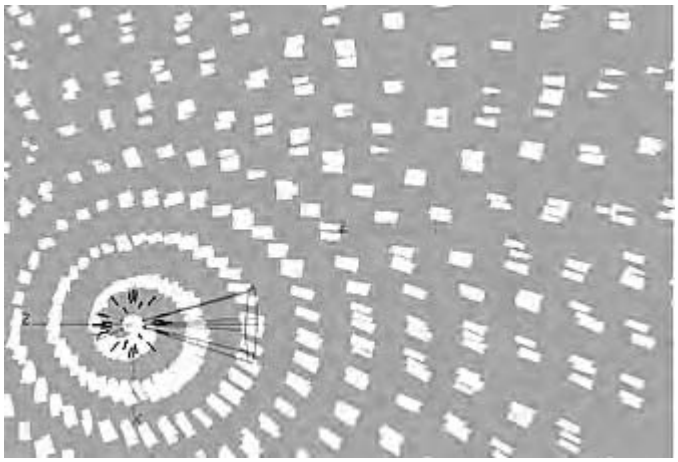
The following example illustrates how to spin particles using the expression:

speed = speed + (cos(frame),sin(frame),0)

The settings in the Particle Manipulator menu, Particle Generator menu, and the resulting effect are shown. Notice that the particle Timestep value is increased to improve the accuracy of the particle stream.







Arithmetic functions that have two or more arguments are listed in the following table.

Symbol	Description
atan(*x,y*)	Arctangent of *y* over *x*
pow(*x,y*)	*x* to the power of *y*
mod(*x,y*)	Returns the remainder of dividing *x* by *y*
min(*x,y*)	Minimum value of *x* and *y*
max(*x,y*)	Maximum value of *x* and *y*
step(*x,y*)	Returns 0 if *x* < *y*, 1 if *x* >= *y*
dot(*v1*, *v2*)	Dot product of two vectors; returns a scalar
cross(*v1*, *v2*)	Cross product of two vectors; returns a vector
turbulence(*v*, *o*)	Turbulence of vector *v* and octave *o*; returns a float
turbulence3(*v*, *o*)	Turbulence of vector *v* and octave *o*; returns a vector
smoothstep (*min,max, x*)	Returns 0 if *x* < *min*, 1if *x* >= *max*; if neither are true, returns a hermite interpolation between 0 and 1
clamp(*x, min, max*)	*x* clamped to the range [*min,max*]

The following example illustrates how to create animated turbulence using the expression:

pos = pos + turbulence3(pos, 1) * power



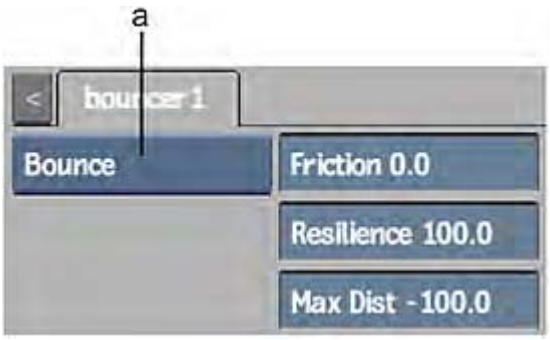


Bouncing Particles

You can bounce a particle stream off either an image or a bouncer. When particles bounce off an image, they bounce off the surface wherever its matte is white or grey. Particles only pass through the areas of the image where its matte is pure black. If a surface's matte is turned off, particles bounce off the entire surface.

When you use a bouncer, it must be the parent of an axis or a light source. A bouncer shows up as a sphere in the Action scene when you use it with a light source. Otherwise, all other types of surfaces create flat bouncers.

Whether you use a bouncer or an image, bouncing particles are controlled using the Particle Bouncer menu.



(a) Bounce Type box

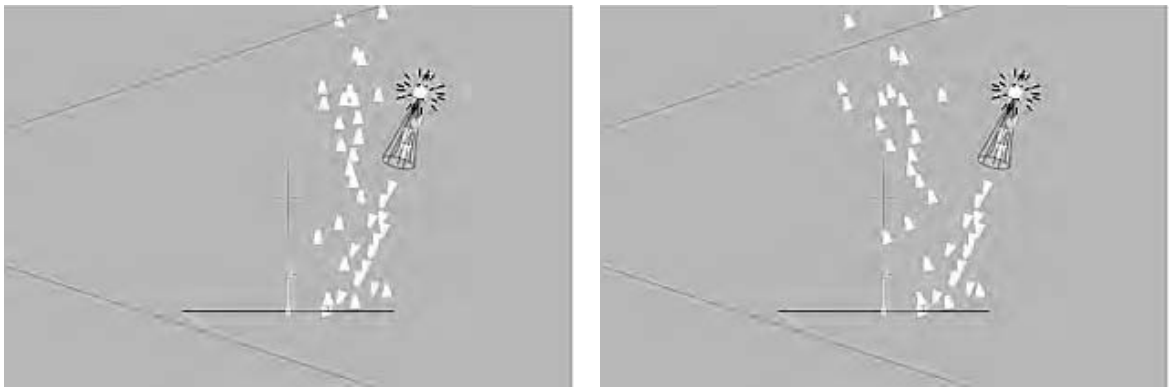
Bounce Type box Select a bounce behaviour.

Select:	To:
Bounce	Bounce particles off images or bouncers. All bounce properties work with this type of bounce behaviour.
Extinct	Make particles disappear when they reach a surface. Only the Max Dist parameter works with this type of bounce.

Select:	To:
Generator	Create an additional particle stream when particles bounce. When using this bouncing behaviour, you must add another particle generator. See Combining Particle Streams (page 786).
Generator+ Extinct	Make particles disappear and create an additional particle stream when particles reach a parented surface. Only the Max Dist parameter works with this type of bounce unless the particles end in another particle stream. See Combining Particle Streams (page 786).

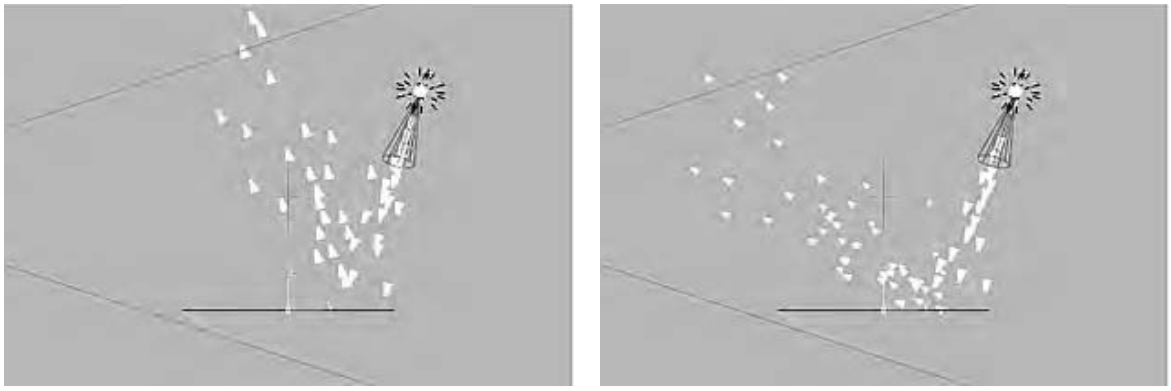
Friction field Displays a percentage to influence the angle of bouncing particles. Editable.

The greater the percentage, the more particles bounce straight. For example, in the left figure, Friction is set to 100%, which bounces particles almost perfectly straight. In the figure on the right, Friction is set to 20%, which causes the particles to deviate once they make contact (bounce).



Resilience field Displays the amount of energy lost with each bounce. Editable.

For example, at a Resilience of 100%, the figure on the left shows there is no energy lost between bounces. Each bounce is at the same height as the last bounce. When Resilience is set to 80%, 20% of energy is lost between each bounce. At 50%, the figure on the right shows the height of each bounce is reduced by 50%. Note that Friction is set to 0 for both examples.



Maximum Distance field Displays the distance a particle must travel past an image for it to still bounce or become extinct. Editable.

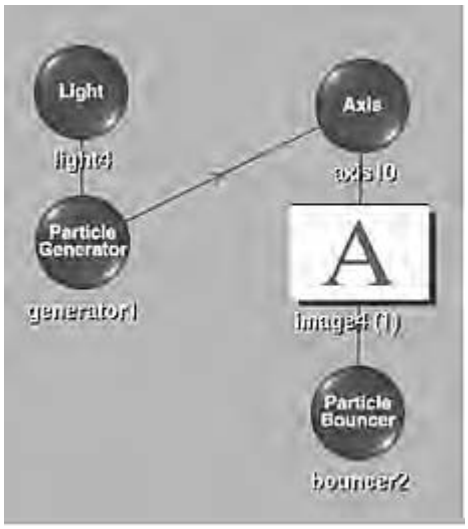
Because an image is only a pixel in width, it is unlikely that fast moving particles will hit the image directly. Max Dist acts like a buffer zone after the surface. If particles fall through the surface, set the Max Dist to -100. Particles that miss the surface by 100 pixels will still bounce.

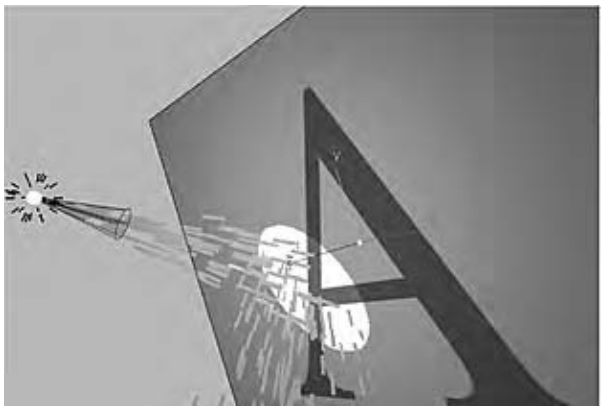
If particles still fall through the surface, increase the timestep to increase the accuracy of the particle stream.

Bouncing Particles Off an Image

You can bounce particles off an image surface only. Particles will bounce off bilinear and bicubic surfaces, but only on the original flat surface and not on any changes in shape.

To create the bouncing particle effect, you start with a particle generator. When you determine the image off which you want to bounce particles, you must parent the particle generator to the image; otherwise, the particles will ignore the image. The following figure shows the schematic and the result of bouncing particles off an image using a light source as a generator.





To bounce particles off an image:

1. Add an image off which you want to bounce particles.
2. Create a particle generator using a light source, 3D object, or surface as a particle generator. See Generating Particles (page 749).
3. Parent the particle generator to the image's axis.
4. Select the image's axis and, in the Axis menu, enable Free.

   The parent-to-child relationship between the particle generator and the image is severed, allowing you to place the image in the scene more easily. Any transformations and animations from the particle generator are not passed to the image.
5. Move the image in the scene until the particle stream passes through it. To do this, you may have to switch views or orbit the camera.
6. In the schematic, select the image.
7. From the Node bin, double-click the Particle Bouncer node.

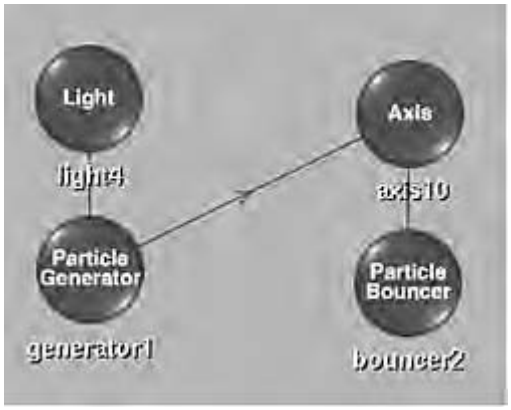
   A particle bouncer is added and parented by the image.
8. To access the Particle Bouncer menu, double-click the particle bouncer node in the schematic, or follow the tab population rules for the Object menu (see Populating Menu Tabs of Selected Objects (page 415)).

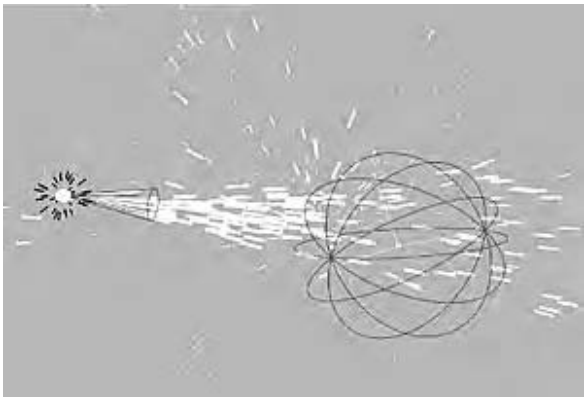
Bouncing Particles Off a Bouncer

A particle bouncer element is added and automatically parented to a light source. The particle bouncer may not be parented by the right light source or axis. The bouncer must be parented by the light or axis added in the last step.

NOTE The shape of the bouncer depends on the particle generator type.

The following figure shows the schematic and the result of bouncing particles off a bouncer.





To bounce particles off a bouncer:

1. Create a particle generator using a light source, 3D object, or surface as a particle generator. See Generating Particles (page 749).
2. Add an axis or a light source to the scene.
3. From the Node bin, double-click the Particle Bouncer node.
4. If you are using an axis to set the position of the particle bouncer, select the axis and go to step 5. Otherwise, go to step 7.
5. In the Axis menu, enable Free to sever the parent-to-child relationship between the particle generator and the axis.

   Any transformations and animations from the particle generator are not passed to the axis. This makes it easier to place the particle bouncer in the scene.
6. Parent the particle generator to the particle bouncer's axis or light source.

   You must parent the particle generator to the axis or light source; otherwise, the particles will ignore the particle bouncer.
7. In the Particle Bouncer menu, select Bounce from the Bounce Type box and set the appropriate bounce properties.

NOTE If the particle bouncer is not parented correctly, unparent the particle bouncer and make the axis or light source added in the last step its parent.

Combining Particle Streams

You can combine two or more particle streams by parenting them together. This has two possible results depending on whether the first particle stream ends or bounces off an object.

NOTE Multiple particles can be parented, but only in a straight line, that is, a particle can only be the particle of one other particle. For example Particle 1 can parent Particle 2, and Particle 2 can parent Particle 3 in the same scene.

Particles Generating Another Particle Stream

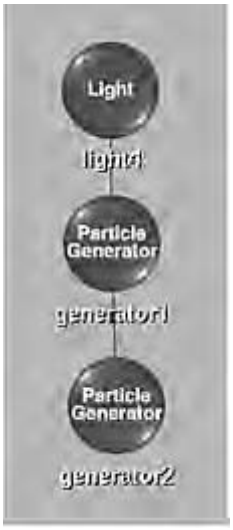
When each particle in the particle stream ends, the second particle stream begins. This can be used, for example, to create fireworks.

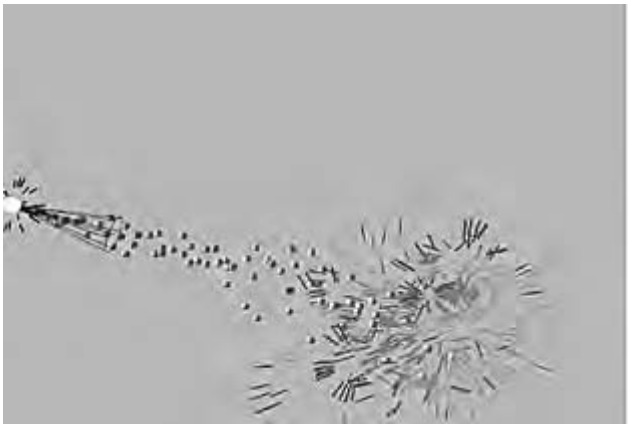
To make particles generate other particles:

1. Create a particle generator. You can use a light source, 3D object, or surface as a particle generator. See Generating Particles (page 749).
2. In the schematic, select the Particle Generator.
3. Add another particle generator by double-clicking the Particle Generator node in the Node bin.
4. In the Particle Generator menu, set the second particle generator's Spread and other properties as appropriate. See Setting Particle Properties (page 754).

   NOTE In the Channel Editor's folder for the second particle generator, you can set the Spread_V channel to add variance to the value in the Spread field.

   The following figure shows the schematic and the result of a particle stream that generates a second particle stream. The first particle stream generates spheres and the second particle stream generates quads.





Particles Bouncing into Another Particle Stream

When bouncing particle off a bouncer, you can generate a second stream of particles. For example, use this technique to give the effect of water or sparks bouncing off an image.

There are two different effects that you can create depending on your bouncer's settings. The following example illustrates the Generator bounce type.

To make bouncing particles generate another particle stream:

1. Create a schematic that bounces particles off an image or a bouncer. See Bouncing Particles (page 783).

2 In the schematic, select the Bouncer object.

3 In the Particle Bouncer menu, select Generator from the Bounce Type box.

4 Add another particle generator by double-clicking the Particle Generator node in the Node bin.

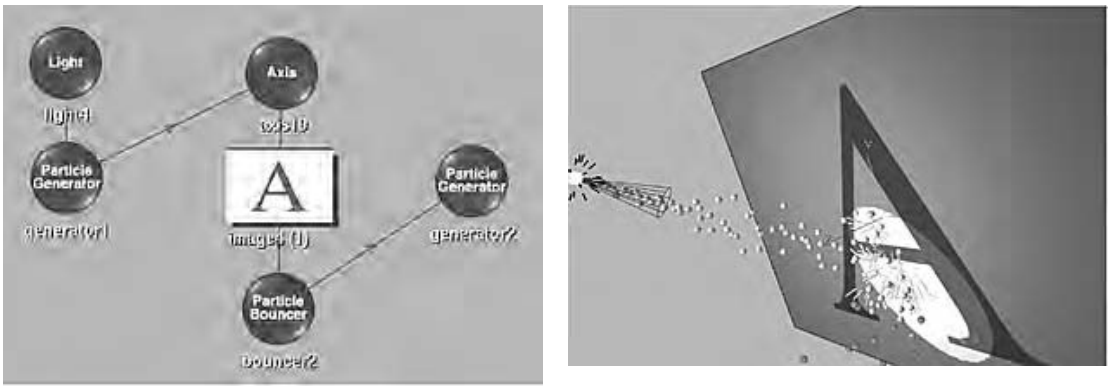
5 In the Particle Generator menu, set the particle generator's other properties as appropriate. See Generating Particles (page 749).

6 In the Channel Editor, open the second particle generator folder to reveal the properties folder.

7 Set the Spread and Spread_V channels to control the spread of the second generator.

If you set the Spread to less than 180, you can also control the rotation of the second particle stream using the Friction field in the Particle Bouncer menu.

The following figure shows the schematic and the result of bouncing particles into another particle stream where the first particle stream continues after bouncing off an image surface. The first particle stream generates spheres and the second particle stream generates quads.



Sample Particle Setups

Ten particle setups are provided in the directory */usr/discreet/<product_home>/examples*. To load one of the setup files, open Action using a black frame as the front, back, and matte clips.

A brief description of each example particle setup is provided in the following table.

Setup File Name	Description
Bounce Example	Uses a gravity manipulator and a bouncer attached to an axis. The bouncer appears as a sphere in Camera view. The particles fall down and bounce off the bouncer.
Bounce Example 2	Uses a gravity manipulator and two image surface bouncers.
Bounce Example 3	Uses an Accpoint manipulator and a bouncer linked to an axis. The bouncer follows the Accpoint manipulator.
Explode Example	An example of exploding 3D text. The Geom node is hidden in Schematic view. The setup uses an Explode generator and Polygon particles. To add 3D text (Geom node), go to the Text menu, click the Text field and type the text string, then click Create.

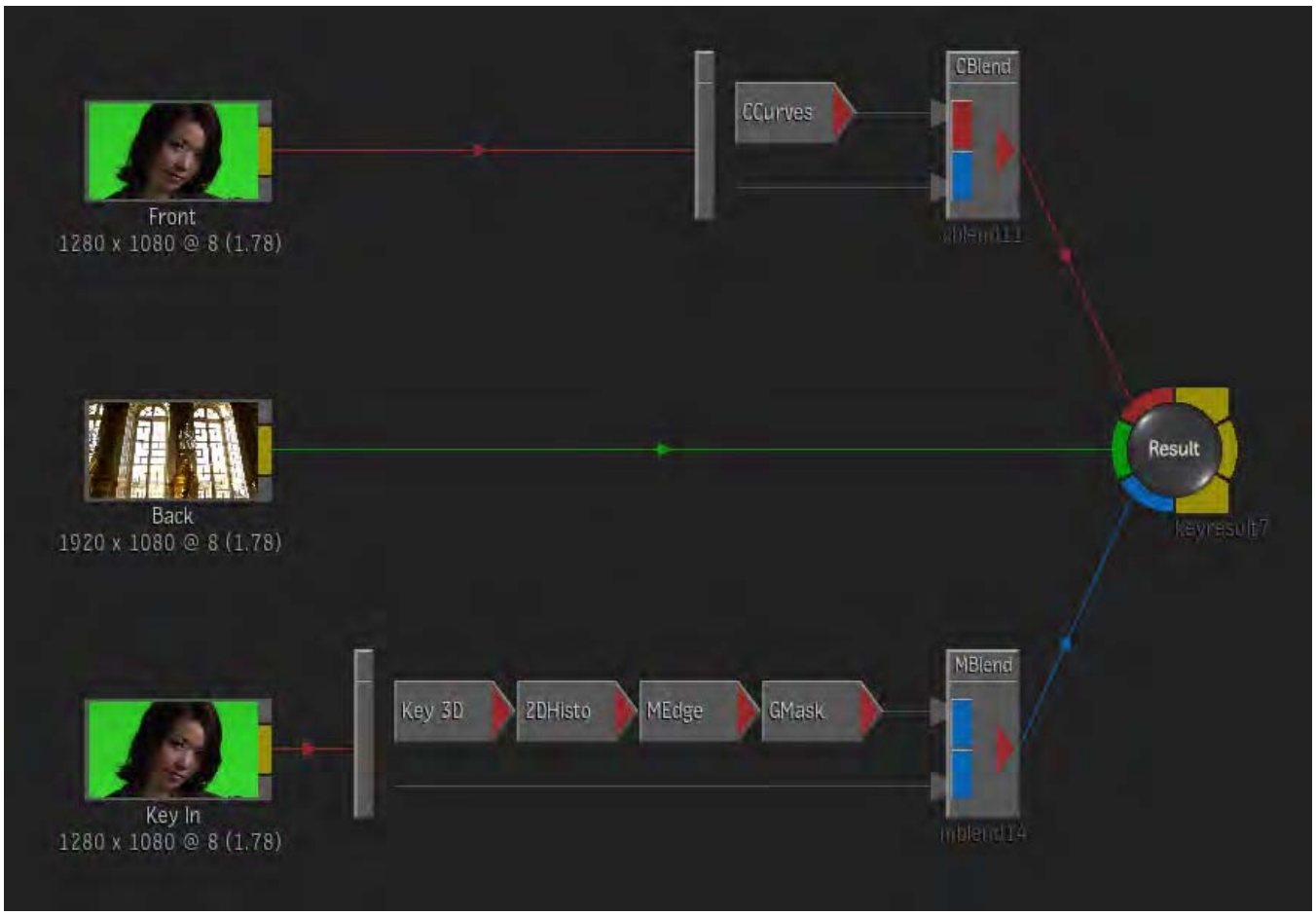
Setup File Name	Description
Explode Example 2	An example of exploding an image surface. The image surface is hidden. Media1 is applied to the generator as a texture and uses Wrap mapping mode. Replace Media 1 with an image of your choice.
FireWorks Example	Uses two linked generators. The first generator generates few particles at each frame. When these reach the end of their lifetime, the second generator is activated.
Function Example	Uses a function manipulator that increases the particles' transparency as they reach the end of their lifetime.
Function Example 2	Uses a function manipulator with sine and cosine functions to modify the speed of the particles.
Node Example	Uses 3D text for the particles. The Geom node (the 3D text) must be named “ParticleDraw”. The generator is parented to the axis of the Geom node, and the selected particle type is Node.
Transform Example	Uses a transform manipulator.

Chroma Keying and Working with Green Screens

18

The Modular Keyer is a schematic environment, like Batch FX, that is designed specifically keying. Within the Modular Keyer, you can choose different Start Modes or presets, which generate node schematics adapted to the different keys you want to pull, such as 3D keys, chroma keys, etc.

When you start the Modular Keyer for the first time, Flame builds a default schematic that includes the 3D Keyer node, in addition to a number of other nodes, that all work well together to create and refine a basic key. This is the 3D Keyer Start Mode.



NOTE If you want to reload a keyer setup or BFX from Flame Premium 2013 or prior, consider the following:

- If you enabled the Flame Reactor rendering engine when you created your project, make sure the media in the Setup/BFX is 8 bit or 16 bit floating point, as the Flame Reactor engine only supports 8 bit and 16 bit floating point media.
- If you enabled the Legacy Rendering engine when you created your project, the media in the Setup/BFX can be 8, 10 or 12 bit but not 16 bit floating point as the legacy keyer does not support 16 bit floating point media.

Within the Modular Keyer, the keyers are represented as processing pipelines, which are collections of interconnected nodes. As you perform keying tasks in the Modular Keyer, you navigate from node to node to accomplish the final key, displayed in the Result node.

As a general example, you could proceed as follows:

- Key out the selected colour in the Keyer node;
- Adjust your blacks and your whites in the 2D Histogram node;
- Refine your matte in the Matte Edge node;
- Mask out troublesome areas of your matte in the GMask node;
- Perform colour correction of the front clip in the Colour Curves node;
- View the result in the Result node.

Because it is node-based, you have a lot of flexibility with the Modular Keyer. You can:

- Choose a different Start Mode to start with a different default keying pipeline. Options are:
  - Channel Keyer
  - Default MK (3D Keyer)
  - HLS Keyer
  - Luminance Keyer
  - Master Keyer
  - RGB Keyer
  - RGBCMYL Keyer
  - YUV Keyer

  IMPORTANT When you select another Start Mode, any work in progress will be lost and the selected keyer is loaded with the default settings.
- Add other effects nodes from the node bin to further refine your key.

NOTE All of the legacy keyers had a colour section available in the settings. This colour section of the legacy keyers is only carried over to the Modular Keyer if the legacy keyer setup used the Master Keyer. If any other legacy keyer was used (RGB, Luminance, etc.), the colour section is discarded when the setup is loaded. To retain the colour section of these legacy keyer setups, enable Classic Engine in the Projects and Users dialog box. Also, motion blur data, from the GMask section of the legacy keyers is not carried over to the Modular Keyer.

Setting Up the Nodes and Media to Pull a Key

To pull a key, you must first set up your composite in the Modular Keyer.

There are two ways to set up the composite in the Modular Keyer:

- From the Timeline.
- From the Tools menu.

To set up the composite in the Modular Keyer from the Timeline:

1. From the timeline, select the segment you want to use as your front.
2. Click FX and select Create Batch FX

   You are taken to the Batch FX view. The schematic is displayed and your front clip is connected to the output node.
3. From the Batch FX node bin, drag the Modular Keyer node between the front clip and the output node and release it when the connection goes from red to yellow.

   The front clip is connected to the front (red) input of the Modular Keyer node, which is connected to the output node.
4. Double-click the back (green) input of the Modular Keyer node.

   You are taken to the Viewing panel and the cursor becomes green and prompts you to select a back clip.
5. In the Viewing panel or the Media panel, click on the clip you want as your back.

   The cursor becomes blue and prompts you to select a matte clip.
6. In the Viewing panel or the Media panel, click on the clip you want as your matte.

   You are taken back to the Batch FX schematic and your front, back and matte clips are connected to the front, back and matte inputs of the Modular Keyer node. You are ready to start creating your key.
7. Double-click the Modular Keyer node.
8. Click the Edit button that appears at the bottom of the screen.

   The 3D Keyer processing pipeline appears with your front, back and matte clips connected to the front, back and matte inputs of the 3D Keyer.. The 3D Keyer is the default keyer.

   NOTE Once inside the Modular Keyer, the Modular Keyer node bin is displayed at the bottom of the screen. Though very similar to the Batch FX Node bin, the Modular Keyer node bin contains only nodes that are relevant for keying. To return to Batch FX view, click Return.
9. Double-click the 3D Keyer node to display the 3D Keyer menu at the bottom of the screen.

To set up the composite in the Modular Keyer from the Tools menu:

1. Select the Tools tab, on the bottom of the window.

   The Tools menu is displayed.
2. Under the Composite tab, click Modular Keyer.

   The cursor becomes red and prompts you to select a front clip.
3. From the Media panel or the Viewing panel, click the clip you want as your front.

   The cursor becomes green and prompts you to select a back clip.
4. From the Media panel or the Viewing panel, click the clip you want as your back.

   The cursor becomes blue and prompts you to select a matte clip.
5. From the Media panel or the Viewing panel, click the clip you want as your back.

   The cursor becomes white and prompts you to select a render location in your workspace for your resulting key.
6. Click on a grey area of the Viewing panel or in a folder in the Media panel to select the render location.

   You are taken inside the Modular Keyer and the 3D Keyer processing pipeline is displayed, with your front, back and matte clips connected to the front, back and matte inputs of the 3D Keyer. The 3D Keyer is the default keyer.

   NOTE Once inside the Modular Keyer, the Modular Keyer node bin is displayed at the bottom of the screen. Though very similar to the Batch FX Node bin, the Modular Keyer node bin contains only nodes that are relevant for keying. To return to Batch FX view, click Return.

7 Double-click the 3D Keyer node to display the 3D Keyer menu at the bottom of the screen.

NOTE You can also access the Modular Keyer, as well as all the other Keyers in the application through Batch, where each Keyer is represented as an effects node. See Using Batch and ConnectFX (page 345).

After you have set up the nodes and media, set up the viewports for keying (page 794).

Setting Up Viewports for Keying

After you have set up your processing pipeline and your media (page 792), you want to set up the viewports for keying.

When keying, you want to set the viewport to 2-up view, to display the schematic and the result views simultaneously, enabling you to view the result as you work.

To set up the viewports for keying:

1 From the Layout box, select 2-up.

The view area is split in two. The schematic view is displayed on the left, by default.

2 Click in the right-side viewer to select it.

A white frame appears around the viewer to show that it is selected.

3 From the View box, select Result.

The result of your key, in its current state, is displayed in the right-side viewer.

About the 3D Keyer

Like traditional keying, the 3D Keyer in the Modular Keyer and Batch is used to select a range of colours to key out, such as the blue or green background in a clip. In the 3D Keyer, however, this range of colours is represented visually in an RGB viewer. You can use the RGB viewer to analyse the colour areas in a clip, and then refine the key with increased accuracy.

The 3D Keyer can be used for many purposes other than pulling a key from the key-in clip. For example, in the Modular Keyer, you could use the Keyer-3D node in the Matte pipe of the CBlend node to isolate a region of the front clip to which you want to apply a particular colour correction.

NOTE It is recommended that you be familiar with the use and terminology associated with the Modular Keyer before you use the 3D Keyer.

3D Keying Workflow

The following table shows the recommended workflow for the 3D Keyer.

Step:	Refer to:
1. Open the Modular Keyer, select the clips, and then open the 3D Keyer.	Setting Up the Nodes and Media to Pull a Key (page 792).
2. Sample for tolerance.	Setting the Tolerance (page 796).
3. Sample for softness.	Setting the Softness (page 799).

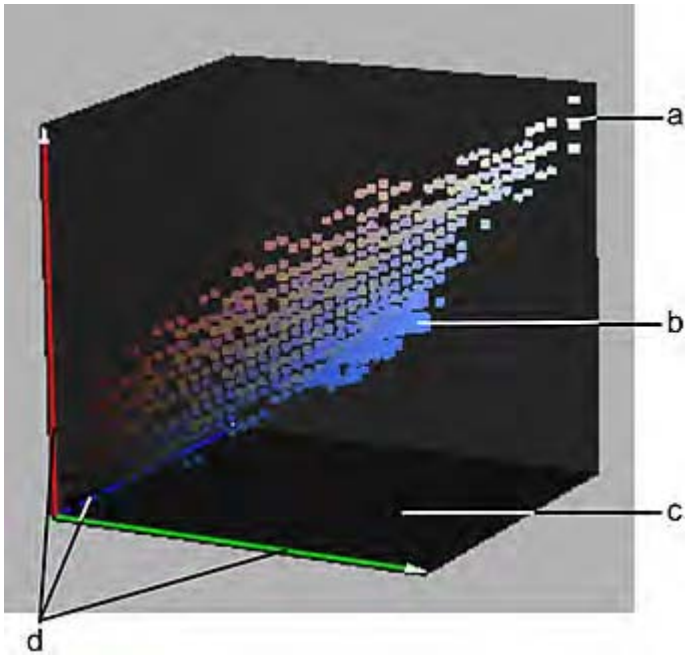
Step:	Refer to:
4. Minimize the noise in softened areas.	Removing Noise from Softened Areas (page 801).
5. Remove grey areas from the key using negative sampling and patches, garbage masks, or RGB colour and softness adjustment.	Removing Grey Areas in the Key (page 804), About Garbage Masks (page 891), Techniques for Adjusting Softness (page 825).
6. Adjust the luminance of the key.	Adjusting the Luminance of the Key (page 842).
7. Adjust the edges of the key.	Modifying the Edges of the Key (page 848).
8. Remove colour spill.	Adjusting Spill Controls with the Colour Curves Node (page 851).
9. Adjust front and matte curves.	Adjusting the Front and Matte Luminance Curves (page 853).
10. Render the clip.	Rendering Your Key (page 856).

Creating a Precise Key in the 3D Keyer

This section describes the tools you use to create a precise key with the 3D Keyer. You can learn other techniques for perfecting your key in More Keying Techniques (page 825).

Using the RGB Viewer

The RGB viewer is a 3D colour model for the key-in clip. Its X, Y, and Z axes represent the red, green, and blue components of the key-in clip's colour space. The cubes of colour show the distribution of all the colours of the key-in clip in RGB colour space. This 3D colour map is referred to as the 3D histogram.



(a) The colours in the key-in clip are represented by the colour cubes of the 3D histogram (b) More prevalent colours are represented by larger cubes (c) Black canvas background (d) Red, green and blue axes

When you scroll through a clip, the 3D histogram cubes update to display the colour distribution of each frame.

Use the 3D histogram to analyse the colour mapping of the key-in clip and to locate the colours to be keyed out. When you sample for tolerance and softness, the sampled areas are represented as ellipsoids in the RGB viewer. You can manipulate sampled objects in the viewer to fine-tune the key. When you plot a colour to key in the image, its location is plotted in the RGB viewer.

You can turn the display of the histogram and the black canvas background on and off and set options to control their appearance. Change the display any time as you create the key to help you best visualize the contents of your image and work with the RGB viewer. To learn more about controlling the display of the RGB viewer, see Setting the 3D Histogram Display (page 810).

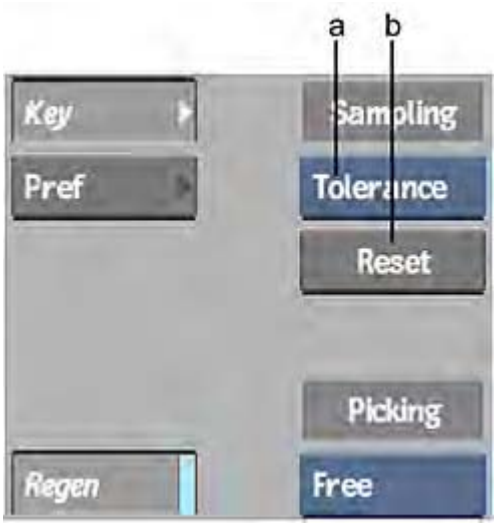
Setting the Tolerance

The first step in pulling a key with the 3D Keyer is to set a range of colours to be keyed out from the front clip. This is called the tolerance range. All pixels in the key-in clip in the tolerance range are black in the matte.

To specify the tolerance range, sample the area that you want to key out in the image window. Take one or more samples until you are satisfied with the result. The tolerance range must not be too large or the edge of the composite will be too hard and the subject in the front clip will appear to be pasted into the back clip.

To set the tolerance range:

1. Click Key.
   The 3D Key menu appears.



(a) Sampling box (b) Sampling Reset button

2 Make sure you are at frame 1. If you move to a different frame while sampling, you will set a second tolerance keyframe (at the new frame) and animate the tolerance.

NOTE To avoid setting keyframes, disable Auto Key.

3 In the Sampling box, set the sample type to Tolerance.

4 Press `Ctrl` and drag the cursor in the area to be keyed out.

A red rectangle appears. The pixel values in the rectangle are used to define the tolerance range. Notice that the rectangle has a maximum size. This ensures that the result is computed quickly.

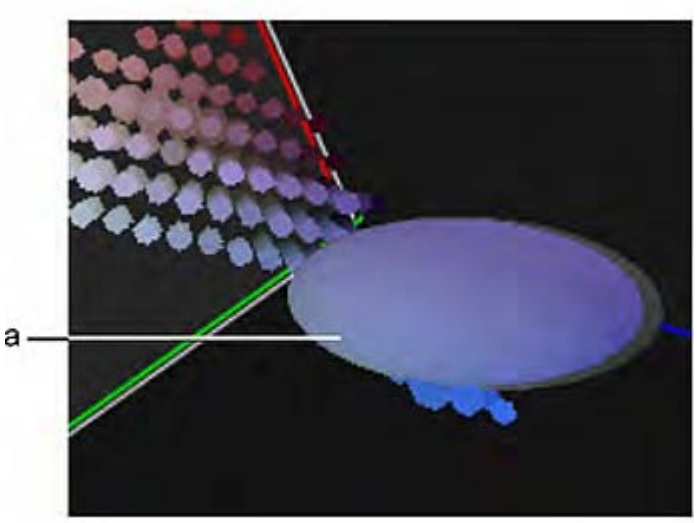
NOTE You can also take a sample by clicking the image and dragging the colour picker in the area to be keyed out. All pixel values that the colour picker samples are used to set the tolerance range.

5 If needed, add to the tolerance range by sampling several more rectangles. The ellipsoid gets bigger as you add to the tolerance.

NOTE If you are not satisfied with the result of the last sample, click Undo. The last sample you performed is cancelled. To reset the entire tolerance sample, click the Sampling Reset button. To remove the entire sample range at the current frame and start over with a new sample, press `Ctrl+Alt` and drag the cursor in the area to be keyed out.

Viewing the Tolerance Sample

The tolerance sample in the RGB viewer is represented as an ellipsoid by default. The ellipsoid shows you the location, in RGB colour space, of the range of colours included in the sample. All colours in the tolerance ellipsoid are black on the matte.



(a) Tolerance ellipsoid

When the histogram is displayed, you can see the location of the sample in relation to the other colours in the image. When it is hidden, you can see the sample more clearly. As you work, show or hide the histogram using the following techniques:

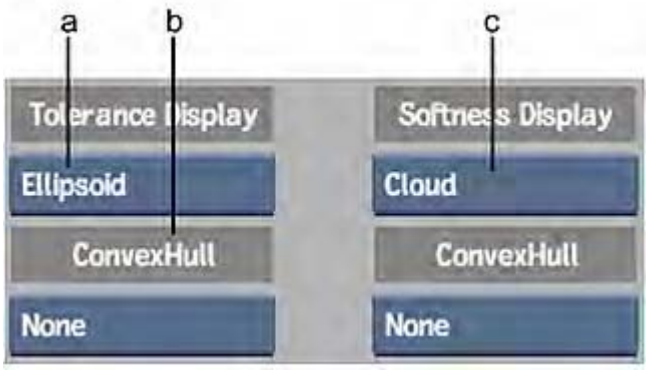
- To display or hide the histogram, press `H`. See Setting the 3D Histogram Display (page 810).
- To view the sample more closely, rotate the RGB viewer by pressing `Ctrl` and dragging it. To zoom in on the RGB viewer, press `Shift` and drag inside it from left to right. To move the RGB viewer, press `Alt` and drag it.

Viewing the Convex Hull

The tolerance ellipsoid surrounds the exact colours sampled, which are connected in RGB space to form a 3D convex hull. You can see the convex hull in the ellipsoid by changing the tolerance display.

To view the convex hull:

1. Press `H` to remove the histogram.
2. In the Tolerance Display box, set the tolerance display to Cloud, which is semi-transparent.



(a) Tolerance Convex Hull box (b) Tolerance Display box (c) Softness Display box

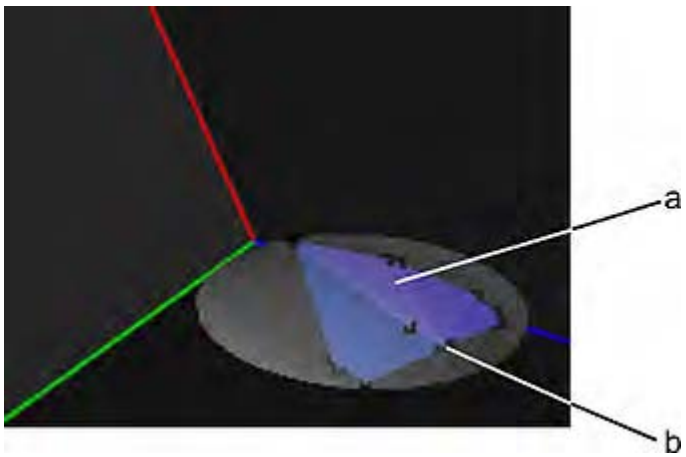
NOTE To learn more about controlling the display of the tolerance sample, see Changing the Display of Key Elements (page 812).

3. In the Softness Display box, set the softness display to None.

NOTE When you set the tolerance range, the 3D Keyer automatically creates a minimum softness range that is equivalent to the tolerance range. This creates a softness ellipsoid of equal size to the tolerance ellipsoid in the RGB viewer. Sometimes the softness ellipsoid is visible at this point, depending on the way the ellipsoids are displayed.

4 Set the Tolerance Convex Hull box to Vert&Surf.

You can now see the convex hull, with vertices indicating the exact colour areas sampled.



(a) Convex hull (b) Convex hull vertices

The convex hull contains the exact colour values of the tolerance sample, while the ellipsoid represents the actual area in which corresponding pixels will be black in the matte. The 3D Keyer uses the ellipsoid rather than the convex hull to determine the keyed out area because it is much faster to calculate.

You can change the shape of the convex hull directly in the RGB viewer to adjust the tolerance. The ellipsoid re-orients itself according to the new shape. See Reshaping the Convex Hull (page 824).

5 For the next step, return the softness display to Cloud.

Setting the Softness

The next step is to set a range of colours in the key-in clip to be partially transparent so as to soften the transition between the front and back clips in the composite. This is called the softness range. To specify the softness range, sample the areas to be softened in the image window. The pixels in the key-in clip in the softness range are grey in the matte. The key should have the greatest possible softness value.

As shown in the following steps, the softness range can be set using the RGB or YUV colour models. RGB Softness is the default setting.

To sample for softness:

1 Make sure you are at frame 1. If you move to a different frame while sampling, you will set a second softness keyframe (at the new frame) and animate the softness.

NOTE To avoid setting keyframes, disable Auto Key.

2 From the Sampling box, select Softness.



3 From the View box, select Front to view the input image for the 3D Keyer.

4 Identify an area where softness is needed. For example, if the clip consists of a talent in front of a blue or green screen, softness is needed at the edge of the talent to soften the transition between the front and back clips. If the clip contains transparency, softness is needed in the semi-transparent area.

5 From the View box, select the current result view (displayed as <Currently Selected Node> Result, in the View box) to monitor the result as softness is applied to the matte while you sample.

TIP You can switch views as you sample the image. The current result view (displayed as <Currently Selected Node> Result, in the View box), shows the result of the processing pipeline up to the selected node. To see the Front Source, select Front from the View box.

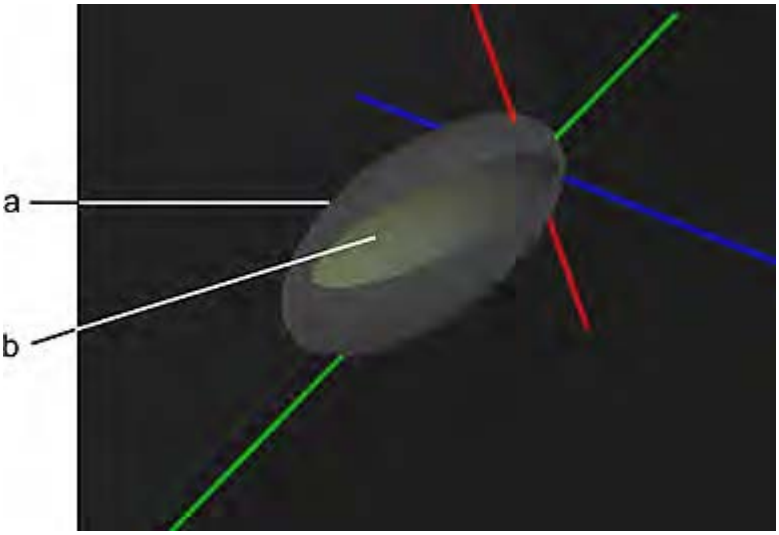
6 Zoom in to see the image more closely.

7 Click in the area you identified.

The cursor changes to a colour picker.

8 Drag the colour picker over the area in the image to be softened. Avoid sampling any area that should not be softened, such as the key itself; however, there is no need to avoid areas that have been keyed out. For transparencies, drag the colour picker in the area that you want to be semi-transparent.

A softness ellipsoid appears in the RGB viewer. Notice the tolerance ellipsoid in the softness ellipsoid.



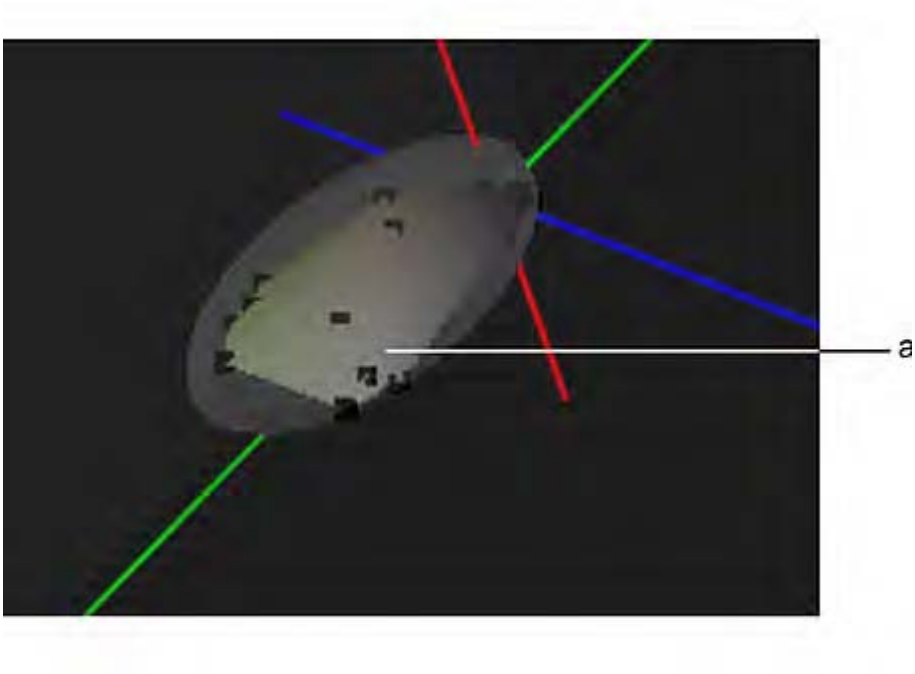
(a) Softness ellipsoid (b) Tolerance ellipsoid

All colours in the softness ellipsoid (except those inside the tolerance ellipsoid) will be grey on the matte. In the softness ellipsoid, the closer a point is to the tolerance ellipsoid, the darker the grey will be. For example, the mid-point between the edge of the tolerance ellipsoid and the edge of the softness ellipsoid is always the same mid-grey.

Notice how the softness ellipsoid increases in size as you sample the edges.

NOTE The softness is rendered interactively on the matte - you see the results immediately. For information on rendering the softness only after releasing the cursor, see Regen (page 828).

9 Like the tolerance ellipsoid, the softness ellipsoid has a convex hull. To see it clearly, set the Tolerance Display and Tolerance ConvexHull display to None, and set the Softness ConvexHull display to Vert&Surf.



(a) Softness convex hull

10 If needed, take another sample elsewhere on the edges or transparency. Each time you sample, the pixels touched by the colour picker contribute to the previously calculated softness range.

NOTE If you are not satisfied with the result of a sample, click Undo. To reset the entire softness sample, click the R button. To remove the entire sample range and start over with a new sample, press `Ctrl+Alt` and drag the cursor in the area to be softened.

11 From the View box, select MK Schematic and, in the schematic, click the Result node.

12 From the View box, select the current result view (displayed as <Currently Selected Node> Result, in the View box) to view the result.

The edges are softened but may be grainy.

Removing Noise from Softened Areas

Once you sample for softness, use the Minimize Noise option to remove grain from the softened areas. Graininess, also known as noise, can occur at the edges of the key, or in the semi-transparent areas such as water or glass.

First, sample an area containing noise. The 3D Keyer analyses the sampled area. Using Minimize Noise, you can then scale the softness in such a way as to minimize the noise in the softened areas.

Minimize Noise is especially useful for semi-transparent areas and edges requiring a lot of softness, such as smoke, reflection, and shadows.

TIP If the entire clip is grainy, or if you are not satisfied with the results after scaling the softness, consider using another approach. For example, degrain the clip with Degrain before keying with the 3D Keyer. You can add a Degrain node to the beginning of the Matte branch anytime. Additionally, use Minimize Noise on a degrained clip if needed.

To remove noise:

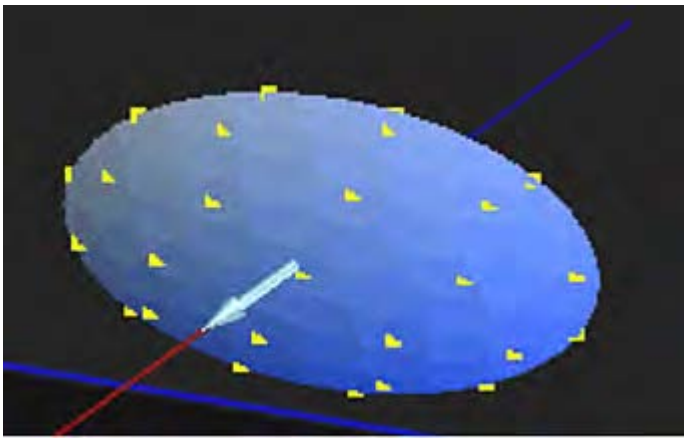
1. Sample for softness. See Setting the Softness (page 799).
2. Zoom in and choose the area you want to analyse. Look for graininess in the softened areas. For transparencies, look for graininess in the semi-transparent areas.
3. Press N (Noise) and drag a rectangle in the selected area.

   The 3D Keyer analyses the pixels in the rectangle.
4. Select Min Noise from the Softness Scaling box.



(a) Softness Scaling box (b) Scaling field

An arrow appears in the softness ellipsoid, showing the direction in which the softness needs to be increased to reduce graininess in the sampled area.



5 Increase the softness using one of the following methods:

- Drag the cursor over the Scaling field to the right. The softness is increased based on the results of the analysis from the grainy region. Observe that the softness ellipsoid is scaled in the direction of the arrow.

  NOTE Each time you change the Scaling value, it is returned to 1.00.

- Use a colour value plotted in the image window as the basis for virtual point scaling of softness. To do this, use the `O` keyboard shortcut to plot a point in the image window, then press `Ctrl+V`, and click and drag in the Player.

6 If some edges or areas are still not softened, analyse again in that area and repeat the procedure.

7 Return the Softness Scaling to Prop (proportional scaling).

Alternatively, press `Alt` + `N` to scale softness based on the Minimize Noise analysis. This keyboard shortcut is the equivalent of selecting Minimize Noise in the Softness Scaling box, then scaling the softness using the Scaling field.

NOTE When Minimize Noise scaling is selected, it controls the arrow in the RGB viewer. To manually control the arrow, you must return to proportional scaling.

To scale softness using `Alt+N`:

1 Perform a noise analysis.

NOTE `Alt` + `N` only works if an analysis for the noise has been done.

2 Press `Alt` + `N` and click and hold the cursor anywhere in the image window. Drag the cursor to the left to decrease the noise or to the right to increase it.

The softness is scaled according to the noise analysis. Notice that Minimize Noise appears in the Softness Scaling box as you use the keyboard shortcut. When you release the cursor, the Softness Scaling option returns to Prop.

TIP For transparencies, use the `V` keyboard shortcut to remove unwanted grey areas in the matte, then use `Alt+N` to reduce noise in areas you chose to soften (the transparency). Perform the two procedures alternately until you achieve the best result.

Using YUV Softness

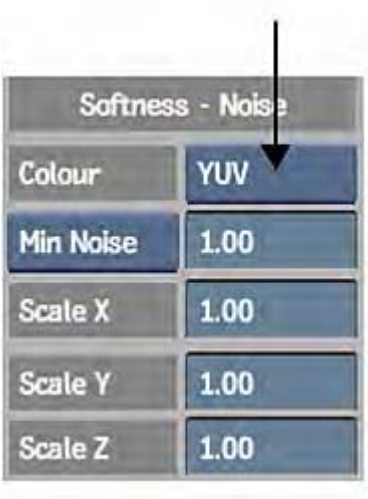
If you are not satisfied with the softness result—for example, if graininess still remains in the softened areas, try using YUV Softness. This option calculates the softness according to the YUV colour space.

YUV Softness often creates softened areas that are smoother and is typically useful for shadows and transparencies. However, since it removes more noise, the resulting softness range is less precise.

NOTE You can switch between RGB and YUV Softness and retain their settings.

To use YUV Softness:

1 Select YUV from the Softness box.

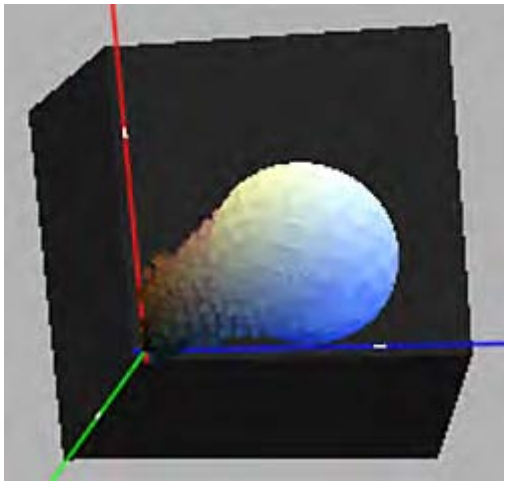


2 Sample for softness as described in Setting the Softness (page 799).

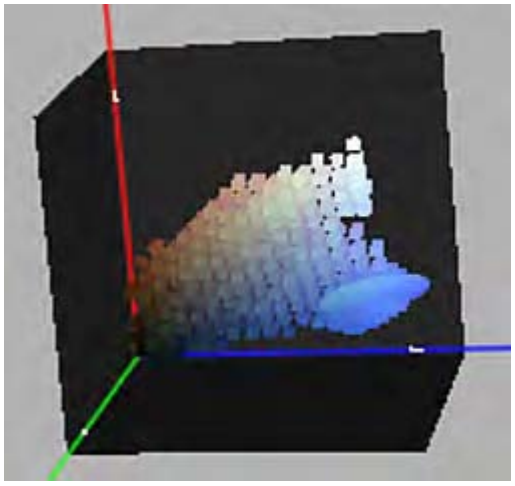
3 Minimize the noise as described in Removing Noise from Softened Areas (page 801).

Compare YUV with RGB Softness

Select RGB Softness from the Softness box. The previous RGB Softness range is still in effect. Compare the results with the YUV Softness. Generally, the softness ellipsoid is larger with YUV Softness, as shown in the following example.



With RGB Softness, the softness ellipsoid is bound by the convex hull



With YUV Softness, the softness ellipsoid is scaled according to the Y component of YUV colour space

If you are satisfied with the result, use YUV Softness. If not, return to RGB softness.

Removing Grey Areas in the Key

Once you have added softness, check to see if there are any grey areas in the key resulting from the setting. If so, there are several techniques to remove the grey areas:

- Perform negative sampling on the areas.
- Use patches.

- Use garbage masks.
- Manipulate the softness element in the RGB viewer.
- Use the Softness fields.

In this section, two methods are shown: using Negative Sampling to refine the softness range, and using the Patch tool to create "selective keys" in the white areas of the matte. To learn other ways to remove unwanted grey areas, see Techniques for Adjusting Softness (page 825).

Negative Sampling

This is probably the simplest method of removing greys from the matte. You sample in the unwanted grey area to subtract those colour values from the softness range.

Try this method first to see if it solves the problem. However, if the colour values in the foreground subject are too similar to those at the edges, you may remove too much of the softness.

To perform negative sampling:

1. From the View box, select the current result view (displayed as <Currently Selected Node> Result, in the View box). Identify the unwanted grey areas in the matte.
2. Select Sample Softness from the Sampling box.
3. Press `Alt` and click a grey area. Gently drag the cursor over the area.

   If the negative sampling removed too much of the desired softness or brought back graininess, click Undo and try another method.

Adding a Patch

Patches are another type of sample you can take in the image window. Unlike tolerance and softness, you can specify the colour that a patch sample renders on the matte. Patch samples can be rendered as white, black, or any shade of grey on the matte. You can also set the opacity of a patch.

One use for patches is to remove unwanted grey areas from the matte. Use a patch to isolate a range of colours from those that have been included in the softness range, then set the colour you want the pixels to have on the matte. Sampling a patch to remove grey areas in the key is similar to creating a garbage mask by keying instead of drawing.

You can use up to three different patches in a key.

To sample a patch:

1. From the View box, select the current result view (displayed as <Currently Selected Node> Result, in the View box).
2. Identify the unwanted grey areas in the matte.
3. In the Sampling box, set the sample type to Patch1.



4 In the Patches box, select Patch 1.



(a) Patches box (b) Active button (c) Patches Softness field

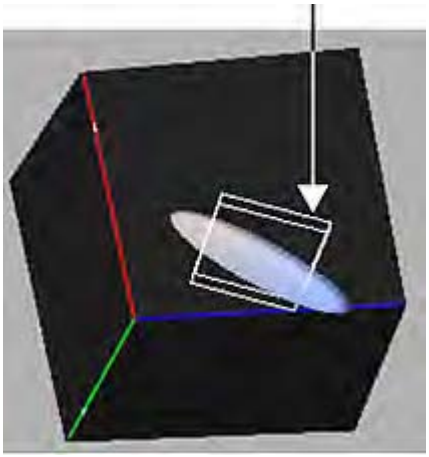
5 Enable Active.

6 Specify the colour in which you want to render the patch. In this example, make the patch white by setting the P1(Patch 1) Colour to 100 (this is the default value). To render the sampled values as black, enter a value of 0. Use a value between 0 and 100 for any shade of grey.

7 Sample the unwanted grey area by dragging the cursor over it.

The grey areas of the matte corresponding to the colour values sampled are changed to white.

All the colour values sampled are included in the patch. Notice the patch in the RGB viewer, represented by a white wireframe box. All colour values in the box will be rendered white on the matte.



8 If necessary, sample again to add to the patch. Watch the edges to make sure they are not affected by the sample.

TIP To remove colour values from the sample, press `Alt` as you sample. To undo a sample, click Undo.

9 If needed, soften the edges of the patch by dragging the cursor to the right or left in the Patches Softness field.

A negative softness value softens the edges of the patch inwards from the edge. A positive value softens the edges outwards beyond the edge, adding softened pixel values to the patch.

TIP You can view softness as a red wireframe box when the patch is displayed in Box mode. See Changing the Patches Display (page 813).

10 If needed, adjust the opacity of the patch using the P1 Opacity field A value of 100% renders the patch as fully opaque. A value of 0% renders the patch as fully transparent.

The RGB viewer now contains three key elements:

- The tolerance sample (as an ellipsoid)
- The softness sample (as an ellipsoid)
- The patch (as a white wireframe box)

For instructions on completing the key using the basic keying technique, see the sections referred to in 3D Keying Workflow (page 794).

Animating Key Elements

In a live-action clip, you can animate the tolerance, softness, and patch ranges as the subject in the clip moves. You can also animate the colour, opacity, and softness of patches.

Animating the Range of Key Elements

You animate the tolerance, softness, and patch ranges by changing the range at different frames. In the Channel Editor, range changes appear in the Shape channel. You can set a Shape keyframe by:

- Adding or subtracting pixel values from the range by sampling again in the image.
- Adjusting the key elements in the RGB viewer. All transformations made to a key element are applied to its Shape keyframe, including translation, rotation, scaling, and noise analysis.
- Changing the values in the Softness Scaling X, Y, and Z fields.

If you display the convex hull while animating a key element, the convex hull fills the ellipsoid in between keyframes. The sample is recreated to fill the ellipsoid on interpolated frames. This ensures the smoothest possible transition between keyframes.

To display the Channel Editor, click the Animation button.

NOTE To animate key elements, enable Auto Key.

Shape Keyframes

The shape of ellipsoids and patches is defined by their translation, rotation, and scaling values, so the Shape keyframe is actually composed of multiple parameters. The Y value in the Channel Editor does not have the same significance as it does for a single parameter (for example, opacity). Instead, the Y value is composed of sequential numbers that represent keyframes that have been set. Each consecutive keyframe is assigned a sequential Y value: the first keyframe has a Y value of 1, the second has a Y value of 2, and so on. Although the Y value does not represent a single value, you can still adjust the curve to tweak the shape.

To animate the range of a key element:

1. Perfect the key at frame 1.
2. Enable Auto Key.
3. Scroll to other frames and, where necessary, adjust the range using any of the techniques provided in this chapter.

   Keyframes are added at each frame where you change the tolerance, softness, or patch range.

   NOTE When you set a tolerance keyframe, a softness keyframe is also set. This is because the softness range is always at least as large as the tolerance range. When you set the tolerance range, the 3D Keyer automatically creates a minimum softness range that is equivalent to the tolerance range. When you adjust the tolerance, the softness range is also adjusted to accommodate the change.
4. Optionally, tweak the animation by adjusting the shape curve in the Channel Editor. The image updates as you make changes.

Changing the Interpolation

Parameter values in between keyframes are interpolated. Change the type of interpolation using the Channel Editor.

The default interpolation for the shape curve is Bézier, which creates a smooth transition between keyframes. If your clip has a very sudden change in the colour values (for example, if a light was switched on at a particular frame), use Constant interpolation between the two keyframes where the change occurs. See Setting Interpolation (page 2179).

Animating the Patch Parameters

Animate the colour, softness, and opacity of patches by changing the values in the Patches menu. Channels for each of these parameters are located in the Patch folders of the Channel Editor.

To animate the patch parameters:

1. Go to the appropriate frame.
2. Enable Auto Key.
3. Change the values in the Colour, Opacity or Softness fields as needed.

Resetting a Channel

There are two ways to reset a channel:

- Using the Reset button under he Sampling box.
- Using the Rst Chn button in the Animation controls.

Mastering the RGB Viewer

The RGB viewer gives you a graphic representation of what you are doing in the key. In this section, learn how to:

- Move, rotate, and zoom in on the RGB viewer.
- Change the display of the 3D histogram and canvas.
- Change the appearance of elements in the RGB viewer.
- Select key elements in the RGB viewer.
- Plot pixels in the image and in the RGB viewer.
- Translate, rotate, and scale key elements in the RGB viewer.
- Reshape the convex hull by manipulating its vertices.

Moving, Rotating, and Zooming in on the RGB Viewer

The RGB viewer represents the 3D colour space of a key-in clip. As you work on the precision of a key, you will want to change the position, orientation, and size of the RGB viewer. Use `Alt`, `Shift`, and `Ctrl` to move, rotate, and zoom in on the RGB viewer.

To:	Do this:
Move the viewer	`Alt`-click the viewer and drag it.
Zoom in on the viewer	`Shift`-click the viewer and move the cursor to the left or right.
Rotate the viewer	`Ctrl`-click the viewer and move the cursor in any direction. By default this rotates the viewer around its centre. Using Rotation Axis options in the Preferences menu, you can also rotate the viewer around a plotted point, or around the centre of the tolerance or softness ellipsoid. This makes it easier to zoom in on specific elements in the viewer.

Resetting the RGB Viewer

To reset the viewer's position, orientation, and size, click Home in the lower-left corner of the Key, Patches, or Preferences menu.

NOTE Other settings, such as resolution or canvas light, are not reset.

Setting the Display of the RGB Viewer

As you work with sampled elements in the RGB viewer, set the display of the 3D histogram, canvas background, and border lines to suit the operation you are performing. For example, when modifying the softness range, consider hiding the 3D histogram to see the softness ellipsoid more clearly.

Setting the 3D Histogram Display

The 3D histogram provides a visual representation of the colours in the key-in clip. It shows how the colours in your clip are distributed in RGB colour space, with exposure and contrast settings taken into account.

The most prevalent colours are represented by larger cubes; colours that are less prevalent are represented by smaller cubes.

NOTE The size of the cubes approximates the colour usage in the image, but it is not intended as an exact representation. You can increase the resolution of the cubes using the histogram grid.

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



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

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

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

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

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

Change other display elements of the histogram using the Preferences menu. To access the Preferences menu, click Pref.



Histogram Enable to show or hide the histogram. Alternatively, use the H keyboard shortcut.



Histogram grid Enable to displays the resolution for the cubes:

- x16 (the default)
- x8 displays fewer and larger cubes
- x32 displays smaller cubes

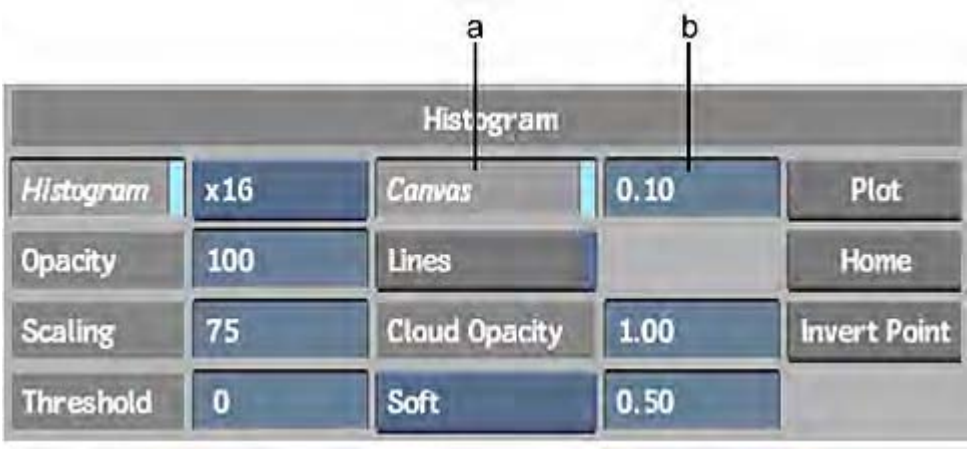
Opacity field Displays the opacity of the cubes. As you adjust the opacity, smaller cubes become transparent at a faster rate than larger ones. The default setting is 100% opacity.

Scaling field Resizes the cubes. Cubes representing less prevalent colours (the smaller cubes) are scaled down more than those representing more prevalent colours. The cube of the most prevalent colour remains the same size (as set in the Histogram Grid box). By increasing the size difference between the colours of greater and lesser prevalence, you can see the main components of the colour space more clearly.

Threshold field Removes the display of colours that are less prevalent in the image. As you increase the threshold, less prevalent colours are increasingly removed from the histogram.

Setting the Canvas Display

The canvas is the black background of the RGB viewer. Control the display of the canvas using the Preferences menu.



(a) Canvas button (b) Canvas Light field

Canvas button Enable to show or hide the canvas. Alternatively, use the C keyboard shortcut.

Canvas Light field Displays the amount of lighting in the RGB viewer. The light source emanates from behind the viewer.

Setting the Line Display

You can show or hide a white outline of the borders of the RGB viewer, as well as green wireframe boxes around selected ellipsoids. Show the boxes to quickly identify the selected element, or hide them to reduce "clutter" in the viewer.

To show or hide lines in the RGB viewer:

1. Enable or disable the Lines button in the Setup menu.

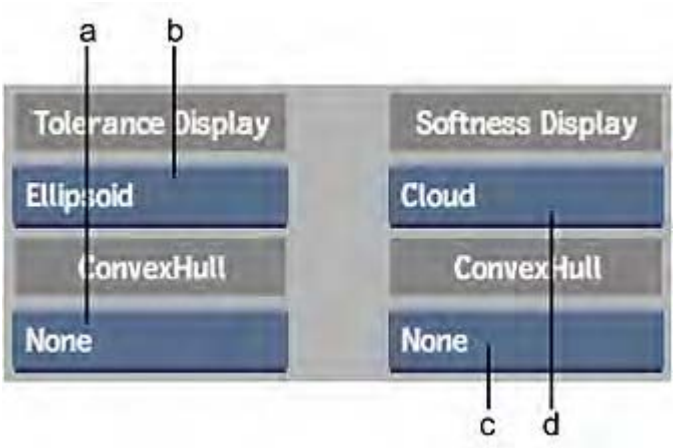
Changing the Display of Key Elements

You can change the display of the tolerance ellipsoid, softness ellipsoid, and patches as you work on the key to suit the operation you are performing. For example, to adjust the tolerance range after setting the softness, you could set the softness display to wireframe, or remove its display altogether, for a better view of the tolerance. See Setting the Tolerance (page 796) and Setting the Softness (page 799).

Setting the Tolerance and Softness Ellipsoids

As you sample the tolerance and softness in a key, corresponding key elements appear in the RGB viewer. Set the appearance of the tolerance and softness ellipsoids to make it easier to work with them while refining the precision of the key.

Use the Key menu to set the display of the tolerance and softness in the RGB viewer.



(a) Tolerance Convex Hull box (b) Tolerance Display box (c) Softness Convex Hull box (d) Softness Display box

Tolerance and Softness Display boxes Select an option to set the appearance of the tolerance and softness ellipsoids.

Select:	To display:
Ellipsoid	An opaque ellipsoid of the exact colours sampled.
Cloud	A semi-transparent grey ellipsoid. You can set the transparency of the ellipsoid using the Cloud Opacity field in the Preferences menu. See Setting the Cloud Opacity (page 814).
Wireframe	The ellipsoid as a wireframe outline
None	No ellipsoid. Note that when an ellipsoid is not displayed, it still has an effect on the image

Tolerance and Softness Convex Hull boxes Select an option to set the appearance of the tolerance and softness convex hulls.

Select:	To display:
Vertices	Only the vertices of the convex hull.
Surface	Only the surface of the convex hull.
Vert&Surf	Both the surface and vertices of the convex hull.
None	No convex hull.

NOTE Vertices do not appear on keyed elements when Tolerance Display or Softness Display is set to Ellipsoid (because an ellipsoid is opaque).

Changing the Patches Display

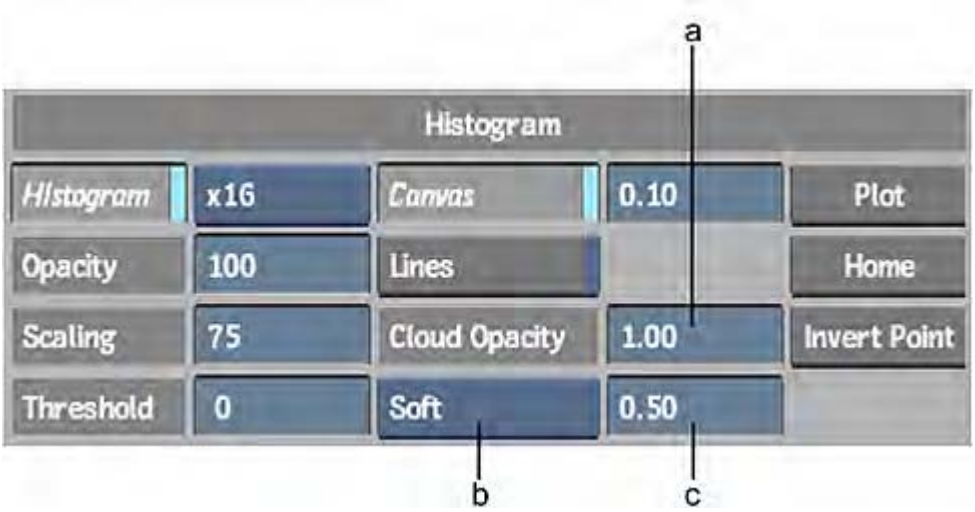
You set the display of each of the three patches individually. In the Patches menu, use the Display boxes to set the display of patches 1, 2 and 3.

Display box Select an option to set the appearance of the current patch.

Select:	To display:
Box	The patch as a wireframe box.
Surface	The convex hull of the patch.
Box&Surf	Both the wireframe box and convex hull of the patch.
Cloud	The patch as a semi-transparent grey box. You can set the transparency of the box using the Cloud Opacity field in the Preferences menu. See Setting the Cloud Opacity (page 814).
None	No patch. When a patch is not displayed, it still has an effect on the image.

Setting the Cloud Opacity

You can control the opacity of the Cloud setting of ellipsoids and patches as they appear in the RGB viewer using the Cloud Opacity fields in the Preferences menu.



(a) Overall Opacity field (b) Key Element box (c) Individual Opacity field

You can adjust the opacity for all key elements or for individual key elements:

- To set the cloud opacity for all ellipses and patches, use the Overall Opacity field.
- To set the cloud opacity for an individual key element, select the element from the Key Element box, then set the opacity in the Individual Opacity field.

Displaying Plotted Points and Regions

When you plot colour values and ranges of colour values in the key image, you can display or hide the plotted points and regions in the RGB viewer.

To show or hide plotted points:

1. Enable or disable the Plot Point button in the Preferences menu.



(a) Plot Point button (b) Plot button

NOTE When you plot a colour value, the Plot Point button is automatically enabled to display the colour value in the viewer.

To show or hide Plot Regions 1, 2, or 3:

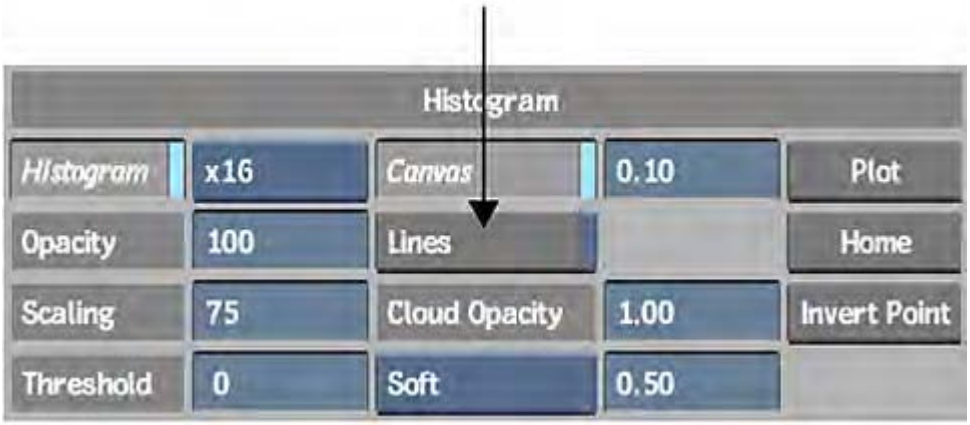
1 Enable or disable the corresponding buttons in the Preferences menu.

Selecting Key Elements in the RGB Viewer

As you translate, scale, and rotate key elements in the RGB viewer, you must select them individually. The 3D Keyer provides two methods for selecting key elements—clicking on them in the viewer or using the Picking box. The Picking box enables precise selection especially when tolerance, softness, and patch elements overlap each other in the RGB viewer.



The Preferences menu contains the Lines button. Use this button to display lines around key elements when they are selected in the viewer. This can make it easier to identify the selected key element.



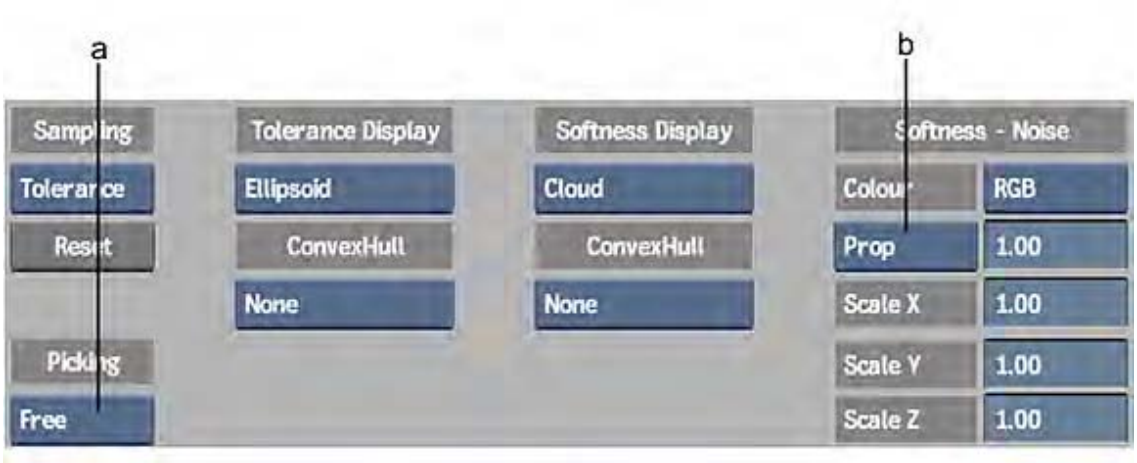
Clicking Key Elements

When you click key elements in the RGB viewer, a directional arrow appears through the selected element. The arrow points in a particular direction according to where you click on the element. See Controlling Elements with the Directional Arrow (page 818).

The Picking box must be set to Free to select an element by clicking it. Also, the Softness Scaling must be set to Prop not Minimize Noise (Minimize Noise selects softness and controls the arrow direction).

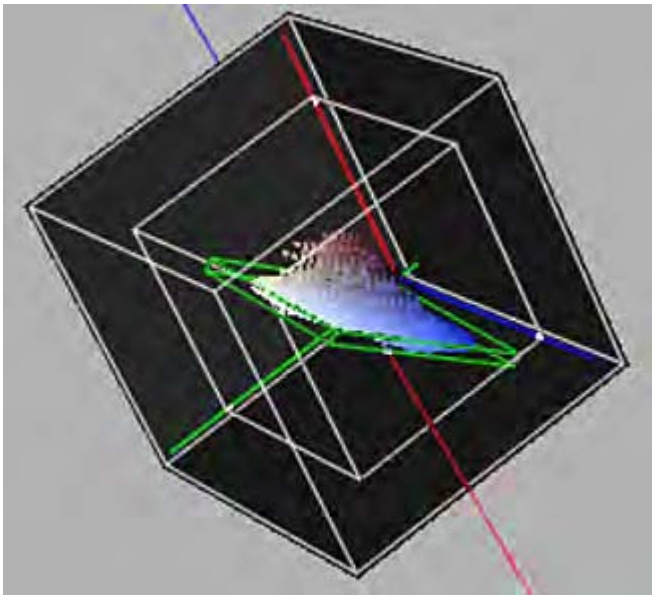
To select key elements by clicking:

1 In the Key menu, select Free from the Picking box and make sure that Prop is selected in the Softness Scaling box.



(a) Softness Scaling box (b) Picking box

2 Select a key element in the RGB viewer by clicking it.



A directional arrow appears through the selected key element and, if the Lines button is enabled, a green wireframe box appears around the selected key element.

3 At this point, you are ready to proceed with modifying the selected key element. See Translating, Scaling, and Rotating Key Elements (page 822) and Reshaping the Convex Hull (page 824).

If you are unable to select the element, the element may be behind or inside another element. You can hide the element that is in the way by setting its display to None. See Changing the Display of Key Elements (page 812). Alternatively, use the Picking box to select the element, as described in the next procedure.

Using the Picking Box

When an element is selected with the Picking box, you cannot select other elements in the viewer by clicking on them. This helps prevent other key elements from being selected by accident while you work with a particular key element.

The Picking box is also useful because it allows you to select a key element without changing its current arrow position. For example, if you accidentally click elsewhere in the viewer and lose the selection, and you want to re-select the element without changing the arrow position, use the Picking box.

NOTE When you use the Picking box to select a key element that has not been previously selected, the arrow is not displayed. You must actually click the element to set the arrow direction.

To select key elements using the Picking box:

1 In the Key menu, verify that the Softness Scaling is set to Prop (Minimize Noise forces the softness to be selected).

2 Select the key element from the Picking box.



If lines are displayed in the viewer, a selected ellipsoid has a green wireframe box around it and patch boxes change from white to green. If the element's arrow direction has been set, the arrow is displayed.

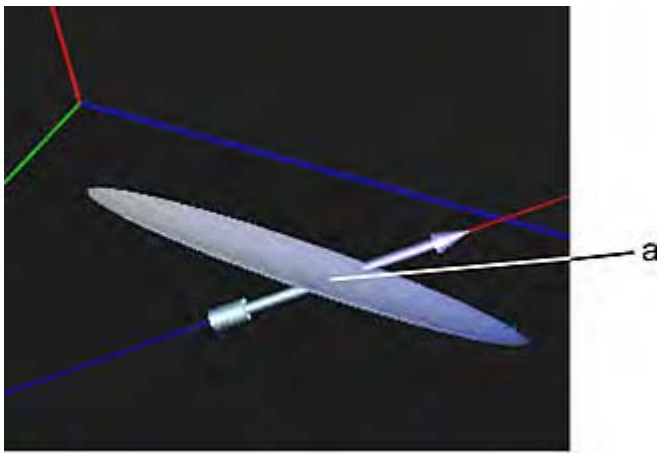
3 To display the element's arrow and set its direction, click the element.

At this point, you are ready to proceed with adjusting the range of the selected key element.

Controlling Elements with the Directional Arrow

Each key element (tolerance, softness, and patches) has a directional arrow that passes through its centre point. Use this arrow to indicate the direction in which you want to perform a particular action, such as scaling or translating the element.

- To display an element's arrow, select the element by clicking on it.
- To set the arrow direction manually, click inside the key element anywhere between its centre and the area towards which you want the arrow to point.



(a) Click in the element to set the arrow direction.

NOTE You can only click on a part of the element's surface that is visible. If necessary, rotate the RGB viewer so that the correct area of the surface is visible.

- To point the arrow towards a plotted point, simply select the element (either by clicking on it or using the Picking box) before plotting. The arrow will automatically point towards the plotted point.
- To reverse the direction of the arrow, click the Reverse Point button in the Key menu.

NOTE When Minimize Noise is enabled, this tool controls the arrow direction. See Removing Noise from Softened Areas (page 801).

Plotting Colour Values

You can plot pixels in the key image and view the location of the pixel colour value in the RGB viewer. You can plot a single colour value or a range of colour values. These plotted areas do not affect the key at all—they simply identify colour values in the viewer.

You can plot a colour value to view its location in relation to key elements in the viewer. Additionally, you can employ a number of techniques to modify a key element in relation to a plotted colour value or range of values. For example, you could plot a colour value corresponding to an unwanted grey area in the foreground subject, and then translate or scale the softness ellipsoid such that the colour value is outside the ellipsoid.

Plotting Single Colour Values

Plotting a pixel in the key image sets the exact location of the colour value in the RGB viewer. The colour value is identified by a small, white wireframe cube in the viewer. As an alternative, you can use the Plot tool to select a colour value directly in the viewer by clicking on a 3D histogram cube.

If a key element in the viewer is selected before you plot the colour value, the element's directional arrow moves to point towards that colour. Plotting pixels in this way allows you to quickly refine the matte by translating or scaling a key element with respect to the plotted colour.

To plot a pixel on the key image:

1. In the RGB viewer, select the key element that you want to modify in relation to a plotted colour value. For example, click a patch key element.

   If you have problems selecting the element, see Selecting Key Elements in the RGB Viewer (page 815).
2. If needed, zoom in on the key image so that you can select a pixel more easily.
3. Click Plot.



   Plot is enabled.
4. Click a pixel in the key image. You can also drag the cursor in the image, then release the cursor when the colour picker is over the desired pixel.

   In the RGB viewer, the colour value of the selected pixel is plotted and the arrow of the selected key element moves to point towards the plotted colour value.

If needed, rotate the viewer by pressing Ctrl and dragging it to get a better view of the key element with respect to the plotted value.

5 Proceed with modifying the selected key element. Using the keyboard shortcuts, you can move the selected key element to refine the key.

Press:	To:
V-click	Sample pixels where you click in the image. The arrow is repositioned in the RGB viewer. Drag left or right to scale along this axis.
6 and drag key element	Move the selected key element in the direction of the arrow.
7 and drag key element	Rotate the selected key element in any direction around its centre point.
8 and drag key element	Scale the selected key element proportionally from its centre. Drag left to scale down and right to scale up.
9 and drag key element	Scale the selected key element in the direction of the arrow (non-proportionally) from its centre. Drag left to scale down and right to scale up.
0 and drag key element	Scale the selected key element in the direction of the arrow (non-proportionally), starting from the point on the key element's surface where the back of the arrow remains anchored in place.

As you move the key element away from the plotted pixel, you can see the effect on the key image or the matte.

To select a colour value in the RGB viewer:

1 In the RGB viewer, select the key element that you want to modify in relation to a plotted colour value. For example, click a patch key element.

If you have problems selecting the element, see Selecting Key Elements in the RGB Viewer (page 815).

2 Press H to display the 3D histogram.

3 If needed, zoom in on the viewer or rotate it so that you can select a histogram cube more easily.

4 Click Plot.



Plot mode is activated and the cursor changes to a colour picker.

5 Select a cube in the histogram of the RGB viewer. If needed, drag the colour picker around the viewer until it is pointing to the correct colour value.

The selected cube is surrounded by a white wireframe box and the direction of the key element arrow moves toward the selected cube.

Plotting a Range of Colour Values

When you plot a range of colour values in the image, the resulting plotted region in the RGB viewer appears as a 3D convex hull. You can plot up to three colour ranges in the image. The plotted regions do not in themselves affect the key image in any way.

A plotted region allows you to compare the intersection of a key element with a particular colour range in the image, providing you with yet another alternative for adding precision to your key.

For example, use this tool to plot a colour range that you do not want to be softened. Using the plotting region as the reference, you translate and scale the softness ellipsoid away from the plotted region to ensure the softness precision of the key. Plotted regions also help you to determine whether or not to use a garbage mask. If a colour that you want to soften is the exact colour that you want key out, then you should apply a garbage mask.

To plot a region:

1 In the Preferences menu, enable the Plot Regions 1 button.

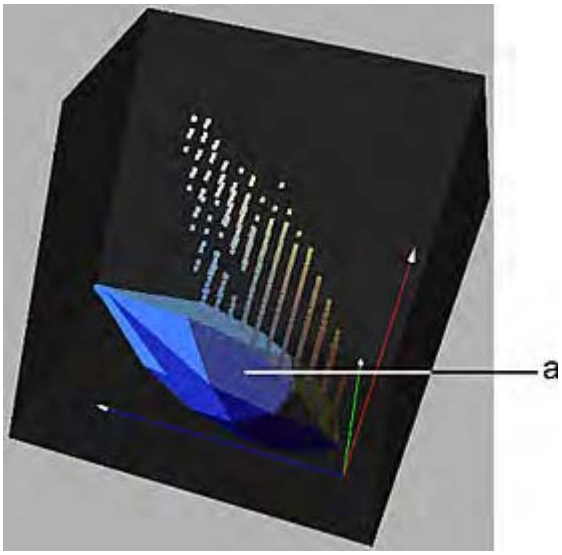
The display of Plot region 1 is enabled.

2 In the Sampling box, select Plot1.



3 In the key image, sample one or more pixels using the same methods that you use for sampling tolerance or softness by dragging in the image.

A 3D convex hull appears in the RGB viewer.



(a) Each vertex contributes to the convex hull that represents the plotted region.

4 Continue sampling pixels in the key image until you are finished plotting the region.

Plotting Several Regions You can plot a second and third region, and compare the positions of the convex hulls. To plot a second region, use the procedure described above, selecting Plot 2 from the Sampling box and enabling the Plot Regions 2 button.

For example, sample one plot region in an area of the matte containing unwanted grey areas, and another plot region where the grey area is intentional. You can then observe where the colours of the two regions are the same (where the plotted regions intersect), and adjust the softness accordingly.

To:	Do this:
Remove the grey areas from the foreground subject.	Move the softness key element away from the plotted region intersection.
Prevent the matte from becoming over softened or transparent.	Use a patch.
Keep part of the key that has the exact colour as the area being softened.	Use a garbage mask.

Translating, Scaling, and Rotating Key Elements

You can fine-tune the key by translating, scaling, or rotating the tolerance ellipsoid, softness ellipsoid, and patches in the RGB viewer. For example, scale the softness ellipsoid in a particular direction to increase the softness. You use the same methods for all three key element types.

To translate or scale an element in a particular direction, you use its arrow to specify the direction. When scaling an element along the axis of the arrow, you have the option of scaling it equally in both directions from its centre, or scaling it only in the forward direction of the arrow. You can also scale an element proportionally in all directions (X, Y and Z directions).

Before performing the following operations, set up the RGB viewer to best display the elements you are working with. Zoom in on the element, decide if you want to show the histogram, and hide elements that are in the way. See Changing the Display of Key Elements (page 812).

TIP Display the 3D histogram as you make modifications to see the areas of colour that you are working in. Try reducing the size and opacity of histogram cubes to see more clearly. See Setting the 3D Histogram Display (page 810).

To modify the tolerance or softness using these techniques, the ellipsoid must be displayed. The Tolerance Display or Softness Display must be set to Ellipsoid, Wireframe, or Cloud. Likewise, when reshaping patches, the patch box must be displayed. The box is displayed when the Display is set to Cloud, Box, or Box&Surf.

See More Keying Techniques (page 825) to learn ways to apply these techniques in specific situations.

To translate a key element:

1. Select the element. See Selecting Key Elements in the RGB Viewer (page 815).
2. Identify the direction in which you want to translate the element and set the arrow in this direction. You can do this manually. See Controlling Elements with the Directional Arrow (page 818).
3. Alternatively, you can also plot a pixel on the image or a histogram cube (see Plotting Single Colour Values (page 819)). When you plot a pixel on the image or a histogram cube, the arrow of the selected key element automatically points towards the plotted pixel.
4. Press 6 and drag the cursor in the RGB viewer. Drag to the right to translate the element forward along the arrow axis, or to the left to translate it backward.

To rotate a key element:

1. Select the element. See Selecting Key Elements in the RGB Viewer (page 815).
2. Press 7 and drag the cursor in the key element.

   The key element rotates in any direction around its centre point.

To scale a key element proportionally:

1. Select the element. See Selecting Key Elements in the RGB Viewer (page 815).
2. Press 8 on the keyboard and drag in the RGB viewer. Drag to the right to increase its size, or to the left to make it smaller.

To scale a key element from its centre:

1. Select the element. See Selecting Key Elements in the RGB Viewer (page 815).
2. Set the arrow in the desired direction. You can do this manually. See Controlling Elements with the Directional Arrow (page 818).

   You can also set the arrow by plotting a point on either the image or a histogram cube. When you plot a point on the image or a histogram cube, the arrow of the selected key element automatically points towards the plotted pixel. See More Keying Techniques (page 825).
3. Press 9 and drag the cursor in the RGB viewer. Drag to the right to scale it larger along the axis of the arrow, and to the left to scale it smaller.

   The element is scaled equally in the forward and backward directions of the arrow.

To scale a key element in one arrow direction:

1. Select the element. See Selecting Key Elements in the RGB Viewer (page 815).
2. Set the arrow in the desired direction. You can do this manually. See Controlling Elements with the Directional Arrow (page 818).
3. You can also set the arrow by plotting a point on either the image or a histogram cube. When you plot a point on the image or a histogram cube, the arrow of the selected key element automatically points towards the plotted pixel. See More Keying Techniques (page 825).

4 Press 0 and drag the cursor in the RGB viewer. Drag to the right to scale it larger in the forward direction of the arrow, or to the left to scale it smaller towards the centre of the element.

The element is scaled in the forward direction of the arrow only.

Reshaping the Convex Hull

You can exclude or add specific ranges of colour to a sample by modifying the shape of the convex hull of the softness ellipsoid, tolerance ellipsoid, and patches. You do this by manipulating the vertices on the hull. The ellipsoid or patch box changes shape according to the new shape of the convex hull.

This can be an accurate way of modifying the sampled ranges since you can see the area of RGB colour space in which you are working and because you can plot specific parts of the image and add them to the hull. Also, scaling the ellipsoid using one vertex of the convex hull provides a more precise (localized) result than scaling using the arrow alone (using 0).

To change the shape of the convex hull, you can add and move vertices, or delete vertices from the convex hull entirely.

Before performing the following operations, set up the RGB viewer to best display the elements you are working with. Zoom in on the element, decide if you want to show the histogram, and hide elements that are in the way.

To reshape the convex hull of the tolerance or softness, the ellipsoid must be displayed. It must be set to Ellipsoid, Wireframe, or Cloud. Likewise, when reshaping patches, the patch box must be displayed. The box is displayed when the Display is set to Cloud, Box, or Box&Surf.

It is best to show the convex hull. Although it is not necessary to perform the operation—unless you are deleting vertices, you can see what you are doing more clearly. See previous sections in this chapter, such as Changing the Display of Key Elements (page 812).

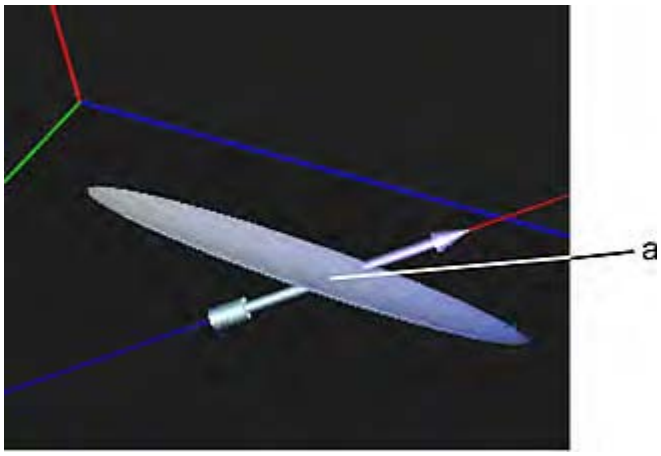
To add vertices to the hull and move them:

1 Select the key element by clicking it.

2 Position the arrow of the selected element in the direction in which you want the ellipsoid or patch to grow or shrink. For example, point the arrow towards colours you want to include in your sample. Alternatively, plot a point to position the arrow. See Controlling Elements with the Directional Arrow (page 818).

3 Press and hold V on the keyboard and click anywhere in the RGB viewer.

A vertex is added to the hull at the intersection of the arrow and the edge of the ellipsoid or patch.



(a) Vertex is added to the hull

NOTE When using V in the RGB viewer, you can scale any convex hull (tolerance, softness or patches). If you use V in the image window, it will apply to softness only.

4 To add colours to the sample, drag the cursor to the right. This moves the vertex in the forward direction of the arrow.

To remove colours from the sample, drag the cursor to the left. This moves the vertex backward towards the centre of the element.

When you are satisfied with the result in the image window, release the cursor.

The hull is reshaped as you move the vertex. If you did not display the convex hull, it is temporarily displayed while you are moving the vertex. The direction of the arrow may change slightly as you move the vertex, but the direction it moves in is always the direction in which the arrow was initially pointing.

NOTE You can release the V key and the cursor, then press them again to scale the same vertex. You retain control over the new vertex until you change the position of the arrow.

To remove a vertex from the convex hull:

1 Display the convex hull of the key element, including the vertices. See Changing the Display of Key Elements (page 812).

NOTE You do not have to select a key element to delete a vertex from its convex hull.

2 If necessary, zoom in on the element to distinguish the vertices more clearly.

3 Press and hold the - sign on the keypad (not the one on the keyboard) and click on the vertex that you want to remove.

See Techniques for Adjusting Softness (page 825).

More Keying Techniques

In this section, learn how to adjust the key using various methods to manipulate the key elements (tolerance, softness, and patches) in the RGB viewer, as well as the Softness Scaling controls in the Key menu.

It is recommended to read Creating a Precise Key in the 3D Keyer (page 795) and Mastering the RGB Viewer (page 809) before proceeding.

Techniques for Adjusting Softness

This section includes three additional methods for adjusting the softness of the key.

Scaling the Softness Ellipsoid to Remove Grey Areas

Increasing the softness may create unwanted grey areas in the foreground subject. There are several ways to remove these grey areas. One method is to plot a pixel in a grey area, then translate, scale, or rotate the softness ellipsoid such that the plotted region is no longer in the ellipsoid. In effect, you will no longer be softening the plotted region.

If you translate, rotate, or proportionally scale the softness ellipsoid, all areas of the ellipsoid are affected to a degree. By contrast, when you use vertex scaling, you can affect a more localized area of the ellipsoid, leaving the rest of the ellipsoid unchanged. Vertex scaling is therefore often the most accurate method to use.

To scale the softness ellipsoid:

1. Show the canvas (press C) and hide the histogram (press H).
2. Use the Soft Display box to change the display of the softness ellipsoid to Cloud.
3. Select the softness ellipsoid in the RGB viewer by clicking on it or selecting Softness from the Picking box.
4. From the View box, select Keyer 3D Result to see the matte.
5. Zoom in to see the grey pixels up close.
6. Click Pref.
7. In the Preferences menu, click Plot.



8. Plot a grey pixel in the foreground subject by clicking it.

   The colour value of the pixel you clicked is plotted in the RGB viewer. The softness ellipsoid arrow passes through the plotted point.
9. To view the exact location of the plotted colour value in relation to the softness ellipsoid, rotate and scale the viewer.

   The plotted point inside the softened region is represented by the softness ellipsoid.
10. Press V, click in the viewer and drag the cursor to the left to scale the ellipsoid down along the axis of the arrow. Watch the image update interactively. As soon as the grey area in the subject disappears, release the cursor to stop scaling the ellipsoid.

    NOTE If you are not satisfied with the result, click Undo to return the ellipsoid to its previous shape, then scale again.
11. If some grey areas remain, plot those areas and repeat the procedure.

Scaling the Softness Ellipsoid to Add to Softness

You can also use the technique described in the previous section to increase the softness at the edges of the key. Start by identifying the region where you want to introduce more softness, and then plot a pixel in that area. You see the colour value in the viewer. Press the V keyboard shortcut, then either enlarge or decrease the size of the softness ellipsoid minimally by dragging the cursor to the right or left. Include the colours you want and exclude the rest.

TIP Display the 3D histogram to help you figure out the direction in which you want to scale the tolerance or softness.

Using the Softness X, Y, and Z Fields

You can also adjust the softness in the key by changing the values in the Softness Scaling X, Y and Z fields. Drag the cursor over the fields, watching the result in the image window. You can often get a good result just by experimenting.



NOTE The Sampling box does not have to be set to Softness to use these fields.

Adjusting Transparencies with Patches

When keying transparencies, the transparency is sometimes either not visible enough or too prominent in the final composite. You can use patches to lighten or darken transparencies.

To adjust a transparency using a patch:

1. Use the basic keying technique to get the transparency as soft as possible.
2. Sample a patch in the transparency. See Adding a Patch (page 805).
3. View the final composite by displaying the CurResult view.
4. Do one of the following:
   - To lighten the transparency, set the Colour of the patch to 0 and the Opacity to 100, then lower the Opacity.
   - To darken the transparency, set the Colour of the patch to 100 and the Opacity to 0, then raise the Opacity.

Setting 3D Keyer Preferences

The following section explains the available preferences for working in the 3D Keyer.

Auto Key

Enable this button to set keyframes to animate the key. See Animating Key Elements (page 807) for details.

Regen

When you sample (softness, tolerance, and so on) using the colour picker or the numeric fields, you have the option of having the image update interactively as you move the cursor, or having the image update only when you release the cursor.

It is useful to update the image only after sampling if interaction during sampling is slow. This can happen if you are working with high resolution images, or if you are sampling while viewing a context from further along the pipeline. This can also occur in the current result view (displayed as <Currently Selected Node> Result, in the View box), where a large amount of processing is required to display the result.

To update the image interactively:

1 Enable Regen.

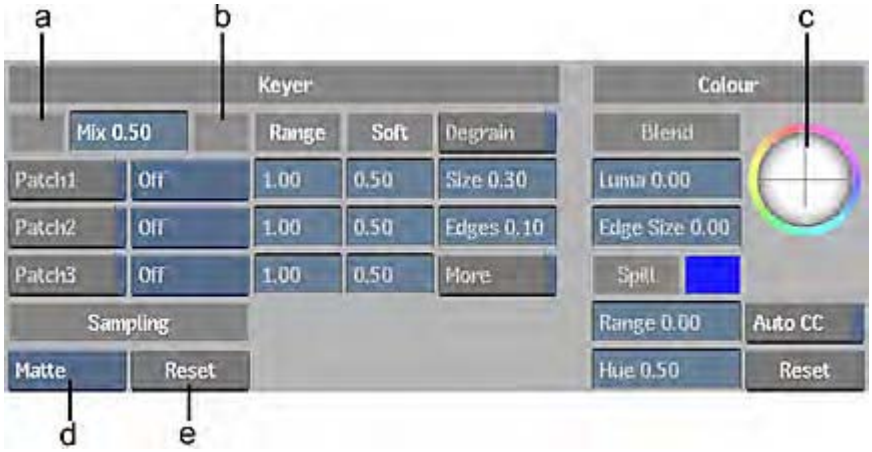
To update the image after completing the sample:

1 Disable Regen.

Creating and Refining a Key in the Master Keyer

Use the Master Keyer method to create a complete key—extract colours to generate a matte and then refine the result—using only the Master Keyer menu. The Master Keyer uses an algorithm that quickly isolates a colour and is very good for chroma keying.

With the Master Keyer, you can refine the key by gesturally modifying the matte, removing colour spill, blending edges, applying patches, and removing grain.



(a) Primary Sample colour pot (b) Secondary Sample colour pot (c) Edge Balance trackball (d) Sampling box (e) Reset button

The following procedure is a recommended workflow for keying a clip with the Master Keyer. You may not need to complete all the procedures. You may also revisit procedures as you develop the key.

To create a complete key:

1 Generate and refine a matte using the Matte controls.
2 Remove any colour spill.
3 Improve the blend between the front and back clips.
4 If the image is grainy, apply the Degrain algorithm.

5 Create any necessary patches to remove unwanted grey areas from the matte.

Generating Mattes

Use the Matte controls in the Master Keyer to generate and refine a matte for your chroma key.

To generate and refine a matte:

1 In the Master Keyer menu, select Matte from the Sampling box.



2 Set the view to Result or Matte view.

3 Click the Primary Sample colour pot and then sample the image. Click the image to sample a single pixel.

The Auto CC button is enabled and the initial pure colour value for the key is set.

4 Click the Secondary Sample colour pot and then sample the image. Sample an area where you do not want any softness in the matte.

5 Drag in the Mix field to adjust the mix between the primary and secondary sample. Drag right to include more of the secondary sample or left to include less.



TIP You can adjust the mix at any time as you develop the key.

6 Gesturally refine the matte. In the image window, click an area of the matte that you want to refine—only those parameters that pertain to the area you click appear. Parameters are displayed in order of importance, from top to bottom (those that are brightest and at the top have the greatest effect on the image). You can then modify a parameter by dragging its highlighted slider.

- To add softness, drag a slider to the right.
- To remove softness, drag a slider to the left.

The red indicator shows the original value and the yellow indicator shows the current value.

7 To modify more than one parameter, move the mouse between the parameters to highlight a parameter, or drag the pen vertically. When you highlight the parameter you want to adjust, drag the slider.

TIP If you do not like the result, you can click Undo to reset parameters directly after you complete the operation. There is only one level of undo in the Keyer menu.

8 When you are finished modifying the displayed parameters, click another area of the image without highlighting a parameter to hide them. Alternatively, you can press any key, such as `spacebar` or `Esc`.

The parameters are no longer displayed.

9 Repeat steps 6 to 8 in other parts of the matte to further refine it. Only the parameters that apply to the problem area will appear.

10 To scroll through the image and display the pertinent parameters, Alt-drag the image without clicking it. The parameters update as you drag. You can then modify the displayed parameters by clicking the image and dragging the highlighted slider.

NOTE To reset matte parameters, click the Reset button, next to the sampling box. All matte parameters are reset, except the Mix field and the key colour.

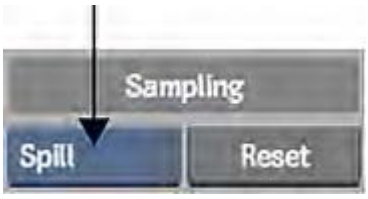
After you have generated your matte, perform basic colour spill removal (page 830).

Basic Colour Spill Removal

After you create and refine your matte (page 829) for the key, you can perform basic colour spill removal by gesturally modifying the Spill parameters.

To gesturally remove colour spill:

1 In the Master Keyer menu, select Spill from the Sampling box.



2 Set the view to Result view.

3 Enable Auto CC to apply the Spill settings to the front clip.



4 Remove any colour spill. Click anywhere in the image window, and then modify the parameters that appear by dragging the sliders—you do not need to click a specific area.

Drag:	To:
Range	Set the range for the removal of colour spill along the edges of the key. Drag right to soften the edge and remove colour spill further into the key. Drag left to harden, or create a thinner, edge.

Drag:	To:
Hue	Modify and suppress colours that are adjacent to the primary sample.

The red indicator shows the original value and the yellow indicator shows the current value. The values also appear in the Spill fields. The original values should be good, so make small adjustments only.

5 To modify both parameters, move the mouse between them to highlight a parameter, or drag the pen vertically. When you highlight the parameter you want to adjust, drag the slider.

If you do not like the result, you can click Undo to reset parameters directly after you complete an operation. There is only one level of undo in the Modular Keyer.

6 When you are finished modifying the displayed parameters, click another area of the image without highlighting a parameter to hide them. Alternatively, you can press any key, such as `spacebar` or `Esc`.

The parameters are no longer displayed.

NOTE To reset Spill parameters, click the Reset button, next to the sampling box.

If you are satisfied with your colour spill removal, move on to blending your front and back clips (page 832).

If you want to perform additional colour spill suppressions, see Advanced Colour Spill Removal (page 831).

Advanced Colour Spill Removal

If you want to suppress colour spill beyond what was shown in Basic Colour Spill Removal (page 830), you can use the Spill controls in the Colour menu to eliminate and disguise the colour spill in the Master Keyer. You can sample the colour you want to remove, and then adjust the Range and Hue fields to suppress the selected colour and shift its adjacent colours.



(a) Spill Colour pot (b) Auto CC button in Range menu (c) Spill fields

When you sample an image to generate a key from a Range menu, the sampled colour in the Spill colour pot is also updated. However, the inverse is not true—if you sample a colour using the Spill colour pot, sampled colours used to generate keys are not updated. You can therefore sample colour spill without affecting the original sample.

NOTE To apply these settings to the key, the Auto CC button must be enabled.

To remove colour spill from a key:

1. Click the Result node so that you can see the changes in the right side viewer. If you do not see the result, see Setting Up Viewports for Keying (page 794).
2. Enable Auto CC in the Colour menu to apply the Spill settings to the front clip.
3. The Spill colour pot displays the colour that will be suppressed in the clip. By default, the original colour extracted from the key-in clip is displayed. In many cases, you can use this colour because the colour spill is the same colour as the original colour. However, if the colour spill is not the same, change the colour sample. Click the colour pot, use the colour picker to sample the colour spill in the image window, and then click the colour pot again.

   NOTE When you select the colour you want to key from the Master Keyer menu, the colour that appears in the Spill colour pot is automatically updated to match the key colour. However, the inverse is not true—when you sample a colour from the Spill colour pot, the sampled colour used to generate a key is not updated.

4. Drag in the Spill fields to remove the colour spill.

Drag:	To:
Range	Suppress the primary sample colour where there is colour spill (along the edges of the key).
Hue	Modify colours that are adjacent to the primary sample and further remove colour spill.

   NOTE You can reset the Spill parameters, Blend parameters, and the Edge Balance trackball by clicking Reset, located to the right of the Hue field.

When you are satisfied with your colour spill removal, move on to blending your front and back clips (page 832).

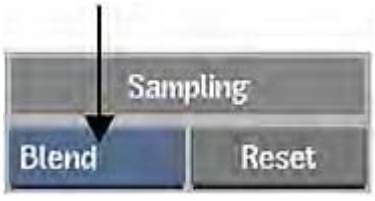
Blending Front and Back Clips

After you remove colour spill (page 830), you can use the Blend parameters in the Master Keyer to gesturally modify the luminance at the edge of the key so that it blends with the luminance in the background clip. For example, when the front clip is darker than the back clip, you can use the Master Keyer's Blend parameters to lighten the edge of the key.

When you blend front and back clips, you can add a cast to the edge of the key and further merge the clips.

To blend the front and back clips:

1. In the Master Keyer menu, select Blend from the Sampling box.



2. Set the view to Result view.

3 Enable Auto CC to apply the Spill settings to the front clip.



4 Blend the front and back clip. Click anywhere in the image window, and then modify the parameters that appear by dragging the sliders—you do not need to click a specific area.

Drag:	To:
Luma	Darken or lighten the edge of the key. Luma only affects the luma of the edge.
Edge Size	Set the range for the blend. Drag right to soften the edge and blend further into the key. Drag left to harden, or create a thinner, edge. Edge Size affects both the Luma field and the Edge Balance trackball.

TIP To drag faster, apply more pressure with the pen, or hold Alt+Spacebar while you drag.

The red indicator shows the original value and the yellow indicator shows the current value. The values also appear in the Blend fields.



5 To modify both parameters, move the mouse between them to highlight a parameter, or drag the pen vertically. When you highlight the parameter you want to adjust, drag the slider.

TIP If you do not like the result, you can click Undo to reset parameters.

6 When you are finished modifying the displayed parameters, click another area of the image without highlighting a parameter to hide them. Alternatively, you can press any key, such as `spacebar` or `Esc`.

The parameters are no longer displayed.

7 To add a cast to the edge of the key and improve the overall look by matching the edge with a colour cast in the back clip, drag the Edge Balance trackball toward the colour you want to add. The trackball only affects the chroma of the edge.



NOTE To reset Blend parameters, click the Reset button, next to the sampling box. All blend parameters are reset, except the Edge Balance trackball. Control-click the Edge Balance trackball to reset it.

After you have set your blend parameters for the front and back clip, remove grain from your clips (page 834).

Removing Grain

After you have set your blend parameters for the front and back clips (page 832), you may want to remove grain from your clips. Graininess can make it difficult to pull a clean and effective key. Use the Degrain tools in the Master Keyer to remove grain from clips.

NOTE Degrain parameters are animatable. Degrain samples are also animatable and appear in the Channel Editor as the Degrain_Matrix channel, located in the Key_Degrain folder. See Animating Keyframes (page 1471).

To remove grain:

1 In the Master Keyer menu, select Degrain from the Sampling box.



2 Set the view to Result view.

3 Sample a grainy area of the image. To sample a single pixel, click the image. To sample an area of the image, Control-drag a selection box.

The Degrain button is enabled and the algorithm is applied to the image—grain is removed from the image.

NOTE The Degrain button is either enabled or disabled for the entire clip—you cannot turn it on and off for different frames in the clip.

4 Drag in the Degrain fields to modify the grain size and restore edge sharpness.



(a) Size field (b) Edges field

Drag:	To:
Size	Estimate the size of the grain in the image.
Edges	De-sharpen the edge of the image. By default, Degrain sharpens the edges. Use the Edges field to restore the natural look of the edges in the image.

5 If you are not satisfied with the result, you can start over with a new sample, and then adjust the Degrain fields. To resample an area of the image, Control+Alt-drag the image.

The sample is outlined in green, indicating that you are resampling the image.

NOTE You can also reset Degrain parameters. To reset Degrain parameters, click the Reset button, next to the sampling box.

6 If resampling the image and adjusting the Degrain fields does not sufficiently remove grain, enable More to increase the overall effect of Degrain.

NOTE The More button is either enabled or disabled for the entire clip—you cannot turn it on and off for different frames in the clip. When you enable More, processing speed slows down.

After you have removed grain from your clips, you may still have to remove unwanted greys (page 835).

Removing Unwanted Greys

If you have unwanted grey areas in the matte, you can use up to three patches in the Master Keyer to isolate a range of colours to be included in, or excluded from, the key.

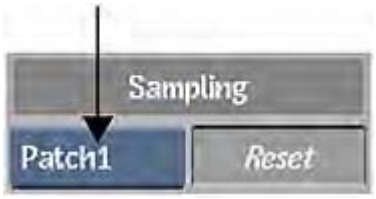
NOTE Patch parameters are animatable. Patch samples are also animatable and appear in the Channel Editor as a Matrix channel, located in the Patch folder.

There are three types of patches in the Master Keyer, that are applied to different areas of the image.

Selected patch:	Is applied to:
Black	Areas of the image to be included in the black part of the matte.
White	Areas of the image to be included in the white part of the matte.
Edge Analysis	Areas of the image that are along the edge of the key. Edge Analysis is useful when there is a specific edge you want to erode but cannot do so with the Matte parameters. You can then increase or decrease the softness of this patch using the Soft field.

To remove unwanted greys:

1 In the Master Keyer menu, select Patch1, Patch2, or Patch3 from the Sampling box.



2 Set the view to Matte view or Result view.

3 Sample the image where you want to apply the patch. To sample a single pixel, click the image. To sample an area of the image, Control-drag a selection box.

When you Control-drag to sample, the sample is outlined in red. In the Master Keyer menu, the Patch button is enabled and the appropriate patch appears in the Patch box and is applied to the image.

NOTE A Patch button is either enabled or disabled for the entire clip—you cannot turn it on and off for different frames in the clip.



(a) Patch buttons (b) Patch controls (c) Patch boxes

4 To add more colour to the patch, resample the image.

5 To use the same patch but start with a new sample, Control+Alt-drag the image.

The sample is outlined in green, indicating that you are resampling the current patch.

6 To manually select a patch type, select it from the Patch box.

NOTE If you want to reset the patch so that you can automatically select the patch type, you must disable the patch and set the patch type to Off. You can then resample an area in the image and generate a patch type.

7 To improve the patch, use the Patch controls.

Drag:	To:
Range	Increase or decrease the colour range that is included in the patch.
Soft	Soften the edge.

NOTE To reset Patch parameters, click the Reset button, next to the sampling box.

Creating a Key by Extracting a Single Colour with the Channel Keyer

You can extract one of the three primary colours (red, green, or blue), or a custom colour from the key-in clip. This method is useful for clips containing transparencies such as glass or smoke.

Extracting a Primary Colour

Extract one of the three primary colours from the key-in clip when you have a front clip shot in front of a blue, red, or green screen.

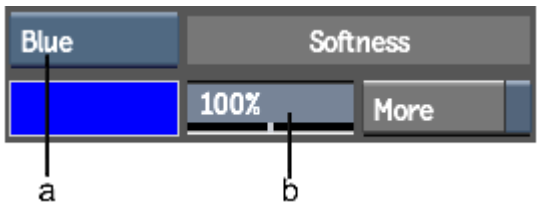
To create a key by extracting a primary colour:

1 From the Start Mode dropdown list, select Reset to Channel and click Confirm.

The Channel keyer processing pipeline is displayed in the schematic view.

2 Double-click the Channel Keyer node.

The Channel controls appear.



(a) Channel Mode box (b) Softness field

3 Select one of the primary colour options (Red, Green, Blue) from the Channel Mode box.

4 Enable More to enhance the keying effect.

The More option extends the range of the colour to be extracted from the key-in clip.

5 Increase or decrease the softness for the key using the Softness field.

The softness value determines how much of the key-in clip is partially transparent in the matte. Softness creates a smoother transition between the front and back clips in the composite clip.

6 Click Result to see the result of your key.

Extracting a Custom Colour

Use the Custom colour channel option in the Channel Keyer to select the colour to be extracted from the key-in clip. This option can give good results when keying transparencies, particularly by experimenting with the Softness and Colour Spread values.

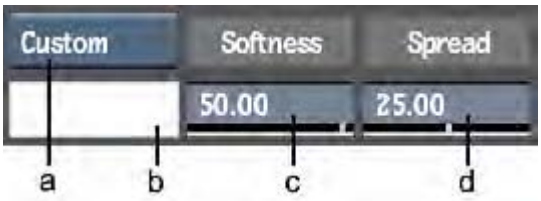
To create a key by extracting a custom colour:

1 From the Start Mode dropdown list, select Reset to Channel and click Confirm.

The Channel keyer processing pipeline is displayed in the schematic view.

2 Double-click the Channel Keyer node.

The Channel controls appear.



(a) Channel Mode box (b) Average Colour pot (c) Softness field (d) Colour Spread field

3 Select Custom from the Channel Mode box.

4 Click the Average Colour pot.

The cursor changes to a colour picker.

5 Click the image to select a single colour, or drag the colour picker across a region to obtain the average of the colours encountered by the colour picker. Try selecting different colours in the background area to get the best result.

TIP To keep shadows, click just outside them. To remove shadows, click inside them.

6 Adjust the softness for the key using the Softness field.

Increasing softness raises the level of grey in the matte. Lowering it makes the matte sharper.

7 Adjust the colour spread for the key using the Spread field.

Increasing the colour spread value extends the range of colours extracted from the key-in clip.

8 Click the Result node so that you can see the changes in the right side viewer. If you do not see the result, see Setting Up Viewports for Keying (page 794).

NOTE To improve the key at this point, try adjusting it with the histogram in the 2D Histogram node. See Adjusting the Luminance of the Key (page 842).

Creating a Key by Extracting a Range of Colours with the HLS, YUV, RGB and RGBCMYL Keyers

Another technique for creating a key is to extract a range of colours from the key-in clip. Use this technique for clips where the colour you are extracting contains impurities.

When you build a key by defining a colour range, you can use one of four colour models: RGB, YUV, HLS, or RGBCMYL. Each model interprets the key-in clip differently and gives a slightly different result.

After you choose a colour model, you set a range of colours to become partially transparent in the key-in clip to soften the transition between the front and back clips. This is called the *softness range*. All pixels in the front clip within the softness range become grey in the matte. The key should have the greatest possible softness value.

After setting the softness range, you set a range of colours to be keyed out in the key-in clip. This is called the *tolerance range*. All pixels in the front clip within the tolerance range become black in the matte. The tolerance range must not be too large or the edge of the composite will be too hard and the subject in the front clip will appear to be pasted into the back clip.

About the Different Colour Models

There are four colour models you can use.

HLS

In the HLS Keyer menu, you set the softness and tolerance ranges using the hue, luminance, and saturation channels.

YUV

In the YUV Keyer menu, you set the softness and tolerance ranges using the luma and chroma signals of YUV component video.

RGB

In the RGB Keyer menu, you set the softness and tolerance ranges using the red, green, and blue channels.

RGBCMYL

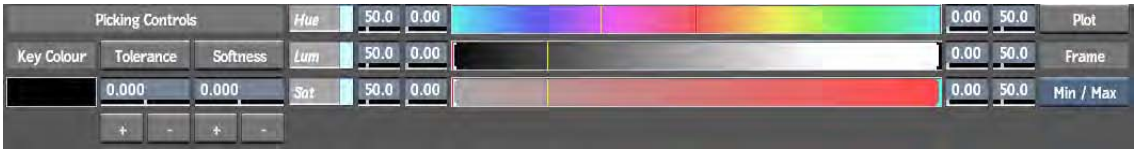
In the RGBCMYL Keyer menu, you set the softness and tolerance ranges using the red, green, blue, cyan, magenta, yellow, and luminance channels. This colour model provides subpixel resolution.

Selecting a Colour Model Based Component Keyer

To determine which component keyer will give you the best key, try creating a sample matte for each keyer.

To create a sample matte for each keyer:

1. From the Start Mode dropdown list, select one of the following: Reset to RGB, Reset to YUV, Reset to HLS, Reset to RGBCMYL.
2. Select Key In in the View box to view the key-in clip.
3. Double-click the Keyer node in the schematic.



The softness for each model is set automatically to 50, which helps you to quickly gauge the model that will create the best key for your clip.

4 At frame 1, click the Average Colour pot.

5 Drag the colour picker around the area you want to key out.

The average colour sampled by the colour picker appears in the Average Colour pot. In each channel of the colour model, the channel value of the average colour appears as a white line. All pixels in the key-in clip with the selected colour value are keyed out.

The yellow lines in the colour model channels indicate the limits of the softness range. The pixels in the front clip with colour values at the centre of the range are black (transparent) in the matte. As you move away from the centre, the pixels become more opaque.

6 From the View box, select Matte.

7 Repeat this procedure to plot the average colour using the other colour models, and then view each resulting matte to determine which model gives the best result. After you decide on a colour model, use the tools described in the next sections to refine your matte.

After you have selected a colour model and set the average colour, set the softness range (page 840).

Setting the Softness Range

After you choose a colour model based component keyer and set the average colour (page 839), adjust the softness range. The softness at the edges of the matte increases as you increase the softness range. To set the softness, you can use the colour picker, the numeric fields, or the Master Softness field.

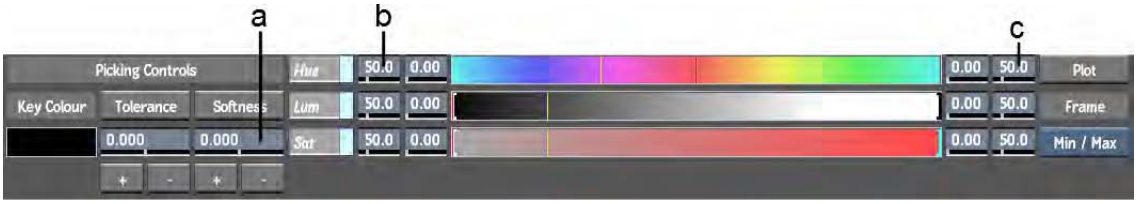
To set the softness using the colour picker:

1 Zoom in to see the edges of the matte more clearly.

2 Click Softness in the selected colour model menu.

The cursor changes to a colour picker.

3 Position the colour picker at the edge of the matte. To increase the softness, click and slowly drag the colour picker toward the centre of the matte. The values within the area you selected are used to adjust the maximum and minimum values for the softness range. The positions of the yellow lines change as you drag the cursor on the image.



(a) Master Softness field (b) Minimum Softness field (c) Maximum Softness field

4 To increase or decrease the softness range, click the + or - button beside the Softness button and then click an area of the image.

To set the softness range using the numeric fields:

1 Set the minimum value for the softness range using the Minimum Softness field on the left side of the colour bar.

2 Set the maximum value for the softness range using the Maximum Softness field on the right side of the colour bar.

NOTE You can also set the softness range by entering a numeric value in the Master Softness field.

TIP As a reference, when setting softness and tolerance ranges, you can display the colour value for any pixel in the key-in clip using the Plot tool. You can then adjust the tolerance or softness range so the pixel falls within one of the ranges. For example, plot pixels at the edges of the matte to check for softness, or plot pixels in the background to check for tolerance.

To plot a pixel's colour values:

1. Click Plot.

   The cursor changes to a colour picker.
2. Select a pixel in the image area.

   A red bar appears in each colour gradient showing the colour value of the pixel.

After you have adjusted the softness range, set the tolerance range (page 841).

Setting the Tolerance Range

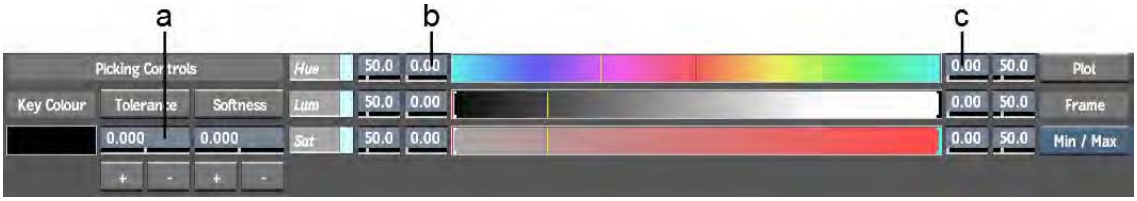
After you define the softness range in a colour model based component keyer, you can set the tolerance range to remove the greys outside the key shape. The maximum and minimum tolerance values define the range of colours to be keyed out in the key-in clip.

To set the tolerance range using the colour picker:

1. Click Tolerance.

   The cursor changes to a colour picker.
2. Drag the colour picker around the area of the image that you want to key out.

   The colour values that the colour picker samples are used to set the maximum and minimum values for the tolerance range. The white lines define the limits of the tolerance range. All colour values between the white lines are extracted from the key-in clip.



(a) Master Tolerance field (b) Minimum Tolerance field (c) Maximum Tolerance field

3. To increase or decrease the tolerance range, click the + or - button beside the Tolerance button.

   NOTE An alternative method of setting the tolerance range is to press Control and draw a rectangle in the area of the image you want to key out.

To set the tolerance range using the numeric fields:

1. Set the minimum value for the tolerance range using the Minimum Tolerance field on the left side of the colour bar.
2. Set the maximum value for the tolerance range using the Maximum Tolerance field on the right side of the colour bar.

NOTE You can also set the tolerance range by entering a numeric value in the Master Tolerance field.

TIP As a reference, when setting softness and tolerance ranges, you can display the colour value for any pixel in the key-in clip using the Plot tool. You can then adjust the tolerance or softness range so the pixel falls within one of the ranges. For example, plot pixels at the edges of the matte to check for softness, or plot pixels in the background to check for tolerance.

To plot a pixel's colour values:

1. Click Plot.
   The cursor changes to a colour picker.
2. Select a pixel in the image area.
   A red bar appears in each colour gradient showing the colour value of the pixel.

Creating a Key by Setting the Luminance

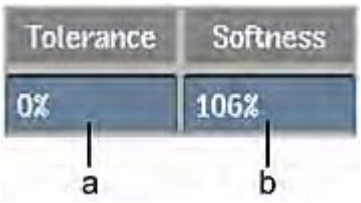
You can create a key using only the Luminance channel. The softness and tolerance values are expressed as percentages.

This technique is useful for clips with high contrast or filmed against a black background. You can also use the Luminance channel to adjust a matte that has already been rendered. Load the matte as the key-in clip, and then adjust it in the Luminance menu.

NOTE When you load a matte as the key-in clip and open the Luminance menu with default menu values, the resulting Keyer matte is identical to the original matte.

To create a key by setting the luminance:

1. From the Start Mode dropdown list, select Reset to Luminance and click Confirm.
   The Luminance Keyer processing pipeline is displayed in the schematic view.
2. Double-click the Luminance Keyer node.
   The Luminance controls appear.



(a) Master Tolerance field (b) Master Softness field

3. Set the softness in the Master Softness field.
4. Set the tolerance in the Master Tolerance field.
   A value of 100 for the tolerance creates an entirely opaque matte.

Refining Your Key

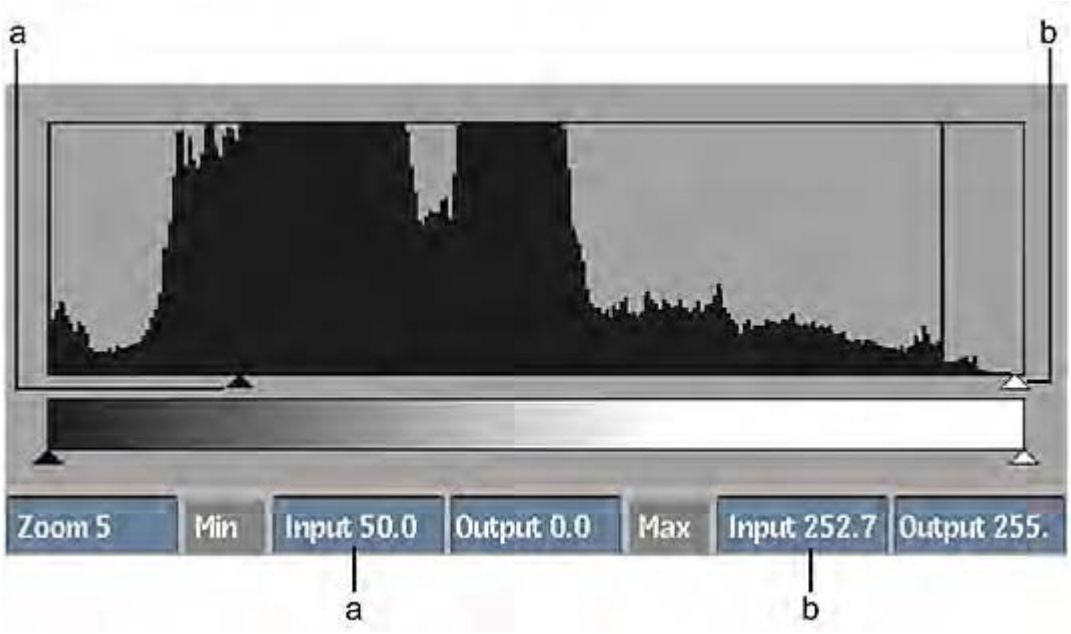
Adjusting the Luminance of the Key

After creating a key, you can adjust luminance values to fine-tune the result. The 2D Histogram node in the schematic displays the distribution of luminance values within the matte. The horizontal axis of the histogram

represents the range of luminance values in the matte and spans from 0 (black) to 255 (white). The vertical axis shows the number of pixels at each luminance value.

Setting the Range of the Luminance Values

Use the List and Gain fields with the Input Level controls, in the 2D Histogram node in the schematic, to set the range of luminance values in the matte. You can darken black areas of the matte or remove grey from white areas of the matte.



(a) Minimum Input level (b) Maximum Input level

Removing Grey from the Black Areas of the Matte

The Minimum Input level sets the start of the range of luminance values. Pixels with luminance values below the Minimum Input level are mapped to black (0).

You can set the Minimum Input level by dragging the black triangle, or by setting a value in the Minimum Input Level field.



The matte before adjusting the Input levels



The matte after lowering the Minimum Input level

Removing Grey from the White Areas of the Matte

The Maximum Input level sets the end of the range of luminance values. Pixels with luminance values greater than the Maximum Input level are mapped to white (255).

You can set the Maximum Input level by dragging the white triangle, or by setting a value in the Maximum Input Level field.



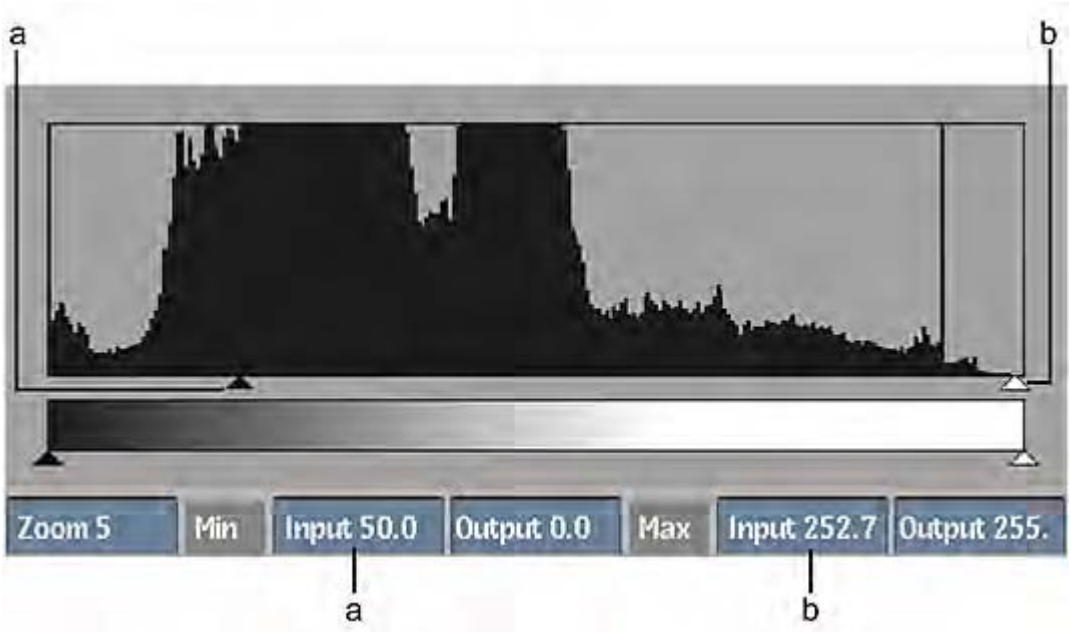
The matte before adjusting the Input levels



The matte after lowering the Maximum Input level

Remapping Black and White

Use the Output levels in the 2D Histogram node in the schematic to remap the luminance values for black (0) and white (255). You can brighten the dark areas of the matte or darken the white areas.



(a) Minimum Output level (b) Maximum Output level

Brightening the Matte

The Minimum Output level sets the luminance value of all black (0) pixels in the matte. Black pixels in the matte are mapped to the luminance value set by the Minimum Output level.

You can set the Minimum Output level by dragging the black triangle, or by setting the value in the Minimum Output Level field.



The matte before adjusting the Output levels



The matte after lowering the Minimum Output level

Darkening the Matte

The Maximum Output level sets the luminance value of all white (255) pixels in the matte. White pixels in the matte are mapped to the luminance value set by the Maximum Output level.

You can set the Maximum Output level by dragging the white triangle or by setting the value in the Maximum Output Level field.



The matte before adjusting the Output levels



The matte after lowering the Maximum Output level

Boosting the Luminance of the Key Using Gain and Lift

You can remove grey from the key by increasing the gain and lowering the lift values in the 2D Histogram node in the schematic. Increase the gain to eliminate the light greys that may be in the white area of the matte, and decrease the lift to eliminate dark greys in the black area of the matte.

NOTE Adjusting these values increases the contrast and may harden the edges of the matte.

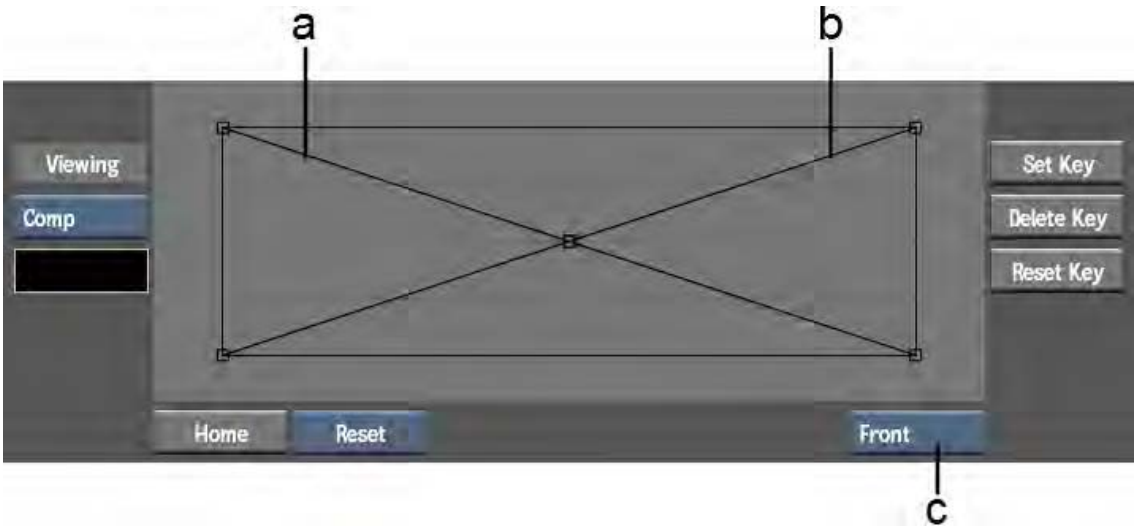
Using the Luminance Curves

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

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

To adjust the luminance curve:

1. Double-click the Result node in the schematic.
   The Luminance curve appears.



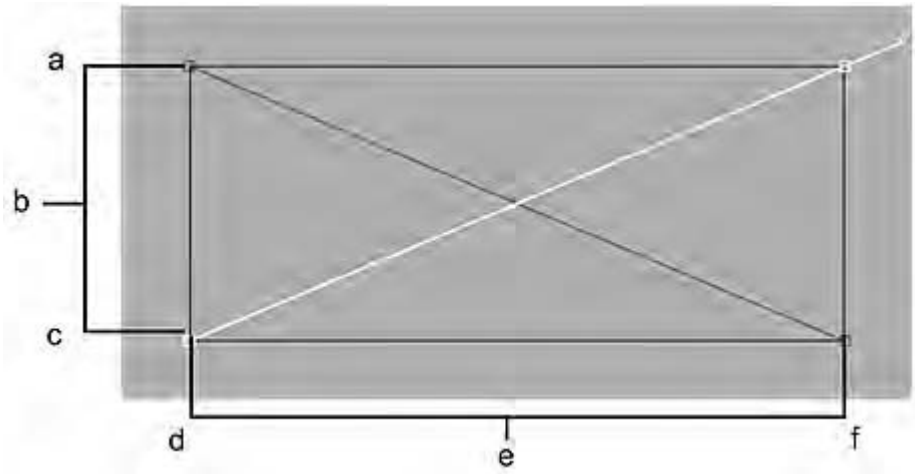
(a) Front matte curve (b) Back matte curve (c) Front/Back box

2 Use the Viewing box to select the image you want to view as you adjust the curves.

Select:	To view:
Result	The composite clip.
Matte	The front matte. You cannot see changes made to the back matte when this view is selected.
Bmatte	The back matte. You cannot see changes made to the front matte when this view is selected.
Comp	The composite with a coloured background. The default colour is black. To select a different colour, click the colour pot to the right of the Comp button. The colour picker appears.
Key In	The image that was used to pull the key.
Back	The background image that is revealed when the front is keyed.
Front	The image to which you applied the key.

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

The selected curve changes to white.



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

4 Click a point to display its tangent handle and drag the handle to adjust the curve. Use other modes in the Tools box to further adjust the curve, adding or deleting points or breaking tangent handles, as needed.

Modifying the Edges of the Key

Use the Shrink, Erode, and Blur controls in the Matte Edge node in the schematic, to enhance the edge of the keyed image.

Use:	To:
Edges	Detect the edges of your matte and fine-tune the edges with tolerance controls. This is useful for cleaning up difficult mattes.
Shrink	Remove pixels from the edge of the matte. This control should not be used when the object in the front clip has soft edges, such as hair.
Erode	Blend the light and dark edges of the matte.
Blur	Apply a softening filter to the edge of the matte. You can select either a Gaussian filter or Box filter.

To access the Edge Matte controls:

1 Double-click the Matte Edge node in the schematic.

The Edge Matte controls appear.

To detect the edges of your matte:

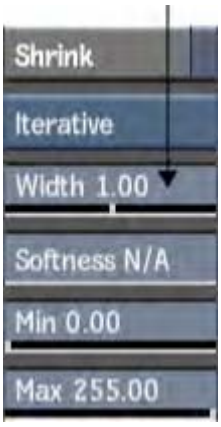
1 In the Matte Edge menu, enable Edges.

The edges af your matte are detected and displayed in the viewer.

2 Fine-tune the edges by modifying the Min and Max fields (tolerance) and the Width field, which determines the width, in number of pixels, of the detected edges.

To shrink the edge of the matte:

1. In the Matte Edge menu, enable Shrink.
2. Set a value in the Shrink Width field.



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



The matte before enabling the Shrink filter



The matte after setting the shrink width value to 1.00

To erode the edge of the matte:

1. In the Edge Matte menu, enable Erode.
2. Set a value in the Erode Width field.



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



The matte before enabling the Erode filter



The matte after setting the erode width value to 1.00

To blur the edge of the matte:

1. In the Edge Matte menu, enable Blur.
2. Set values in the Blur Width and Height fields.



These values specify the width and height of the Blur filter applied to the edge of the matte.



The matte before enabling the Blur filter



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

Adjusting Spill Controls with the Colour Curves Node

After you create a key and key out any trouble areas, some of the background colour may have spilled over at the edge of the key. Colour spill suppression in the component keyers is done through the Colour Curves node. The Colour Curves menu lets you sample the colour you want to suppress and then suppress that colour where necessary, using the suppression curve.

- Adjust the Suppression curve in the Colour Curves node to suppress a selected colour.
- Adjust the Hue Shift curve in the Colour Curves node to perform a hue shift on a selected colour.

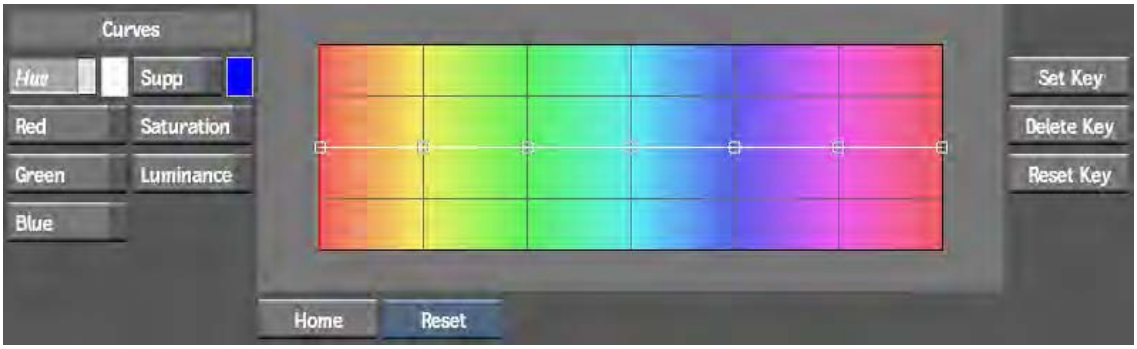
NOTE To remove colour spill in the Master Keyer, see Basic Colour Spill Removal (page 830) and Advanced Colour Spill Removal (page 831). Alternatively, you can connect a Colour Curves node between the front clip and the front input of the Master Keyer and follow the procedures below.

Adjusting the Suppression Curve

To suppress colour spill using the suppression curve:

1. Set the view to Result, so that you can see the changes as you make them.
2. Double-click the Colour Curves node in the schematic.

   The Colour menu appears.



The Colour menu displays colour curves over a hue spectrum. When you modify the shape of a curve over a region of the spectrum, only those colours are affected.

3 Click Supp to modify the Suppression curve.

4 The colour pot next to the Suppress button displays the colour that will be suppressed in the clip when you modify the curve. By default, the blue colour is displayed. To change the colour sample, click the colour pot, use the colour picker to sample the colour spill in the image window, and then click the colour pot again.

5 Click the colour pot next to the Supp button.

The cursor turns into a colour picker.

6 Select a pixel within the spill.

A red vertical bar appears in the hue spectrum identifying the colour to be suppressed.

7 Use the cursor to move the points along the Suppression curve.

8 On the Suppression curve, drag the point closest to the plotted colour down to a value of 25, intersecting the plotted colour.

The colour spill is suppressed.

9 Continue modifying the shape of the curve until you are satisfied with the result.

10 Click Saturation to adjust the saturation of the spill.

NOTE Once you remove the saturation from a spill, you may want to increase the values for the other curves (for example, red and green if you removed a blue spill) to reconstruct some of the natural colours at the edge of the keyed image.

11 Click Red, Green, or Blue to edit individual colour curves.

12 Click Luminance to adjust the luminance of the spill.

Adjusting the Hue Shift Curve

You can disguise colour spill by shifting its hue so the colour blends better with the background. Sample the colour to which you want to shift the colour spill and then adjust the Hue Shift curve to shift the colour spill accordingly.

To Create a Hue Shift:

1 Set the view to Result, so that you can see the changes as you make them.

2 Double-click the Colour Curves node in the schematic.

The Colour menu appears.

The Colour menu displays colour curves over a hue spectrum. When you modify the shape of a curve over a region of the spectrum, only those colours are affected.

3 Enable Hue to modify the Hue Shift curve.

4 Select a pixel within the spill in the Result image.

   A red vertical bar appears in the hue spectrum identifying the colour to be shifted.

5 Click the colour pot next to the Hue button.

   The colour picker appears.

6 Select or pick the colour you want to shift the spill to.

7 Use the cursor to move the points along the Suppression curve.

8 On the Hue Shift curve, drag the point closest to the plotted colour down to a value of 75, intersecting the plotted colour.

   The colour spill is shifted toward the Hue colour.

9 Continue modifying the shape of the curve until you are satisfied with the result.

Inverting a Matte

Use the Negative node to invert a matte. By inserting the Negative node between the Matte Edge and the GMask nodes in the processing pipeline, the matte is automatically inverted. The Negative node has no settings to configure.

To invert a matte:

1 From the Modular Keyer Node bin, drag the Negative node to the schematic.

2 Hold Shift and drag the Negative node to the Matte Edge node so their tabs touch and repeat for the GMask node.

   The Negative node is inserted to the schematic and your matte is inverted.

Adjusting the Front and Matte Luminance Curves

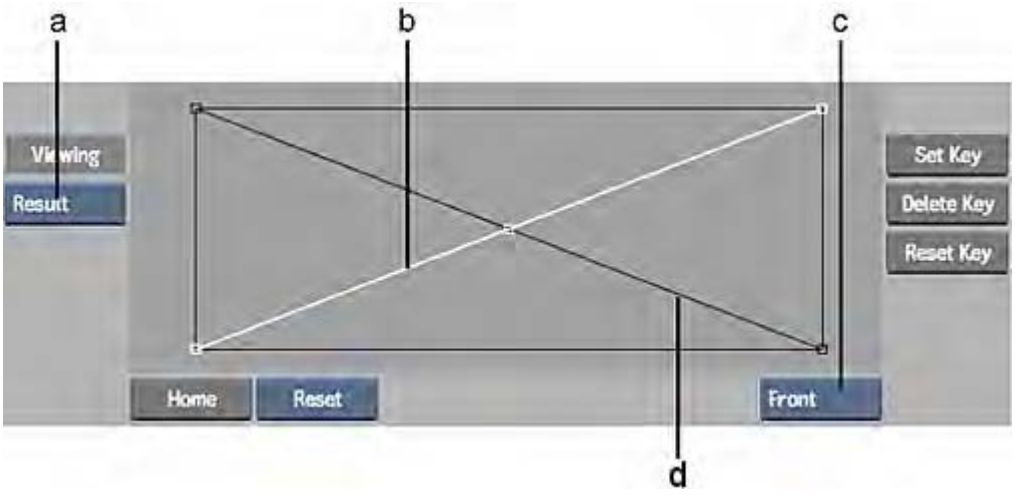
When you create a matte for the front clip, the Modular Keyer automatically creates a matte for the back clip to specify which part of the back clip is used for the composite. By default, the back matte is the inverse of the front matte.

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

To adjust the luminance curves:

1 Click the Result node in the pipeline.

   The Matte Curves menu appears.



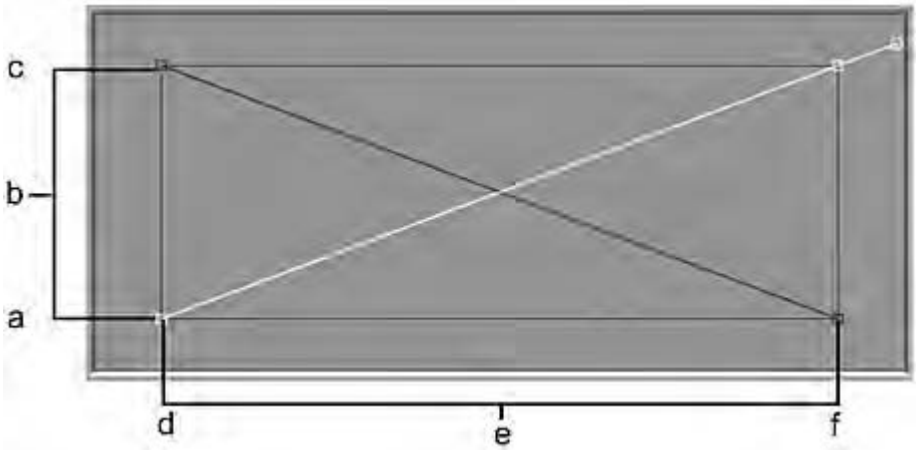
(a) Result box (b) Front matte curve (c) Matte box (d) View box (e) Back matte curve

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

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

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

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



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

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

Viewing a Key with a Solid Colour Background

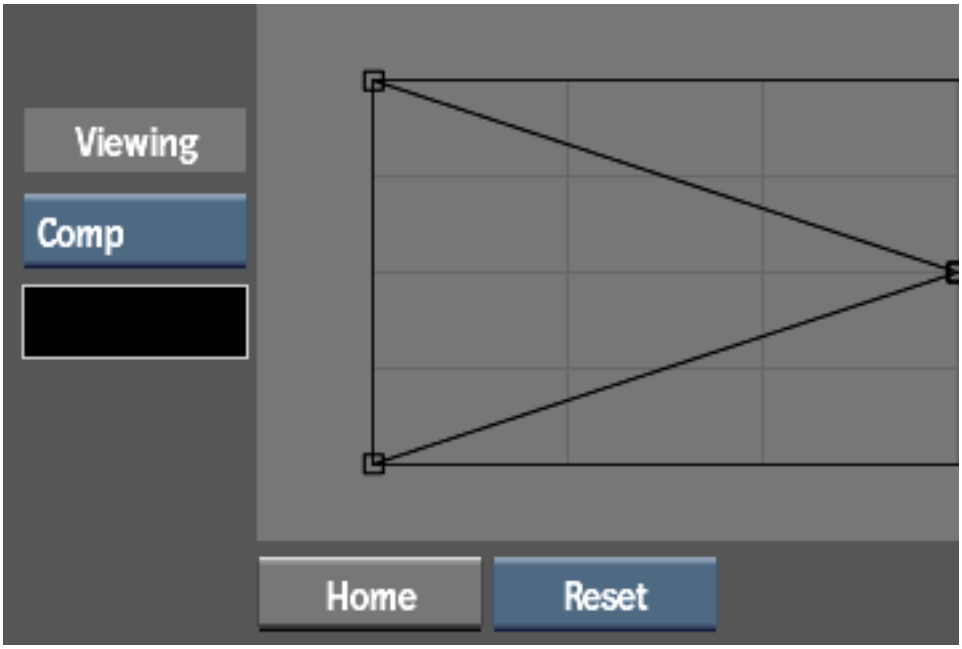
The following optional technique can help you create cleaner keys more quickly.

While creating the key, you can view the composite using a solid colour for the background in place of the back clip. This can help you see details in the image, such as colour spill, that you may otherwise miss. You can use the default colour (black) or select a colour using the colour picker.

NOTE This view does not affect the final render.

To use a solid colour background:

1. Double-click the Result node.
   The Matte Luminance Curves menu appears.
2. In the Result box, select Comp.
   A colour pot appears.



3. Click the colour pot under the Comp view output option.
   The colour picker appears.
4. Select a colour for the background using the colour picker.
   Your key is displayed on a solid colour background in the Result view.

Animating Your Key

The following parameters of a key can be animated:

- The average colour selected using the Average Colour pot in the Keyer menu

- The minimum and maximum Tolerance and Softness values
- The Lift, Gain, Shrink, and Erode values
- The blur factors for the matte and key-in clip
- The Maximum and Minimum Input and Output level values for the histogram

To display the Animation controls in the Keyers, click Animation to the left of the keyer menus. Using the Channel Editor, animate the parameters.

Resetting Your Key

You can reset the individual node within the Modular Keyer processing piepline. You can also reset the entire Modular Keyer processing pipeline.

To reset individual node within the Modular Keyer processing pipeline:

1. Select the node you want to reset.
2. Right-click the node and select Reset.
   The selected node is reset.

   NOTE You can reset multiple nodes by holding the Control key, selecting the nodes and right-clicking to reset.

To reset the entire Modular Keyer processing pipeline:

1. Shift+click the Result node.
2. Right-click the Result node and select Reset.
   The entire Modular Keyer processing pipeline is reset.

Rendering Your Key

To view your key in the timeline, you must render it. You can render the key inside the Batch FX view or in the timeline.

To render your key in the Batch FX view:

1. When you are satisfied with your key, exit the Modular Keyer by clicking the Return button.
   You are taken to the Batch FX view.
2. In the Batch FX view, click the Render Combo box and select Render. If the Render option is already displayed, simply click Render.
   The rendering starts. A progress bar appears at the bottom of the screen.
3. After the rendering is complete, click EXIT BFX to return to the timeline.
   - If you accessed the Modular Keyer from the timeline, your original source clip is updated in the timeline and in the Viewing panel and displays the result of your key.
   - If you accessed the Modular Keyer from the Tools tab, the result of your key is rendered as new media and is displayed in the Viewing panel.

   A small BFX icon appears on the clip and the timeline segment.

To render your key in the timeline:

1. Once you are happy with your key, exit the Modular Keyer by clicking the Return.

   You are taken to the Batch FX view.

2. In the Batch FX view, click EXIT BFX to return to the timeline.

   Your key is displayed in the timeline. A dotted line is displayed on the segment and you cannot view the result, as the segment needs to be rendered.

3. From the Editing panel, click the Render Combo box and select Render. If the Render option is already displayed, simply click Render.

   The rendering starts. A progress bar appears at the bottom of the screen.

4. Once the rendering is complete, the dotted line becomes a full line and you can view the result of your key. Your original source clip is updated in the Viewing panel and displays the result of your key. A small BFX icon appears on the clip and the timeline segment.

Stabilizing and Tracking 19

Use the Stabilizer to remove camera instability and motion jitter, and to track reference points in your clips. You can also use the Stabilizer to produce 2D motion or lock a bilinear surface to the clip's background. With tracking, a point or points on the clip are tracked as they move through the scene. You can then apply the resulting motion path to an object on another layer so that it follows the same path as the object you tracked.

Stabilizing is the inverse of tracking. With stabilizing, the motion path is used to shift the scene so that the point that is tracked remains fixed at one position.

Tracking and stabilizing are often processes of trial and error. It is recommended that you track or stabilize using the default settings. If the tracker box strays from its original point, you can fine-tune the analysis.

Accessing the Stabilizer

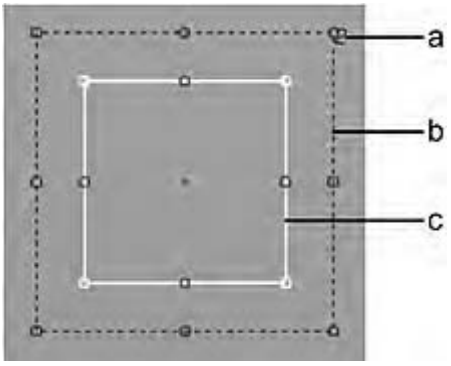
You access the Stabilizer differently, depending on how you want to track or stabilize a clip. For example, when you access the Stabilizer from the Tools tab, you stabilize with one tracker. When you access from Action, you have the option to use two trackers. You need two trackers when the clip you want to stabilize has a camera roll or zoom—the second tracker enables you to track the rotation and zoom of the camera.

Access the Stabilizer from:	**To:**
The Tools tab	Stabilize.
Action/ Timeline Effects Axis	Track or stabilize.
Keyer/ GMask	Track a garbage mask or the vertices of a GMask.
Distort	Track vertices or the axis of a spline when warping or morphing.
Warper	Track points or an axis of a mesh when warping or morphing.
3D Tracker	Provide a 2D tracking path for 3D manual tracking.
Paint	Track an AutoPaint stroke.
Blur, Glow	Track the center point of the radial blur.

Access the Stabilizer from:	To:
2D Transform	Provide correction for 2D motion, rotation and scaling.

How the Stabilizer Works

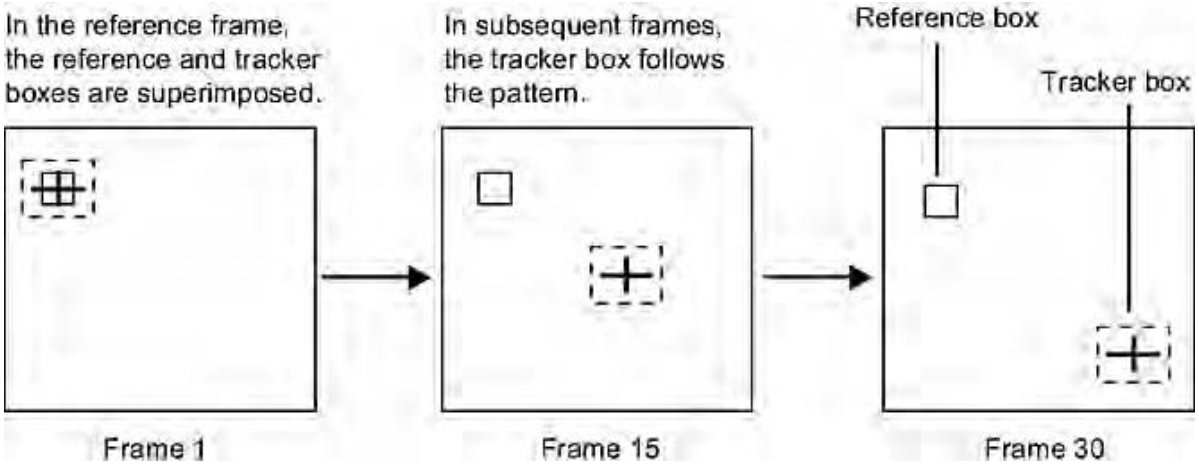
The Stabilizer uses trackers to generate tracking data. Each tracker consists of a solid box, called the *reference box*, and a dashed box, called the *tracker box*. The reference box establishes the reference point (the feature to track or stabilize) in any frame of the sequence. The tracker box indicates to the Stabilizer where to locate the reference point. The tracker box follows the frame-to-frame movement of the reference point.



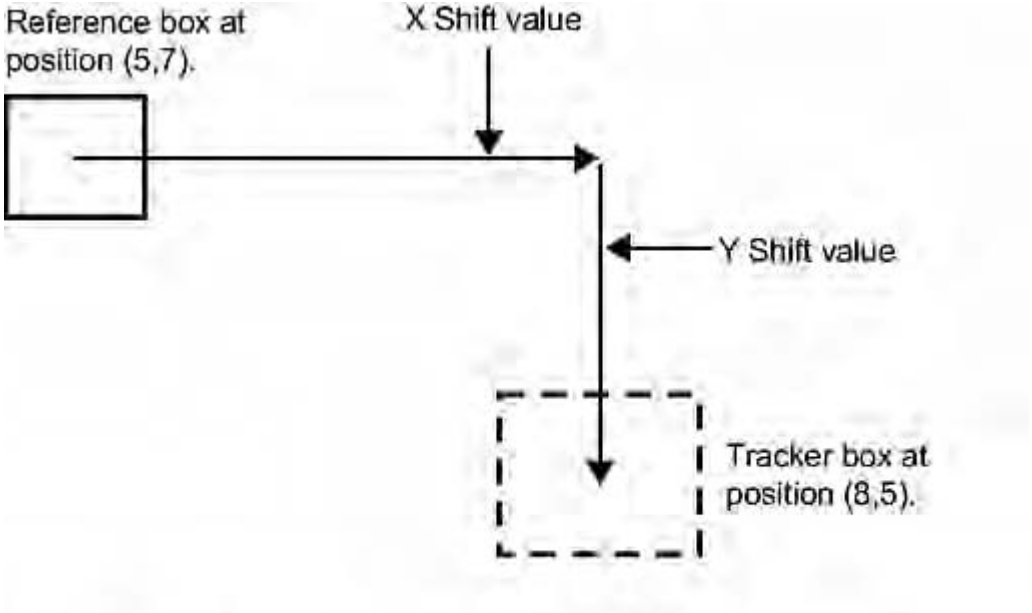
(a) Tracker number (b) Tracker box (c) Reference box

You start by selecting one or more reference points on your clip. Locate the first frame containing the movement to be tracked (the reference frame). In general, the reference frame is the first frame of the sequence. The choice of the reference point depends on whether you are tracking or stabilizing. When tracking, the reference point is a feature you want to track; when stabilizing, the reference point represents the point around which the image is stabilized. See Selecting a Reference Point (page 862) for details. Place the reference box(es) around the selected feature(s).

Once you have set the tracker positions, start the tracking process, also referred to as analyzing the clip. During the analysis, the tracker box associated with each tracker moves as the Stabilizer looks for a pattern that matches the reference in each frame of the clip.



The Stabilizer calculates the difference between the position of the tracker box and the position of the reference box to produce X and Y Shift values. Shift values represent a measurement in pixels and subpixels of how much the reference point has moved.

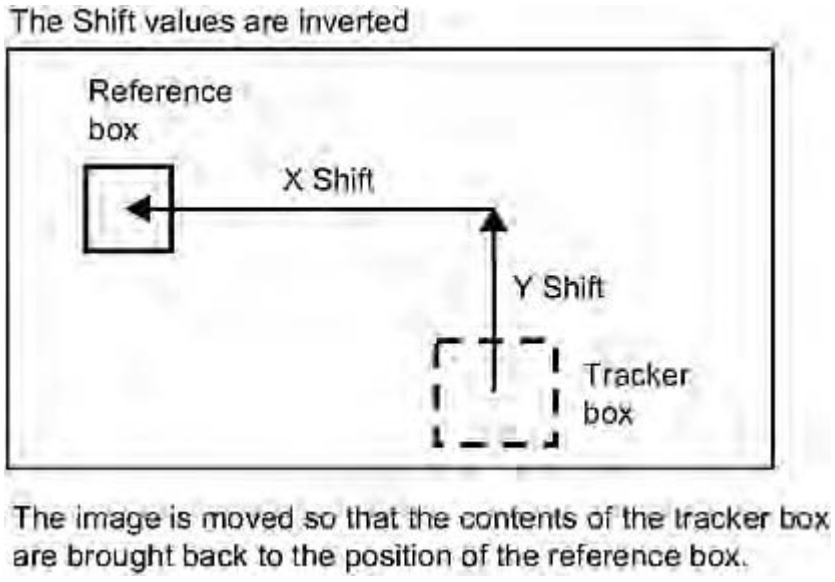


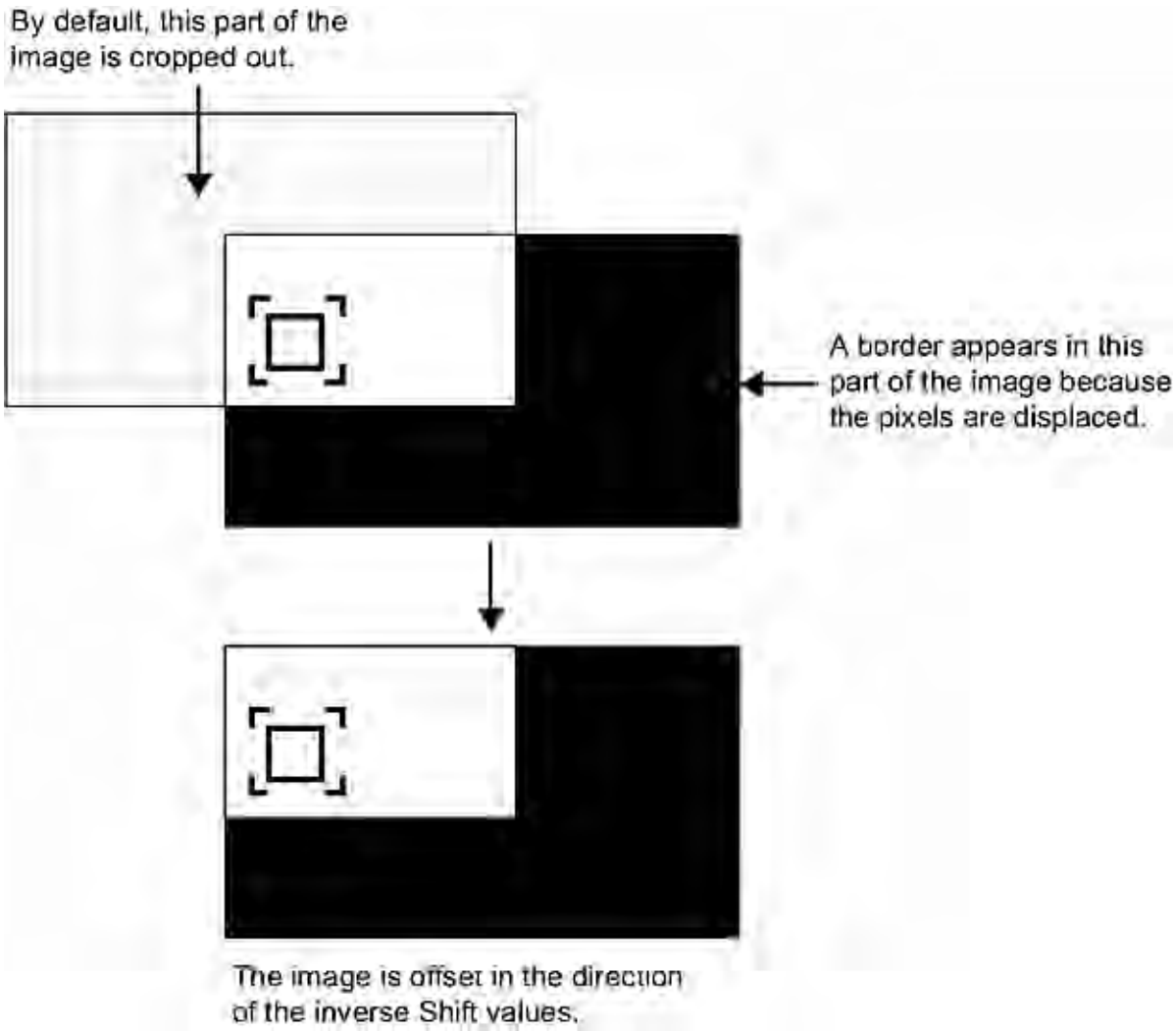
Reference position in X –	8
Tracker position in X	5
Shift value	3

Reference position in Y –	5
Tracker position in Y	7-
YShift value	2

When the analysis is complete, you fine-tune it if a tracker box has strayed from the reference it was supposed to follow. Once you are satisfied with the results, you can apply the data to the clip.

To track, the Stabilizer applies the Shift values "as is." To stabilize, the Stabilizer inverts the X and Y Shift values in each frame of the sequence, and moves the image according to these values. This gives the impression that the reference point stays in the same position throughout the sequence. Because the image is moved during stabilization, a border appears on one or more edges, which means that you lose some pixels. The following illustrations summarize the process.





Working with Trackers

This section provides information that is common to many procedures. It is recommended that you first read the procedure you want to perform in Selecting a Stabilizer Method (page 868) or Tracking (page 878), and then consult this section when needed.

Selecting a Reference Point

A good reference point is a high-contrast pattern that has good definition both vertically and horizontally, which allows for perfect registration in both directions.

Selecting a good reference point is a process of trial and error. Play the clip several times to become familiar with the material. Ideally, you should try to find a pattern that is present in every frame. In some cases, this is not always possible and you may have to track two different patterns, track an object that disappears behind another one, or track an object that moves out of the frame. For more information, see Tracking Difficult Shots and Correcting Errors (page 881).

The frame you use for the reference point should be the frame where the reference image is most representative in terms of shape, size, and rotation. For this reason, you can select the reference point in the middle frame if the pattern you want to use as a reference undergoes severe rotation or scaling. In most other cases, you will position the reference box over the reference point in the first frame of the sequence.

Selecting a Tracker

You can use any of the following methods to select a tracker:

- Select any tracker by selecting Move or Select in the Edit Mode box and click the tracker box, the tracking path, or the reference box of the tracker in the image window.
- Select a tracker by clicking its corresponding button (Tracker1, Tracker2, and so on).
- Select a tracker by pressing the corresponding number key on your keyboard. You must use the number keys on the alphanumeric keyboard, not the numeric keypad.
- Select any tracker by pressing the `up` and `down arrow` keys on your keyboard.
- Select any tracker by selecting its channel in the Channel Editor.

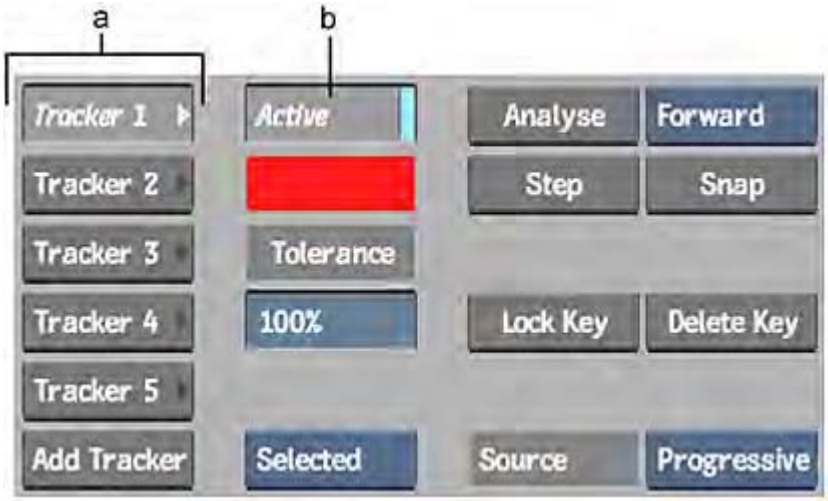
When you select a reference box, a tracker box, or a keyframe in the image window, the timeline automatically goes to the frame where the keyframe was set.

Positioning the Reference and Tracker Boxes

When you position the reference box, the tracker box automatically follows. You can leave the boxes together if you place the reference box in the first or last frame of the clip. However, if you start the analysis on a different frame from where you position the reference box, you should position the tracker box in the first frame of the clip on the pattern you want to track. This way, the Stabilizer knows where to look for the pattern to track when you analyse the clip.

To position the reference and tracker boxes:

1. Go to the frame where you want to position the reference and tracker boxes.
2. If the tracker and reference boxes do not appear on the image, click the appropriate Tracker button and enable Active.



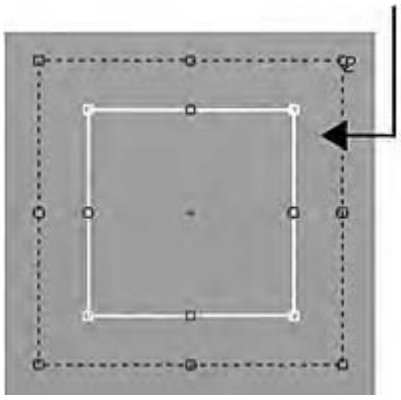
(a) Tracker buttons (b) Active button

NOTE Each tracker is automatically assigned a different colour. However, you can customize the tracker colour at any time.

3 Click inside the reference box to select it, and drag it over the pattern you want to track.

The reference box changes into a magnifying glass. By default, the tracker box follows the reference box.

4 To position the tracker box on a different frame from the reference box, go to that frame and move the tracker box over the pattern to track. To move only the tracker box, click anywhere outside the reference box.



Moving the Reference

When you select the reference box in the image window, the timeline automatically goes to the frame where you set the reference. For example, if you move to frame 1 and select the reference box, if the reference is set in frame 10, frame 10 appears automatically in the image window.

To move the reference, you can:

- Reset the reference box. See Resetting the Reference and Tracker Boxes (page 864).
- Select Add in the Edit Mode box to add a new reference.

Resetting the Reference and Tracker Boxes

By default, the tracker box moves with the reference box. If you move the tracker box by clicking in the area outside the reference box, it no longer follows the reference box. The new position is recorded in the Track X and Track Y channels in the Channel Editor and has precedence over the Shift value when the tracker box appears. If you want the tracker box to follow the reference box, you can delete the new keyframe. You can also use the Reset box to reset the reference and tracker boxes to their default position and size.

To reset the tracker box:

1 From the Edit Mode box, select Delete.

2 Click the tracker box.

The keyframe for the tracker box is deleted and the tracker box moves to the current position of the reference box.

3 From the Edit Mode box, select Move.

The tracker box now follows the reference box.

To use the Reset box:

1 Select either Reset Ref or Reset Track from the Reset box.

The reference or tracker box is reset to its default position and size.

NOTE You can also select the tracker box, enable Active for the tracker, and click Delete in the Animation menu to reset the tracker box.

Resizing the Reference and Tracker Boxes

Resize the reference box to make it surround the feature you chose to track more closely. By only including the recognizable feature in the box with no other details that may change throughout the clip, you minimize the possibility of the tracker losing the reference point.

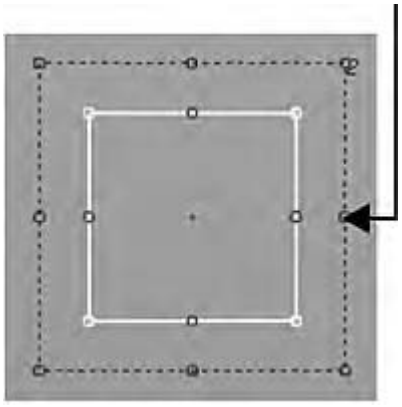
TIP Sometimes a small reference point does not give good results. Try enlarging the box to include more of the selected feature. Note that the larger the box, the slower the analysing speed.

Resize the tracker box so that it is large enough to accommodate the most frame-to-frame movement of the reference point. For example, if the movement of the reference point is mostly horizontal, you can increase the width and reduce the height of the tracker box. If there is a large amount of movement both horizontally and vertically, increase both the width and the height of the tracker box. Note that the smaller the tracker box, the faster the analysis.

To resize the reference and tracker boxes:

1 Do one of the following:

- To resize the reference box and tracker box interactively on the frame, press a resize handle on the box and drag.



- To use the menu to move or resize the reference or tracker box for the current tracker, enter values in the Reference or Track fields.



TIP Use the image window Zoom command to get a closer view of the tracker and reference boxes.

NOTE To reset the reference or tracker box to its default position and size, select either Reset Ref or Reset Track from the Reset box.

Working with Multiple Trackers

You can change a parameter for all active trackers at once. For example, you can change the dimensions of the tracker box for all trackers or set Fixed to off for all trackers. It can be useful to hide all but a selected tracker if you want to edit its tracking path.



(a) Tracker Selection box

Choose:	To affect:
Selected	Only the selected tracker and show all trackers in the image window.
Solo	Only the selected tracker and hide all other trackers.
Gang	All the active trackers, except when changing the colour of the trackers.

Locating a Tracker or a Keyframe

To find the number of a tracker, click any part of its tracking path in the image window. To find the frame number associated with a particular keyframe, click the keyframe. The timeline moves to the corresponding frame.



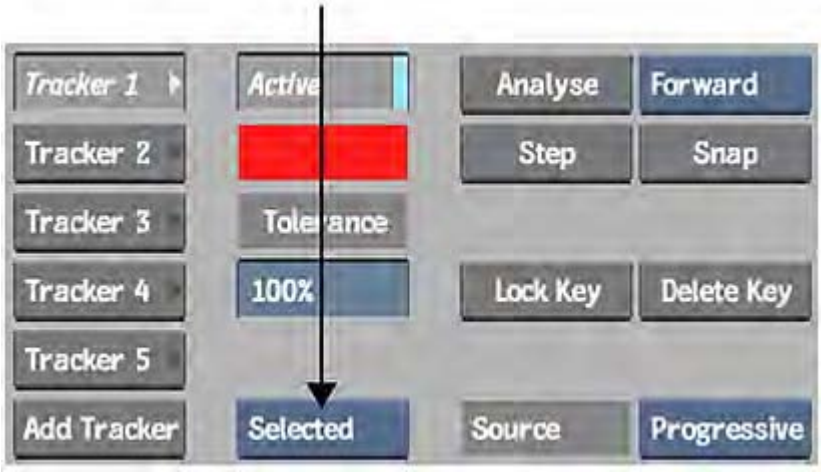
(a) When you select a tracking path, the tracker number appears here.

Locating a Shift Channel

Use the Expand/Collapse button to locate a Shift channel quickly.

To locate a Shift channel:

1. Select Gang in the Tracker Selection box.



   You can now change a parameter for all active trackers at once.
2. Click on the tracking path in the image window.
3. Toggle the Expand/Collapse button to Expand.



4 Click Animation to display the Channel Editor.

5 Scroll up or down to display the Shift folder.

The Shift curve of the tracker appears in the Channel Editor.

Speeding Up the Analysis

To speed up the analysis, disable the Icons and Path buttons in the Setup menu before starting. This turns off the display of the tracker and reference boxes, as well as the tracking path. Re-enable the buttons to fine-tune the analysis.

Selecting a Stabilizer Method

Use the stabilizer to remove unwanted shaky motion and to smooth the camera motion. It can separately smooth motion with respect to translation, rotation and scaling in an image sequence to produce a stabilized image.

Stabilizing a clip is divided into two parts. First, the movie is scanned frame by frame, while the motion within the movie is tracked, and the related tracked motion data is recorded. With trackers installed, you can play the stabilized clip.

The first step to stabilizing a clip is to select an appropriate stabilization method.

Stabilization Methods

Auto Stabilize

- No way to add manual trackers.
- Requires the least work to stabilize a clip.
- Can stabilize simple 2D motion, scaling and rotation, and perspective transformation.
- Tries to do best effort.
- Can smooth or pin the detected motion.
- Can remove jitter but keeps camera motion.
- Can adapt resolution and advanced padding options.

Where possible, start with this stabilization method.

Stabilizer

- Requires you to add manual trackers.
- Enable simple padding and texture repeat options.
- Can only stabilize simple 2D motion.

2D Transform

- Can stabilize simple 2D motion, scaling and rotation.
- Can adapt resolution and advanced padding options.

NOTE To access any of the above stabilizers:

1. Select the Transform tab in the Tools tab.
2. Enable the preferred stabilizer (Auto Stabilize, Stabilizer, or 2D Transform).
3. Pick the clip, then click on an empty part of the Thumbnail view.
   You are in the selected stabilizer editor.

Workflow

Step1: Consider your goal and select a method of stabilizing the clip.

Goal: To Stabilize Simple 2D motion (click here). (page 869)

- Auto Stablize
- Stablizer
- 2D Transform

Goal: To Stabilize Motion or Scaling or Rotation (click here). (page 871)

- Auto Stabilize
- 2D Transform

Goal: To Stabilize a Perspective Transformation (click here). (page 873)

- Auto Stabilize

Step 2: Determine the quality of the result of the stabilization method. Change stabilization method if necessary.

Depending on the goal of the stabilization, not all methods may be available. For example, if doing perspective transformation, Auto Stabilize is the only option available.

1. If Auto Stabilize does not provide adequate results, then use Stabilizer.
2. If Stabilizer does not provide adequate results, then use 2D Transform.

Simple Stabilization

Use simple stabilization to stabilize a clip where there is no pan or tilt.

To stabilize when there is no pan or tilt:

1. Select **Tools ➤ Composite ➤ Stabilizer**.
   The cursor changes to Pick Front.
2. Select the front clip.
   The cursor changes to Render Here.
3. Click on any free (or a grey) area on the workspace.
   You are in Stabilizer.
4. Switch to Front view.
5. Position Tracker1 over a pattern that you want to stabilize.
6. Click Analyze to generate the stabilization data.
7. Fine-tune the data if necessary. See Tracking Difficult Shots and Correcting Errors (page 881).
8. Switch to Result view to see the stabilized shot.
9. Adjust the framing options as you need (Roll, Fill, Crop Edges, Letterbox, Shift).
10. Once you are satisfied with the stabilization, go to frame 1, make sure Tracker 1 is still active, and click Render.

To stabilize using Auto Stabilize:

1. Select **Tools ➤ Composite ➤ Auto Stabilize**.
   The cursor changes to Pick Front.
2. Select the front clip.
   The cursor changes to Render Here.
3. Click on any free (or a grey) area on the workspace.
   You are in Auto Stabilize.
4. Select 2D Stabilization from the Stabilization Method box.
   You are performing a two-dimensional analysis.
5. Switch to Front View.
6. **Optional:** Enable the Region Of Interest button.
   Indicate the region you want to analyze on the front clip.
7. Click the Analyze button.
   Analysis takes place. This may take some time however you will see the percentage of progress in the analysis in the Progress field.
8. Switch to Result view to see the stabilized shot.
9. Enable the Components (Position X, Position Y) that you want to stabilize.
10. For these components select either Fixed or Smooth.
    - Fixed pins the component during the analysis.
    - Smooth removes the jittering with 0% being the original motion and 100% removing the highest frequency jitter.

    The analysis is complete.

NOTE The default reference frame is the first frame of the clip. You can change the reference frame of the analysis by defining a new reference point.

To define a new reference point:

1. Position the positioner to the desired frame.
2. Click the Set Frame button.

   The analysis will readapt the timing of the transformation; stabilization will occur before and after the set frame.

To stabilize using 2D Transform:

1. Select **Tools ➤ Composite ➤ 2D Transform**.

   The cursor changes to Pick Front.
2. Select the front clip.

   The cursor changes to Render Here.
3. Click on any free (or a grey) area on the workspace.
4. Select Pan & Scan from the Transform Type box.

   You are now limited to modifying only the position and scale.
5. In the Stabilization tab, select the Enter Stabilizer button.

   You are in Stabilizer.
6. Position Tracker 1 over a pattern that you want to stabilize.
7. Click Analyze to generate the stabilization data.
8. Fine-tune the data if necessary. See Tracking Difficult Shots and Correcting Errors (page 881).
9. Once you are satisfied with the stabilization, exit back to 2D Transform.
10. Enable the Components (Position X, Position Y) that you want to stabilize.
11. Adjust the framing options as you need (Roll, Fill, Crop Edges, Letterbox, Shift) .

NOTE The default reference frame is the first frame of the clip. You can change the reference frame of the analysis by defining a new reference point.

To define a new reference point:

1. Position the positioner to the desired frame.
2. Click the Set Frame button.

   The analysis will readapt the timing of the transformation; stabilization will occur before and after the set frame.

Stabilizing Motion, Scaling, and Rotation

To stabilize Motion, Scaling and Rotation using Auto Stabilize:

1. Select **Tools ➤ Composite ➤ Auto Stabilize**.

   The cursor changes to Pick Front.
2. Select the front clip.

   The cursor changes to Render Here.
3. Click on any free (or a grey) area on the workspace.

You are in Auto Stabilize.

4 Select 2D Stabilization from the Stabilization Method box.

You are performing a two-dimensional analysis.

5 Switch to Front View.

6 **Optional:** Enable the Region Of Interest button.

Indicate the region you want to analyze on the front clip.

7 Click the Analyze button.

Analysis takes place. This may take some time however you will see the percentage of progress in the analysis in the Progress field.

8 Switch to Result view to see the stabilized shot.

9 Enable the Components (Position X, Position Y, Scaling, Rotation) that you want to stabilize.

10 For these components select either Fixed or Smooth.

- Fixed pins the component during the analysis.
- Smooth removes the jittering with 0% being the original motion and 100% removing the highest frequency jitter.

The analysis is complete.

NOTE The default reference frame is the first frame of the clip. You can change the reference frame of the analysis by defining a new reference point.

To define a new reference point:

1 Position the positioner to the desired frame.

2 Click the Set Frame button.

The analysis will readapt the timing of the transformation; stabilization will occur before and after the set frame.

To stabilize Motion, Scaling and Rotation using 2D Transform:

1 Select **Tools ➤ Composite ➤ 2D Transform**.

The cursor changes to Pick Front.

2 Select the front clip.

The cursor changes to Render Here.

3 Click on any free (or a grey) area on the workspace.

4 Select Pan & Scan from the Transform Type box.

5 In the Stabilization tab, select the Enter Stabilizer button.

You are in Stabilizer.

6 Position Tracker 1 over a pattern that you want to stabilize.

7 For stabilizing scaling and rotation you must position Tracker 2 over a pattern.

8 Click Analyze to generate the stabilization data.

9 Fine-tune the data if necessary. See Tracking Difficult Shots and Correcting Errors (page 881).

10 Once you are satisfied with the stabilization, exit back to 2D Transform.

11 Enable the Components (Position X, Position Y, Scaling, Rotation)that you want to stabilize.

12 Adjust the framing options as you need (Roll, Fill, Crop Edges, Letterbox, Shift).

NOTE The default reference frame is the first frame of the clip. You can change the reference frame of the analysis by defining a new reference point.

To define a new reference point:

1. Position the positioner to the desired frame.
2. Click the Set Frame button.

   The analysis will readapt the timing of the transformation; stabilization will occur before and after the set frame.

Stabilizing a Perspective Transformation

To stabilize Motion, Scaling and Rotation using Auto Stabilize:

1. Select **Tools ➤ Composite ➤ Auto Stabilize**.

   The cursor changes to Pick Front.
2. Select the front clip.

   The cursor changes to Render Here.
3. Click on any free (or a grey) area on the workspace.

   You are in Auto Stabilize.
4. Select Perspective from the Stabilization Method box.

   The Perspective button is enabled.
5. Switch to Front View.
6. **Optional:** Enable the Region Of Interest button.

   Indicate the region you want to analyze on the front clip.
7. Click the Analyze button.

   Analysis takes place. This may take some time however you will see the percentage of progress in the analysis in the Progress field.
8. Switch to Result view to see the stabilized shot.
9. For the Perspective component select either Fixed or Smooth.
   - Fixed pins the component during the analysis.
   - Smooth removes the jittering with 0% being the original motion and 100% removing the highest frequency jitter.

   The analysis is complete.

NOTE The default reference frame is the first frame of the clip. You can change the reference frame of the analysis by defining a new reference point.

To define a new reference point:

1. Position the positioner to the desired frame.
2. Click the Set Frame button.

   The analysis will readapt the timing of the transformation; stabilization will occur before and after the set frame.

Manually Stabilizing 2D Motion

To manually stabilize 2D motion:

1. Select **Tools ➤ Composite ➤ Stabilizer**
   The cursor will change to Pick Front.
2. Select the front clip.
   Your source clip is rendered (if needed).
   The cursor will change to Render Here.
3. Click on any free (or a grey) area on the workspace.
   You are now in the stabilizer.

Stabilizing a Clip from Action

You can import stabilizing data to lock the position of an axis in relation to a reference point on the front clip. This means that any movement in the front clip is matched by the axis. You can enter the Stabilizer from Action to stabilize an image so that the axis changes to keep the reference point fixed against the background.

To stabilize a clip from Action:

1. Load the clips into Action.
   When selecting clips, keep in mind that the front clip is the image that you want to stabilize and the back clip contains the reference point you want to track. Also, the clips must be of the same resolution.
2. In the Axis menu for the selected media, make sure that the motion path is disabled (Path button).



(a) Stabilizer button (b) Stabilizer Option box (c) Tracking Rotation Option box (d) Tracking Scale Option box (e) Motion Path button

NOTE Do not move the axis from its default position at the centre of the image window before entering the Stabilizer, or else the tracking data will be overwritten. Use the offset axis to add an offset to the tracker movement instead.

3. In the Stabilizer Option box, select Stabilize and click the Stabilizer button.
   The front clip is automatically loaded into the Stabilizer and the Stabilizer menu appears.
4. In the Stabilizer, position Tracker1 over the reference point that you want to track.

5 To help you view your tracking result without any objects blocking the view, enable the Context button. While similar to a result view, the context view allows you to see all of the Action scene, except for the selected node (and any children of the selected node).



6 Depending on the performance of your system and the complexity of your Action setup, you can enable Linetest in the setup menu to display the context view at a lower resolution.



When you play the clip in Action context, you can see the front image follow the inverse movement of the reference point in the front clip.

7 Click Analyse to generate the translation data.

8 Once the analysis is complete, make sure that Tracker1 is still active, and click Return.

When you exit the Stabilizer, the X and Y Shift values for the reference position are automatically copied into the X and Y translation channels for the selected axis in Action.

Removing Jitter While Keeping Overall Motion

Use the Jitter option to remove the jitter from a clip while retaining the overall motion. The Stabilizer calculates the difference between an average applied to the channel by using the current Over value and the original tracking data. Applying this curve leaves only the jitter values as keyframes. Copying these curves can also be useful if you want to extract the jitter values to apply to another clip.

The Jitter option removes jitter in a clip on both the X and Y axes simultaneously, and averages the pan over time so that it appears even. Jitter control offers more flexibility than Fixed X and Fixed Y, which remove motion in one direction only. Use Fixed X and Fixed Y in simple situations, or to produce a result quickly.

Removing jitter is a process of trial and error. Try different Over values until you find one that yields good results. As a general rule, start with a large Over value over *n* frames to remove slow jitter, and start with a small Over value to remove fast jitter.

To remove jitter and keep overall motion:

1 Select **Tools ➤ Composite ➤ Stabilizer**.

The cursor changes to Pick Front.

2 Select the front clip.

The cursor changes to Render Here.

3 Click on any free (or a grey) area on the workspace.

You are now in Stabilizer.

4 Position Tracker1 over a pattern you want to stabilize.

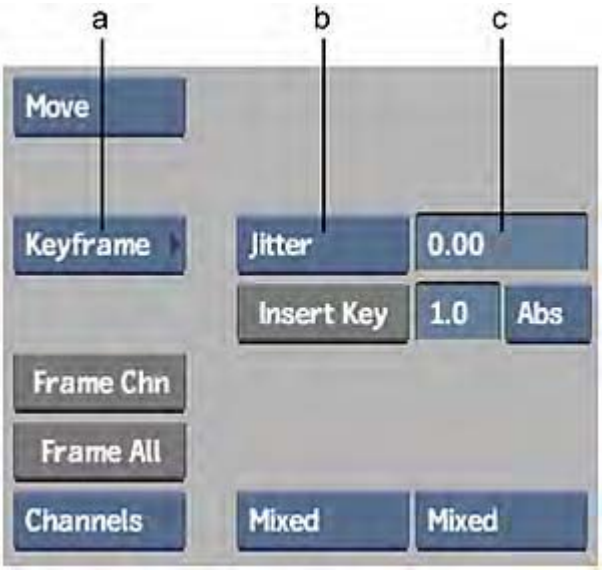
5 To automatically remove the border at the edges of the clip, select Crop Edges from the Scaling and Shifting box.

6 Go to the reference frame, and click Analyze to generate the Shift data.

7 Click Animation to display the Channel Editor. Select the tracker number and expand the Shift folder.

The Shift curves turn white when the Shift channel is selected.

8 Select Keyframe from the Animation Controls box, and then enter an Over value in the Curve Value field.



(a) Animation Controls box (b) Curve option box (c) Curve Value field

9 Select Jitter from the Curve option box. If Jitter was already displayed when you entered the Over value, select it again to apply the Over value.

The transformation is applied to the curves. The curves show the amount of motion that the Stabilizer will remove. They should wrap around the zero point and should not be entirely flat; otherwise, the Stabilizer will not remove any jitter.

10 Click Result and play the clip to determine if it is stable enough. If the clip is stable enough, click Exit. To revert to the original shift data, click Paste.



11 Repeat from step 7 with different Over values until you are satisfied with the results.

Removing Jitter with Fixed X and Fixed Y

The Fixed X and Fixed Y buttons remove motion in one direction only. Enable the Fixed X button to remove vertical jitter only. Movement on the horizontal (X) axis, such as a camera pan, is not affected. Enable the Fixed Y button to remove horizontal jitter. Movement on the vertical (Y) axis is not affected.



Smoothing Out Camera Motion

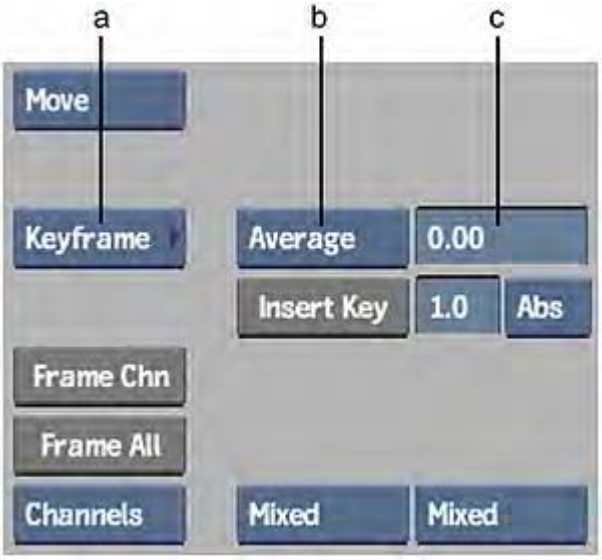
Use the Average options to smooth out uneven camera motion. For example, if the camera moves at a certain rate and suddenly drops or increases in speed, the Average option can stabilize the camera motion.

The Average option smooths camera motion over a group of keyframes. Use the Curve Value field to determine how much stabilizing is applied to the clip. A larger value averages camera motion over more keyframes and increases camera smoothness. Use Average to affect the Stabilizer's Shift values since Shift represents the amount of movement in a clip.

Analyse your clip to generate the initial channel data, then select the channel to apply the average to.

To smooth out camera motion:

1 Position the tracker(s) on your image and click Analyse.
2 Click Animation to open the Channel Editor and expand the Shift folder for the track(s).
3 Select the X or Y Shift folder and enter an average value in the Over field.
4 Select Keyframe from the Animation Controls box, and then enter an average value in the Curve Value field.



(a) Animation Controls box (b) Curve option box (c) Curve Value field

5 Select Average from the Curve option box.

Tracking

You can use the Stabilizer to make a clip or part of a clip track an object in another clip. For example, you can paste a logo to the side of a moving car. You can track a clip using one, two, or four reference points:

- In one-point tracking, one reference point is tracked to produce 2D motion without rotation or scaling.
- In two-point tracking, two reference points are tracked to generate translation, scaling, and rotation data that is applied to the foreground composite.
- In four-point tracking, the tracking data generated from four reference points is used to lock the four corners of a bilinear surface to the background in Action.

Since tracking involves compositing at least two clips, you open the Stabilizer from Action. When loading the clips in Action, you load the clip that contains the pattern you want to track as the back clip. The method for selecting the object that follows the pattern on the back clip is different for four-point tracking than it is for one and two-point tracking. Both methods are explained in the following sections.

One-Point and Two-Point Tracking

In both one-point and two-point tracking, you select the object that follows the pattern on the back clip by assigning the tracking data to its axis in Action.

For one-point tracking, you use only one tracker (usually Tracker1) to generate position information. For two-point tracking, you use a second tracker (usually Tracker2) to generate rotation and/or scaling information. The Stabilizer obtains this information by comparing the position of Tracker2 to that of Tracker1.

Before you select the two reference points on the back clip, note the task of each tracker:

- Tracker1 follows the horizontal and vertical translation of the reference point. You should position Tracker1 over a point on the pattern that you want to track.
- Tracker2 tracks the rotation and/or the change in size of the pattern. In the first frame, the rotation is always 0 and the scaling factor is always 100%. In subsequent frames, a rotation and/or scaling factor is

added if the relative position of the two trackers changes. You should position Tracker2 over a point that represents the rotation or change in size of the pattern.

When you open the Stabilizer from Action for two-point tracking, you can select whether you want rotation information, scaling information, or both. The choice depends on the movement of the object to track and on the camera movement in the clip.

To perform one-point or two-point tracking in Action:

1. Load a front and back clip in Action.

   The front clip contains the object and the back clip supplies the pattern that you want to track.
2. In the Axis menu for the selected media, make sure that the motion path is disabled (Path button).



(a) Stabilizer button (b) Stabilizer Option box (c) Tracking Rotation Option box (d) Tracking Scale Option box (e) Motion Path button

NOTE Do not move the axis from its default position at the centre of the image window before entering the Stabilizer, or else the tracking data will be overwritten. Use the offset axis to add an offset to the tracker movement instead.

3. In the Stabilizer Option box, select Track.
4. From the Tracking Rotation option box and Tracking Scale option box, set the tracking options.

Select:	To:
Rotation Off and Scaling Off	Do one-point tracking.
Rotation On and/or Scaling On	Do two-point tracking.
Rotation Inv	Invert the rotation data.
Scale Inv	Invert the scaling data.

5. Click the Stabilizer button.

   The Stabilizer opens and the back clip appears.

   NOTE If the wrong clip appears in the image window, return to Action and change the back clip to the clip that contains the pattern you want to track.

6 Position the tracker(s) over the pattern(s) that you want to track, and click Analyse to generate the tracking data.

NOTE You can fine-tune the tracking data if necessary. See Tracking Difficult Shots and Correcting Errors (page 881).

7 Once you are satisfied with the tracking, make sure the trackers you used are still active, and then click Return.

The Axis menu in Action reappears. The tracking data is applied to the front clip.

Four-Point Tracking

With four-point tracking, you use four trackers in the Stabilizer to generate tracking data for anchoring the four corners of a bilinear surface to the background clip. The bilinear surface then tracks the horizontal and vertical translation of the reference points on the background clip.

Because four-point tracking applies to surfaces, you select the object that follows the pattern on the back clip by assigning the tracking data to its bilinear surface in Action.

The markers you use must be well defined and should be planned when shooting the sequence. It is not always possible to do four-point tracking when the markers are not well defined.

To perform four-point tracking:

1 Load front and back clips in Action.

The front clip supplies the surface and the back clip supplies the four anchor points that you want to track.

2 Double-click the image node in the schematic to display the Surface menu.

3 Select Bilinear from the Shape box.

4 Click the Vertices tab to display the Stabilizer button and tracking options.

5 In the Stabilizer Option box, select Track

6 Click the Stabilizer button.

The Stabilizer opens and the back clip appears.

NOTE If the wrong clip appears in the image window, return to Action and change the back clip to the clip that contains the pattern you want to track.

7 Position the four trackers on the background clip, and click Analyse to generate the tracking data.

NOTE The trackers should be positioned in order; otherwise, the result will be inverted or displaced. Place Tracker1 in the upper left area of the image to be tracked, Tracker2 in the upper right, Tracker3 in the lower left and Tracker4 in the lower right.

8 After the analysis is complete, make sure the trackers are still active, then click Return.

The Surface menu in Action reappears. The tracking data is automatically applied to the four corners of the bilinear surface.

Using Offsets

Instead of anchoring the entire front image to the back clip, you may only want to use a small region of the image. You can move the four corner anchor points anywhere inside or outside the front image to define the area of the front image that you want to use, and animate the resulting offsets.

If the anchor point offsets need to change from frame to frame to track a pattern on a bilinear surface, you can use the Stabilizer to generate the offsets automatically. Note that the image can be distorted when you use this method.

To generate the offsets using the Stabilizer:

1. Track the four anchor points on the back clip.
2. Display the Action Surface menu for the bilinear surface that you tracked.
3. In the Vertices tab, enable Edit Offsets and use the Offsets controls to edit the offsets to match the corners of your square.
4. Click the Stabilizer button.

   The front clip appears in the Stabilizer.
5. Position the four trackers on the front clip in the same order as the four anchor points.
6. Click Analyse to generate the tracking data. After the analysis is complete, make sure the trackers are still active, then click Return.

   The Surface menu in Action appears. The tracking data is automatically applied to the front clip.

You can also animate the offsets. The animation channels for the anchor point offsets are in the Offsets channel of the bilinear surface in Action Animation menu. To maintain resolution independence, each X and Y value is expressed as a percentage relative to its corresponding corner. Each anchor point offset starts with a default value of zero. For example, if you change the upper-left X offset value to 20, then the upper-left anchor point is moved inside the bilinear surface by 20%. Negative values offset the anchor points outside the bilinear surface.

You can also use the Copy and Paste buttons in the Channel Editor to copy one anchor point offset setup to another.

Tracking Difficult Shots and Correcting Errors

If the tracker box strays from the reference point that it is supposed to be tracking, incorrect shift keyframes result. If such an error occurs, you can stop the analysis, correct it, and restart it at any frame.

This section covers the different strategies you can use to track difficult shots. After trying any of the strategies, you will need to redo the analysis. Click Analyse to generate new keyframes based on the updated information you provide.

Resetting the Shift Data

Although it is not necessary, you can erase old keyframes before redoing the analysis.

To reset the shift data:

1. Select Reset Shift from the Reset box.



The shift value reverts to 0.

Adjusting the Size of the Tracker Box

If the Stabilizer cannot find the reference point within the boundaries of the tracker box during analysis, the tracker box will stray from the reference point and produce incorrect keyframes. Although you can manually correct these keyframes, it is easier to make the tracker box large enough to accommodate the movement of the reference point. Note that processing time increases as the size of the tracker box increases. See Resizing the Reference and Tracker Boxes (page 865).

To adjust the size of the tracker box:

1. Press on the pen or hold down the left mouse button to stop the analysis.

   NOTE Make sure to press below the timeline to avoid moving the tracker box.

2. Go back to the last good frame before the tracker box strayed from the reference point.
3. Adjust the size of the tracker box so that it is large enough to accommodate the largest frame-to-frame movement of the reference point.
4. Click Snap, and then click Analyse.

Positioning the Tracker Box Manually

When the reference point is temporarily covered by another object, position the tracker box manually.

When you manually position the tracker box, the new position is recorded in the Track X and Track Y channels of the Channel Editor. These values have precedence over the Shift X and Shift Y values.

To manually position the tracker box:

1. Press on the pen or hold down the left mouse button to stop the analysis.

   NOTE Make sure to press below the timeline to avoid moving the tracker box.

2. Advance the clip to the frame where the reference point becomes visible again, and reposition the tracker box over the reference point. Click Analyse to restart the analysis at this frame.

   Since the X and Y shifts are recorded as keyframes in channels, the Stabilizer calculates the translation values for the frames in which the reference point was covered. The final result will be a smooth motion.

Locking Keyframes in Place

You can lock Shift keyframes so that they stay in place even if you try to move them manually or perform an analysis. This is useful, for example, when you set keyframes manually and want to prevent Flame Premium from overwriting them when you redo the analysis.

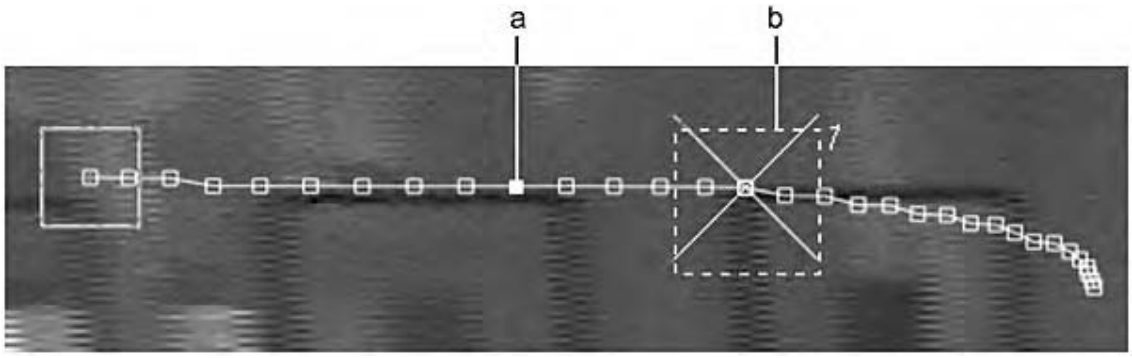
In the following procedure, you select the keyframes directly in the image window. However, you can also select the Shift keyframes in the Channel Editor.

NOTE You cannot lock several keyframes at once. You must lock each keyframe individually.

To lock keyframes in place:

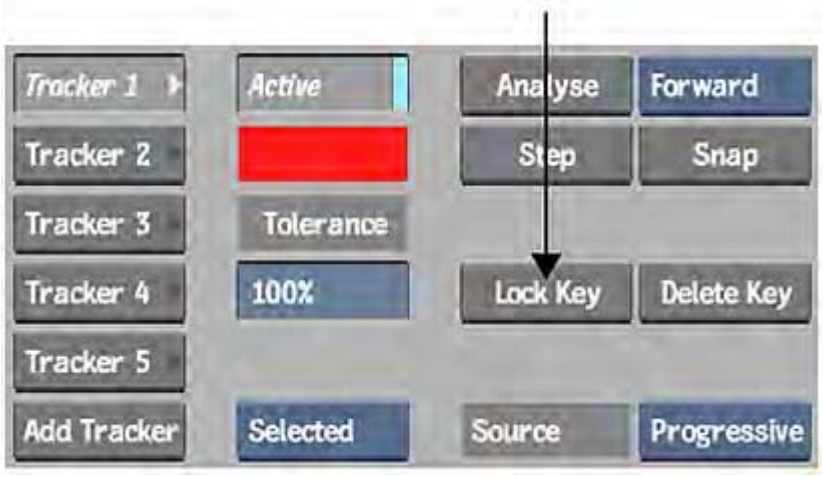
1. If the tracking path does not appear in the image window, enable Motion Path in the Setup menu.
2. Zoom in on the image until you can see the points clearly and select the point you want to lock on the tracking path.

   The tracking path becomes white when you click it, and the frame that corresponds to the keyframe appears in the image window.



(a) The inside of locked keyframes are filled (b) The tracker box appears when you select a keyframe

3. Click Lock Key.



4. To unlock a keyframe, select it and click Unlock Key.

Changing the Reference Point

Change the reference point to another feature if the feature you have been tracking moves out of the frame, or alters such that it is no longer trackable. You should do this before the original reference point moves out of the frame. This allows the Stabilizer to calculate the offset between the two reference points and predict the position of the original reference point. If the Path button in the Setup menu is enabled, you will see the extrapolated path of the original reference point.

Change the reference point before analysing the clip.

To change the reference point:

1. Click Setup and then enable Path.
2. Select the tracker for which you want to change the reference point.
3. Display the desired frame in the image window.
4. Select Add from the Edit Mode box.
5. `Ctrl+Shift`-drag the reference box and position it on the new reference point.
6. Go to the first frame of the sequence and click Analyse.

Finding the Best Tolerance Value

The Tolerance value determines how much discrimination the Stabilizer uses in matching the reference point from frame to frame and in setting a keyframe for that frame. At 100% tolerance (no discrimination), almost anything is considered as a match and a keyframe is set. At 0% tolerance (complete discrimination), only a perfect match is accepted.

If the reference point becomes hidden by another object, you can use the Tolerance value to make the tracker ignore the reference point in parts of the clip where it is hidden, then continue tracking it normally when it reappears. Choosing a good Tolerance value reduces the need to manually reposition the tracker box during analysis.

TIP You can also adjust the tolerance to make a desirable keyframe that has been filtered out reappear.

To find the best Tolerance value:

1. Click Setup and then enable Path to view the tracking path.
2. Analyse the clip with full (100%) tolerance.



3 Go to the first frame where the reference point is hidden by another object and adjust the tolerance value until the crosshair reappears in the tracker box.

   The crosshair reappears when there is a keyframe.

4 Go to the first frame in the sequence and click Analyse.

Once you determine the optimal tolerance value, further analysis automatically discards the undesirable keyframes, and the position of the reference point is extrapolated until the reference point reappears.

Analysing Backward

Analyse the clip backward when the pattern you want to track grows larger or when it is off screen at the beginning of the clip.

NOTE You cannot analyse backward if you need to change or snap the reference. In this case, reverse the clip without interpolation before entering the Stabilizer and analyse it forward.

To analyse backward:

1 Select Backward from the Direction box.

2 Go to the last frame of the sequence.

3 Position the reference and tracker boxes.

4 Click Analyse.

Tracking Manually

Track manually when the pattern you are tracking disappears behind an object for several frames, moves out of the frame, or is extremely difficult to track.

Tracking a Pattern that Disappears

Tracking a pattern that disappears is easier to do when the Fixed option is off. However, the results may not be as accurate because any small errors you make in positioning the tracker box are compounded when Fixed is off. Leave Fixed on when possible.

To track a pattern that disappears:

1. Go to the last keyframe before the reference point disappears.
2. Click Snap to redefine the reference point on this frame.



3. Select Add in the Edit Mode box.
4. `Ctrl+Shift`-drag the tracker box to the required position in the current frame.

   Use the reference image you see in transparency as a guide to find the new position. A keyframe is set and the timeline advances one frame forward or backward when you release the pen. The direction of the timeline depends on the setting in the Direction box.

   NOTE If you do not see the reference image in transparency, set the Opacity option in the Setup menu to approximately 50%.

5. Repeat step 4 as necessary.

   TIP Lock the keyframes as you add them.

Tracking an Erratic Pattern Manually

Using this method, you create part of the tracking path manually. You can either add a keyframe in each frame, or add keyframes in significant frames of the clip and extrapolate the curve for the other frames.

You must delete unwanted keyframes before you can add new ones.

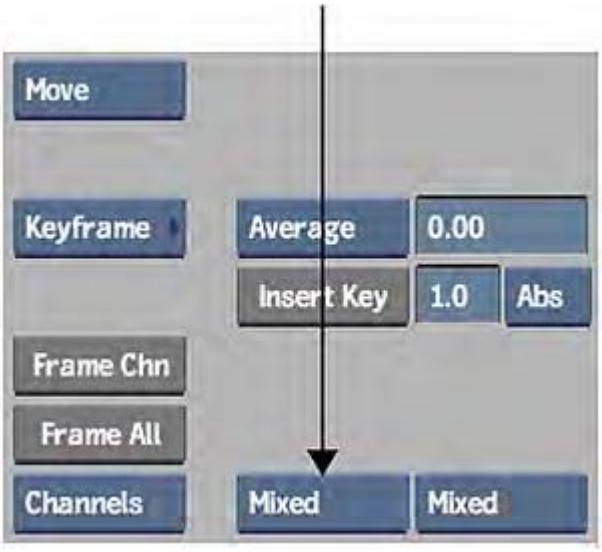
TIP To go to the desired frame, select Move in the Edit Mode box and click on the corresponding keyframe in the image window.

To track an erratic pattern manually:

1. Erase any bad keyframes by selecting Delete in the Edit Mode box and clicking the bad keyframes in the image window.
2. Go to the last good frame before the frame where you want to start tracking manually.
3. Click Snap to redefine the reference on this frame.

   NOTE If you are tracking a whole clip manually, position the reference box on the pattern to track.

4. Select Add from the Edit Mode box.
5. Select one of the following options from the Interpolation Mode box.



- To set critical keyframes only, select Bézier.
- To set keyframes on every frame, select Linear.

6 Add a keyframe by holding `Shift` and dragging the tracker box over the reference point.

You should see the reference image in transparency. The keyframe is added in the following frame when you release the mouse button or lift the pen.

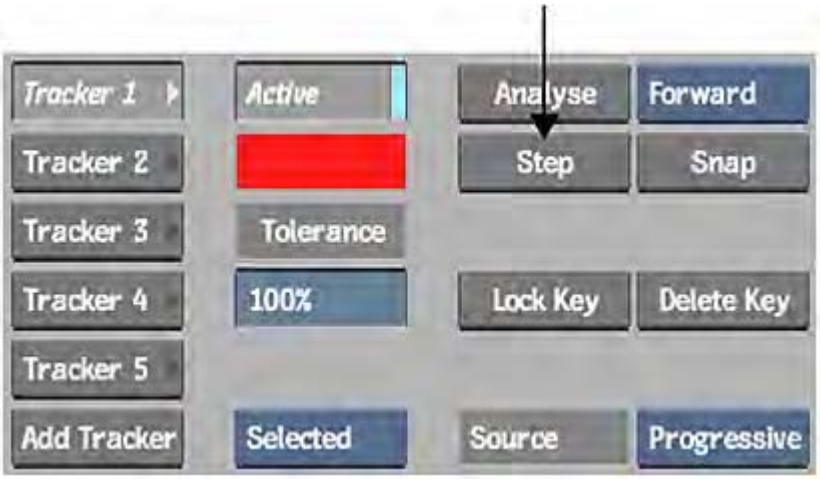
NOTE If you do not see the reference image in transparency, make sure you have defined a reference and check that the Opacity is set to approximately 50% in the Setup menu.

7 Repeat step 6 for every frame where you want to define a keyframe.

8 Play the clip and adjust the tracking path by moving or adding keyframes.

Analysing One Frame at a Time

Analysing one frame at a time is useful for difficult shots because you can adjust the position of the tracker after each frame. Click the Step button to analyse a single frame and advance to the next frame.



Editing the Stabilizer Channels on the Image

You can edit the tracker channels directly on the image instead of editing them in the Channel Editor. Note that the curve on the image shows the X and Y components together and not separately like in the Channel Editor. The tracking path on the image shows the actual displacement of the tracker box.

To edit the curve directly in the image window, Path must be enabled in the Setup menu. When you select a keyframe on the curve, the timeline goes to the frame that corresponds to the keyframe. When you select the reference box, the timeline goes to the frame where you set the reference.

When you edit the tracking path in the image window, all the options in the Edit Mode box are available except Rect Zoom, Zoom, and Pan.

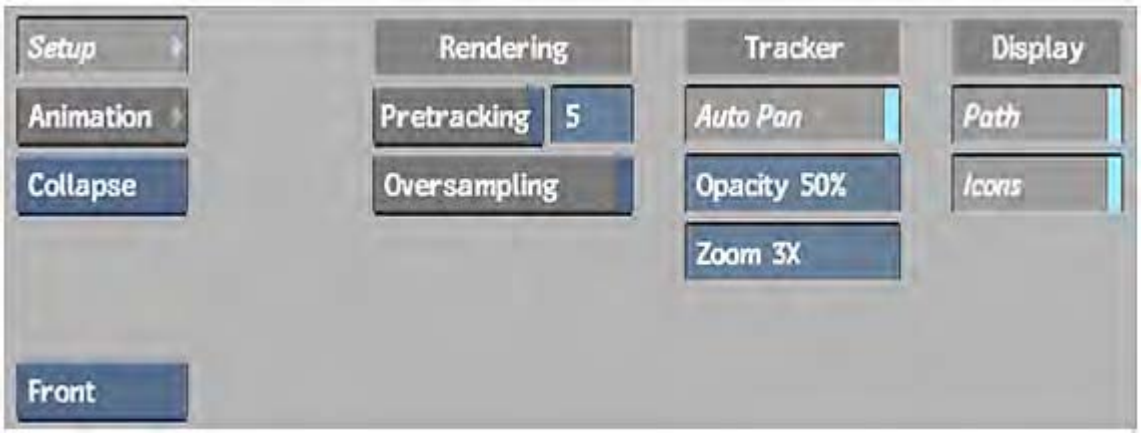
Tracking a Degraded Copy of the Clip

If other options do not work, you can try degrading a copy of the image until the contrast of the pattern to track is high enough. You then generate tracking data from this copy, save the setup, and apply it to the original clip. Try the following methods for degrading the clip:

- Saturate the colours in the Colour Corrector.
- Increase the contrast in the Colour Corrector.
- Apply filters (for example, Emboss).
- Remove film grain using Degrain.

Setup Options

Use the options in the Stabilizer menu to specify preferences for tracking and the user interface. Use the options in the Setup menu to specify general and rendering preferences.



Changing the Colour of Trackers

When you add a new tracker, the system automatically assigns it a unique colour so that you can easily distinguish between multiple trackers. You can change the colour of an individual tracker or of all the trackers at once.

To change the colour of the tracker:

1. Click the Tracker button that corresponds to the tracker you want to change.

   TIP To change all trackers, select Gang in the Tracker Selection box.

2. Click the Tracker colour pot.
3. Select a colour with the colour picker.
4. Click in the Tracker colour pot to apply the new colour to the tracker.

Pretracking a Clip

Before analysing, you can use the Pretracking option in the Setup menu to preview the motion path for a specified number of frames. You can then adjust the tracker position, if necessary, to find the best reference point. The Pretracking option applies only to the selected tracker, regardless of whether you selected Solo, Selected, or Gang in the Tracker Selection box. If you move or resize a tracker with Pretracking enabled, the next frames are analysed.

To pretrack a clip:

1. Position and select a tracker.
2. In the Setup menu, make sure that Path is enabled. If it is disabled, Pretracking is not available.
3. Enter the number of frames to pretrack.

   NOTE If you enter a number larger than the length of the clip, only existing frames are pre-tracked.

4. Enable Pretracking.
   The motion path appears.

   NOTE Frames, not fields, are pretracked. If you are in Fields mode, frames will be pretracked.

Copying Shift Channels

Using the Channel Editor, you can copy a tracker's Shift channel values and paste them into any other channel in any module. For example, you can copy jitter values to the X and Y position channels of an axis to add realism to a static scene, or to a channel to add noise.

There are two Copy buttons in the Stabilizer. One takes the aspect ratio of the clip into account and the other one does not.



(a) This Copy button takes the aspect ratio into account (b) This Copy button does not take the aspect ratio into account

The coordinate system that the Stabilizer uses for mapping pixels on the image differs from the system used by other tools. In the Stabilizer, the pixel with coordinates (0,0) is in the lower-left corner, while in most other modules it is in the centre of the screen. When pasting shift curves to the channels of an axis, use the Copy button that takes the aspect ratio into account. Since this button also accounts for the differences in pixel coordinates between tools, it ensures that the values are mapped properly.

To copy and paste a shift channel:

1. In the Channel Editor, open the folder of the tracker whose shift values you want to copy.
2. Select the X or Y Shift channel, or the entire Shift folder to select both X and Y Shift channels.
3. Click one of the Copy buttons, depending on whether or not you want to maintain the aspect ratio.
4. If necessary, load the clip to which you want to apply the Shift values into the appropriate tool.
5. Open the folder of the channel to receive the copied shift values and select the appropriate channel(s).
6. Click Paste.

Importing and Exporting Data

Import and export tracking and shift data values to an ASCII file using the Import and Export buttons located in the Track and Shift areas of the Stabilizer menu.



The tracking data specifies the X and Y position of the tracker box in relation to the origin point of the image; these are absolute coordinate values. The origin point has coordinates (0,0), and is located at the lower-left corner of the image.

The shift data specifies the difference between the position of the reference box (0,0) and the position of the tracker box in the current frame. These are relative coordinate values.

Both the Tracking values and the Shift values are formatted as follows:

frame#: X position, Y position

NOTE If the position of the reference box is changed during the analysis, the offset is compensated for and the exported tracking data still shows a continuous tracker path.

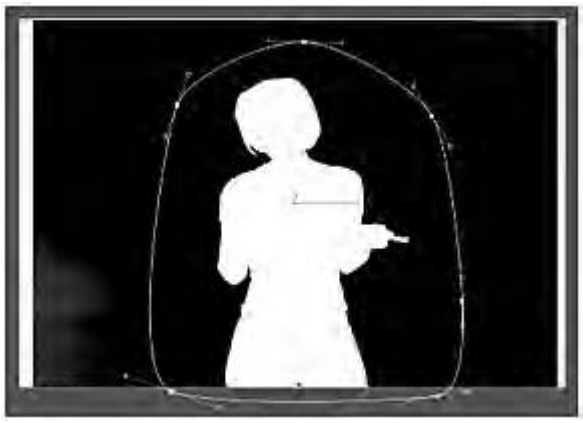
Masking and Rotoscoping

20

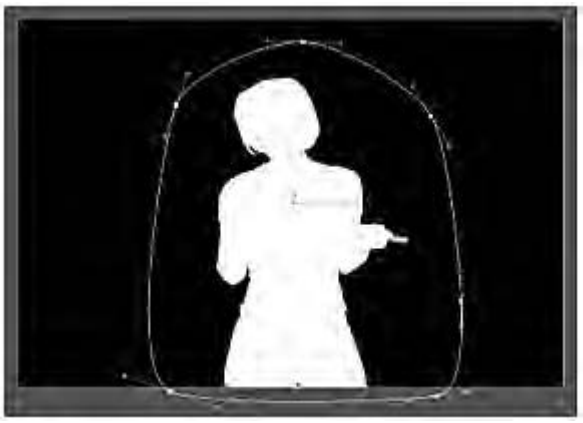
When pulling a key, use garbage masks to isolate particular areas of an image to include with, or exclude from, the opaque area of the matte. Garbage masks are spline-based objects you draw directly on a clip or image. Depending on what you want to accomplish, you can set the area inside each garbage mask to be either opaque (white), transparent (black), or semi-transparent (any shade of grey).

Garbage masks are useful in many situations, such as when removing unwanted elements like equipment or people from a blue-screen shot, when removing unwanted borders from an image, or when creating complex keys by blending portions of the front and back clips. You can animate the shape of a garbage mask, apply motion blur to it, and even use the Stabilizer to make a garbage mask follow a moving element in a clip.

The following example illustrates keying out an unwanted white border using a garbage mask.



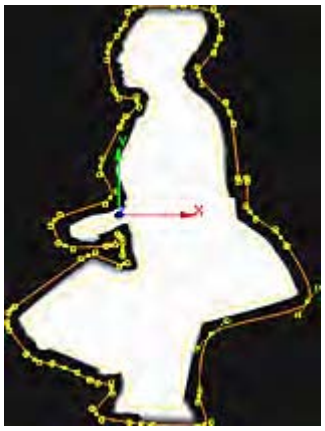
Matte before garbage mask is applied



Matte after garbage mask is applied

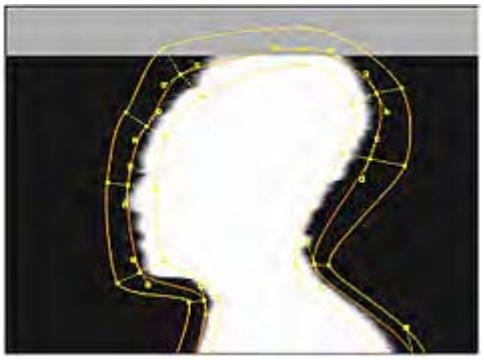
You can also apply softness to the edge of a mask using a uniform or advanced gradient. Instead of having an abrupt transition from white to black in the matte, you can control the gradations of grey between the key and the background. For example, softness can compensate for uneven edges of a key by blending the front and back clip. You can apply uniform softness around the edge of a mask or vary the softness range for different parts of the mask.

An advanced gradient adds two splines around the garbage mask. By moving points on these splines, you can control the gradient according to how far the surrounding spline is offset from the garbage mask.



Softness applied around edge of garbage mask spline

Image courtesy of Behavior Communications Inc.



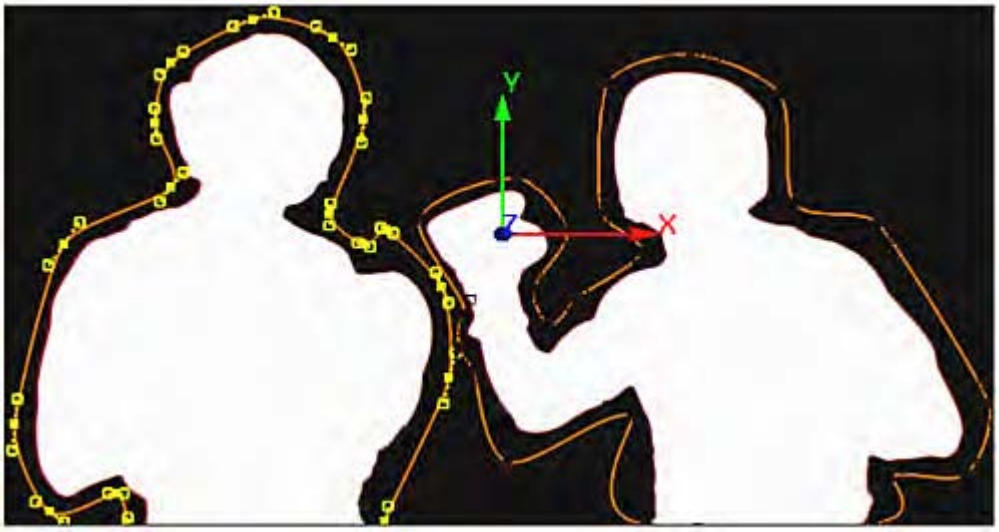
Advanced gradient varies softness for different parts of the garbage mask

Image courtesy of Behavior Communications Inc.

Using Garbage Masks in Batch or Batch FX and the Modular Keyer

Certain garbage mask features are supported only in the GMask node in Batch or Batch FX and the Modular Keyer.

You can use a parameter called Region of Interest (ROI) when drawing multiple garbage masks on a clip to remove the portion of the matte that falls outside of each mask while respecting the area defined by all masks.



ROI enables you to reveal multiple parts of the matte.

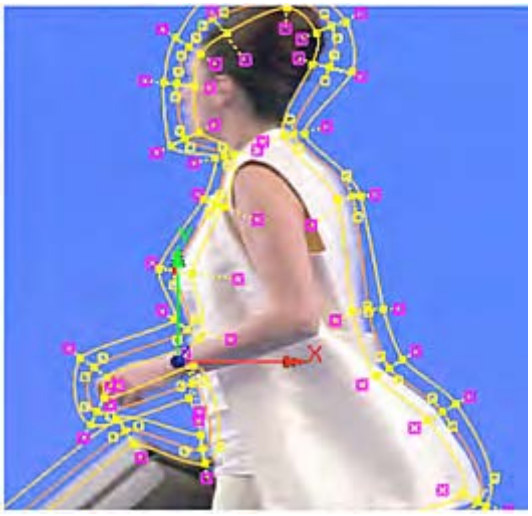
Image courtesy of The Post Group

You also have the option of using the Tracer with an advanced gradient. The Tracer is an additional softness tool you can use to pull a key from scratch. This allows you to key images with a lot of detail at the edges. It uses a system of localized keys, called *pickers*, which analyse the colour information both inside and outside

the mask to derive a key for the mask edge. You can use both the Tracer and the advanced gradient within the same garbage mask.

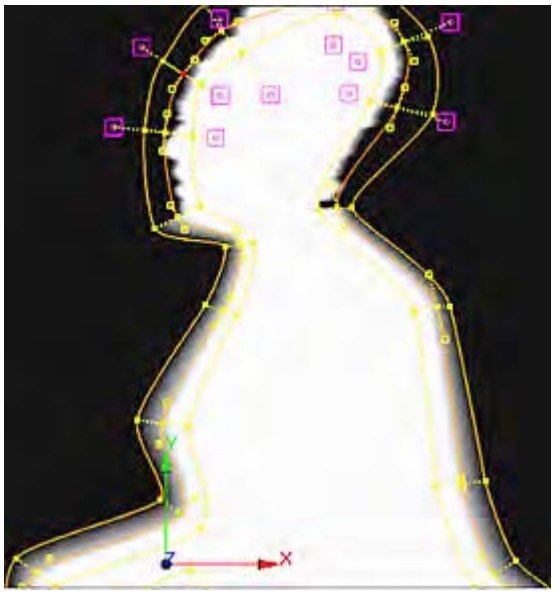
The Tracer is particularly useful for keying challenges such as:

- Very fine, wispy talent edges (hair, fur, or lace) where a blue or green screen may show through.
- An object not shot on a blue or green screen.



Garbage mask with pickers sampling image for more advanced softness

Image courtesy of Behavior Communications Inc.



Garbage mask with pickers applied to only select vertices, the remainder using an advanced gradient

Image courtesy of Behavior Communications Inc.

The Stabilizer and pickers are not available when this type of input is attached to the node.

Accessing the Garbage Mask Menu

Access the Garbage Mask menu from the Modular Keyer or the GMask node when you want to use multiple garbage masks or use the Tracer for detailed masks.

Use the Garbage Mask to isolate particular areas of an image to include with, or exclude from, the opaque area of the matte.



To access the Garbage Mask menu, use:

- Batch, then select a node from the Node bin. (page 894)
- Timeline, then use Batch FX (page 894).
- Modular Keyer, then select a node from the Node bin. (page 894)

This node accepts a front and a matte clip as input, and outputs a result.

The Garbage Mask menu includes the Tracer and the region of Interest (ROI) functions, which can also be found in the Modular Keyer. The node processes gaps in clips set to No Media as black frames. An unconnected front clip will return an error, while and unconnected back clip will process black frames. You can save and load GMask setups directly in Batch or Batch FX.

Accessing the Garbage Mask from Batch

To access the Garbage Mask from Batch;

1. Go to Batch.
2. Double-click on, or drag, the GMask Node from the node bar and add it to the schematic.
3. Double-click the GMask Node.
   The GMask Node editor opens.

Accessing the Garbage Mask from the Batch FX

To access the Garbage Mask from the Timeline through Batch FX;

1. Select Timeline.
2. Select the segment.
3. Click the FX button.
4. Click the Create Batch FX button.
5. Double-click on, or drag, a GMask node into the schematic.
   The GMask node is now in the schematic.
6. Double-click the GMask node.
   You are in the GMask node editor.

Accessing the Garbage Mask from Modular Keyer

To access the Garbage Mask from the Modular Keyer;

1. Double-click on, or drag, a GMask node into the schematic from the Modular Keyer editor.
   The GMask node is now in the schematic.
2. Double-click the GMask node.
   You are in the GMask node editor.

About Garbage Mask Options

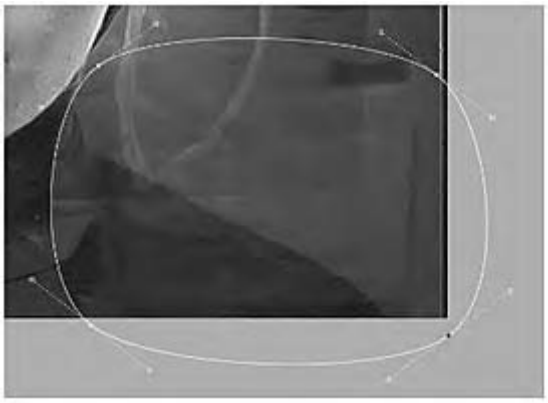
You can customize a mask's properties in the Garbage Mask menu.

Render Mask button Turns masks on or off as you work. A mask that is turned off can be seen in the image window, but it has no effect on the image. Masks are turned on by default.garbage masks:turning on and offConstant shape, for garbage masks

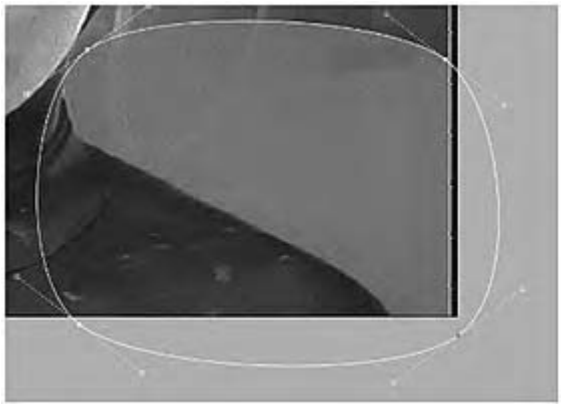
NOTE This parameter cannot be animated.

Outside button Applies the effect to the part of the image that is outside the mask.

Colour field Sets the blend value between the front and back image in the area that the mask is affecting (that is, either outside or inside the mask). A value of 50% is a 50/50 blend between the front and back clip. A value of 100% is the front clip. A value of 0% is the back clip.

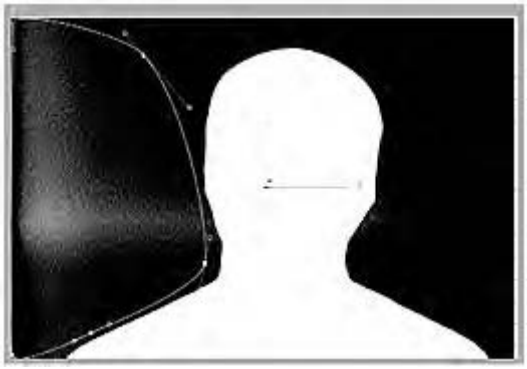


Mask with 25% colour

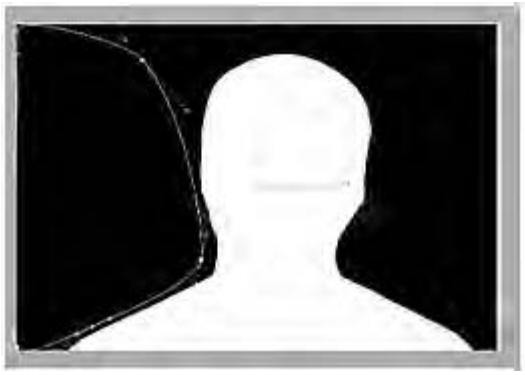


Mask with 75% colour

Opacity field Sets the opacity of the matte in the area affected by the mask (that is, either inside or outside the mask). A value of 100% means the matte is completely opaque; 50% means the matte is 50% transparent; and 0% has no effect on the image.



Mask with 0% opacity



Mask with 100% opacity

Lasso Fit field Increase or decrease the number of points in the segments of the mask that are drawn freehand. See Adjusting the Number of Points in Freehand Segments (page 901).

Edge Softness box Apply a uniform gradient or use an advanced gradient to set different levels of softness for different parts of the mask edge. In the GMask node in Batch or Batch FX or the Modular Keyer, use the Tracer to set variable softness around the mask edge using pickers. See Refining the Mask (page 916).

Alpha field Defines the transparency of the gradient from the edge of the mask.

Offset field Defines the border position of the gradient from the edge of the mask.

Inner Edge, **Outer Edge**, **Distance fields** Applies the distance and degree of change between the opaque and transparent part of the mask edge.

Linear Interp Enable to use linear interpolation of the mask border between keyframes. Disable to use rounded interpolation.

Constant Shape button Enable to modify the mask's shape without setting keyframes. This forces all animatable parameters (except the Tracer parameter Sample On) to be set for the whole clip rather than for only the current frame. It also removes any existing keyframes and applies the shape of the current frame to the rest of the clip.

Splines button Enable the display of softness, borders and pickers.

Borders button Enable the display of borders. The Border button is active when the Advanced Gradient and Tracer menus are displayed.

Pickers button Enable the display of pickers. This element is only available when accessing the Tracer tool in the Garbage Mask menu from the GMask node in Batch or Batch FX or the Modular Keyer.

X/Y fields Defines the mask's X and Y offset from its axis.

Setting Mask Options

You can control how a mask affects the matte. For example:

- Set the opacity, colour, and edge softness of the mask.
- Specify whether the effect is applied to the inside or outside of the mask.
- Adjust the offset of the mask from its axis.
- Turn a mask on or off.

To set mask options:

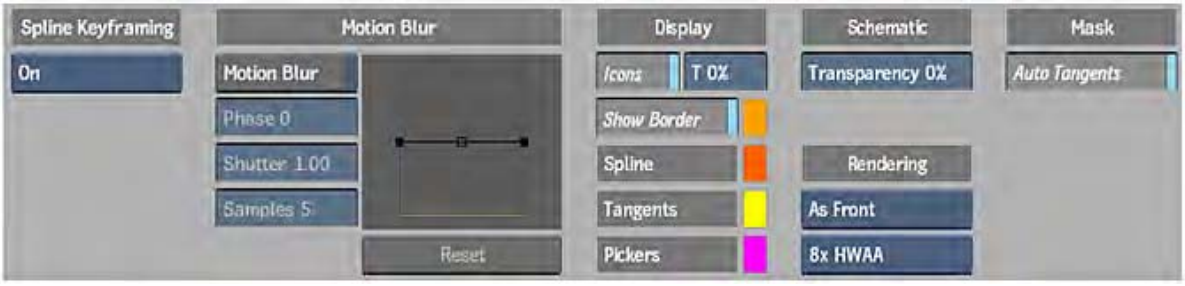
1. Display the Matte or Result view.



2. Select the mask in the image window:
   - Click anywhere on the garbage mask spline.
   - Click the Geom node in the schematic.
3. Access the Garbage Mask menu.
4. Set the mask options. See About Garbage Mask Options (page 894).

About Drawing Options

Use the Mask Setup menu to access mask drawing options.



Spline Keyframing box Determines under what conditions keys are set, and for which parameters, when you are animating a garbage mask spline. See Animating Masks Using Spline Keyframing (page 907).

Motion Blur settings Controls motion blur applied to garbage masks that can be used to match the movement of objects in a clip. See Applying Motion Blur to Garbage Masks (page 912).

NOTE There are additional setup options to do with animating garbage masks. These only appear when you disable Shape Animation in the Garbage Mask menu. For more information, see Animating Masks Using Spline Keyframing (page 907).

Icons button Globally displays or hides the garbage mask splines and axes.

NOTE If you hide garbage maks splines and axes with the Icons button, the Widget All / Widget Sel button in the View menu has no effect.

Transparency field Sets the transparency for the garbage mask splines and axes.

Show Border button Displays the Softness Offset wireframe border, defined in the Shape menu.wireframe:in garbage masks

Show Border colour pot Sets the colour for the Softness Offset wireframe.

Spline colour pot Sets the colour for the wireframe of the garbage mask.

Tangents colour pot Sets the colour of the garbage mask tangents and vertices.

Pickers Display colour pot Sets the colour of the Tracer's pickers. This element is only available when accessing the Garbage Mask Setup menu from the GMask node.

Schematic Display Transparency field Sets the transparency of the nodes in the garbage mask schematic.

Rendering box Select the rendering method.

Hardware Anti-aliasing Sample box Select a hardware anti-aliasing sampling level to accelerate edge anti-aliasing with no performance penalty. The graphics hardware automatically renders the image at full speed with approximately the equivalent of up to 32 samples of anti-aliasing (depending on your graphics card and project graphic bit depth). Hardware anti-aliasing also gives anti-aliasing during normal interaction instead of only while rendering.

Auto Tangents button Positions a tangent for each vertex set to create a smooth curve between the vertices. When Auto Tangents is disabled, the tangents are positioned under the vertex, resulting in straight lines between vertices. When you draw freehand segments in a mask with Auto Tangents off, vertices are added with broken tangents, allowing the mask to follow your cursor movement.

Drawing a Mask

After setting the drawing options, you can draw the mask.

To draw a garbage mask:

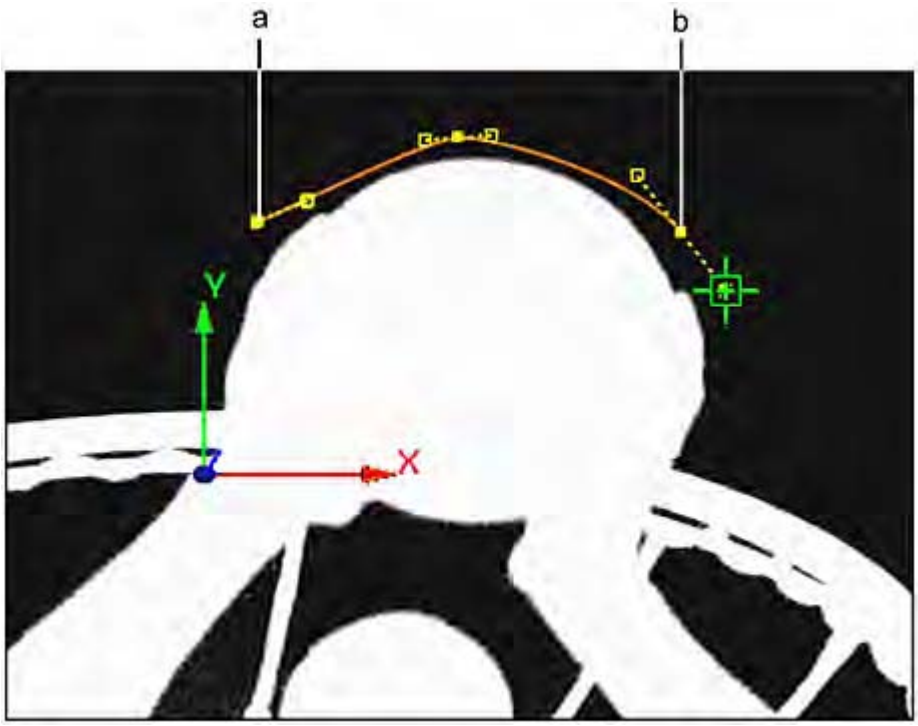
1. Access the Garbage Mask menu.
2. Select the view where you want to draw the mask from the View box.



Which view you use depends on the context of your key and why you are drawing the mask. For example, if you want to crop the edges of a clip by drawing an opaque mask around the subject and then turning on Outside, try drawing the mask using the Matte view. Or, if you are rotoscoping a subject that has not been shot on a blue or green screen, try either the Front or Result view.

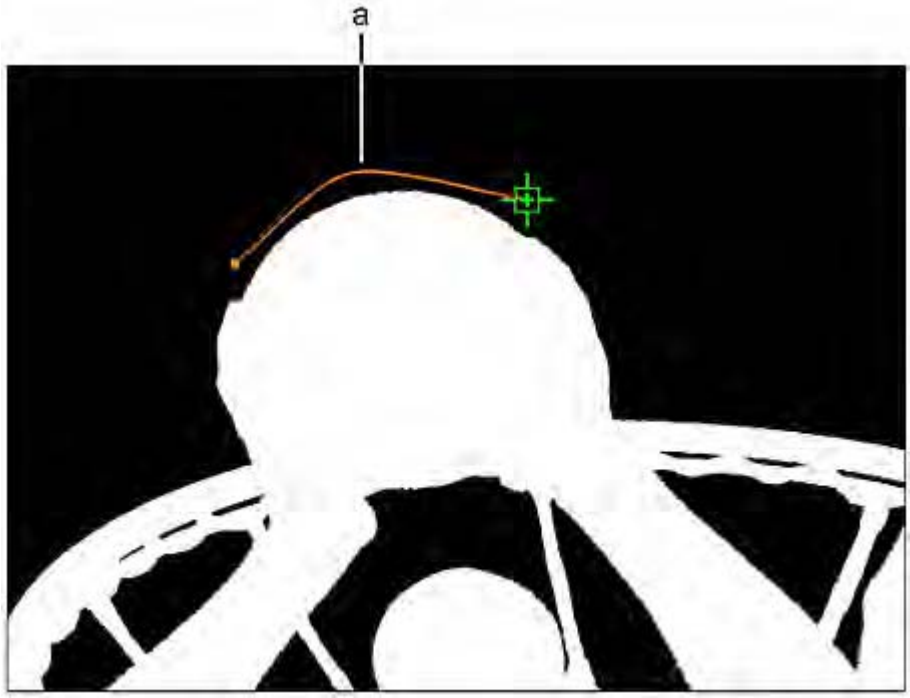
TIP In Batch or Batch FX or the Modular Keyer, you can use multiple viewports as you draw the garbage mask. For example, you can draw the mask in Matte view, while also displaying Result view. Using multiple viewports, you can immediately see how your mask affects the clip.

3. Do one of the following:
   - Click Add.
   - Select Create from the Edit Mode box.
   - Press N.
4. Do any of the following to draw the garbage mask:
   - Click to add vertices.



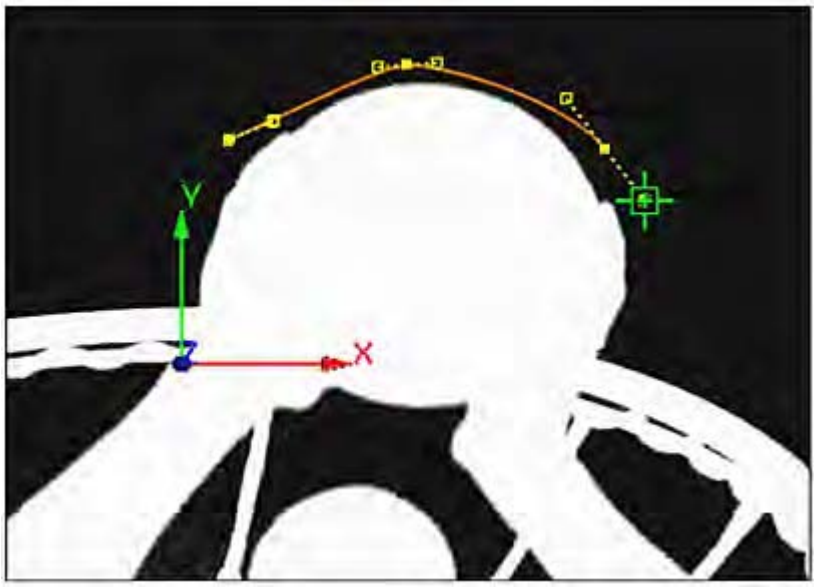
(a) Vertex added by first click (b) Vertex added by third click

- Press `Shift` to temporarily hide the vertices and tangents as you are drawing the mask.



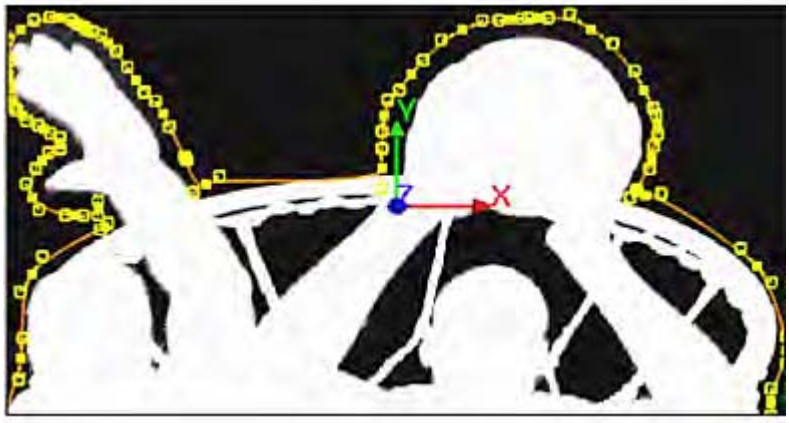
(a) Vertices are temporarily hidden while holding Shift

- Click-drag to control the tangency of any vertex as you are drawing the mask. When you release, you can continue to add vertices.



Tangency handles extend as you drag from a vertex

- `Shift`-drag to add freehand segments to the mask. The vertices and tangents of the mask you are drawing are temporarily hidden. Vertices are added where you drag, and appear when you release `Shift`. After closing the mask, you can use the Lasso Fit parameter to increase or decrease the number of vertices that define the freehand segments of the mask. See Adjusting the Number of Points in Freehand Segments (page 901).



Segments of garbage mask drawn freehand

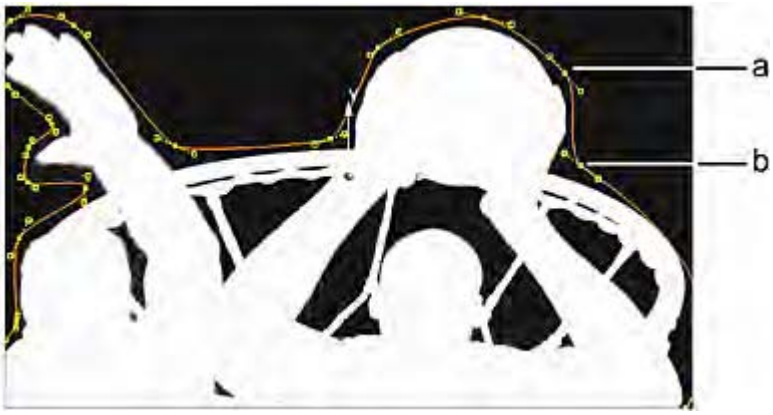
NOTE If you are drawing the mask with Auto Tangents turned off, freehand segments are drawn with broken tangents. See Breaking Tangents (page 904).

5 To close the mask, do one of the following:

- Click Close.
- Click the first vertex.
- Drag to the first vertex.

TIP You can also close the mask by pressing the keyboard shortcut for the edit mode you want to use next: `M` for Move, `S` for Select, or `B` for Break. The mask closes and you are ready to work in that mode.

When the garbage mask is closed, its vertices and tangents can be edited and animated.



(a) Vertex (b) Tangent

If you want to view the nodes that are added with each new garbage mask, select Schematic from the World View box. You can use the Schematic view to access a menu, create parent-child relationships between masks and axes, delete masks, as well as perform other organizational tasks.



(a) World View box

Adjusting the Number of Points in Freehand Segments

You can increase or decrease the number of points in freehand segments of a mask by adjusting the Lasso Fit value. An entire garbage mask can consist of freehand segments, in which case the Lasso Fit value affects the whole garbage mask. Otherwise, the Lasso Fit value only affects the part of the mask drawn freehand.

The Lasso Fit parameter loses its influence over freehand segments of a mask if you have edited points as follows:

- Added or deleted points using the Add or Delete Edit modes
- Saved and loaded a garbage mask
- Started a new session with the application

To adjust the number of vertices using Lasso Fit:

1. If necessary, select the garbage mask you want to affect.
2. Drag the Lasso Fit field. Increase the value to decrease the number of points in the freehand segments of the mask. Decrease the value to increase the number of points in the freehand segments.

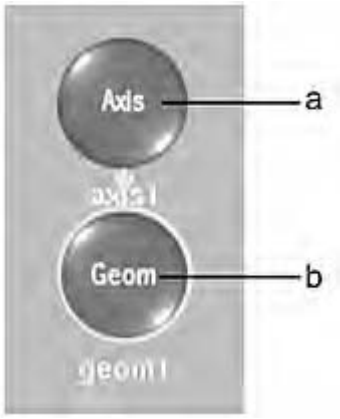
Segments of the mask that have been drawn freehand are affected, while segments created by simple clicks remain unaffected.

NOTE If you move vertices of a garbage mask and then change the Lasso Fit value, all the vertices snap back to the original position where they were first drawn, and all animation is lost. This occurs only if the Lasso Fit value retains its influence on the garbage mask.

Garbage Mask Nodes

When you create a garbage mask, an Axis node and a Geom node are added to the schematic. The Axis node contains rotation, position and scaling data. The Geom node contains information about how the mask will affect the image. There is also a Camera node, which is always part of the scene. The Camera node can parent the Axis node for greater precision over the garbage mask, particularly if you want to rotate or orbit the camera with respect to the garbage mask.

To view the schematic, select Schematic from the World View box to toggle between the Schematic view and the previous view.



(a) Axis node (b) Geom node

Changing the Priority Order of Masks

When you create several garbage masks on an image, you can change the order in which they are drawn, or layered in the scene. This affects the resulting image because a mask with a higher priority has precedence over one with a lower priority. You can use the priority order to create complex effects involving multiple garbage masks.

To set the priority of a mask:

1. Select the mask.
2. Click Push to move the mask down a layer; click Pop to move the mask up a layer.



NOTE You cannot animate the priority by setting Push and Pop at different frames. When you set a mask's priority with Push or Pop, it is set for the entire clip.

Transforming the Mask

You can apply transformations such as translation, rotation, scaling, shearing, and centre changes to a garbage mask. The transformation data for the mask is stored in its Axis node. You apply transformations to the mask using the Axis controls.

All transformation parameters in the Axis menu can be animated. You can use a motion path to animate the position of a mask.

You can also apply motion blur to a garbage mask. See Applying Motion Blur to Garbage Masks (page 912).

To transform a mask:

1 Access the Garbage Mask menu.
The Axis controls appear.

NOTE The Axis controls are similar to the Axis menu in Action.

2 Use the Transformation fields to modify the position, shape, and size of the mask.

Manipulating Vertices and Tangents

Use the options in the Edit Mode box to manipulate the vertices on the mask. You can edit the mask most easily in Front, Key-In, or Matte view.

Selecting Vertices and Tangents

You can select vertices and tangent handles when Select, Move, or Scale is selected in the Edit Mode box:

- To select an individual vertex or tangent handle, click the vertex or tangent handle.
- To select multiple vertices, `Ctrl`-drag a selection box over a series of vertices.
- To select multiple vertices using the pen, press the pen button and drag a selection box over a series of vertices.
- To add another vertex to a multiple selection, `Shift`-click the vertex.

- To add several more vertices, press Shift+Ctrl and drag a selection box over the additional vertices.
- To deselect all vertices, click anywhere outside the mask.

Selecting Pickers and Softness Vertices

You can select pickers or softness vertices using the following techniques. Pickers are an element of the Tracer tool, which is only available in the GMask node in Batch or Batch FX and the Modular Keyer. See Applying Softness Using Pickers (page 918).

To select a single picker or softness vertex:

1. Click the picker or vertex that you want to select.

To select multiple pickers or softness vertices:

1. Select the mask vertices that the pickers or softness vertices are associated with (see previous section).
2. Click a picker or softness vertex corresponding to one of the selected mask vertices.
   The pickers or softness vertices are selected for all the selected mask vertices.

Moving Vertices and Tangents

You can move vertices and tangents using the Move and Scale options of the Edit Mode box:

- To move a vertex or tangent in any direction, select Move. Click the vertex or tangent and drag.

  TIP If you have trouble selecting a tangent handle that is very close to the vertex, hold Q then select it.

- To move selected vertices in any direction, select Move. Click one of the selected vertices and drag.
- To move one or more selected vertices in the direction perpendicular to their tangents, select Scale. Click one of the selected vertices and drag.
- To ensure better continuity on a garbage mask, automatically adjust tangents as you move vertices or scale the mask. Press and hold G and then drag a vertex. See Automatically Adjusting Tangent Handles of Adjacent Vertices (page 905).

Breaking Tangents

You can separate two tangent handles (“break” the tangent) and move them separately using the Break option from the Edit Mode box:

- To break and move a tangent handle, select Break and click the tangent handle. The tangent is displayed as a solid line, indicating it is “broken.”
- To reconnect two broken tangent handles, select Auto and click either of the two tangent handles. The tangent is displayed as a dotted line.
- To change the position of an individual tangent handle after releasing the cursor, select Move.

Removing and Adding Tangents

You can also use the Break option to remove tangents from vertices by clicking the vertex while in Break mode. When you remove a tangent, the curve defined by the tangent is removed.

The shape of the border line differs depending on whether adjacent vertices are broken:

- If adjacent vertices are unbroken, the border line curves as it approaches the vertices.
- If adjacent vertices are broken, the border line is straight as it approaches the vertices.

You can create a garbage mask composed entirely of straight edges by removing the tangents from all the vertices:

- To remove the tangent of a vertex, select Break and click the vertex.
- To add a tangent back to a vertex, select Auto and click the vertex. Alternatively, using the Break option, click the vertex and "drag" the tangent out again.

Automatically Adjusting Tangent Handles of Adjacent Vertices

When you move vertices, the tangent handles of the two adjacent vertices remain fixed in their current position. You can make them automatically adjust to create smooth curves between the vertices by pressing the G keyboard shortcut as you move vertices.

You can reverse the behaviour so that dragging vertices automatically adjusts tangents without using the keyboard shortcut. You reverse the behaviour by setting the following environment variable:

```
DL_GMASK_AUTO_MOVE_MODE
```

To reverse functionality of the G keyboard shortcut:

1. From the home directory of the application, open the *.cshrc* file in a text editor.
2. Add the following line to the end of the *.cshrc* file:

   ```
   setenv DL_GMASK_AUTO_MOVE_MODE
   ```
3. Save and close the *.cshrc* file.
4. In the home directory, type:

   source .cshrc
5. Launch or restart the application.

   When you click or move a vertex on a garbage mask, the tangents are automatically repositioned. When you press **G** and click or move a vertex, the tangents are not repositioned.

   TIP Alternatively, you can set the environment variable in the shell pointing to the home directory of the application. When you set an environment variable in the shell, the environment variable is enabled for the current session only.

Creating Right-Angled Vertices

You can create right-angled vertices using the following technique.

Press:	To:
Ctrl+A	Select all vertices.
B	Break apart tangents.
G	Create right-angled vertices of selected vertices.

Animating Garbage Masks

You can animate a garbage mask using the Shape channel in the Channel Editor or using spline keyframing, which animates individual Vertex channels. Vertex channels correspond to the vertices—or handles—that appear along the perimeter of the garbage mask. The Vertex channel names are contained in the Spline folder in the Channel Editor.

You can apply tracking data to a mask's axis to animate a garbage mask. Existing keyframes are disregarded, after applying tracking data, you can animate individual vertices and make adjustments.

Animating Masks Using the Shape Channel

The Shape channel shows when the shape of the garbage mask changes during the animation.

To animate a garbage mask using the Shape channel:

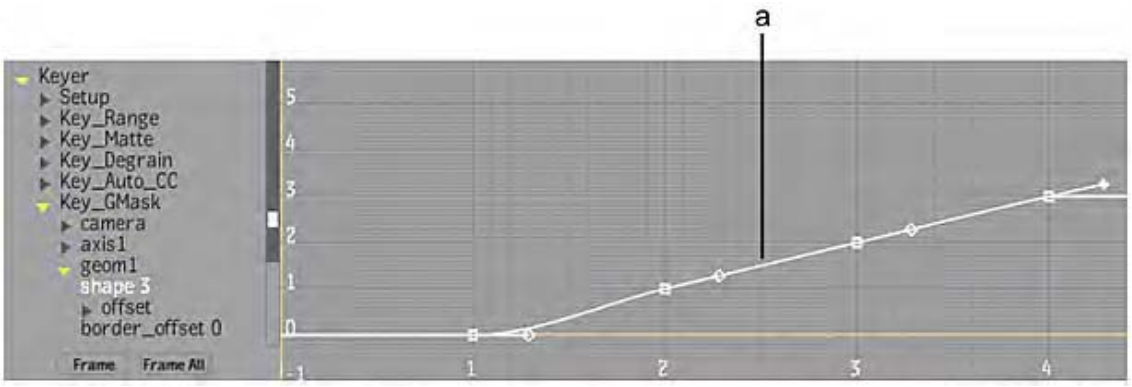
1. In the Garbage Mask menu, click Mask Setup.
2. In the Garbage Mask Setup menu, turn on Spline Keyframing.
   Each time you move a vertex, a shape key is added at the current frame.

   NOTE Disable Spline Keyframing if you do not want a keyframe added to the Shape channel.
3. Click Mask Setup again.
   The Garbage Mask menu appears.



(a) Shape Animation button (b) Linear Interpolation button

4. Enable Shape Animation.
   If keyframes are already set for specific vertex channels, the following message appears: “Convert explicit channels to a single shape channel?”
5. Click Confirm.
   The Shape channel appears in the Channel Editor and any Vertex channel keyframes are converted.



(a) Shape channel with 4 keyframes

NOTE Enabling Shape Animation disables the Vertex Keyframing and Channel Selecting boxes in the Garbage Mask Setup menu. These contain options for animating individual vertex channels. See Animating Masks Using Spline Keyframing (page 907).

6 Depending on how you want to animate the garbage mask, use the Linear Interpolation and Constant Shape buttons as follows.

Enable:	To:
Linear Interpolation	Make the garbage mask trajectory linear. Enable this button when rotoscoping. For smooth garbage mask animation, disable this button.
Constant Shape	Keep the garbage mask shape the same for all frames. When you enable this button, the following message appears: "Remove all keyframes except current one?" Click Confirm.

Animating Masks Using Spline Keyframing

You can animate the shape of a mask by moving the vertices of a garbage mask spline at different frames throughout the clip. Each vertex has its own X, Y and Z position channels in the Channel Editor, as well as several other parameters, as shown in the following table.

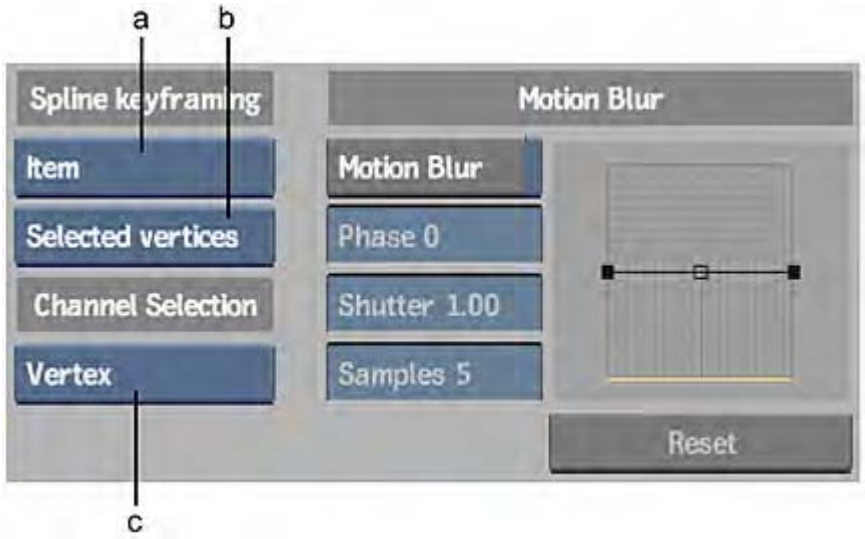
Parameter	Channel Editor Folder & Name(s)	Channel Editor Values
The position of the mask vertices.	*position: X, Y, Z*	
The position of the left and right tangent handles relative to the mask vertex.	*tangents: left, right*	
The tangent continuity (handles broken or unbroken).	*tangents: continuity*	0 = broken handles 1 = unbroken handles
The active/inactive status of mask vertices. See Adding and Removing Vertices (page 909).	*Vertex_x: active*	0= inactive 1 = active

Before you start to animate the mask, set the Spline Keyframing options in the Garbage Mask Setup menu. In the GMask node in Batch or Batch FX, these options are located in the Node Setup menu. Use these options to specify which elements will get a keyframe as you animate the mask.

As you adjust the mask, you can opt to apply keyframes to selected vertices or to all vertices. To quickly animate a mask, keyframe all vertices every time you adjust a single vertex. This way, what you see at a given frame will be exactly what you get no matter how you animate the "surrounding" frames.

To have more control over the shape of the mask frame by frame, animate one vertex at a time. This method is useful for tracking garbage masks and keeping control over tangent animation. See Animating Masks Using Tracking (page 911).

By combining the Spline and Vertex Keyframing options in different ways, you can set an animation mode to suit the task at hand. For example, by selecting Item and All Vertices, you could animate just the tangent handles of all vertices on the mask.



(a) Spline Keyframing box (b) Vertex Keyframing box (c) Channel Selecting box

Spline Keyframing box Sets the conditions under which keyframes are set, and for which parameters.

NOTE The Spline & Tracer, Spline only, and Item options are only available if you disable Shape Animation in the Garbage Mask menu.

Select:	To:
Spline & Tracer	Set a keyframe for all the parameters of the vertex or vertices and related elements, with the exception of the Sample On parameter. This option is only available when accessing the Garbage Mask Setup menu from the GMask node.
Spline only	Set a keyframe for one or more vertices, or related elements, for the specified parameter, as well as for the mask vertex position and the tangent position channels.
Item	Set a keyframe for a parameter of a vertex, or of a related element. Only the particular parameter gets a keyframe.
On	Set a keyframe for the shape channel when a change to a vertex is made (such as moving a vertex or tangent handle, breaking tangent handles, or changing the Active/Inactive status of a vertex). This option is only available when Shape Animation is enabled in the Garbage Mask menu.

Select:	To:
Off	No keyframes are set.

Vertex Keyframing box Sets the conditions under which parameters are applied to specified vertices in the mask.

NOTE The Vertex Keyframing box is only available if you disable Shape Animation from the Garbage Mask menu.

Select:	To:
All Vertices	Set a keyframe for the parameter you are animating for all the vertices of the mask. Related elements, such as tangents, also all get a keyframe when you animate a parameter on the mask.
Selected Vertices	Set a keyframe for the parameter you are animating for all selected vertices on the mask. Related elements, such as tangents, that are selected, also all get a keyframe when you animate one on the mask.

Channel Selecting box Determines which channels are selected in the Channel Editor when you select one or more vertices.

NOTE The Channel Selecting box is only available if you disable Shape Animation from the Garbage Mask menu.

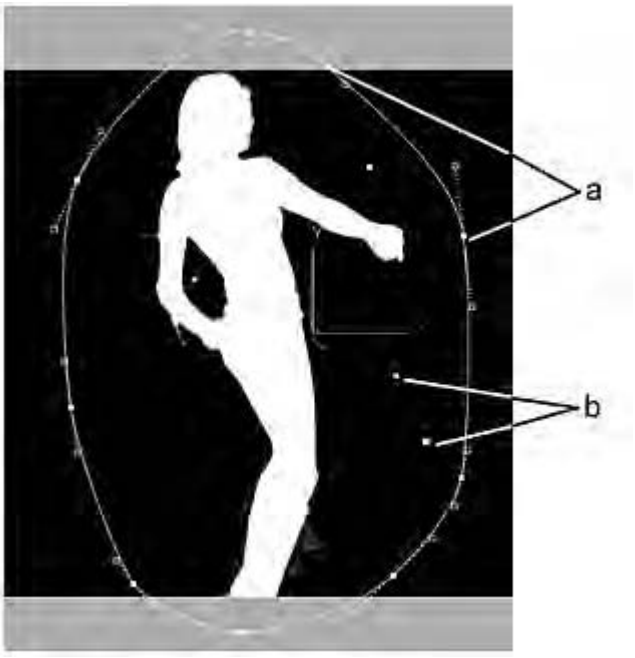
Select:	To:
Vertex	Select all the channels related to the vertices you select on the garbage mask. For example, whether you select a vertex or one of its tangents, its active, position, and tangent channels are also selected in the Channel Editor.
Item	Select only the particular channel corresponding to the selected vertices. For example, if you select the right tangent handle of a vertex, only the corresponding right channel is selected in the Channel Editor.

Adding and Removing Vertices

To further control the shape of a mask throughout the clip, you can add and remove vertices from your mask at any frame of the clip. For example, if the shape changes radically or gets bigger part way through the clip, you can add vertices to reshape the mask at that frame. When you add a vertex part way through the clip, it is added to the whole clip but respects the existing shape of the mask in all previous and subsequent frames. Therefore, any reshaping that you have done to the mask is not affected by the new vertices.

NOTE Adding or deleting vertices on masks with freehand segments causes the Lasso Fit value to lose influence. See Adjusting the Number of Points in Freehand Segments (page 901).

If you delete a vertex, it becomes “inactive” from that frame to the next active/inactive keyframe set for that vertex, or, if there are no keyframes, to the end of the clip. An inactive vertex has no effect on the shape of the mask. On previous frames, the vertex is still active, so the mask shape is unchanged by the deletion. To delete the vertex completely (that is, throughout the whole clip), click it a second time.



(a) Active vertices (b) Inactive vertices

Use the options in the Edit Mode box to add and delete vertices from the mask, and to control which vertices are active and inactive.

To add a vertex to the entire mask:

1. Select Add from the Edit Mode box.
2. On any frame in the clip, click the mask where you want to add the vertex.

   The new vertex is added and is active throughout the clip. It does not affect the existing mask animation, regardless of where in the clip you add the vertex.

To make an active vertex inactive:

1. Switch to Delete mode.
2. Click the vertex.

   The vertex is inactive from the current frame to the next frame where an Active/Inactive keyframe has been set, or, if there are no keyframes, to the end of the clip. The status of the vertex on previous frames is unchanged.

   NOTE Alternatively, use the Active button in the Shape menu to make vertices inactive. Select the vertex you want to make inactive (you must be in Select, Move, or Scale mode), and then click Active.

To make an inactive vertex active:

1. Switch to Add mode.
2. Click the inactive vertex.

   The vertex is active from the current frame to the next frame where an Active/Inactive keyframe has been set, or, if there are no keyframes, to the end of the clip. The status of the vertex on previous frames is unchanged.

   NOTE Alternatively, use the Active button in the Shape menu to make vertices active. Select the vertex you want to make active (you must be in Select, Move, or Scale mode), and then click Active.

To delete a vertex from the mask:

1. At any frame, switch to Delete mode.
2. Click the vertex.

   It becomes inactive.
3. Click the vertex a second time.

   It is removed from the mask for the duration of the clip.

   NOTE You cannot delete the last three vertices of a mask, as they are the minimum necessary to define it. If you want to delete the mask, you delete its Geom node in the Garbage Mask Schematic.

Animating Masks Using Tracking

You can animate a mask by applying tracking data to it. This is useful when you want a garbage mask to follow a moving element in a clip. To do this, you can either:

- Animate the entire mask by applying the data to the mask's axis.

  You can track an object on the front or back clip and apply the tracking data to the axis of a mask, or to a hierarchy of masks.
- Animate selected vertices according to reference points that you set in the Stabilizer.

  Each vertex you select is assigned a tracker in the Stabilizer. The vertices are repositioned according to the reference points that you set in the Stabilizer.

When you apply tracking data to a mask, only the mask axis, or position of selected vertices, is animated. Any other keyframes set for the mask are disregarded. You can, however, animate individual vertices after applying tracking data to your mask's axis or to individual vertices. It is especially useful to manually adjust the tangent handles of the vertices at different frames where needed.

To animate an entire mask with tracking:

1. Access the Garbage Mask menu.

   The Garbage Mask menu appears.
2. Select the mask.
3. From the Clip box, select the clip you want to track.
4. If necessary, adjust the mask using the Position, Rotation, Scale, Shear, and Centre fields.
5. Set the Rotation and Scaling options. To track rotation, select Rotation On; to track scaling, select Scaling On.
6. Select an option from the Adjust box.

Select:	If the selected mask is:
Adj Offset	Parented to one axis.
Adj Axis	Parented to a hierarchy of objects.

7. Go to the frame where you want to start tracking.
8. Click S.

   The Stabilizer appears.
9. Position of the trackers.

NOTE The first tracker is for position data and the second tracker is for rotation and scaling data. If rotation and scaling were not selected, the second tracker has no effect.

10 Click Analyse.

Fine-tune your analysis if necessary. For more information, see Stabilizing and Tracking (page 859).

11 Click Return.

The Garbage Mask menu reappears. The tracking data is applied to the mask.

12 Fine-tune your mask if necessary.

To animate individual vertices on a mask with tracking:

1 Select a vertex or a group of vertices on a mask.

2 Access the Garbage Mask menu.

The Garbage Mask menu appears.



(a) Clip box (b) Adjust box

3 From the Clip box, select the clip you want to track.

NOTE When tracking vertices on a mask using the Garbage Mask menu in the GMask node in Batch or Batch FX or the Modular Keyer, you can only track the front clip.

4 From the Adjust box, select Adj Tangents.

The tangents for the selected vertices are adjusted while the points are being tracked.

5 Go to the frame where you want to start tracking.

6 Click S.

The Stabilizer appears. The trackers are automatically placed in the position of the selected vertices. Reposition if needed.

7 Click Analyse.

Fine-tune your analysis if necessary.

8 Click Return.

The Garbage Mask menu reappears. The tracking data is applied to the selected vertices on the mask.

9 Fine-tune your mask if necessary.

Applying Motion Blur to Garbage Masks

Use motion blur on a garbage mask to compensate for the movement of an object in a clip. Motion blur is created according to the animated movement of a mask's vertices. The amount of blurring is affected by the speed of the mask's movement.

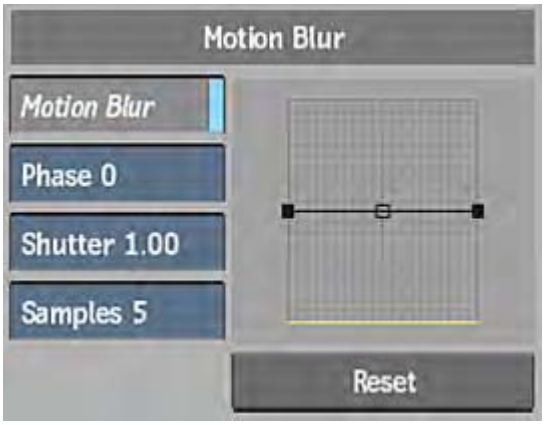
Motion blur can account for the natural blurring of an object as it moves or rotates in space. For example, assume you are drawing a garbage mask around a car as it turns a corner. Because you are using an overhead shot, the edges of the car where motion blur occurs change as the car accelerates and rounds the corner. By

applying motion blur to the mask, which has animated position and rotation values to match the car's movement, appropriate blurring of the car's edges into the background clip occurs.

TIP Besides the practical use of blurring a garbage mask, that is, to match the movement of an object you are keying, you can try blurring masks for artistic effects in multi-layered composites.

To apply motion blur to a garbage mask:

1 Access the Garbage Mask Setup menu.



2 Set the motion blur as follows.

Motion Blur button Enable to apply motion blur to garbage masks.

Phase field Specifies whether the motion blur is based on the movement before or after the current frame. A value of -100 places the motion blur before the frame, while a value of 100 places the motion blur after the frame. A value of 0 is centred, which evenly distributes the motion blur. The default value is 0.

Shutter field Controls the duration of the motion blur at each frame, which affects the size of the motion blur. Increasing Shutter value does not increase the processing time.

Samples field Determines the quality of the motion blur produced by the number of samples taken at each frame. Increasing the Samples value causes the processing time to increase linearly.

NOTE You can animate the Motion blur button, as well as the Phase, Shutter, and Samples fields. They can be found in the Channel Editor under the *motion_blur* folder.

Motion Blur curve Determines the transparency of the samples that create the blurring effect. The left side of the curve controls the samples of the incoming movement of the subject in the frame. The right side of the curve controls the samples of the outgoing movement of the subject in the frame.

For example, to add blur to the left edge of the mask as it travels from left to right in the clip, adjust the motion blur curve so it slopes down. If you want to add a bit of blur on each side of the mask, adjust the motion blur curve so it peaks in the middle.

Select one of the three vertices on the curve to move it. When you select a vertex, its tangent handle is displayed. You use the tangent handle to adjust the slope of the curve.

NOTE The garbage mask motion blur settings are similar to those in the Axis menu in Action.

Motion blur occurs on the mask according to the settings you select and the speed its vertices traverse the clip.

3 Enable Motion Blur. This makes the motion blur available for application on any garbage mask or axis in the setup.

NOTE Motion blur settings are applied uniformly to all garbage masks in the setup.

4 To reset all motion blur settings back to their default settings, click Reset and then click Confirm.

5 To apply motion blur to a garbage mask, access the Garbage Mask Shape controls, select the garbage mask, and then enable MBlur.

The settings specified in the Garbage Mask Setup menu will be used to create the motion blur effect on the garbage mask. You can select one garbage mask at a time.

6 To apply motion blur to an axis, select the axis, and then enable MBlur.

The settings specified in the Garbage Mask Setup menu will be used to create the motion blur effect on the axis. You can select one axis at a time.

Blurring a Stationary Garbage Mask

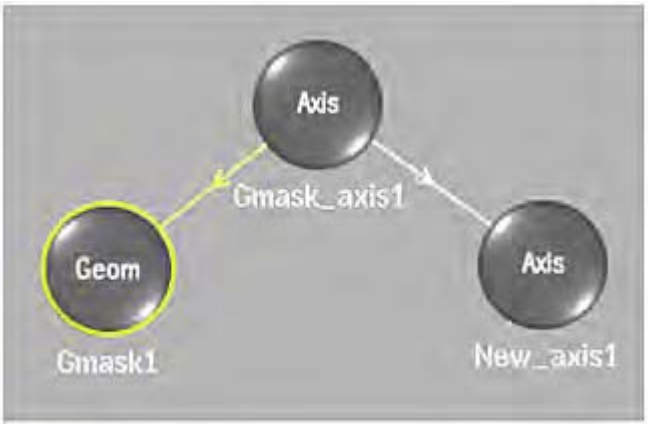
You can create a motion blur effect for a garbage mask without having the mask actually change position. This could be useful, for example, if you would like to simulate movement in a stationary image.

To apply axis motion blur:

1 Toggle to schematic view and add a new axis.

2 Parent the new axis to the garbage mask axis.

The garbage mask and the new axis should now both be parented to the garbage mask axis.



3 Apply motion blur to the new axis.

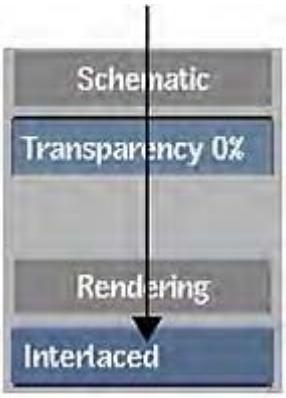
Motion blur is applied to the stationary garbage mask.

Rendering Garbage Masks in Interlaced Mode

Using the GMask node, you can render garbage masks in Interlaced mode to accommodate video material.

To render garbage masks in Interlaced mode:

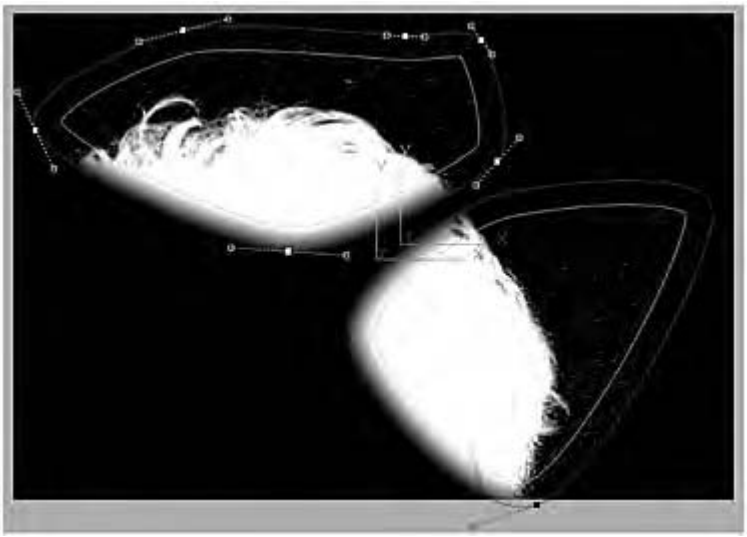
1 From the GMask node's menu, select Interlaced from the Rendering box.



Using Multiple Garbage Masks

Use the Region of Interest feature on clips containing multiple garbage masks that each hide a portion of the matte. Region of Interest ensures that multiple overlapping or non-overlapping garbage masks all respect the areas defined by one another. Furthermore, you can individually modify the opacity of each mask.

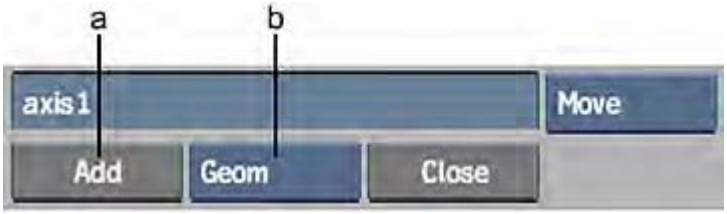
Region of Interest is only available when accessing the Garbage Mask menu from the GMask node in Batch or Batch FX.



Two garbage masks with Region of Interest enabled

To draw more than one garbage mask revealing a matte:

1 In the Garbage Mask menu, select Geom from the Object box and click Add.



(a) Add button (b) Object box

2 Draw the first garbage mask.

3 Switch to Result view.

4 Enable Region of Interest.

Outside is enabled and the Colour is set to 0.0 for the mask.

NOTE Disabling Region of Interest toggles the Colour setting back to what it was before.

5 Adjust offsets and softness as required.

6 To adjust the individual opacity for each mask, select the mask and then adjust the value in the Opacity field.

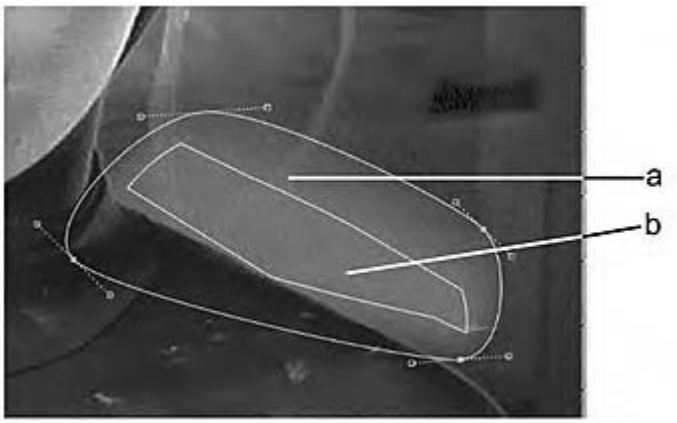
7 Draw any other garbage masks as needed.

NOTE Each time you add a garbage mask with ROI enabled, Result view is temporarily cleared of all masks so that you see the entire matte. This way you have a better view of what you want to reveal and mask.

Refining the Mask

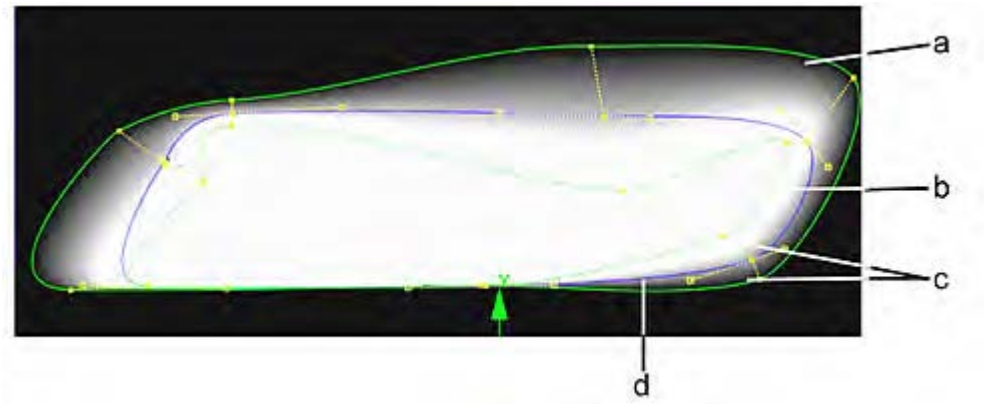
You can adjust the softness gradient of a mask to smoothen its edges. You can smoothen the gradient towards the inside edge, the outside edge, as well as the area where the inside and outside adjustments have an effect. You can create a uniform gradient around the edge of the mask, or use an advanced gradient to control the shape of the gradient at different parts of the mask.

To create a uniform gradient, you define how far you want the gradient to be offset from the edge of the mask and then set its transparency. To vary the shape of the gradient, you move vertices on inner and outer softness splines. The gradient will be based on how far each vertex point is offset from the mask.



(a) Alpha region (b) Inner offset

Advanced gradient provides a versatile method for setting the softness of the mask edge. It applies a gradient according to the distance of the softness borders from the garbage mask spline. It allows you to customize the softness gradient at different parts of the mask. The advanced gradient has two softness borders, one inside and one outside the mask border. It also includes inner and outer softness vertices for each regular mask vertex.



(a) Outer Softness border (b) Inner Softness border (c) Softness vertices (d) Mask border

Some potential uses for the advanced gradient are:

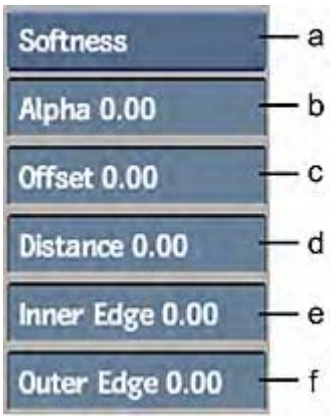
- Creating compositing effects when using Action
- Removing wires
- Pulling a key on areas of the mask where pickers are not needed

When using the Tracer in the GMask node in Batch or Batch FX or the Modular Keyer, you can work in two modes: advanced gradient and pickers. Individual vertices can be set to either of these modes.

You can change the default mode of the Tracer from advanced gradient to pickers, a system of localized keys. Tracer analyses the colour information both inside and outside the mask, then compares the values and uses a keying algorithm (the same one used in the 3D Keyer) to derive a key for the mask edge. This system allows you to key images with a lot of detail at the edges.

NOTE For good results with the Tracer, the object must have sufficient (and relatively consistent) chroma and luma differences between its foreground and background.

To control softness and advanced gradients, use the Garbage Mask Shape controls.



(a) Edge Softness box (b) Alpha field (c) Offset field (d) Distance field (e) Inner Edge field (f) Outer Edge field

Applying Softness Using a Uniform Gradient

Use the Softness fields to apply a constant range of softness to the edge of the mask. The Alpha and Offset fields define the fade-out of the softness gradient from the edge of the mask.

Once gradient transparency and offset have been determined, the degree of smoothness can be set using the Inner Edge and Outer Edge fields. You can then use the Distance field to determine the area where Inner Edge and Outer Edge settings have an effect.

To apply uniform softness:

1. Access the Garbage Mask menu.
2. Select a mask in the image window.
3. If necessary, toggle the Edge Softness box to Softness.

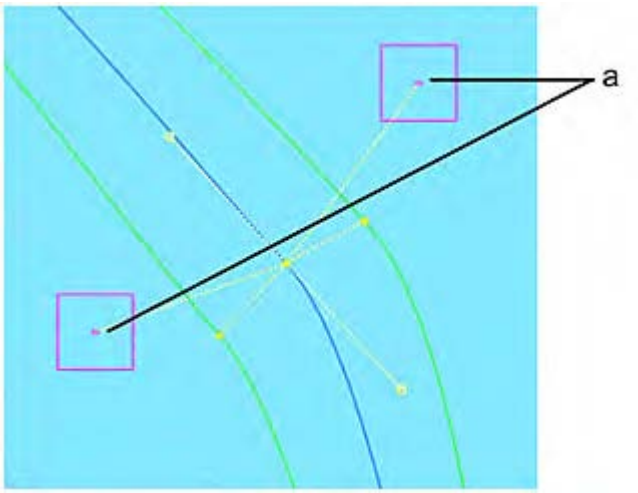


4. Set the border of the softness gradient using the Offset field. The gradient will be affected by how far the softness border is offset from the edge of the mask.
5. Set the transparency of the gradient using the Alpha field.
6. Use the Inner Edge field to smoothen the softness gradient towards the inside.
7. Use the Outer Edge field to smoothen the softness gradient towards the outside. Adjusting this value is especially noticeable if you are smoothening a transition from a black inside edge to a white outside edge.
8. Use the Distance field to modify the area of which the Inner and Outer Edge adjustments have an effect.

Applying Softness Using Pickers

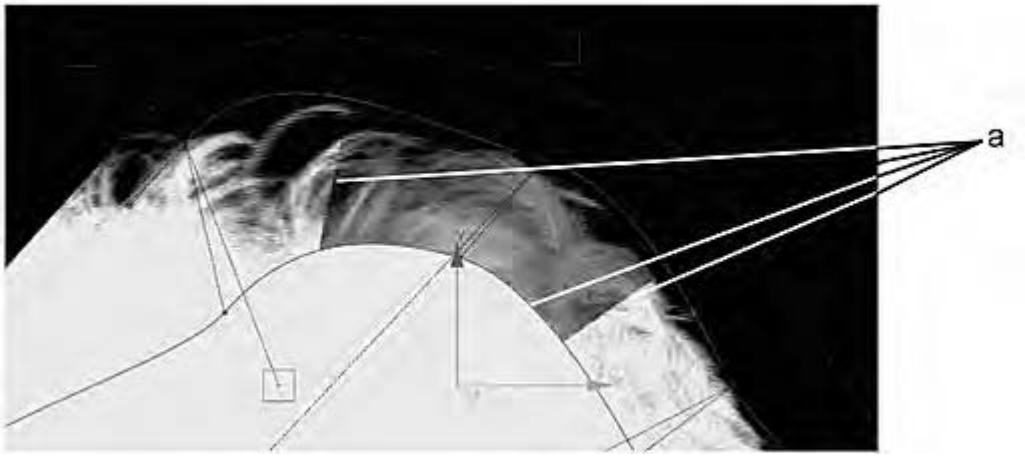
You can change the default mode of the Tracer from advanced gradient to pickers, allowing you to sample colours inside and outside the garbage mask spline, to apply softness according to the colour transition in the clip. Individual vertices can be set to either of these modes.

When great control over the mask edge is needed, for example, for fine edge detail, use pickers to effectively key out the background. The Tracer uses pairs of pickers to do luma and chroma analysis of the area inside and outside of the mask and derives localized edge keys from this information.



(a) Pickers—Colour analysis is done in the area inside the pickers.

The area that each picker affects extends halfway towards the two adjacent mask vertices, and up to the two softness borders, as shown in the following example.



(a) Borders of area of pickers' influence.

Using pickers, you can key objects that otherwise would be extremely difficult to key. Imagine a golden horse with a very fine mane on an unfocused background of various colours. As long as you have some chroma/luma differences in the background (green vegetation, blue sky, black earth or rocks), you can 'force' the outside pickers to sample these colour values. The inside pickers can sample the golden average values, and the Tracer can generate a soft-edged matte based on the difference between the two sets of values.

Pickers can be selected and manipulated independently of the vertices they are associated with. See Selecting Pickers and Softness Vertices (page 904).

Mixing Advanced Gradient and Pickers

When the subject has a mixture of fine, wispy edges and hard, clean edges, use a mix of localized keys and advanced gradients. Pickers are better for the fine edge areas and advanced gradient is sometimes better for the hard edge areas. You can set the state of each vertex to advanced gradient (pickers off) or localized key (pickers on) mode.

- To convert a section of a mask from advanced gradient to pickers, select one or more vertices with no pickers and enable the Picker button. Adjust the pickers as needed.

- To convert a section of a mask from pickers to advanced gradient, select one or more vertices with pickers and disable the Picker button.
- To toggle selected vertices between the two modes, use the 0 (zero) keyboard shortcut.

To completely remove gradient (for hard edges):

1. Select the mask vertices in the area of the mask border where you want to remove the gradient.
2. Click any one softness vertex corresponding to one of the selected mask vertices (either an inner or outer vertex).

   All the corresponding inner or outer softness vertices are selected.
3. From the Edit Mode box, select Scale.
4. Click any one of the selected softness vertices and drag until the softness border is lined up with the mask border.
5. Repeat for the second softness border (inner or outer).

To separate the softness border from the mask border:

1. Click a mask vertex, and then drag to the right to move the softness vertex out.

Applying Softness Using the Tracer

Drawing a Tracer-aided matte entails four basic steps.

To draw a Tracer-aided matte:

1. Analyse the image to determine which areas are appropriate for pickers and which areas are better served by advanced gradients.
2. Draw the garbage mask.
3. Assign pickers to vertices where they are needed.
4. Adjust the softness borders, offsets, tangents, vertices, and pickers (where used).

To analyse the image:

1. Examine the edges of your talent to decide which portions would benefit from advanced gradient treatment and which would respond better to localized key treatment. If your clip has colours that change greatly throughout the clip, using pickers may be difficult.

   For more information, see Applying Softness Using Pickers (page 918).

To draw the garbage mask:

1. In the Modular Keyer or Batch or Batch FX, add a context point further down the processing pipeline or processing tree. You can then use the Context view to see the effect of the matte on the result image.

   NOTE When using the Tracer to pull a key from the Modular Keyer, remove the nodes before the GMask node in the default pipeline.
2. Display the Garbage Mask menu.
3. From the View box, select Reference.

   This view is a reference image of the clip.

4 Draw a garbage mask around the subject and close it by clicking the first vertex drawn. Use as many vertices as required to adequately enclose the subject, but try to avoid excessive vertices. This will minimize unnecessary tweaking and manipulation later.

TIP Place more vertices in areas with variation in the background.

To assign pickers to selected vertices:

1 From the Edge Softness box, select Tracer.

The Tracer menu appears, and two softness borders (the green lines) are added to the mask.



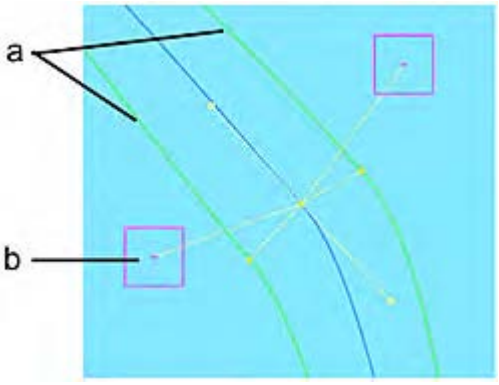
(a) Edge Softness box

2 Select the vertices to which you want to apply pickers. See Selecting Vertices and Tangents (page 903).

3 Enable Pickers.

NOTE When one or more pickers are enabled, mask characteristics are automatically set to Outside disabled and a Colour value of 0 so that the mask will be white inside and black outside. Do not set the Outside button unless you want to reverse the effect.

A pair of pickers is provided for each selected vertex.



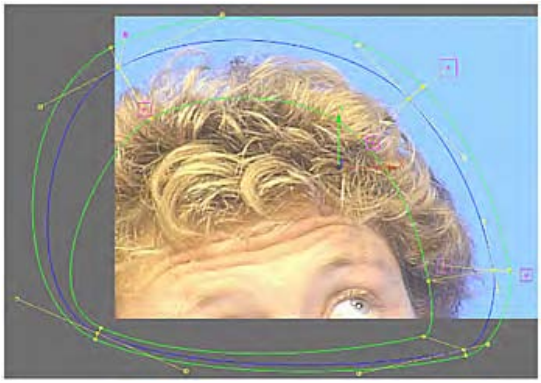
(a) Softness borders (b) Picker

- On vertices without pickers, the softness borders delineate the area where a softness gradient is applied.
- On vertices with pickers, the softness borders delineate the area to which softness is applied according to the picker values.

To fine-tune the matte:

1 Adjust the softness borders and mask border by moving the vertices and tangents:
- The mask border should follow the general contour of the subject as closely as possible.
- The outer softness border should completely surround all details that you want to include in the matte—all wispy details and edges must be within this line.
- The inner softness border should be well within the area where softness control is needed.





When moving tangents, you can opt to have the pickers follow the tangent movement (the default behaviour), or be independent of tangent movement. To make pickers independent of tangents, switch to Break mode and click either picker. You can animate this behaviour. See Animating a Tracer Mask (page 923).

NOTE If the tangents are broken, the pickers are automatically unlinked from the tangents.

2 For each pair of pickers, one requires a sample of the area outside the mask, while the other requires a sample of the mask interior.

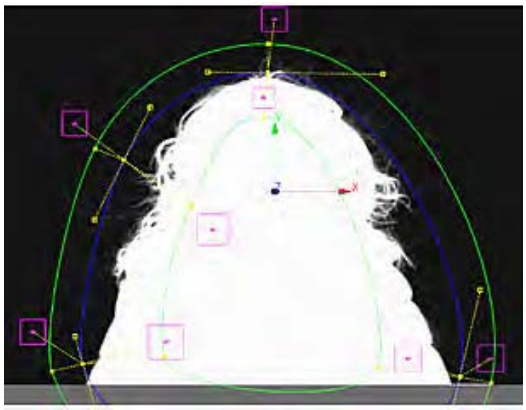
Place one picker outside the subject to sample values you do not want to include in the matte (for example, a blue screen). Place the other picker within the subject for a colour value sample of an area you do want included in the matte. Try to select areas where the colour values do not change too much throughout the clip.

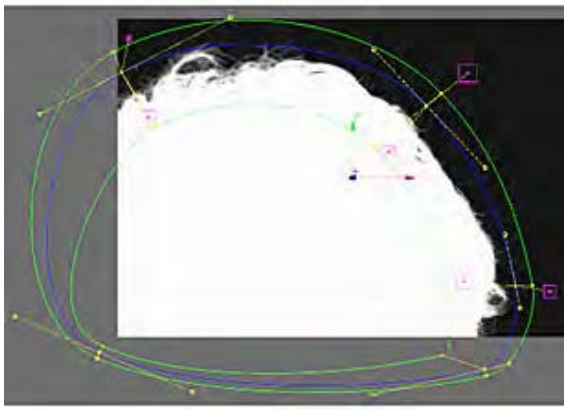
3 Click Matte or Result (Matte will provide a clearer view) to see your progress.

4 Scale the pickers by doing one of the following:
- Select Scale edit mode, click within the picker, and drag to the left to reduce the size or to the right to increase the size.
- Select the picker and then drag the cursor over the Sample Size field.

5 Move the pickers around and enlarge or reduce them to interactively see how you can obtain the best results. This will require toggling between the Front and Matte views.

The following figure illustrates the Matte view result of the pickers placed in the previous figures.





Animating a Tracer Mask

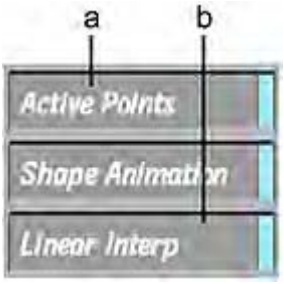
When using the Tracer, you can animate the basic vertex parameters, as well as the following additional parameters.

Parameter	Channel Folder and Name(s)	Channel Values
The position of the two softness vertices relative to the mask vertex.	border: upper = outer vertex* bottom = inner vertex*	
Whether pickers are linked to tangents or not. Linked—Pickers move with tangents. Unlinked—Pickers remain in their current position when tangents are moved.	pickers: fixed	0 = unlinked 1 = linked
The on/off status of pickers.	pickers: active	0 = off, 1 = on
The position of the pickers relative to the mask vertex.	pickers: upper = outer picker* bottom = inner picker*	
The picker size.	pickers: u-size = outer picker size* b-size = inner picker size*	
The Sample on/off status of pickers. See Animating Picker Values (page 924).	sample: active	0 = Sample off 1 = Sample on
The interpolation mode for picker values when Sample is off. See Animating Picker Values (page 924).	sample: interp.	0 = Constant 1 = Linear

*Channel Editor designations for “upper” and “bottom” assume the mask was drawn clockwise.

Animating Picker Values

Two options are available to control how picker values (that is, the sampled colour values) are animated: Sample On and Interpolation mode.

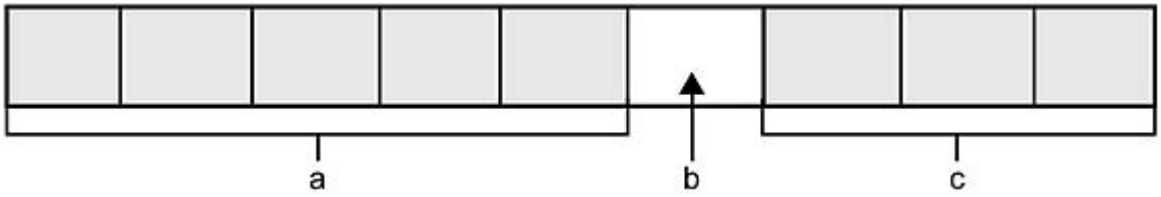


(a) Sample box (b) Interpolation button

Sample box Active resamples picker values at every frame (the default). Passive disables resampling for one or more frames. When Passive at a particular frame, the picker values previously sampled at another frame are used. It is useful to adjust the size and position of pickers at one particular frame until the result is optimal, and then apply these values to other frames. In this way, if movement in the clip in other frames causes the area being sampled to move away from the pickers, you do not need to readjust the pickers to get a good reading.

You can select Active or Passive for one or more selected pickers.

Selecting Passive works best when colour values remain fairly consistent throughout the clip. In this case, one picker sampling will often be enough for the entire clip.



Typical technique for using Sample On: one sample is used for the entire clip

(a) Sample Off (b) Sample On (c) Sample Off

When there is more colour variation in the clip, you may need to take samples at several frames.

To use one picker sampling throughout a clip:

1. Choose a frame in which the colour values are representative of the average colour values found in the clip.
2. Select the pickers you want to sample. See Selecting Pickers and Softness Vertices (page 904).
3. At that frame, make sure Active is selected from the Sample box (the default).
4. Set the picker values by setting their position and size until you get the optimal result.
5. Go to the next frame and select Passive from the Sample box.

   The sample values read at the frame you chose will be used to compute the softness for all other frames in the clip.

To take samples at several frames:

1. Follow the steps in the previous procedure to set a picker sampling for the clip.

2 Starting at the first frame sampled, examine your results frame by frame (either forward or backward through the clip).

3 When you come to an unsatisfactory result, select the pickers you want to re-sample and select Active.

4 Adjust the picker position and size until you get a good result.

5 Go to the next frame and select Passive from the Sample box.

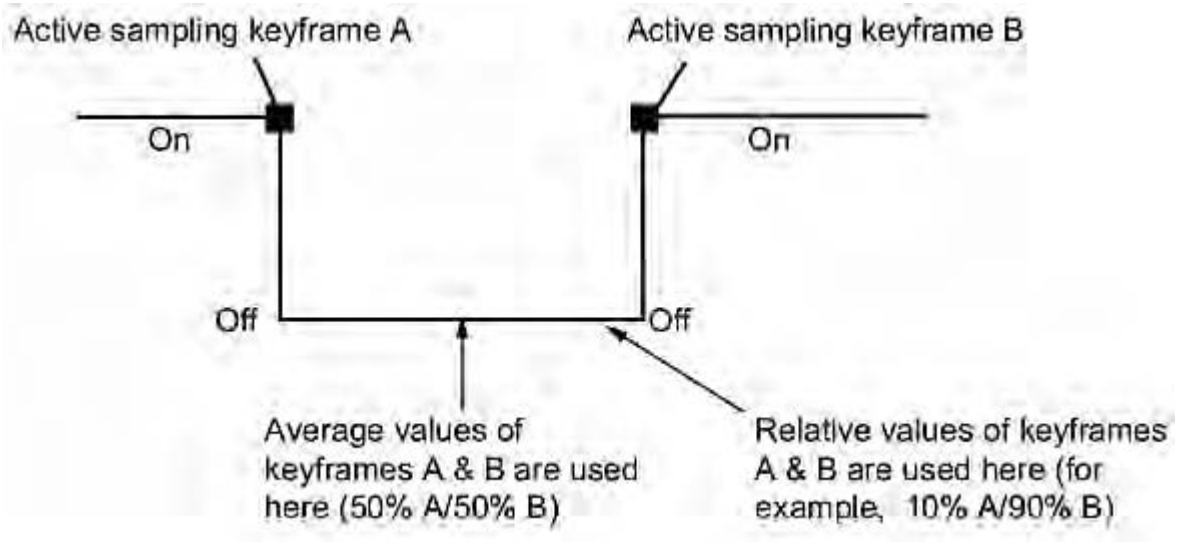
6 Continue verifying frames until you come to one that needs to be adjusted.

7 Repeat steps 3-5.

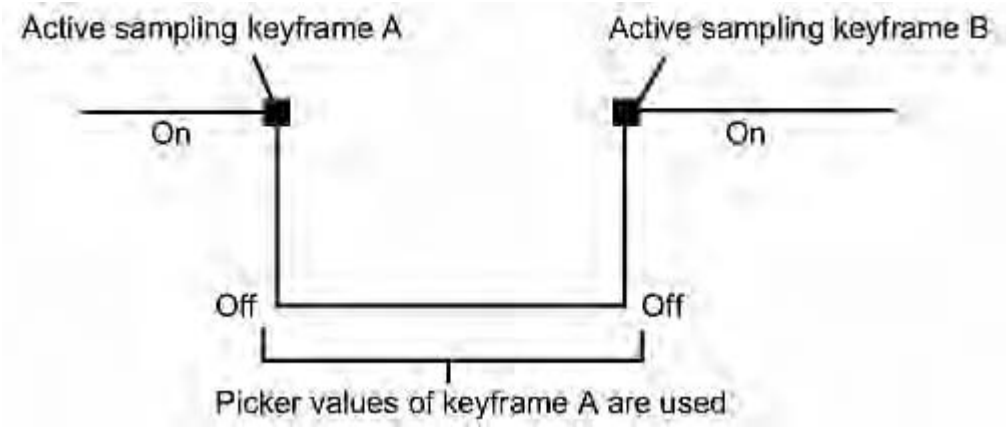
8 Repeat for the remainder of the clip.

Interpolation button Displays the interpolation between keyframes that have Active selected as the Sampling option.

- Linear (enabled): Picker values are interpolated between keyframes with active sampling. The picker values for each frame are computed based on the values of the previous and next active sampling keyframes and the proximity of the frame to those keyframes.



- Constant (disabled): Picker values are fixed between keyframes with active sampling. This means that the picker values at the first active sampling keyframe will be used for all subsequent frames up until the sampling is active again.



Tracking with the Tracer

Tracking with the Tracer is done using the same method as with regular garbage masks. See Animating Masks Using Tracking (page 911).

NOTE When animating selected vertices with the Stabilizer, the vertices do the tracking, not the pickers. Once you obtain the tracking data, you will need to go back through your clip from frame 1 onwards, making adjustments to the pickers where necessary.

Troubleshooting the Tracer

Every clip has its own challenges and requires a certain amount of tweaking and fine-tuning to achieve the best results.

Problem	One of the Tracer segments (localized keys) on my matte shows noisy black and white artefacts and nothing else. What's wrong?
Possible Cause	The two pickers are sampling luma and/or chroma values that are too similar.
Solution	Try moving one of the pickers to a differently coloured area, or enlarge or reduce the size of one or both of the pickers to include more varied colour information. If there is simply no area in the clip that differs enough, you will need to use an advanced gradient for that segment.
Problem	One of the Tracer segments (localized keys) on my matte appears to be showing the black/white matte information reversed, or as a negative. What's wrong?
Possible Cause	You may have reversed the position of the two pickers.
Solution	Try exchanging the inside picker for the outside one and vice-versa.
Problem	I can't get a good colour sample from the area outside the mask edge in a certain portion of the image. What can I do?
Possible Cause	There is simply none (or not enough) of the needed colour in the immediate area of that particular picker.
Solution	You can drag the picker somewhere further away from the vertex to get a better colour sample. The picker can be located anywhere on the image in order to facilitate the best possible colour sample. Do not, however, place a picker outside the image.
Problem	I'm getting really nice edge detail on my matte, but I'm also getting black/white holes in my matte. Is there anything I can do?
Possible Cause	This may be caused by the generation of random noise and/or artefacts.

Solution 1	Try the Clean algorithm by clicking the box labelled Clean. You may want to try several different percentages to find the optimal level of cleaning that doesn't effect your edges (the default percentage is 70%). The Clean algorithm works only with the Tracer and pickers on. If you have advanced gradient segments, they will be unaffected. The Clean command works on all vertices with pickers, whether or not they are selected. The area affected by the Clean command is the same area that the pickers affect (see Applying Softness Using Pickers (page 918)).
Solution 2	Use the inner border to isolate the problem areas.
Solution 3	Add a second garbage mask.
Problem	I'm repositioning and rescaling the pickers and the matte goes from having too much softness (overly transparent) to having too little (overly opaque). What can I do?
Possible Cause	The sampled colour range is either too broad or too narrow.
Solution	Locate the areas that have a tendency to become too softened and the picker that is 'responsible' for this by moving the pickers until the matte results change for the worse. Include a larger portion of these softer areas in a picker box. By sampling a fuller range of chroma/luma in this area, a more uniform, averaged effect is created.

Saving Garbage Mask Setups

Garbage masks can be saved and loaded as setup files, with all vertex and animation information preserved.

To save garbage masks as a setup:

1 In the Garbage Mask menu, click Save to open the file browser.



2 From the GMask Type box, select Save GMask.



3 Type the name of the setup, and click Save.

Loading Garbage Mask Setups

In the file browser, when loading garbage mask setups, you can quickly navigate to either the default gmask directory or a custom directory for the current project.

In Action, you can load a garbage mask as a 3D model. See Using Garbage Masks as 3D Geometry (page 549).

To navigate to the default or custom directory when loading garbage mask setups:

1 Select Default or Custom from the GMask Directory box.



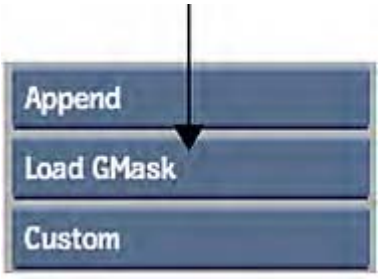
NOTE The custom directory is updated every time you navigate to a new directory.

To load a garbage mask setup:

1 In the Garbage Mask menu, click Load to open the file browser.

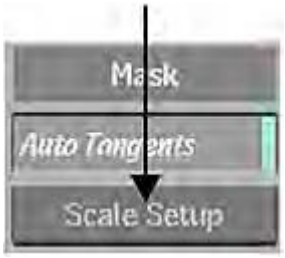


2 From the GMask Type box, select Load GMask.



3 From the file browser and select the name of the setup to load.

If the GMask setup you are loading was created in a different resolution than the current project, click Scale Setup to scale the GMask.



NOTE The Scale Setup button is available on the GMask Setup menu only when accessed from the GMask node in Batch or Batch FX or the Modular Keyer.

Preset Garbage Mask Setups

A library of preset garbage mask setups is available. To load a preset garbage mask, navigate to the */usr/discreet/<product home>/gmask* directory and select one of the following:

- 4_point_star
- 5_point_star
- 6_point_star
- arrowhead
- cat_eye
- circle
- diagonal
- diamond
- heart
- hexagon
- keyhole
- oval
- pentagon
- rectangle
- round_rectangle
- s_shape
- square
- triangle
- v_shape

You can also view proxies for each of these setups using the file browser.

Importing and Exporting Raw Setup Files

You can import and export raw setup files for animated garbage masks. A garbage mask exported as a raw setup file includes the following information:

- The number of keyframes in the animation of the spline
- The number of vertices in the shape
- The x, y, and z position of every vertex at each keyframe

To export a raw setup file:

1. In the Garbage Mask menu, click Save to open the file browser.
2. From the GMask Type box, select Export RAW.
3. Type the name of the file to be exported, and click Save.

To import a raw setup file:

1. In the Garbage Mask menu, click Load to open the file browser.
2. From the GMask Type box, select Import RAW.
3. Navigate to the appropriate directory and select the raw setup file to import.
   The raw setup file is imported into the Keyer.

To import a raw setup file in the Modular Keyer:

1. In the Modular Keyer, click the GMask node in the processing pipeline. See Accessing the Garbage Mask Menu (page 893).
2. In the GMask menu, click Load.

   The Load menu and file browser appear.
3. In the Load menu, select Import Raw and Default from the Load option boxes.



4. Enable Append or Replace.
5. In the file browser, specify the name and path of the file to import.

To export a raw setup file in the Modular Keyer:

1. In the Modular Keyer, click the GMask node in the processing pipeline. See Accessing the Garbage Mask Menu (page 893).
2. In the GMask Setup menu, click Save.

   The Save menu and file browser appear.



(a) Save option box

3. In the Save menu, select Export Raw from the Save option box.
4. In the file browser, specify the name and path of the file to export.
5. Click Save.

Creating Customized Wipes with Garbage Masks

Use masks on the incoming clip to create a customized transition that wipes into the outgoing clip.

Setting the Mask Drawing Options

Before you create a mask, set how the points of the mask are drawn.

To set the drawing options:

1. From the Wipe Editor, click Setup.

2 Specify the setup options.

Enable:	To:
Auto Tangents	Create tangents for every new point you set.
Show Border	View the border defined in the Offset field of the Shape menu. The colour pot next to the Show Border button defines the colour of the border.
Invert	Reverse your wipe. For instance, if you originally create an expanding box wipe, enabling Invert creates a shrinking box.
Spline Keyframing	Allow animation of points on the mask.

3 To change the colour of the mask's wireframe, colour points, or offset border, click a colour pot and pick a colour.

Colour Correcting 21

About Colour Correcting

The Colour Corrector includes tools that provide precise control over colour values. You can modify luma ranges in a clip (shadows, midtones, and highlights), sample colours, and adjust the colour balance. You can rewire colour channels and suppress colours, as well as animate a colour correction by manipulating the animation curve in the Channel Editor.

Accessing the Colour Corrector

To access the Colour Corrector, you must load clips of the same resolution. If the clips you want to load have different resolutions, resize them so that they have the same resolution.

You can load a front clip, a front and back clip, or a front, back, and matte clip for colour correction. Changes in colour are applied to the front clip.

The Colour Corrector can be accessed from the following locations:

- Timeline, then use a Colour Corrector timeline effect. (page 933)
- Batch, then select a node from the Node bin. (page 934)
- Modular Keyer, Action, Paint tool. (page 934)
- Timeline, then use Batch FX. (page 934)

Accessing the Colour Corrector from the Timeline

To access the Colour Corrector from the Timeline:

1. Select Timeline.
2. Right-click on the segment that will be colour corrected.
3. Select Add Effect.
   The Video Effects ribbon appears.
4. Enable CC (colour correct).
5. Click Enter Editor...

You are in the Colour Corrector.

Accessing the Colour Corrector from Batch

To access the Colour Corrector from Batch:

1. Double-click on, or drag, the Colour Correct node from the node bar and place it in the schematic. Make sure you connect to other nodes or clips in the process tree.
2. Click the Colour Correct node.

   The Colour Correct Node editor appears.

Accessing the Colour Corrector from Modular Keyer, Action, or Paint

You can colour correct clips loaded into the MOdular Keyer, Action or Paint.

Back clips and key-in clips cannot be colour corrected.

To access the Colour Corrector from the Modular Keyer:

1. Double-click on, or drag, the Colour Correct node from the node bar to the schematic.
2. Double-click the Colour Correct node.

   You are in the Colour Correct editor.

To access the Colour Corrector from Action:

1. From the Media menu, select the media containing the clip you want to colour correct.
2. Double-click the CC field for the media you want to colour correct.

   The clip is loaded into the Colour Corrector.

To access the Colour Corrector from Paint:

1. Do one of the following:
   - To colour correct a clip, click Setup in the Paint menu, click CutOut, enable CC, and then enable the CC button for the clip you want to colour correct.

     The clip is loaded into the Colour Corrector.
   - To colour correct a cutout, click Cut/Paste in the Paint menu, create a cutout, and then use the Colour Correction field or click CC.

     The cutout is loaded into the Colour Corrector.

Accessing the Colour Correct from Batch FX

To access Colour Correct through Batch FX:

1. Select Timeline.
2. Select the segment.
3. Click the FX button.
4. Click the Create Batch FX button.
5. Double-click on, or drag, a Colour Correct node into the schematic.

The Colour Correct node is now in the schematic.

6 Double-click the Colour Correct node.

You are in the Colour Correct node editor.

Hiding the Colour Corrector Menu

You can hide the Colour Corrector menu while adjusting an image if you find the menu distracting. With the menu hidden you can modify numeric fields, use the colour wheel, or access buttons via their corresponding keyboard shortcuts.

To hide the Colour Corrector menu while modifying a numeric field or the colour wheel:

1 Click and hold any numeric field or the colour wheel. For example, click and hold the Hue field.

2 Press `Alt+'` (on the Tilde key) while continuing to hold the cursor.

The Colour Corrector menu is hidden.

3 Drag the cursor to modify the field value or adjust the colour wheel.

The changes you apply will be visible in the image as you drag if Regen is enabled (see Dynamic Updating (page 941)). Otherwise, the changes appear when you release the mouse.

4 Press `Alt+`` (on the Tilde key) to display the menu.

To hide the Colour Corrector menu while accessing buttons via their corresponding keyboard shortcuts:

1 Press `Alt+`` (on the Tilde key).

2 Use the Colour Corrector keyboard shortcuts.

3 Press `Alt+`` (on the Tilde key) to display the menu.

Overlay User Interface

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 Colour Corrector or Colour Warper from Tools. It is not available when accessing the Colour Corrector or Colour Warper as a node or as a timeline effect.



To toggle between the Overlay user interface and the regular user interface:

1 Press `Ctrl+`` (on the Tilde key).

To hide the Overlay user interface:

1 Press `Alt+`` (on the Tilde key).

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 tool, 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 keyboard shortcuts.

Press:	To:
`Ctrl+Alt+(num pad)+`	Increase the opacity of the Overlay user interface.
`Ctrl+Alt+(num pad)-`	Decrease the opacity of the Overlay user interface.

TIP You can repeatedly press the keyboard shortcuts to increase or decrease opacity, or simply hold down the keyboard shortcuts until the desired opacity is reached.

Enable Invert (or press `Ctrl+Alt+(num pad)*` to switch the Overlay user interface outlines and text from white to grey.

Viewing Clips

You can view a front, back, matte, or result clip in the image window. The front, back, and matte clips show the source clips used. The result clip shows your colour corrections as you apply them.

You can also view the following information for the clip:

- Channel: displays the numerical data used by the components in the clip.
- Reference: displays any referenced clips.
- Tracks: displays when an effect is used in the timeline
- Info: lists the channel, components, any related values, and the number of keys used in each effect in the clip.

To display a clip:

1 From the View box, select the clip you want to view. For example, select Result to view the result clip.

TIP Use the controls below the image window to change the frame displayed in the image window. Use the viewing tools in the upper-right corner of the panel to enlarge and pan images.

To change the clip that is displayed using the keyboard shortcuts:

1 Consult the following table.

To display:	Press:
The front clip	`F1`
The back clip	`F2`
The matte clip	`F3`

To display:	Press:
The result clip	F4
The channel data	F5
The track timeline	F5
The Info table	F5
The Referenced clips	Select Reference from the View box

Colour Corrector Settings

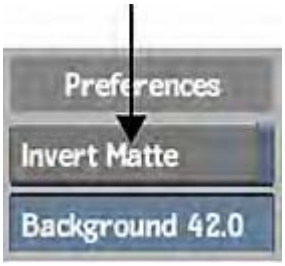
In the Colour Corrector Setup menu, you can change matte clip settings, the way that colour changes are updated, background brightness, and cropping. You can also reset colour corrections made from the Colour Corrector, Colour Warper, or both.

Inverting the Matte

You can invert the matte clip. Inverting the matte swaps the transparent and opaque areas of the matte and redefines the area of the front image that can be colour corrected, as well as the area of the back image that is shown.

To invert the matte clip:

1. Click Setup in the Colour Corrector menu.
2. Enable Invert Matte.



Changing the Background Brightness

You can adjust the brightness of your work area background. Changing the background brightness is a global setting. Changing the brightness through the Colour Corrector menu has the same effect as changing it through the Flame Premium Preferences menu.

To change the background brightness:

1. Click Setup in the Colour Corrector menu.

2 Enter a value in the Background field.



Creating a Crop Box

Use a crop box to view the colour changes to a limited region of the front clip. Colour changes only appear in the area within the crop box. This feature speeds up processing and lets you see the difference between the source clip and the result clip.

For example, while colour correcting, you can use the crop box to split the image window. The split window allows you to view the front clip and the result or matte clip at the same time.

To create a crop box:

1 Do one of the following:
- Hold down the `Ctrl` key or hold the pen button and drag the box across the image.
- From the Setup menu, enable the Crop button and enter the dimensions for the crop box in the Left, Right, Bottom, and Top fields.



The crop box appears in the image window. The display inside and outside the crop box depends on how many clips are loaded.

If you loaded:	Inside the crop box:	Outside the crop box:
A front, back, and matte clip	Shows the front clip in the opaque area of the matte and the back clip in the transparent area of the matte.	Shows the back clip.
A front and back clip	Shows the front clip.	Shows the back clip.
A front clip only	Shows the colour-corrected clip.	Shows the non-colour corrected clip.

2 To adjust the size of the crop box, drag on the corner points of the box, or modify the values in the Left, Right, Bottom, and Top fields in the Setup menu.

3 To move the crop box in the image window, click a side of the box and drag to a new location. Alternatively, press `Ctrl` and redraw the crop box.

TIP If Overlay user interface is enabled, the crop box will extend beneath the Overlay user interface. To access the bottom edge, be sure to grab from an area where there are no user interface elements.

To disable a crop box:

1 From any Colour Corrector menu, disable the Crop button.

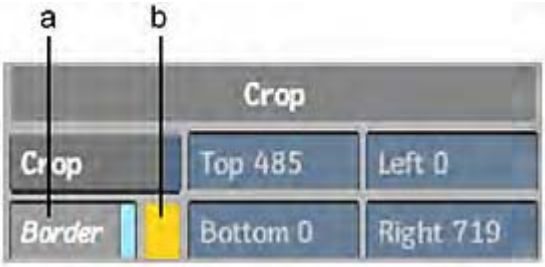
When you disable the Crop button in a Colour Corrector menu, it is also disabled in the Setup menu.

Changing the Colour of the Crop Box

You can change the colour of the crop box so that it is easier to distinguish from the rest of your image.

To change the colour of the crop box:

1 In the Colour Corrector Setup menu, click the colour pot beside Border.



(a) Border button (b) Colour pot

2 Select a colour with the colour picker.

3 Click in the Border colour pot to apply the new colour to the border.

Hiding the Crop Box

You can hide the crop box to prevent it from affecting your colour correction.

To hide the crop box:

1 In the Setup menu, disable Border.

Dynamic Updating

Use the Regen button to enable dynamic updating of changes you make in the Colour Corrector. When Regen is enabled, you see the changes interactively as you adjust the values in the various displays. When Regen is disabled, you do not see changes until values are entered after releasing the cursor, or by clicking the `Enter` key in the calculator.

To enable dynamic updating:

1 Select **Setup ➤ Master**.

2 Enable Regen.



Viewing Reference Clips

You can use the split bar to view and sample a reference clip while you work. Reference clips are particularly useful when you want to match colours or compare colour corrections.



(a) Focus clip (b) Split bar (c) Reference clip

TIP If the Overlay user interface is enabled, the split bar can extend beneath the Overlay user interface. To access it, grab from an area where there are no menu elements.

Resetting Colour Corrections

You can reset colour corrections created with the Colour Corrector, the Colour Warper, or both. You can also reset the corrections you make inside the Colour Corrector using the Reset Basics button, or by using the Range, Histogram, or Curve graphs.

To reset a colour correction:

1 Select an option from the Reset box.



Select:	To reset:
Reset All	The entire tool. Both the Colour Corrector and Colour Warper settings are reset to their default values.
Reset CC	The Colour Corrector. Settings in the Colour Warper are not affected.
Reset CW	The Colour Warper. Settings in the Colour Corrector are not affected.

NOTE If accessing the Colour Corrector or the Colour Warper as a timeline effects, only Reset All is available.

2 Click Confirm.

To reset a colour correction within the Colour Corrector:

1 Select one of the reset buttons depending on what you want to reset.

Select:	To reset:
Reset Basics	The basic properties of the Colour Corrector. Any changes made using the Curve, Histogram, or Range charts are unaffected.
Reset in the Curve menu	The settings defined by the Curve chart. The Basic properties of the Colour Corrector are unaffected.
Reset in the Histogram menu	The settings defined by the Histogram. The Basic properties of the Colour Corrector are unaffected.
Reset in the Range menu	The settings defined by the Range chart. The Basic properties of the Colour Corrector are unaffected.

Saving Colour Correction Setups and Preferences

You can save or load colour correction setups using the Save or Load button. Any setup can be loaded from the file browser for use with another set of clips. You can also save or load colour correction preferences using the Load or Save button.



- To save setups or preferences, click Save.
- To load setups or preferences, click Load.

  TIP Click Revert to revert to the last saved setup. All changes made since the previous Save operation are undone.

When you save colour correction setups, you save the current settings for both the Colour Corrector and the Colour Warper to the setup file. If you overwrite a setup file using the Save button, the current settings in both the Colour Corrector and Colour Warper are saved.

Colour Sampling

You can sample colours from the front and back clips, and instantly see how the front clip will be affected by the colour correction.

Colours are sampled using colour patches. The sampled colours appear in the patches with their colour values. You can display the RGB, HLS, or Y (luma) values for the sampled colours. You can choose to display numerical sample data as either RGB bit values or percentages.

To sample a colour:

1. Below the colour patches, select the sample option you want to use.



Select:	To:
Y	Display the NTSC luma values for the colours sampled. Either NTSC 601 or HD 709 luma values are displayed, depending on the settings in the configuration file.
HLS	Display the hue, lightness, and saturation values for the colours sampled.
RGB	Display the red, green, and blue values for the colours sampled.
Off	Turn off the colour information display.

2 Click the Front colour patch to sample a colour from the front clip. Click the Back colour patch to sample a colour from the back clip. Click both patches to sample front and back clips simultaneously.

The cursor becomes a colour picker.

3 To sample an individual pixel, click a colour in the image.

4 To sample an average colour, `Alt`-drag in the image, or press `Ctrl` and draw a selection box.

TIP Zoom in for more precise colour sampling.

The sampled colour appears in the colour patch. The sampled colour also appears in each menu.

In the:	A sample is displayed:
Colour Wheel	As a reference point on the colour wheel. A black reference point indicates the colour you sampled from the front clip. A white reference point indicates the sample from the back clip.
Histogram menu	As a red vertical line on the gradient bar, indicating the luma value. Only the front sample is displayed.
Curves menu	As red, green, and blue values mapped to the colour curves.
Ranges menu	As a red vertical line on the Luminance gradient bar, indicating the luma value. Only the front sample is displayed.

Colour Sampling and Processing Order

You can use any combination of colour correction commands to modify the front clip. The commands are processed in the following order:

- RGB Rewiring, Monochrome, Negative
- Hue Shifting
- Saturation
- Colour Balancing
- Gamma, Gain, Offset, and Contrast
- Histogram
- Colour Curves
- Chroma Suppression

The processing order is important in determining the colour that appears in the middle (result) section of the colour patch. The middle section displays the colour resulting from the application of all colour correction setups; in other words, the colour at the end of the processing order. This colour is updated as you change any parameter value. The original front colour appears in the left section of the patch.

Matching Colours

You can match and colour correct the colours of the front clip using the back clip as a reference. To perform a match operation, sample colours from the front and back clip and then use the Match button in the Curves menu.

To match colours between two clips:

1. Load a front and back clip from the Thumbnail view.
2. Use the split bar to view the front and back clips in the image window.
3. Click the Curves tab to display the Curves menu.
4. Click the Front colour patch.



The colour picker appears.

5 Using the colour picker, select a colour in the front clip to be corrected.

The colour is added to the Front and Result colour patches.

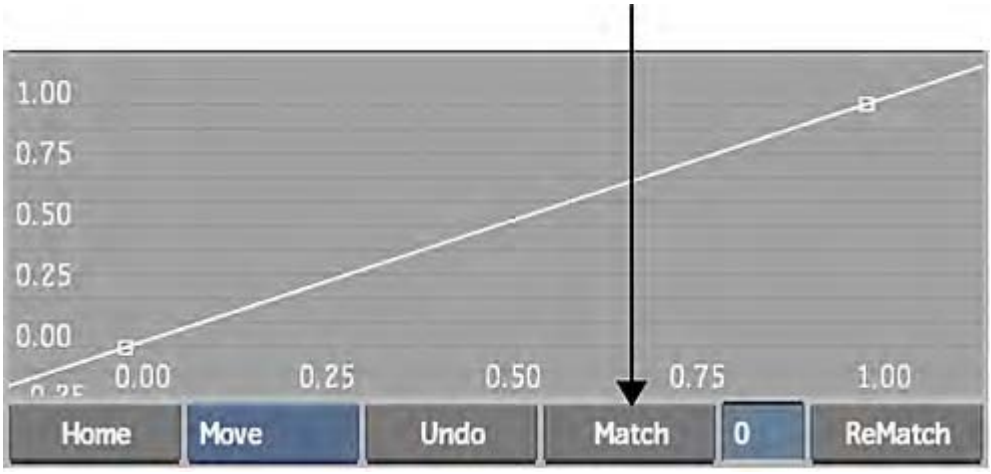
6 Click the Back colour patch.



The colour picker appears.

7 Using the colour picker, select a colour in the back clip to use as a reference colour.

8 Click Match.



The reference colour is applied to the result clip. The reference colour is also added to the Result colour patch and points are added to the colour curves.

When you use Match, a number is displayed in the numerical field beside the Rematch button. The first match you perform is indicated by a 1 in the field, the second by a 2, and so on. To revert to a previous match, click the numerical field and type the number of the match you want to recall. The colour patches and colour curves reset to the values stored in the specified match.

To reset the red, green, blue, or luminance curve, select a curve and click Reset in the Curve menu. The Curves menu also has its own Undo button that is separate from the overall Colour Corrector Undo.

Colour Correcting

Use the colour correction controls to perform many operations. You can:

- Colour correct shadows, midtones, and highlights separately or together.
- Adjust the hue, saturation, or contrast pivot point of an image.
- Rewire the red, green, or blue channels of an image.
- Create a negative or monochrome image.
- Adjust the gamma of the individual red, green, blue, or RGB channels.
- Adjust the red, green, blue, or RGB values in an image by modifying the gain, offset and contrast values for the channel.
- Perform colour suppression.
- Adjust the colour balance.

To reset any of these changes, click Reset Basics. To reset all the changes in the Colour Corrector, click Reset All.

Colour Correcting Shadows, Midtones, and Highlights

When you modify Saturation, Gamma, Gain, Offset, and Contrast, you can colour correct the shadows, midtones, and highlights ranges in the image individually or all together.

To colour correct shadows, midtones, or highlights:

1 Select the luma range you want to modify.



Click:	To modify:
Shadows	Dark areas in the image.
Midtones	Midlevel areas in the image.
Highlights	Light areas in the image.
Master	The entire image. Modification made in the Master range are applied after individual range modifications.

NOTE Monochrome, Negative, Chroma Suppression, and RGB Rewiring always affect the Master tonal range.

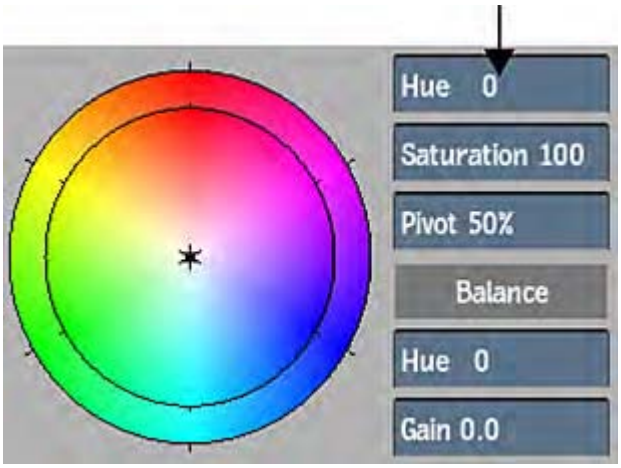
Adjusting the Hue

Hue is the main attribute that distinguishes one colour from another. When you adjust the hue, you also change the colours in the image.

NOTE Hue shift can be applied to individual ranges (highlights, midtones, and shadows) or to the Master tonal range.

To adjust the hue of an image:

1. Select the range you want to modify: Master, Shadows, Midtones, or Highlights.
2. Drag the Hue field to adjust the level of hue and saturation until you are satisfied with the result.



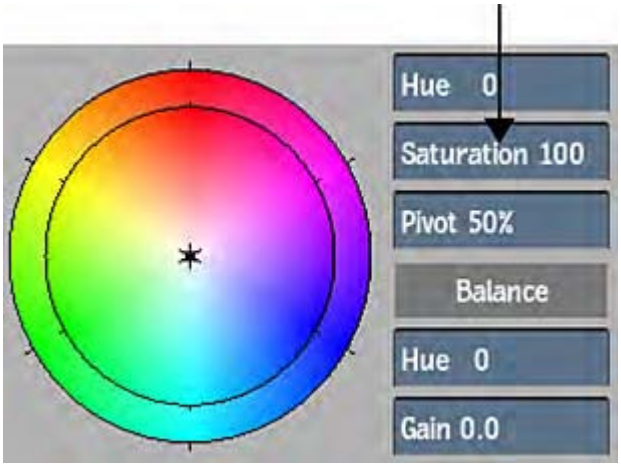
3. Click Process to apply the changes to the clip.

Adjusting the Saturation

You can adjust the purity of colour in an image by adjusting the saturation level. Increasing the level of saturation decreases the amount of grey and produces purer colours. Decreasing the level of saturation increases the amount of grey and reduces the purity of colours.

To adjust the saturation of an image:

1. Select the range you want to modify.
2. Drag the Saturation field to decrease or increase the level of saturation until you are satisfied with the result.



3 Click Process to apply the changes to the clip.

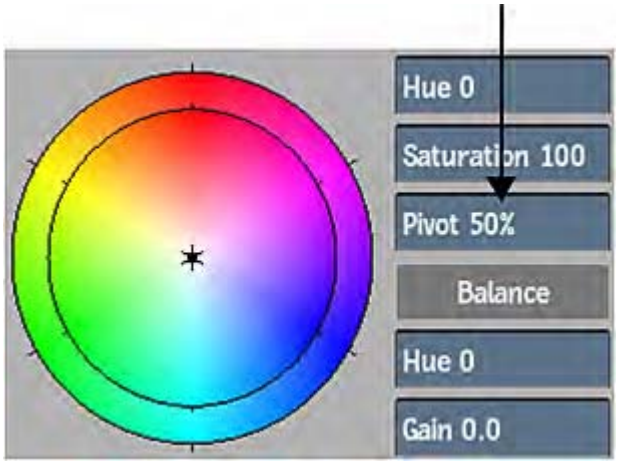
Adjusting the Pivot

Adjust the point at which contrast pixel value is equal to that of the pivot value. Colour values darker than the pivot are further darkened and colours that brighter than the pivot are further brightened.

To adjust the pivot in an image:

1 Select the range you want to modify.

2 Click the Pivot field and adjust the pivot percentage value of the colour range around which the contrast pivots. The default value is 50% for integer input and 18% for floating point input.



3 Click Process to apply the changes to the clip.

Rewiring the RGB Channels

Use the Red, Green, and Blue Channel Rewiring fields to rewire the red, green, and blue channels of an image. Use the Rewire option box to create a monochrome or negative image.

When you rewire a channel, the values for the current colour channel are replaced with those of the new channel. For example, if you select the R ← G option in the Red Channel Rewiring field, the colour values of the red channel are replaced with those of the green channel.

The RGB Rewiring commands can be applied to the Master range only.

To rewire the red channel:

1 Select an option from the Red Channel Rewiring box.



Select:	To:
R←R	Use colour values for the red channel.
R←G	Replace colour values of the red channel with those of the green channel.
R←B	Replace colour values of the red channel with those of the blue channel.
R←Y	Replace colour values of the red channel with the luma of all channels.
R←1-R	Replace colour values of the red channel with its inverse. For example, 1 corresponds to the maximum value of the given colour channel. In 8 bits, this value is 255.

2 Click Process to apply the changes to the clip.

NOTE The green and blue channels are rewired in the same way as the red channel (described in the previous table).

Creating a Negative or Monochrome Image

You can convert a colour image into a monochrome or negative image using the Rewire option box.

TIP You can also convert the image manually by clicking the appropriate Rewire selection box (R, G or B) and selecting the Y option for monochrome or 1- R for a negative image.

To convert a colour image into a monochrome or negative image:

1 Select Mono or Negative from the Rewire option box.



The image is automatically converted to a monochrome or negative image.

2 Click Process to apply the changes to the clip. To cancel the changes, click Reset Basics.

TIP You can also create a monochrome or negative image using the Monochrome and Negative buttons in the Processing menu.

Adjusting the Gamma

You can adjust the grey values of an image by adjusting the gamma. This allows you to brighten or darken an image without greatly affecting the shadows or highlights.

To adjust the gamma of the image:

1 Select the tonal range you want to modify.

2 Click a Gamma field and enter a value. Lower the gamma value to increase the contrast or raise the gamma value to decrease the contrast.



Adjusting the Gain and Offset

You can boost the colours in an image by increasing the Gain and Offset values. The Colour Corrector multiplies the pixel colour values by the Gain and then adds the Offset value. The resulting colour values are clipped at the maximum value of 255 in 8-bit mode, or 4095 in 12-bit mode.

You can also reduce the colours in the image by decreasing the Gain and Offset values. The resulting colour values are clipped at the minimum value of 0.

Gain is expressed as a percentage value. The default value of 100% has no effect on the image since the colour values are multiplied by 1.

NOTE You can adjust the Gain for individual ranges, or across all ranges simultaneously (by using the Gain fields in the Global controls). However, changes in gain mostly affect the highlights in the image.

To boost or reduce the colours in the image:

1. Select the range you want to modify.
2. Enter a value in the Gain field for the colour channel you want to modify.



3. Enter a value in the Offset field for the channel you want to modify.



4. Click Process to apply the changes to the clip.

Adjusting the Contrast

You can control the gradations between the light and dark areas of an image by adjusting the contrast. The pivot value affects the contrast behaviour. See Adjusting the Pivot (page 950).

To adjust the contrast of the image:

1. Select the range you want to modify.
2. Click a Contrast field and enter a value.

Suppressing Colours

You can suppress the RGB or CMYw colours in an image. Chroma suppression is useful for removing blue spill from a blue-screen composite.

To suppress a colour:

1 Click Master.



NOTE Chroma suppression can only be applied to the Master tonal range.

2 Enable the appropriate button for the colour to suppress.



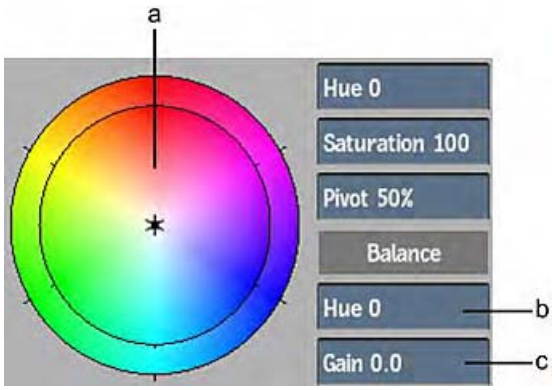
Enable:	To suppress:
R	Red
G	Green
B	Blue
C	Cyan
M	Magenta
Yw	Yellow

The status box on the button is light blue when the button is enabled.

NOTE Suppression of RGB and CMYw always reduces luma.

Adjusting the Colour Balance

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



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

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

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

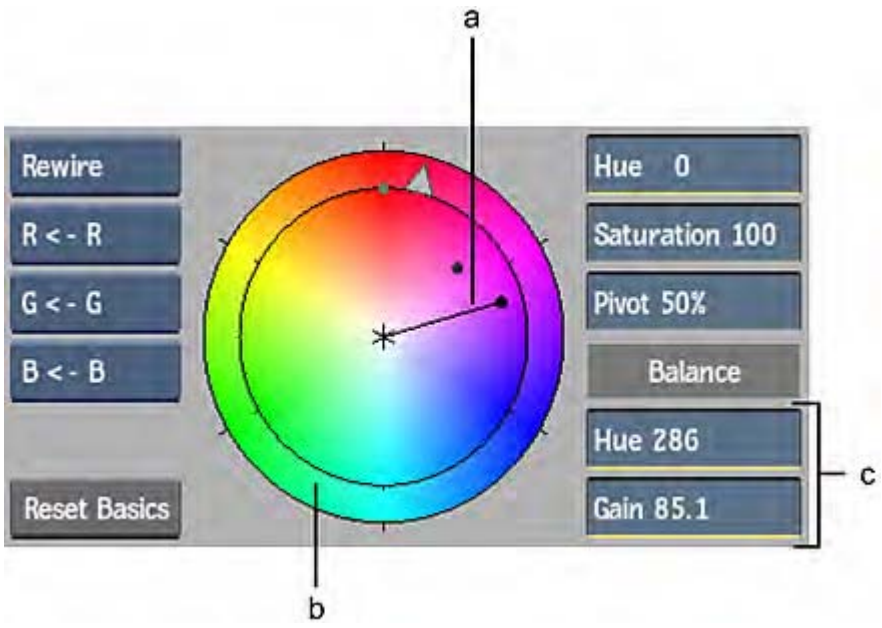
NOTE If Saturation is set to 0, no colour balancing is performed, regardless of the Gain value. In the Channel Editor, Balance channels are labelled hue_balance and gain_balance, respectively.

Creating and Modifying a Colour Balance Point

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

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

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



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

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

To change the colour balance in the image:

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

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

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

   The image dynamically updates as you drag.

   TIP You can interact directly in the image window to change the colour balance. While holding the range keyboard shortcut (`V` for Master; `Z` for Shadows; `X` for Midtones; or `C` for Highlights), drag directly in the image window. The colour wheel and Balance Tools fields update accordingly.
4. If you are not satisfied with the resulting image, click and drag the colour balance point or triangle again. You can continue to move the colour balance point or triangle until the desired effect is achieved.
5. Click Process to apply the changes to the clip.

Adjusting the Colour Range

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

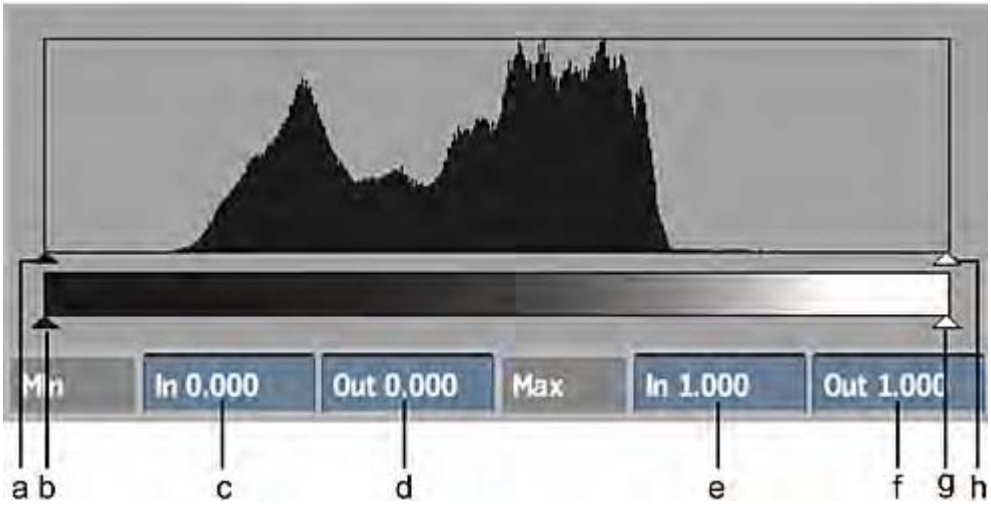
When working with 16-bit floating point images, the values of the histogram are represented on a logarithmic scale. When working with integer images (8, 10, or12-bit), the values of the histogram are represented on a linear scale.

To access the Histogram menu:

1 Click the Histogram tab.



The Histogram menu options are described as follows.



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

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

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



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

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

Increasing Image Contrast

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

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

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

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

To increase the contrast in an image:

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

The value of the lower limit for the colour range appears in the Minimum Input field.

5 Position the cursor on the white triangle below the histogram. Drag left to brighten in the highlight areas in the image.

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

NOTE If you are in Luminance, you can invert the image by reversing the order of the black and white triangles.

6 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.

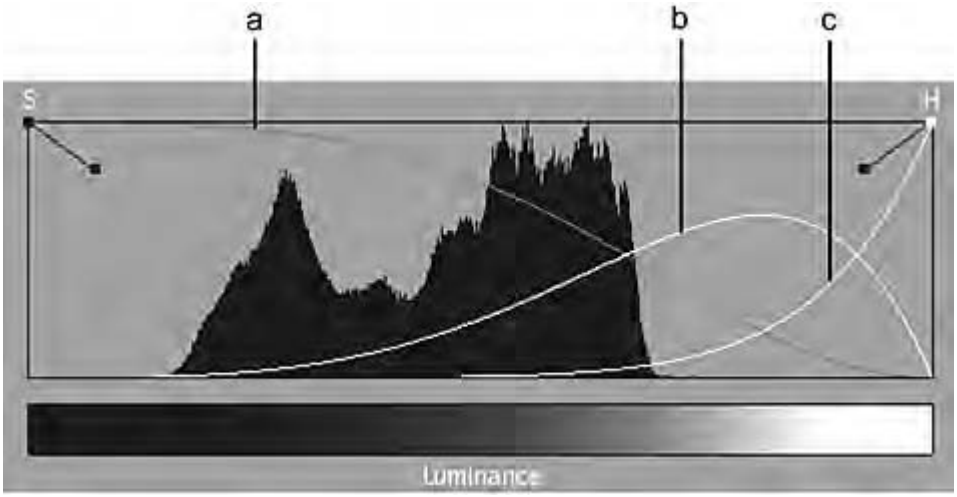
When working with 16-bit floating point images, the values of the histogram are represented on a logarithmic scale. When working with integer images (8, 10, or12-bit), the values of the histogram are represented on a linear scale.

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

To see the effect of adjusting these curves:

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

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

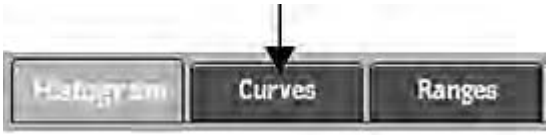
Remapping Colour Values

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

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

To access the Curves menu:

1. Click the Curves tab.



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

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

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

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

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

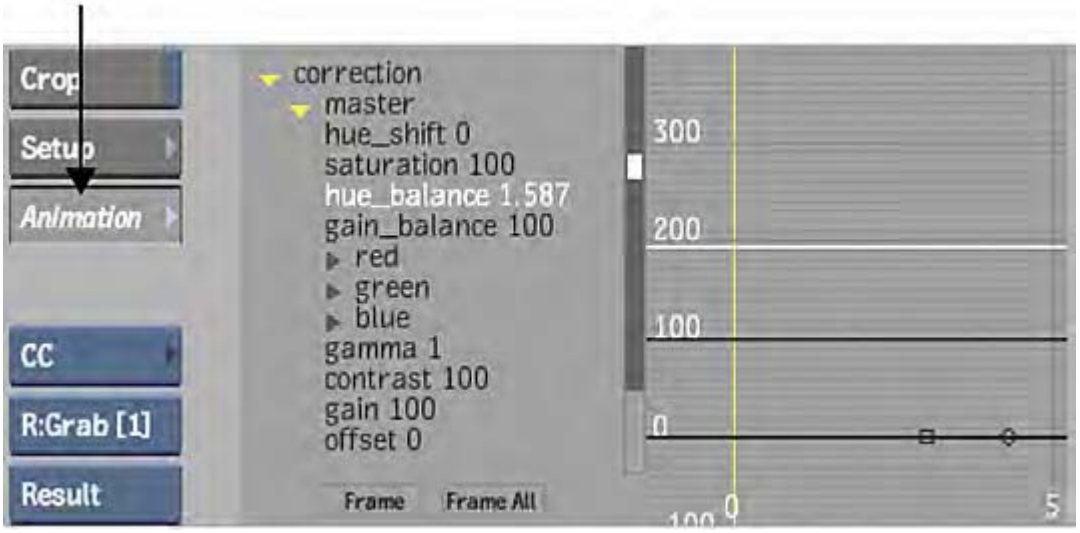
The Curves Editor behaves in much the same way as an animation curve in the Channel Editor. To edit colour curves, select Add, Delete, or Move from the Edit Mode box.

Animating a Colour Correction

Use the Animation controls to animate a colour correction that has been applied to a clip. You can animate any value in the Colour Corrector.

To access the Animation controls:

1 In the Colour Corrector menu, click Animation.



Processing Colour Changes

After you adjust the colours to your satisfaction, you are ready to process the clip. The destination resolution is the same resolution as the source clip.

Before you process the colour changes, you can preview the result clip in the image window.

To apply the colour changes to the front clip:

1 In the Colour Corrector menu, click Process.

The processed clip is saved on the rendered destination reel.

2 Click Exit in the Colour Corrector to return to the Tools tab.

If you entered the Colour Corrector from another tool, you must return to that tool to process the clip. If you are in the timeline, a black line appears on the timeline element to indicate that the clip has been processed.

About the Colour Warper

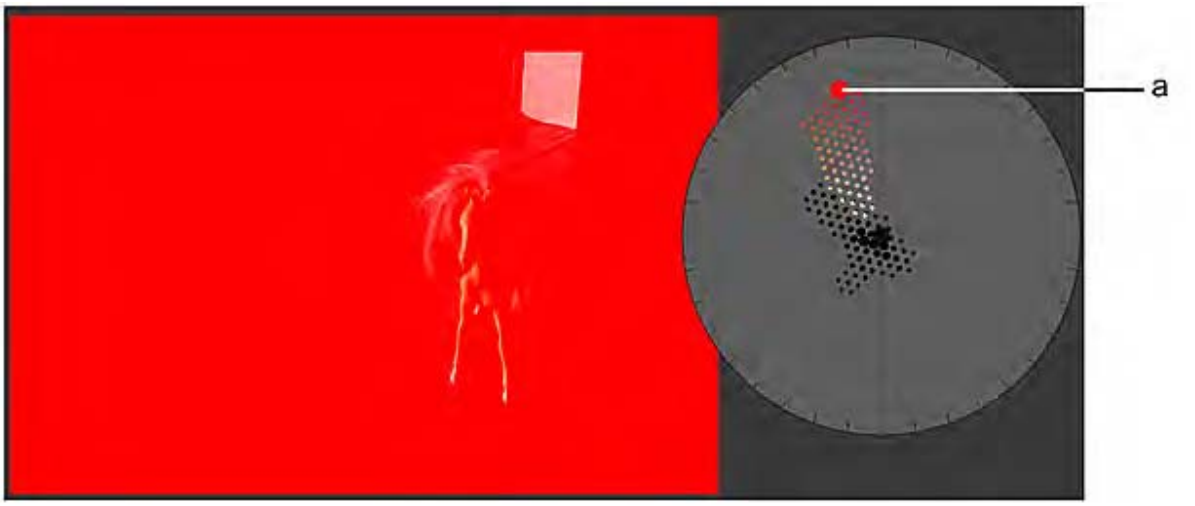
When you colour correct an image or clip, use the Colour Warper to perform advanced colour corrections and create artistic colour effects. The way in which you approach these tasks depends on your goal, the number of clips you are using, and the type of clips being used.

Clips created from source material shot with the same camera equipment under the same lighting conditions may be colour corrected quickly and easily to correct lighting and colour imbalances. Clips created from source material shot at different times of the day, in different seasons, at different locations, or using different

equipment require more work. With the Colour Warper, you can manipulate colours with precision and ease, working on the entire clip as you would with traditional tools or working with a matte to adjust a range of colour in the clip.

Use the Colour Warper to gesturally set black and white levels, adjust specific colours and colour ranges, and accurately match colours in one clip to another. You can also perform hue shifts and suppress colour to remove colour spill or create visual effects such as a colour cast. While you manipulate the colour content of a clip, you can monitor reference clips as well as changes in the colour distribution to ensure that you achieve the result you want. Original data is always preserved, so you can adjust colours without the risk of permanently losing colour information.

When working with 16-bit floating point images in the Colour Warper, you can plot colours outside of the 0 to 1 range. Even when working with integer images, clamped colour information (colours that go beyond the RGB range) can be retrieved using the Colour Warper controls.



(a) Clamped colour



(a) Colour information restored

Accessing the Colour Warper

Use the Colour Warper to modify the colour content in your clips. You can manipulate colour content with intuitive controls that provide precise colour correction, and view histograms that help you visualize your image's colours. You can also output a matte corresponding to selected colours.

You can access the Colour Warper as a Batch or Batch FX node that you drag and drop into the process tree or the Modular Keyer's processing pipeline. Access the Colour Warper from the Modular Keyer to remove colour spill, or access it from Batch or Batch FX to modify the colour content of your clips.

The Colour Warper can be accessed from the following locations:

- The Timeline, using a Colour Correct timeline effect. (page 964)
- Batch, Then select a node from the Node bin. (page 964)
- Modular Keyer, Action or Paint. (page 965)
- Batch FX. (page 965)

When working with large images, you can free up additional screen space with the Overlay user interface. This feature is exclusive to the Colour Corrector and the Colour Warper. The Colour Warper's Overlay user interface is not available when accessing the Colour Warper as a Batch or Batch FX node.

For details on using the Overlay user interface, see Overlay User Interface (page 935).

You can access the following menus from the Colour Warper menu.

NOTE If you are accessing the Colour Warper from Batch or Batch FX or the Modular Keyer, you use the Setup menu for the node to adjust vectorscope and hue cube settings, as well as update colour information. You cannot use the Setup menu for the node to crop the colour correction area or invert a matte.

Accessing Colour Warper from the Timeline

To access Colour Warper from the Timeline:

1. From the Timeline, right-click on the sequence to which you will apply the Colour Warper.
2. Select Add Effect.
   The Video Effects ribbon is displayed.
3. Enable CC (Colour Correct).
4. Click the Colour Box to select Colour Warper.
5. Click the Enter Editor button.
   The Editor screen is displayed.

Accessing the Colour Warper node from Batch

To add a Colour Warper Node in Batch:

1. Go to Batch.
2. Double-click on, or drag, the Colour Warper Node from the node bar and add it to the schematic.
3. Double-click the Colour Warper Node.
   The Colour Warper Node editor opens.

Accessing the Colour Warper from Modular Keyer, Action, or Paint

To access the Colour Warper from the Modular Keyer:

1. Double-click on, or drag, the Colour Warper node from the node bar to the schematic.
2. Double-click the Colour Warper node.
   You are in the Colour Warper editor.

To access the Colour Warper from Action:

1. From the Media menu, select the media containing the clip you want to colour correct.
2. Double-click the CC field for the media you want to colour warp.
3. Click the Colour box.
   The Colour box changes to CW. You are in the Colour Warper.

To access the Colour Warper from Paint:

1. Do one of the following:
   - To colour correct a clip, click Setup in the Paint menu, click CutOut, enable CC, and then enable the CC button for the clip you want to colour correct.
     The clip is loaded into the Colour Warper.
   - To colour correct a cutout, click Cut/Paste in the Paint menu, create a cutout, and then use the Colour Correction field or click CC.
     The cutout is loaded into the Colour Warper.

Accessing the Colour Warper from Batch FX

To access Colour Warper through Batch FX:

1. Select Timeline.
2. Select the segment.
3. Click the FX button.
4. Click the Create Batch FX button.
5. Double-click on, or drag, a Colour Warper node into the schematic.
   The Colour Warper node is now in the schematic.
6. Double-click the Colour Warper node.
   You are in the Colour Warper node editor.

Setting Up Your Work Environment

Set up your work environment to streamline the colour correction process. Use the tools provided to examine the clip or image and efficiently perform advanced colour corrections. As your work progresses, you can modify the Colour Warper environment to suit the task at hand.

You can:

- Display multiple viewports.
- View reference clips.

- View the colour and luma content of a source (front) clip and result clip.
- Apply a colour correction to all or part of a clip.
- Crop the colour correction area (when accessed from the Colour Corrector).
- Invert the matte (when accessed from the Colour Corrector).
- Dynamically update colour information as you work.
- Reset entire colour corrections.
- Clear or reset individual values.
- Save and load colour correction setups and preferences.
- Use the Undo/Redo list.

Multiple Viewports

When working in the Colour Warper, you can display up to four viewports at a time in the image window, including a view of the Channel Editor. Multiple viewports are convenient when you want to modify colour values, create mattes, and compare your result with other images in different viewports simultaneously.

NOTE The Overlay user interface cannot be used with multiple views in the Colour Warper tool.

Reference Clips

You can use the split bar to view a reference clip while you work. Reference clips are useful when you want to match one clip to another. Carefully selecting a suitable reference clip will help you make your colour corrections quickly and efficiently. For example, you can use a reference clip from the Batch or Batch FX schematic to match saturation, whites and blacks, or colours. You can also switch between different references to ensure continuity throughout the project.

NOTE You cannot view reference clips when you access the Colour Warper from the Colour Correction timeline effects. Also, if the Overlay user interface is enabled, the split bar will extend beneath the Overlay user interface. To access it, be sure to grab from an area where there are no user interface elements.

Viewing Colour Information with a Vectorscope

In the Colour Warper, you can use a 2D or 3D vectorscope to help match colours, adjust shadows and highlights, view colour distribution, and ensure that broadcast standards are met. The vectorscopes show the changes you make to colour content in a clip or image.

Each vectorscope can display:

- A histogram showing the distribution of image pixels across luma and hue ranges. The histogram dynamically updates to reflect your changes as you modify colours in the image. Exposure and contrast settings in the image display viewer are taken into account.
- Source and destination colours. Source colours are the colour values in the front clip and destination colours are the colour values in the result clip.
- Plotted and reference colours that are obtained by sampling images in the image window. See Sampling Clips in the Image Window (page 975).

NOTE The 2D and 3D vectorscopes only appear in Result view.

To view colour information in either the 2D or 3D vectorscope:

1. In the Colour Warper menu, enable Scope.

   The vectorscope appears in the image window.

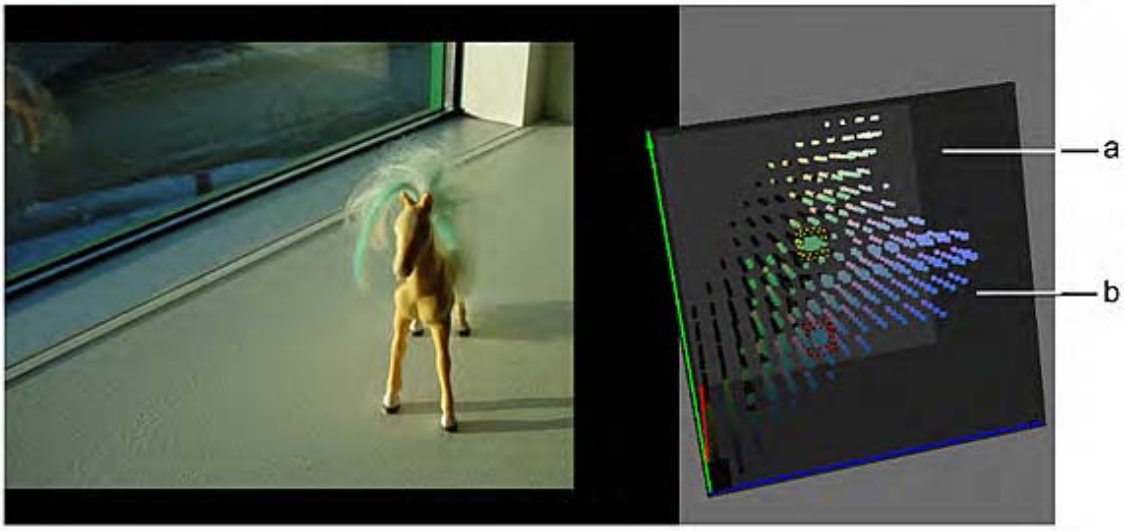
   NOTE You can only view one vectorscope at a time.

2. Click Setup.
3. From the Scope box, select the vectorscope you want to use to view the clip.

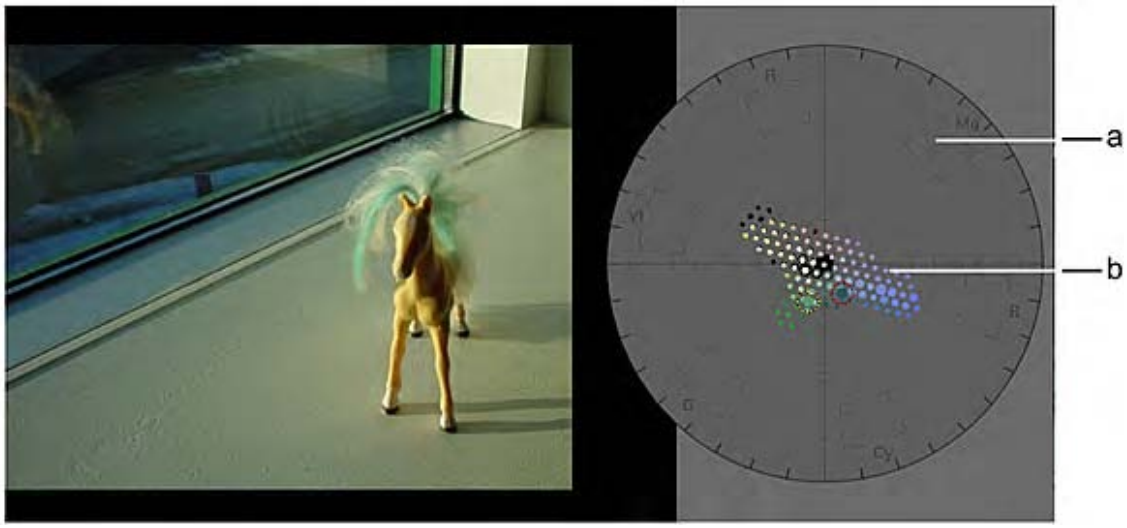
   2DDisplays the 2D vectorscope. Use the 2D vectorscope to analyse the colour content of the clip and locate specific colours in terms of their chroma values. When you view a clip in the 2D vectorscope, you see it in terms of hue and saturation. The size and placement of the colour squares in the 2D vectorscope show the distribution of all the colours in the clip—like a colour wheel; the distance from the centre to the perimeter of the scope maps colour saturation, with the outside edge of the scope delineating pure colour.

   3DDisplays the 3D vectorscope. Use the 3D vectorscope to both analyse the colour content of the clip and to locate specific colours in terms of their chroma and luma values. When you view a clip in the 3D vectorscope, you see it in terms of HLS (hue, lightness, and saturation). Its X, Y, and Z axes represent red, green, and blue, and the size and placement of the colour cubes show the distribution of all the colours of the clip in RGB colour space.

   The selected vectorscope is displayed in the image window. As you scroll through a clip, the histogram is updated to display the colour distribution of each frame.



(a) 3D vectorscope (b) 3D histogram



(a) 2D vectorscope (b) Histogram

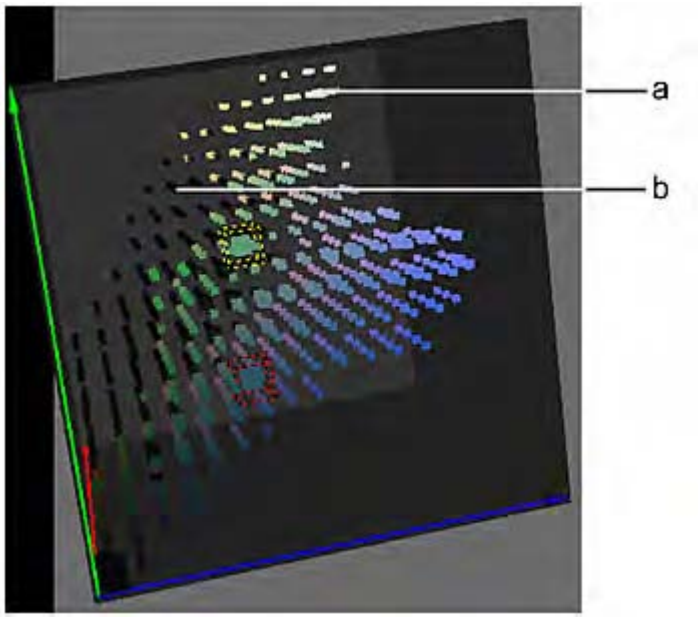
4 Set the following options to customize the display of the 2D or 3D vectorscope.

Enable:	To:
Canvas	Show the vectorscope canvas. When the canvas is off, the vectorscope is transparent, but can be outlined.
Lines	Show the vectorscope outline.

5 Set the following histogram options to show clip colour information.

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

When both Source and Destination are enabled, you see both the source and destination colour values. The source clip's colour values are displayed in black and the result clip's colour values are displayed in colour. When only Source is enabled, the source clip's colour values are displayed in colour.



(a) Result clip (in colour) (b) Source clip (in black)

NOTE You can also toggle the Source and Destination buttons in the Basics menu to show or hide source and destination colours in the histograms.

6 From the Size box, select the size of the coloured squares or cubes that make up the histogram. You can display 8x8, 16x16, or 32x32 coloured dots.

7 To set SMPTE bars for the 2D vectorscope, select an option from the Bars box.

Select:	To match:
Bars 75%	75% SMPTE bars. If the current project is NTSC, the bars are set to 75% by default.
Bars 100%	100% SMPTE bars.

The parameters of the histogram of the 2D vectorscope update accordingly.

8 To toggle between the vectorscopes, double-click the vectorscope in the image window.

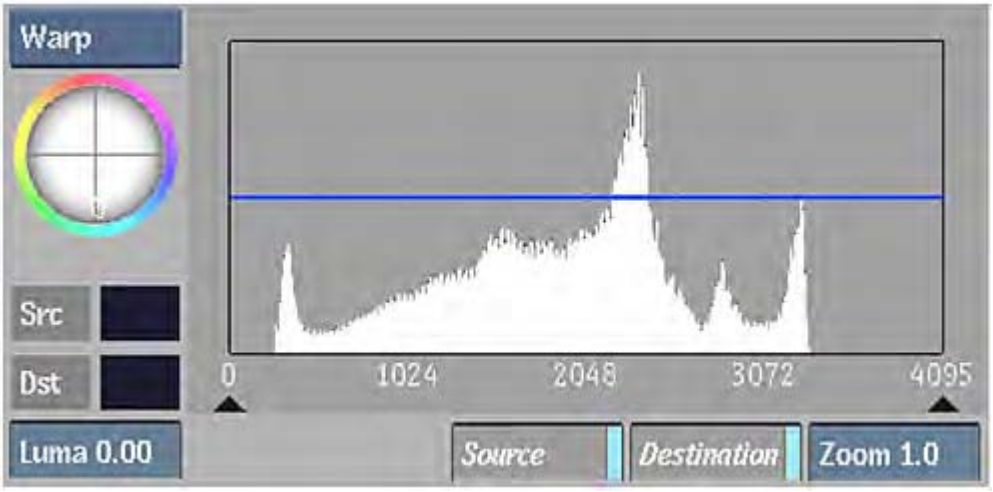
Use these other shortcuts to interact with the vectorscopes.

Press:	To:
`Alt`-drag	Move the 2D or 3D vectorscope
`Shift`-drag	Zoom the 2D or 3D vectorscope
`Ctrl`-drag	Rotate the 3D vectorscope. If you `Ctrl`-drag the 2D vectorscope, the 3D vectorscope appears and rotates.

9 To position a vectorscope at its default location and size, click Home.

Viewing Luma Information with the 2D Luma Histogram

The 2D luma histogram shows the changes you make to luma content in a clip. Use the 2D luma histogram when you want to adjust the lightness of a sample. The 2D luma histogram is displayed in the Basics menu of the Colour Warper.



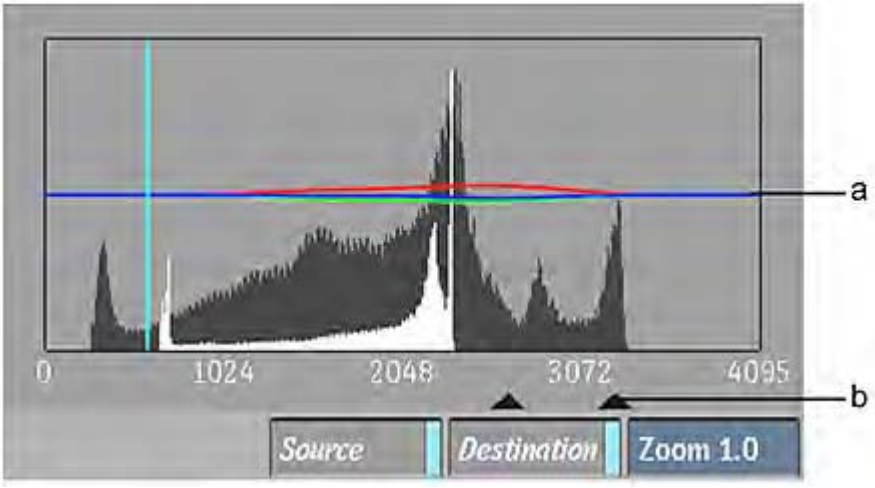
The 2D luma histogram can display:

- A histogram showing the distribution of image pixels across the luma range—the size and distribution of the vertical bars show the luma distribution.
- Source and destination colour values. Source colour values are the luma values in the front clip and destination colours are the luma values in the result clip.
- Plotted and reference colours that are obtained by sampling images in the image window. See Sampling Clips in the Image Window (page 975).

To view luma information in the 2D luma histogram:

1. In the Colour Warper menu, click Basics.

   The 2D luma histogram appears in the Basics menu. R, G, and B values that are modified by the Midtones trackball are displayed as curves in the histogram.



(a) RGB curves (b) Midtone triangle

2 Set the following options to show clip luma information.

Enable:	To:
Source	Show a histogram of the luma values in the front, or source, clip. The source luma values are obtained from the current frame.
Destination	Show a histogram of the luma values in the result clip. The destination colour values are obtained from the current frame.

When both Source and Destination are enabled, you see both the source and destination luma values in the histogram. Source clip colour values are displayed in black and result clip colour values are displayed in off-white.

NOTE You can also toggle the Source and Destination buttons in the Setup menu to show or hide source and destination colours in the histograms.

Colour Correcting All or Part of an Image

Use the options in the Work On box to apply colour corrections to all or part of an image. You can select an option from the Work On box at any time to further modify the image. Use Master to apply colour corrections to the entire image and use a selective to generate a matte and perform selective colour correction—apply the colour correction to a selected range of colour. The Work On box is available in every menu of the Colour Warper.



(a) Work On box

To colour correct all or part of a clip:

1 Select an option from the Work On box.

Select:	To:
Master	Use the Basics menu to modify the entire image. In Master mode, you cannot output a matte.

Select:	To:
Sel. 1, Sel.2, or Sel. 3	Generate a matte from the front clip selective colour correction. You can then use the Basics menu to modify the range of colour defined by the matte. In a selective mode, you can output the matte, selective, or result. See Selecting Colour Ranges for Colour Correction (page 977).

Cropping the Colour Correction Area

When you access the Colour Warper from the Colour Corrector, you can create a crop box to limit the area where the colour correction is applied.

Using a crop box as you colour correct a clip speeds up interactivity, which is especially useful when working at high resolutions. You can also process the clip with a crop box. Everything outside the crop box will be processed as black on the matte.

To use a crop box:

1. In the Colour Warper menu, click Setup.
2. Do one of the following:
   - Hold down the `Ctrl` key or hold the pen button and drag the box across the image.
   - Turn on the Crop button and enter the dimensions for the crop box in the Left, Right, Bottom, and Top fields.



The crop box appears in the image window. By default, the crop box has the same dimensions as the clip. The display inside and outside the crop box depends on how many clips are loaded.

If you loaded:	Inside the crop box:	Outside the crop box:
A front, back, and matte clip	Shows the front clip in the opaque area of the matte and the back clip in the transparent area of the matte.	Shows the back clip.
A front and back clip	Shows the front clip.	Shows the back clip.
A front clip only	Shows the colour-corrected clip.	Shows the non-colour corrected clip.

3. To adjust the size of the crop box, drag on the corner points of the box, or modify the values in the Left, Right, Bottom, and Top fields in the Setup menu.
4. To move the crop box in the image window, click a side of the box and drag to a new location. Alternatively, press `Ctrl` and redraw the crop box.

TIP If the Overlay user interface is enabled, the crop box will extend beneath the Overlay user interface. To access the bottom edge, be sure to grab from an area where there are no user interface elements.

5 You can now use any Colour Warper menu and colour correct the area of the clip that is defined by the crop box. You will only see the results of your changes within the crop box.

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

To disable a crop box:

1 From any Colour Warper menu, disable Crop.

When you disable the Crop button in a Colour Warper menu, the Crop button in the Setup menu is also disabled.

To change the colour of the crop box:

1 In the Colour Warper Setup menu, click the colour pot beside Border.



The colour picker appears.

2 Select a colour with the colour picker.

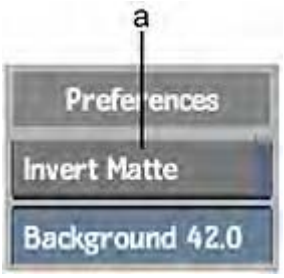
3 Click in the Border colour pot to apply the new colour to the border.

To hide the crop box:

1 In the Setup menu, disable Border.

Inverting the Matte

When you access the Colour Warper from the Colour Corrector and you input a front, back, and matte clip, you can invert the matte to colour correct the region outside the area defined by the matte. Inverting the matte swaps the transparent and opaque areas of the matte and redefines the area of the front image that can be colour corrected, as well as the area of the back image that is shown. To invert a matte, enable the Invert Matte button in the Setup menu.



(a) Invert Matte button

Resetting Colour Corrections

When you access the Colour Warper from the Colour Corrector, you can reset colour corrections created with the Colour Warper, the Colour Corrector, or both.

To reset a colour correction:

1 In the Colour Warper menu, select an option from the Reset box.

Select:	To reset:
Reset All	The entire tool. Both the Colour Corrector and Colour Warper settings are reset to their default values.
Reset CC	The Colour Corrector. Settings in the Colour Warper are not affected.
Reset CW	The Colour Warper. Settings in the Colour Corrector are not affected.

NOTE If accessing the Colour Corrector or the Colour Warper as a timeline FX, only Reset All is available.

2 Click Confirm.

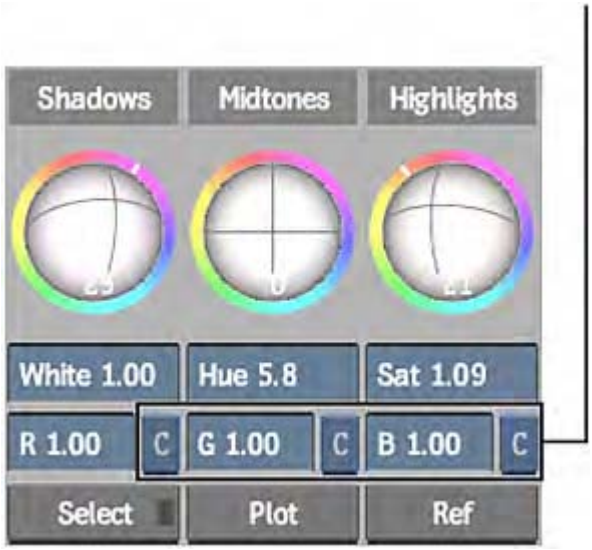
Resetting and Clearing Values

You can clear or reset several Colour Warper values in the Basics menu using individual C/R boxes. C resets the value for the current frame by setting a keyframe. R resets the value for the entire animation curve.

To reset a Colour Warper value:

1 In the Basics menu, do one of the following:

- Select C from the C/R box next to a control.



The value is reset at the current frame.

- Select R from the C/R box.

The value is reset for the entire animation curve.

NOTE Fields that do not have C/R boxes next to them cannot be reset. Use the Undo box instead.

Saving Colour Correction Setups and Preferences

You can save or load colour correction setups using the Save or Load button. Any setup can be loaded from the file browser for use with another set of clips. You can also save or load colour correction preferences using the Load or Save button.



- To save setups or preferences, click Save.
- To load setups or preferences, click Load.

  TIP Click Revert to revert to the last saved setup. All changes made since the previous Save operation are undone.

When you save colour correction setups, you save the current settings for both the Colour Corrector and the Colour Warper to the setup file. If you overwrite a setup file using the Save button, the current settings in both the Colour Corrector and Colour Warper are saved.

Using Undo and Redo

Use the Undo and Redo boxes to remove or redo colour modifications. To undo or redo a modification, click a box and select an option from the list. Set the number of undo levels in the Setup menu.

NOTE You also set the number of undo levels for the Modular Keyer in the Setup menu of Batch or Batch FX.



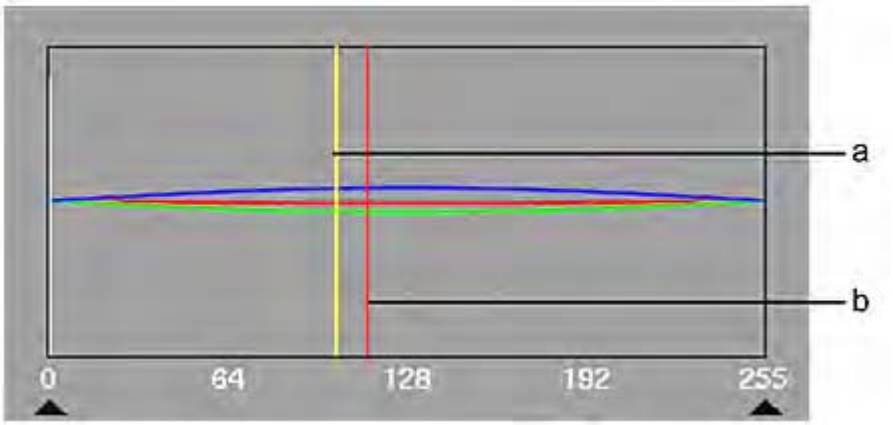
(a) Redo option box

Sampling Clips in the Image Window

When you generate mattes, modify clips, or match colours between clips, you can sample colours in both the result clip and a reference clip. Use Plot to sample a colour from the result clip and Ref to sample one from a reference clip. The Plot and Ref buttons appear on every menu in the Colour Warper.

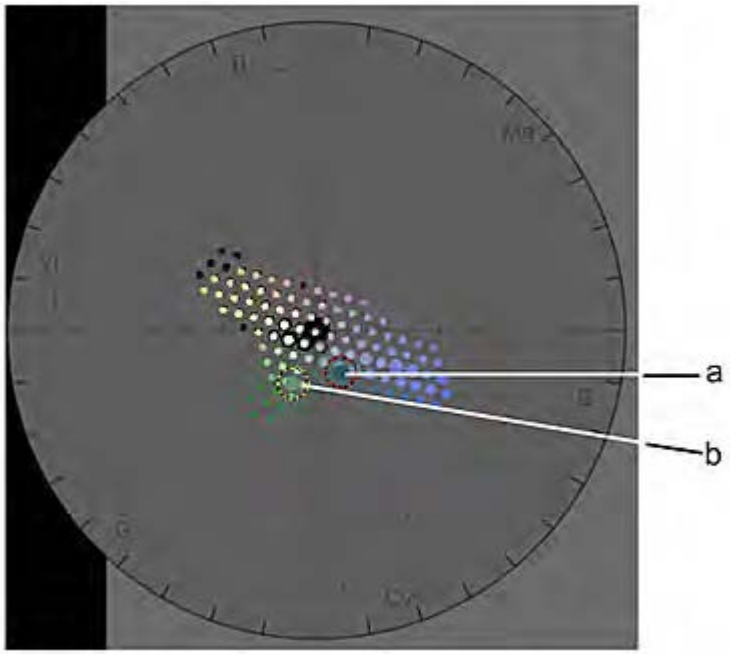
NOTE Ref samples cannot be adjusted.

Samples appear in the histograms. In the 2D luma histogram, the Plot sample is represented by a red bar and the Ref sample is represented by a yellow bar.



(a) Reference colour (yellow vertical line) (b) Plotted colour (red vertical line)

In the vectorscope, the Plot sample is outlined in red and the Ref sample is outlined in yellow.



(a) Plotted colour (outlined in red) (b) Reference colour (outlined in yellow)

You can use Plot to get more information on colours you want to change. When you are adjusting colours and levels and are not sure which controls to use, click Plot and select a colour in the image. You can then view the histograms to determine which control you should use. For example, if the plotted colour appears in the middle of the 2D luma histogram, you can modify the colour in the image using the Midtones trackball while viewing the plotted colour in the 2D vectorscope.

To sample clips:

1 Load a reference clip and position the split bar so that both the reference clip and the result clip appear in the image window.

2 Do one or both of the following:

- Enable Plot and then sample the result clip.

You can:	To:
Click-drag	Sample a single pixel in the image. You can drag through the image until you locate the pixel you want to sample.
`Ctrl`-drag	Sample an average taken from a range of colours in the image.

A red vertical line appears in the 2D luma histogram indicating the luma of the sample. The sampled colour is also outlined in red in the 2D or 3D vectorscope. In the 2D vectorscope, you see the sample in terms of hue and saturation. In the 3D vectorscope, you see it in terms of HLS. If you are plotting a colour in a 16-bit floating point image that is out of the 0:1 range, the red outline appears outside of the vectorscope.

- Enable Ref and then sample the reference clip.

You can:	To:
Click-drag	Sample a single pixel in the image. You can drag through the image until you locate the pixel you want to sample.
`Ctrl`-drag	Sample an average taken from a range of colours in the image.

A yellow vertical line appears in the 2D luma histogram and the sampled colour is outlined in yellow in the 2D or 3D vectorscope. If you are referencing a colour in a 16-bit floating point image that is out of the 0:1 range, the yellow outline appears outside of the vectorscope.

Selecting Colour Ranges for Colour Correction

With some images or clips, you may need to perform selective colour correction—where you modify a range of colour, as opposed to the entire clip or image. Use the tools in the Selective menu to create selectives. Selectives are generated mattes used to isolate colour ranges for selective colour correction.

In the Selective menu, you can:

- Create and refine a matte using tolerance and softness to define the range you want to modify.
- View a matte, a selected colour range, or the result.
- Enable mattes to apply the selective colour correction to the result image. Disable mattes to remove the selective colour correction from the result image.

TIP Set the image window to 2-Up view (`Alt+2`) so that you can create your matte in one viewport while simultaneously monitoring the result in the other.

Generating Mattes

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

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

TIP To get the best key, make sure you have enough processing speed to see the result in real time while interacting with the Tolerance and Softness boundary handles. In the Modular Keyer, you can use the Crop feature to improve interaction.

To generate a matte for selective colour correction:

1. Enable Plot and then sample the image.

   A black dot representing the sample appears on the hue cube. A red line also appears in the luma range and the sample is outlined in the 2D or 3D vectorscope. When you create the matte, you can modify the softness and tolerance boundaries to include or exclude the plotted colour.
2. From the Work On box, select a selective (Sel 1, Sel 2, or Sel 3). With each selective, you define a range for a matte by selecting colours in the front clip.
3. From the Selective View box that appears, select Sel.



   The front clip is displayed as a greyscale image.
4. Click one of the Define controls to set the initial softness and tolerance for the matte.



Click:	To define:
Pick Custom	The tolerance range based on a sample from the image. You must drag the cursor over the image to define initial tolerance.
R, G, B, C, M, or Yw	The tolerance range based on the selected colour channel. For example, click Yw to use the yellow channel to set the tolerance. The luma range is set to default tolerance and softness values.

Click:	To define:
Shadows, Midtones, Highlights	The tolerance range based on the selected luma range. These buttons expand the tolerance and softness boundaries to include all ranges of colours in the image.

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



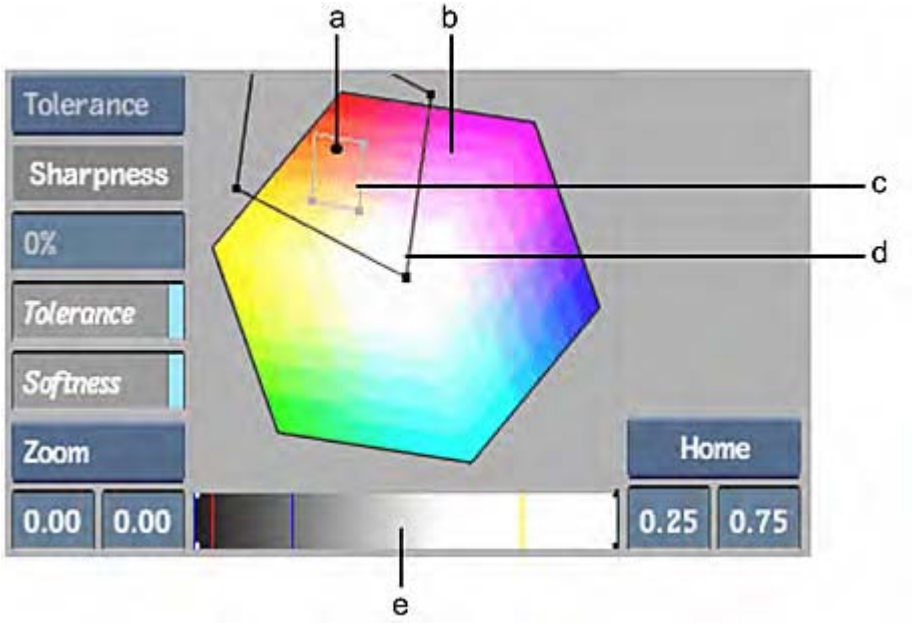
(a) Original colour visible through the greyscale display.

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

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

5 Enable Tolerance and Softness.

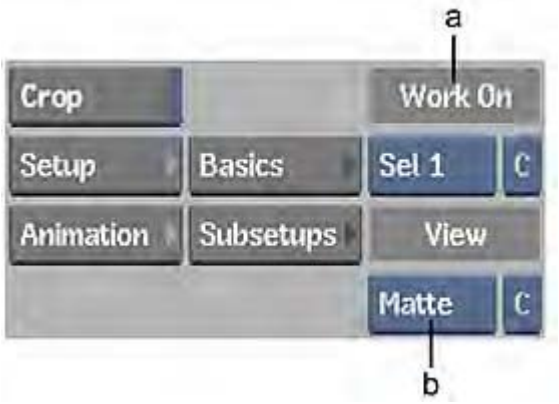
The range of colour used to define the matte is shown on the hue cube with tolerance and softness indicators. The light grey outline shows tolerance and the black outline shows softness. When you define a matte with a luma range, it is shown in the luma range—the white line indicates the tolerance and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



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

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

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



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

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

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

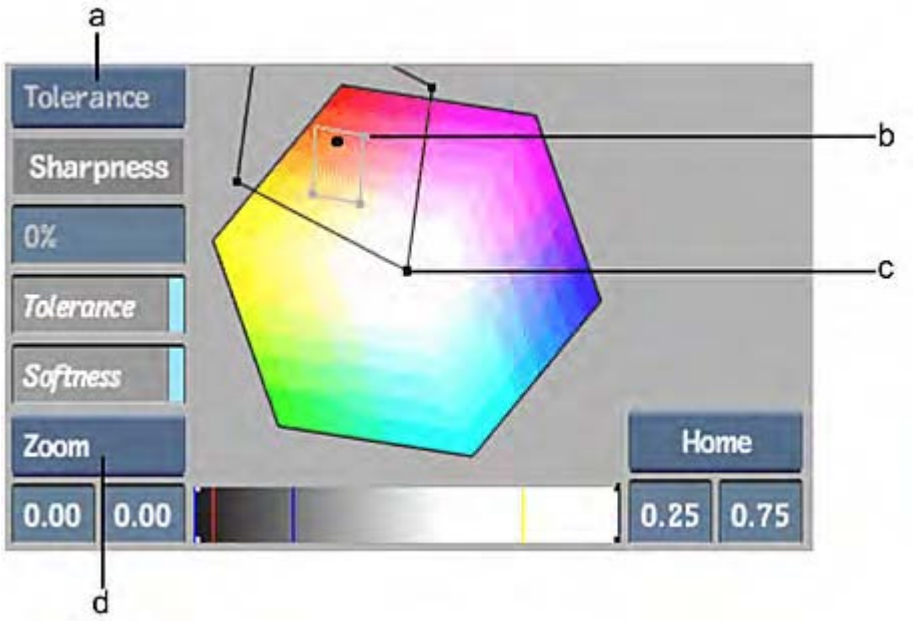
7 To refine the matte, do any of the following:

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

Select:	**To:**
Tolerance	Add tolerance to the matte.
+Softness	Add softness to the matte.

Select:	To:
-Softness	Remove softness from the matte.

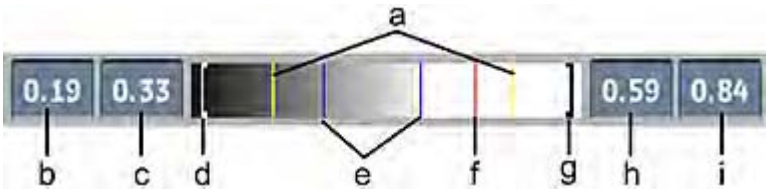
- Adjust softness and tolerance by selecting Move from the Move/Zoom box and then moving the handles of the tolerance or softness outlines on the hue cube. The Tolerance and Softness buttons must also be enabled.



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

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

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



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

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

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

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

9 To invert the matte, enable Inv Selection.

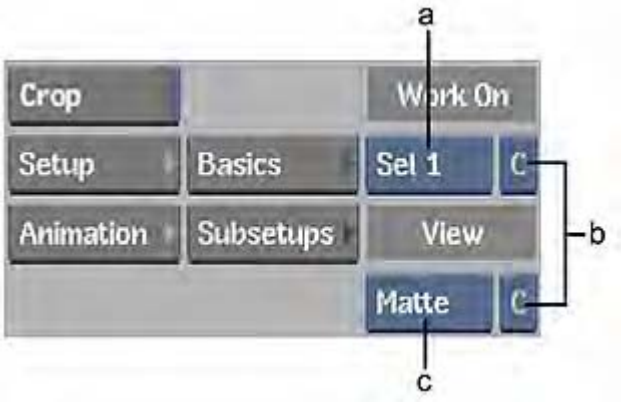
10 Continue fine-tuning the matte until you are satisfied.

You can select the selective in the Work On box from any menu in the Colour Warper to perform advanced colour corrections on the range defined by the matte. You can also change the view in the Selective View box.

TIP When you switch from the Selective menu to any other menu in the Colour Warper, Result appears in the Selective View box. Select Result to view the result clip.

Clearing and Deleting Mattes and Colour Corrections

Use the C/R boxes next to the Work On box and Selective View box to reset a matte or a colour correction. C deletes the matte or colour correction at the current frame by setting a keyframe. R deletes the matte or colour correction for the entire animation curve.



(a) Work On box (b) C/R boxes (c) Selective View box

NOTE If there is no matte or colour correction associated with the correction, the C/R button is disabled.

To delete a matte for the entire animation curve:

1 Select the matte from the Work On box (Sel 1, Sel 2, or Sel 3).

2 Select R from the C/R box next to the Selective View box.

The matte is deleted. All keyframes and colour corrections associated with the matte are also cleared.

NOTE Keyframes associated with the colour correction are not cleared.

To delete a matte at the current frame:

1 Select the matte from the Work On box (Sel 1, Sel 2, or Sel 3).

2 Select C from the C/R box next to the Selective View box.

The matte is deleted at the current frame and a keyframe is set.

To delete a colour correction for the entire animation curve:

1 Select a matte or the entire image from the Work On box (Master, Sel 1, Sel 2, or Sel 3).

2 Select R from the C/R box next to the Work On box.

The colour correction is deleted and all keyframes are cleared. If you selected a matte (Sel 1, Sel 2, or Sel 3) from the Work On box, the matte is unaffected and its keyframes are not cleared.

To delete a colour correction at the current frame:

1. Select a matte or the entire image from the Work On box (Master, Sel 1, Sel 2, or Sel 3).
2. Select C from the C/R box next to the Work On box.

   The colour correction is cleared at the current frame and a keyframe is set. If you selected a matte (Sel 1, Sel 2, or Sel 3) from the Work On box, the matte is unaffected and its keyframes are not cleared.

Correcting Colour Imbalances

During production, differences in lighting and equipment often create colour imbalances in and between clips. When you work on these clips in the Colour Warper, you should perform a basic colour correction to colour balance them. You can balance colours using the tools in the Basics menu. Perform your tasks in the following order:

- Set black and white levels.
- Remove unwanted colour.

Setting Black and White Levels

When you balance the colours in a clip, you should always start with the darkest and lightest parts of the image. You may, for example, need to establish parameters for the black and white (luma) content to ensure that the intensity of the image is in accordance with established broadcast parameters. These parameters are known as *black points* and *white points*. You use the Black and White fields to establish black and white points.



(a) Black field (b) White field

Black field Modifies the luma in the image's shadows without affecting the chroma.

White field Modifies the luma in the image's highlights without affecting the chroma.

To set black and white levels:

1. Sample the highlights in the image. To determine the white point, select a white that appears to be flat rather than reflective so that you obtain a more accurate reading. Do not use a highly reflective

surface such as a window or metal surface as they are often too bright. Enable Plot and then sample the whites in the result clip.

In the Basics menu, a red plot line is displayed in the 2D luma histogram indicating the luma value of the sampled whites in the result clip.

2 While viewing the 2D luma histogram and the image, adjust the white level to modify the luma value and set the white point—drag left to darken the highlights and right to lighten them. The Plot sample is updated as you drag the field.

TIP You can view both the 2D luma histogram and 2D vectorscope to determine what adjustments you need to make to the highlights. For example, if the white point is set to an acceptable level in the 2D luma histogram but the sampled chroma value shows an unwanted colour cast in the 2D vectorscope, use the Highlights trackball to modify the chroma value in the highlights. See Removing Unwanted Colour (page 984).

3 Enable Plot and sample the shadows in the result clip to determine the black level. Sample the darkest part of the image to determine the black point.

A red plot line is displayed in the 2D luma histogram indicating the luma value of the sampled blacks in the result clip.

4 While viewing the 2D luma histogram and the image, adjust the black level to modify the luma value—drag left to darken the shadows and right to lighten them. The Plot sample is updated as you drag the field.

TIP Once the black level is set, you can use the Shadows trackball to adjust the chroma values in the highlights while viewing the plot sample in the 2D vectorscope. See Removing Unwanted Colour (page 984).

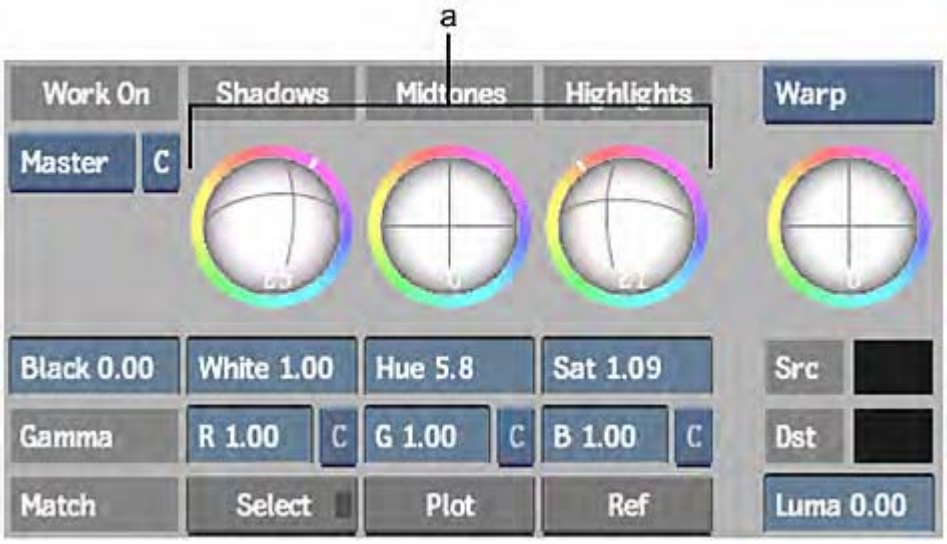
Removing Unwanted Colour

Unwanted colour can be caused by factors such as inconsistent lighting conditions during a shoot or incongruities between the white levels set on a video camera and the given lighting conditions. Factors such as these can result in clips or images that contain unnatural looking colours or one predominant colour, which gives the image an unwanted colour cast. After setting black and white points, you can remove the unwanted colour without affecting the black and white levels. You can:

- Use the Shadows, Midtones, and Highlights trackballs to remove colour from the shadows, midtones, and highlights ranges.
- Use the Suppress trackball to remove a range of colour, such as a colour cast that pervades the entire image.

Not all colour casts detract from the natural appearance of the image. Before you remove a colour cast, it is best to analyse the image or clip, both in relation to other clips in the project and with an eye on maintaining the atmosphere, or mood inherent to the clip. For example, images of sunny summer days should have a reddish cast to convey an atmosphere of warmth. Completely removing the reddish cast may result in a colder feel than you want. In cases where the cast is a complementary one, you may want to refine it to improve the overall effect. See Creating Colour Casts (page 995).

The shadows, midtones, and highlights trackballs in the Basics menu are described as follows.



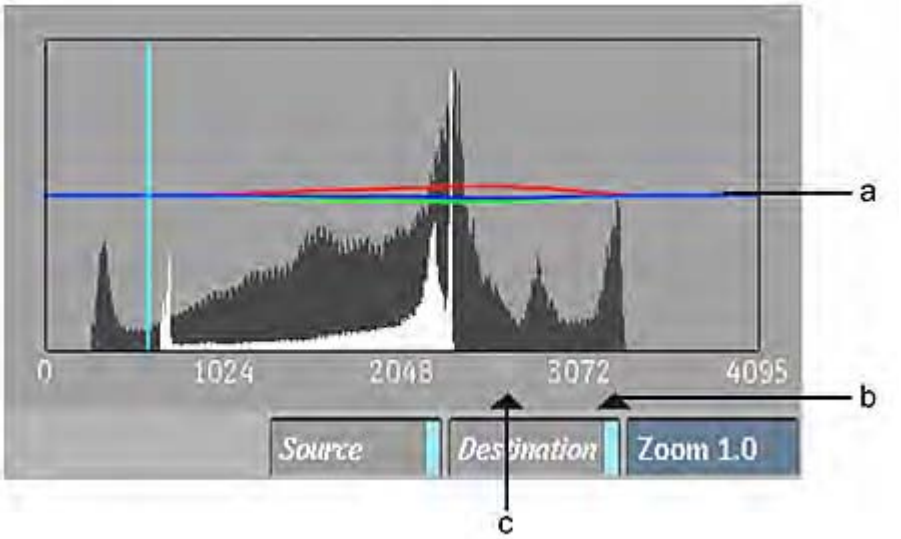
(a) Trackballs

NOTE To make the trackballs respond to changes in pressure, use the Pointer preferences in the Preferences menu.

Shadows trackball Modifies the chroma in the low luminance area without affecting the luma.

Midtones trackball Modifies horizontal RGB curves in the 2D luma histogram. These curves represent the amount of chroma tint you apply to the image. For example, if the red and green curves are above the middle line while the blue curve is below the middle line, you are adding a yellow tint to the midtones components of your image.

By default, the Midtones trackball affects all ranges except white and black, as shown by the histogram's RGB curves. To set the luma range affected by the Midtones trackball, use the two midtones triangles below the 2D luma histogram. Drag the triangles closer together to decrease the range and further apart to increase it. You can modify the range as many times as you like. When working with 16-bit floating point images, the midtones range is set to 0:1, and the midtones triangles can only be moved in this range.



(a) RGB curves (b) Midtones triangle (c) Midtones triangle

Because you can modify the range that the Midtones trackball affects, the trackball will always return to its default reference value—when you release the Midtones trackball and apply the change, it will return to 0.

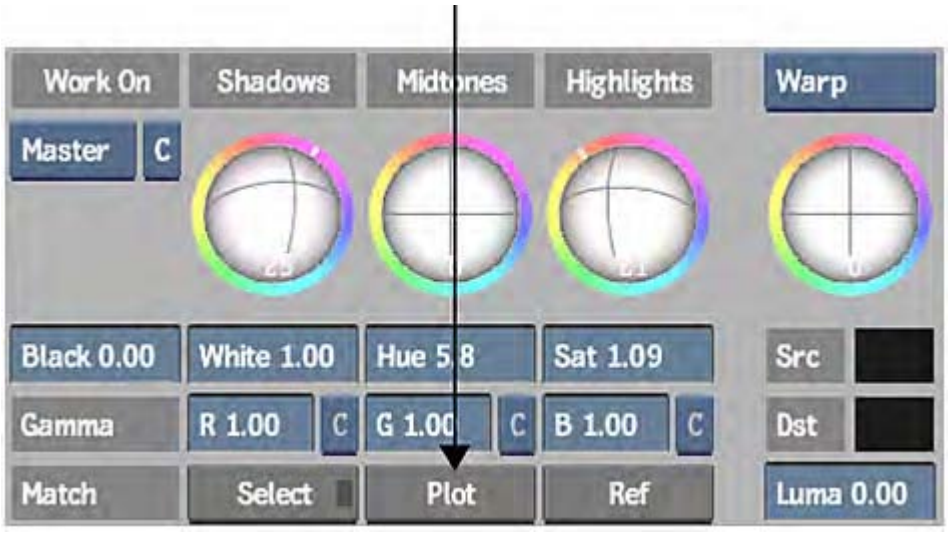
NOTE You can animate the curve using the Shape channel under Midtone in the Channel Editor. Click Animation to view the Channel Editor.

Highlights trackball Modifies the chroma in the high luminance area without affecting the luma.

To remove unwanted colour using the Shadows, Midtones, and Highlights trackballs:

1 Click Basics to display the Basics menu.

2 Enable Plot and then sample the image's highlights.



The sample (outlined in red) appears in the 2D or 3D vectorscope.

TIP You can determine the colour of a sample by dragging right in the Saturation field to temporarily increase the saturation while monitoring the sample in the 2D vectorscope. When you release the mouse, the level that appears in the field resets to its default reference value. Once you determine the cast of the colour, click Undo to reset the saturation level and then resample the image.

3 If necessary, double-click the 3D vectorscope to display the 2D vectorscope.

4 While viewing the 2D vectorscope, drag the Highlights trackball to move the sample towards the centre of the 2D vectorscope.

The chroma in the high luminance area of the clip is modified without affecting the luma.

5 Drag the Midtones trackball to make minor adjustments to the chroma content.

TIP Drag the left midtones triangle to decrease the range affected by the Midtones trackball. Make sure the range includes the sampled colour.

The RGB curves in the 2D luma histogram reflect the changes you make. More of the unwanted colour is removed from the image (while still preserving luma) without affecting the black and white points.

TIP To achieve true white, drag the Highlights and Midtones trackball to move the plot sample to the centre of the 2D vectorscope.

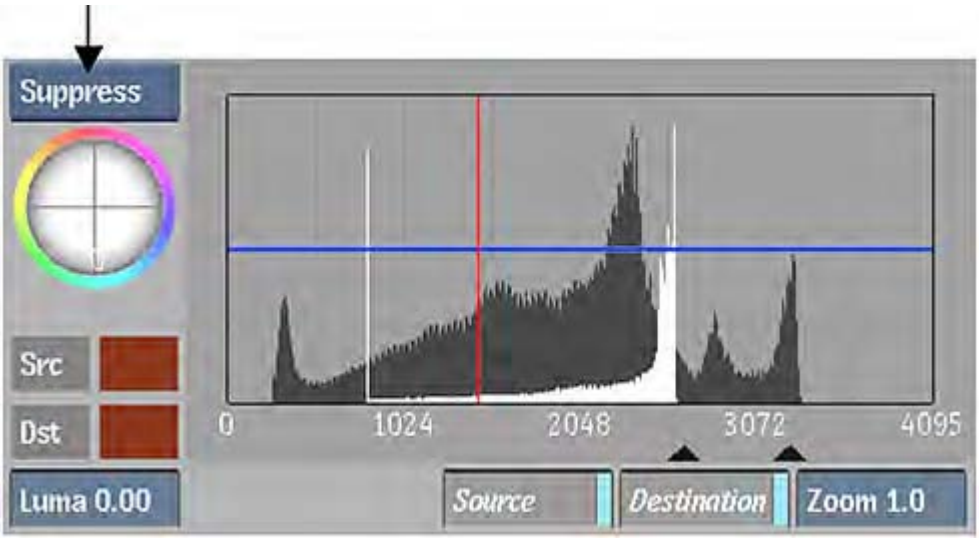
6 Continue modifying the highlights and midtones until you are satisfied with the result.

7 Enable Plot and then sample the image's shadows.

8 Repeat steps 2 to 6 using the Shadows and Midtones trackballs to remove unwanted colour from the shadows (and midtones) in the image.

To remove unwanted colour using the Suppression trackball:

1. Click Basics to display the Basics menu.
2. Select Suppress from the Trackball option box.



3. To sample the colour you want to suppress, enable Plot and then sample the image.

   The sample (outlined in red) appears in the 2D or 3D vectorscope.
4. While monitoring the 2D vectorscope and image, drag the Suppress trackball towards the colour you want to suppress to move it closer to the centre of the 2D vectorscope. For example, to suppress yellows, move the trackball towards yellow.

   TIP Disable Source and Destination to only view the sample.
5. Release the trackball.
6. To further increase the suppression of the same colour, use the trackball to move the colour closer to the centre of the 2D vectorscope.

Improving Contrast

Poor image contrast can be caused by various factors. For example, when you remove a colour cast from a clip, it may become washed out. Washed out clips usually occur when the image contrast and saturation levels are too low. To complete the task of balancing the colours in the clip, use the tools in the Basics menu to correct the poor contrast. To improve contrast and the overall look of the clip, you can:

- Adjust the luma and chroma content in the shadows and highlights, including the black and white point.
- Increase colour saturation without changing luma.
- Modify the luma and chroma content using the Gamma controls.

Adjusting Shadows and Highlights to Improve Contrast

When there is not enough, or too much, black or white in a clip, it will lack contrast. If you want to improve the contrast by changing both the chroma and luma content, including the black and white points, use the controls in the Basics menu to adjust the shadows and highlights in the image.

To adjust the shadows and highlights to improve contrast:

1. Click Basics to display the Basics menu.
2. Enable Plot and then `Ctrl`-drag the cursor over the portion of the result clip containing black or white.

   In the Basics menu, a red plot line is displayed in the 2D luma histogram indicating the luma value of the sampled blacks. In the vectorscope, you can view the saturation and hue.
3. Use the 2D luma histogram and 2D vectorscope to determine which controls you need to use (such as the Black field, and Shadows and Midtones trackballs).
4. While viewing the 2D luma histogram, adjust the black level to modify the luma value. The Plot sample is updated as you drag the field.
5. While viewing the 2D vectorscope, drag the Midtones and Shadows (Highlights) trackballs towards the colour you want to increase in the blacks or whites. The Plot sample is updated as you drag the trackball.

   TIP To achieve true black or white, drag the Midtones and Shadows trackball to move the plot sample to the centre of the 2D vectorscope.
6. Continue modifying the shadows until you are satisfied with the result.

   TIP View the sample in the 3D vectorscope to make more refined modifications.

Increasing Saturation

When an image is desaturated, the colours can appear washed out. To improve contrast, you can increase the colour saturation globally, saturate specific hues, or do both. Increasing saturation levels increases the colour intensity and causes minor changes to the hue. The luma content is virtually unaffected when you modify saturation.

To increase saturation globally:

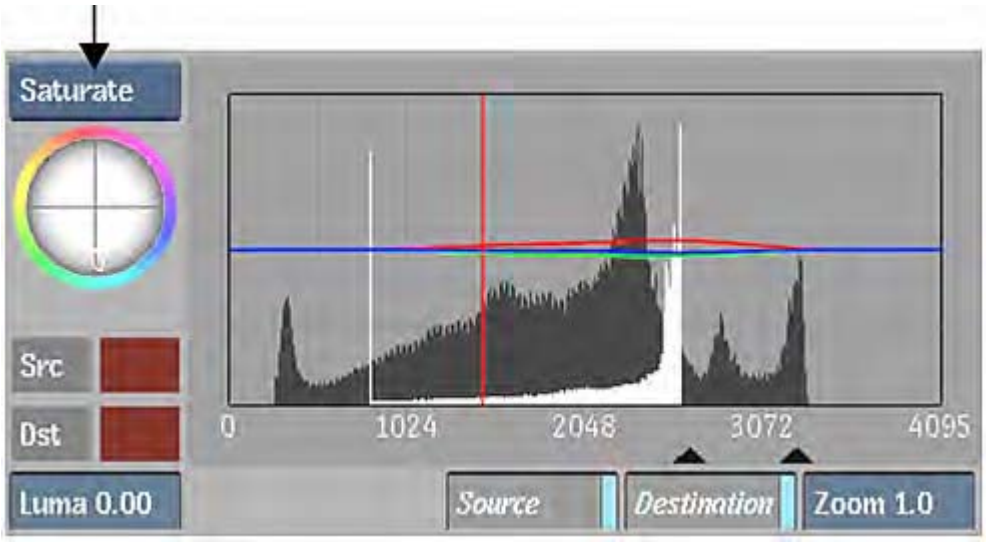
1. Click Basics to display the Basics menu.
2. Drag right in the Saturation field while monitoring the 2D or 3D vectorscope and the image.



The selected colour range (or entire image) moves towards the perimeter of the 2D luma histogram. There is also an increase in colour saturation in the image.

To saturate specific hues of an image:

1. Click Basics to display the Basics menu.
2. Select Saturate from the Trackball option box.



3. Drag the Saturate trackball towards the colour you want to saturate while monitoring the 2D vectorscope. Drag until the colour is closer to the edge of the 2D vectorscope. For example, to saturate reds, move the trackball towards red.

   NOTE To make the trackballs respond to changes in pressure, use the Pointer preferences in the Preferences menu.

4. Release the trackball.
5. To further increase the saturation of the same colour, use the trackball to move the colour closer to the edge of the 2D vectorscope.

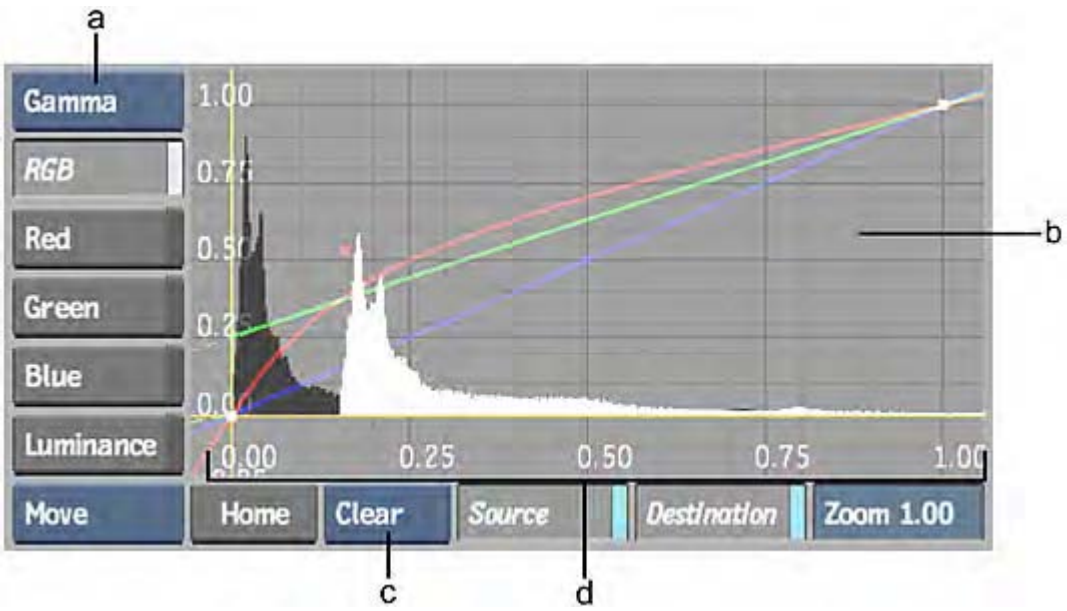
Improving Contrast Using Gamma

When you want to improve contrast by modifying both the chroma and luma content simultaneously without changing the black and white points, use the Gamma controls. You only need to make small adjustments to the RGB gamma curves to improve the contrast.

You can adjust the RGB gamma curves uniformly using the RGB Gamma fields.

You can also use a second set of adjustment curves to tweak each colour channel, as well as the luminance channel, regionally.

After making uniform or regional adjustments, if you are not satisfied with the result, you can reset each channel's gamma or adjustment curve, or clear edited curve values for individual frames.



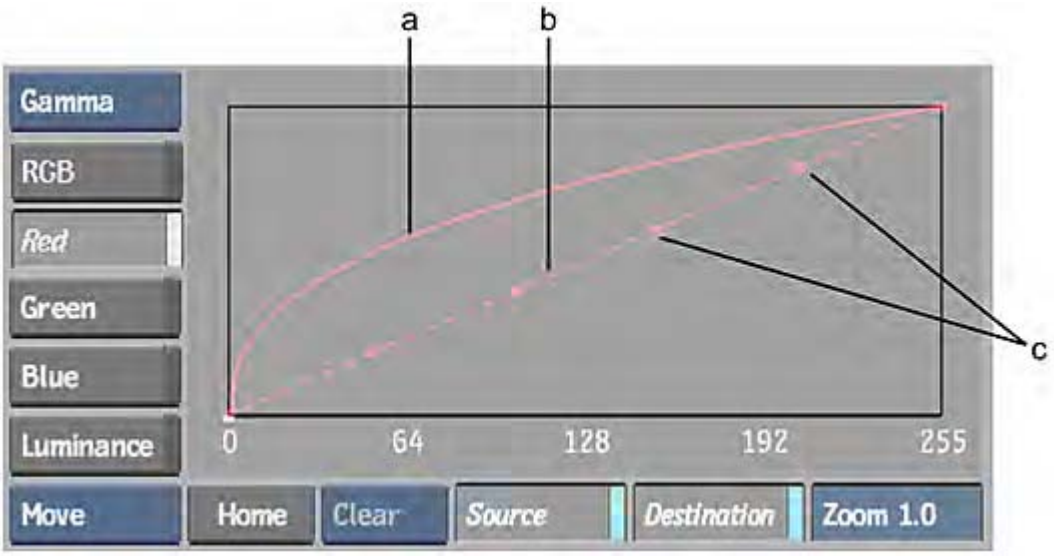
(a) Trackball option box (b) Gamma curves (c) Reset box (d) Luma range

To improve contrast using gamma:

1. Click Basics to display the Basics menu.
2. Select Gamma from the Trackball option box.
3. Display the gamma curve(s) you want to adjust.

Enable:	To display:
RGB	The R, G, and B gamma curves at the same time.
Red, Green, or Blue	The R, G, or B gamma curve individually.
Luminance	The Luminance curve. Use this curve to modify luminance levels locally without affecting the overall luminance.

Two curves are displayed for each channel, the original gamma curve, and an adjustment curve with control points for regional control. Initially these curves overlap.

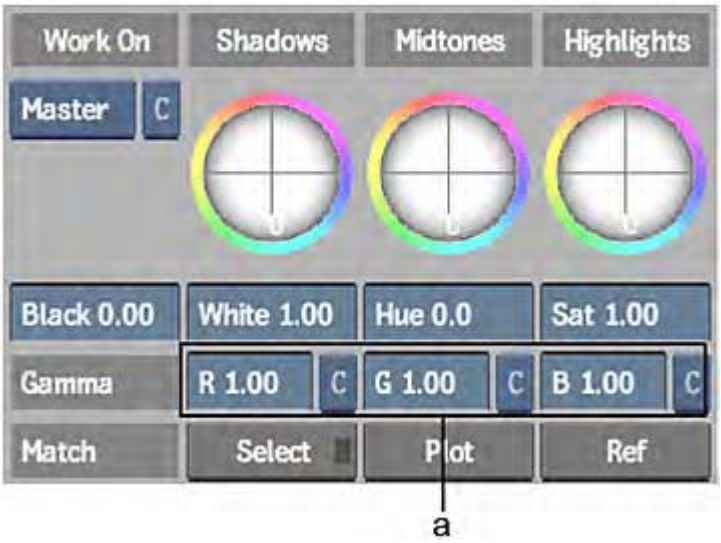


(a) Red channel's gamma curve (b) Red channel's adjustment curve (c) Added control points on adjustment curve

4 While monitoring the 2D vectorscope and image, make a minor adjustment to the gamma curves by doing one of the following:

- To modify R, G, and B gamma values individually across the entire luma range (from black to white), click and drag in the Gamma R, G, and B fields. Drag left to increase contrast or right to decrease contrast. You can also enter a value directly in these fields. Enter a low value to increase contrast or a high value to decrease contrast.

Only the field you modified is updated. The gamma curves are also updated to reflect your changes. The gamma curve is offset from its dotted adjustment curve. Use the adjustment curve to make regional adjustments to the channel.



(a) RGB Gamma Fields

- To modify R, G, and B gamma values proportionally across the entire luma range (from black to white), `Alt`-drag the Gamma R, G, or B field. Drag left to increase contrast or right to decrease contrast.

All three fields are updated. The gamma curves are also updated to reflect your changes. The gamma curves are offset from their dotted adjustment curves. Use the adjustment curve to make regional adjustments to the channel.

TIP To smooth the contrast, you can increase the gamma (raise the curves) and then adjust the Black and White levels in the Basics menu.

- To modify part of the RGB gamma or luminescence range rather than the entire range, use the Edit Mode box to modify the adjustment curves.

Select:	To:
Add	Add control points to an adjustment curve. With Add selected, click either the red, green, blue or luminance adjustment curve to add a control point.
Delete	Delete control points from an adjustment curve. With Delete selected, click a point on the red, green, blue or luminance adjustment curve to delete it.
Move	Move the control points. With Move selected, drag the points to move them.
Zoom	Zoom in on the curves. With Zoom selected, drag over the curves right or left to zoom in or out. You can also press `Ctrl+spacebar` and drag in the curves to zoom.
Rect Zoom	Zoom in on a section of the curves. With Rect Zoom selected, drag a selection box to zoom in on the area of the curves contained by the selection box.
Pan	Pan the curves. With Pan selected, drag over the curves to pan the curves in any direction. You can also press `spacebar` and drag in the curves to pan.

After dragging control points, the image and 2D vectorscope update accordingly.

To reset RGB gamma or luminance curves for all frames in the clip:

1. Do one of the following:
   - To reset individual RGB gamma curves, select R in one of the Channel Reset boxes next to the Gamma R, B, or C field.

   The gamma curve is reset along with its adjustment curve for all frames in the clip. Any control points added to the adjustment curve are removed.

   - To reset the R, G, or B gamma value for the current frame, `Ctrl`-click the GamR, GamB, or GamC field.
   - To reset RGB adjustment or luminance curve values for all frames in the clip, click one of the channel buttons (RGB, Red, Green, or Blue) or the Luminance button to display the curves you want to affect, then select Reset in the Reset box.

   The values for the displayed adjustment curves or luminance curve are reset. Any control points added to the adjustment curves or luminance curve are removed.

To clear RGB gamma or luminance curve values for the current frame:

1. Do one of the following:
   - To clear individual RGB gamma curve values for the current frame, select C in one of the Channel Reset boxes next to the Gamma R,B, or G field.

The gamma curve value is cleared for the current frame, and the gamma curve is updated without affecting its adjustment curve. A keyframe is set with the reset values.

- To clear RGB adjustment or luminance curve values for the current frame, click one of the channel buttons (RGB, Red, Green, or Blue) or the Luminance button to display the curves you want to affect, then select Clear in the Reset box.

The values for the displayed adjustment curves or luminance curve are reset for the current frame. The curves' control points are distributed equidistantly along each curve. A keyframe is set with the reset values.

Matching Colours in Clips

You can match a sample from the result clip to one from a reference clip. Matching clips can improve continuity between clips. You can:

- Match specific colours.
- Match shadows, highlights and overall saturation.

Matching Specific Colours

You can match plot samples to reference samples using the trackballs and by aligning the colours in the vectorscope. For example, you may want to match a red in one image to a different shade of red in another clip. For information on using Plot and Ref controls, see Sampling Clips in the Image Window (page 975).

To match specific colours:

1. Select Result from the View box.

   The result clip is displayed in the image window.
2. If necessary, position the split bar so that both a reference clip and the result clip appear.
3. Enable Plot and then sample a colour in the clip you want to modify (the result clip).

   A red plot line is displayed in the 2D luma histogram indicating the luma value of the sample. In the 2D vectorscope, you can view the saturation and hue of the sample (outlined in red). In the 3D vectorscope, you can view the sample in terms of HLS.
4. Enable Ref and then sample the colour in the reference image you want to match.

   A yellow reference line is displayed in the 2D luma histogram indicating the luma value of the sample. In the 2D vectorscope, you can view the saturation and hue of the sample (outlined in yellow). In the 3D vectorscope, you can view the sample in terms of HLS.
5. Use the 2D luma histogram in the Basics menu and 2D vectorscope in the image window to determine which controls you need to use.

Drag:	When:
White or Black field	You need to adjust the luma values in the highlights or shadows without modifying the chroma. Drag the field so that the plot line in the 2D luma histogram moves towards the reference line.
Highlights, Midtones, or Shadows trackball	You need to adjust the chroma values in the image without modifying the luma. Drag the trackball(s) so that the plot sample in the 2D vectorscope moves towards the reference sample.

6 Use the necessary controls to modify the clip until the plotted sample is aligned with the reference colour in the 2D luma histogram and 2D vectorscope. If necessary, increase or decrease the saturation as well as shift the hue.

TIP View the sample in the 3D vectorscope to make more refined modifications.

Matching Shadows, Highlights, or Overall Saturation

Use the Match feature to match a range of colours in the result clip to a range of colours in a reference clip. You can match highlights, shadows, or the overall saturation of an image.

NOTE Match is not designed to match specific colours.

To match shadows, highlights, or overall saturation:

1 Select Result from the View box.

The result clip is displayed in the image window.

2 Position the split bar so that both a reference clip and the result clip appear.

3 Click Match.

The following message appears in the message bar: "MATCHING: Select an area to be modified."

4 Click and drag on the result clip to draw a box around a range of colours. You can select an area of the image that is predominantly black, white, or contains a wide spectrum of colours. To match the overall saturation of an image, select a larger area of the image.

The blue indicator on the Match button is enabled, indicating that you need to select a destination sample.

5 If necessary, position the split bar to show more of the reference image.

6 Click Match.

The following message appears in the message bar: "MATCHING: Select an area to match to."

7 Click and drag to draw a box around a range of colours in the reference clip. You should match whites with whites, blacks with blacks, or select a broad range of colours on both images to match saturation. Trying to match completely different colours will produce unpredictable results.

The range in the result clip is matched to the range in the reference clip.

8 Repeat these steps as often as necessary to obtain the best possible match between colours. You can use Match repeatedly to make your colour match more precise.

9 To refine the result, use other controls in the Basics menu (such as the Saturation field), and then reuse Match.

To clear a Match operation:

1 Click the Undo box.

To cancel a Match operation in progress:

1 If you begin using Match and want to cancel the matching process, adjust any value using the trackballs or fields.

The blue indicator on the Match button will be disabled, indicating that the matching process has been cancelled.

Performing Hue Shifts

You can use the Hue field in the Basics menu to perform hue shifts on the entire image or a range in the image defined by a selective's matte. Perform hue shifts when you want to change the colour of an object or create other artistic effects. Hue shifts do not affect the luma.

You can also perform hue shifts when you want to suppress colour spill. See Removing Colour Spill (page 997).

To perform a hue shift:

1. If necessary, create a matte for the range of colour you want to change. See Selecting Colour Ranges for Colour Correction (page 977).
2. Click Basics to display the Basics menu.

   If you are using a selective, the Selective View box should display Result when you display the Basics menu.
3. Sample the image to see the hue shift in the 2D luma vectorscope. Enable Plot and then sample the colour range in the image you want to hue shift.

   A red line also appears in the 2D luma histogram and the sample is outlined in the 2D or 3D vectorscope.
4. Drag the Hue field while monitoring the 2D or 3D vectorscope and the image.

   The selected colour content (or entire image) shifts through the colour range.
5. If necessary, you can change the saturation of the hue by dragging the Saturation field—drag right to increase saturation or left to decrease it.

   The selected colour range moves towards the perimeter or centre of the 2D luma histogram.

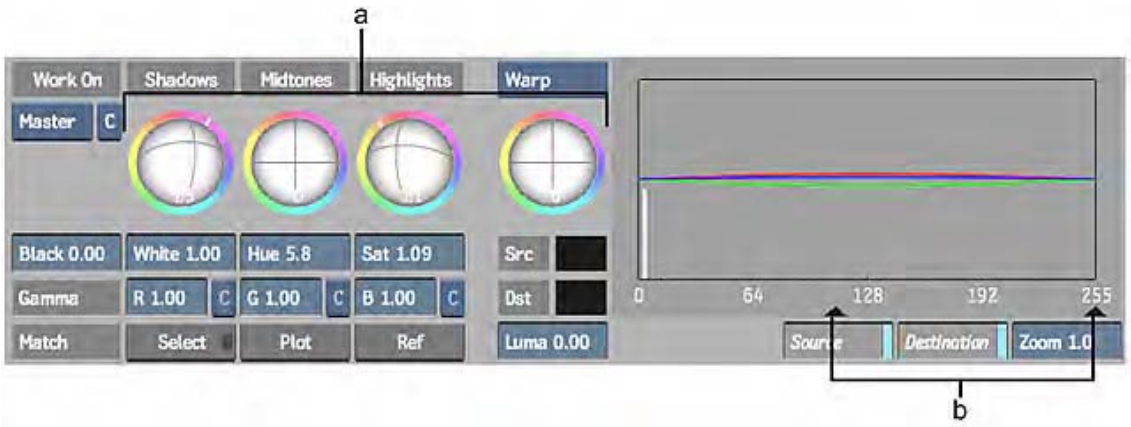
Creating Colour Casts

You can use the Midtones trackball to create a colour cast that affects the chroma in the entire image without affecting the luma. Create a colour cast when you want to emphasize a colour range to create an artistic effect or when you want to refine an existing cast. For example, create or refine a bluish cast to convey a cooler atmosphere.

NOTE Before you create a colour cast, you should correct colour imbalances in the image or clip. See Correcting Colour Imbalances (page 983).

To create a colour cast:

1. Click Basics.



(a) Trackballs (b) Midtones triangles

TIP You can determine the colour of an existing cast by dragging right in the Saturation field to temporarily increase the saturation while monitoring the sample in the 2D vectorscope. Click Undo once you determine the colour of the cast.

2 To add a colour cast to the entire image, drag the midtones triangles so that they span the entire luma range.

3 While viewing a vectorscope and the image, drag the Midtones trackball towards the colour you want to increase in the clip or image. For example, to add more yellow to the image, drag the Midtones trackball towards yellow.

In the 2D or 3D vectorscope, the colours update to reflect your changes. In the image window, the colour in your clip changes according to the direction the trackball is being dragged.

NOTE When you drag the Midtones trackball, the RGB curves in the 2D luma histogram also reflect the changes you make.

4 To increase a range of colour in the highlights or shadows of the image, drag the Highlights or Shadows trackball.

5 To increase or decrease the saturation of the cast, drag the Saturation field—drag right to increase saturation and left to decrease it.

TIP You can make slight modifications to the hue of the cast using the Hue field.

Readjusting Colour After a Colour Correction

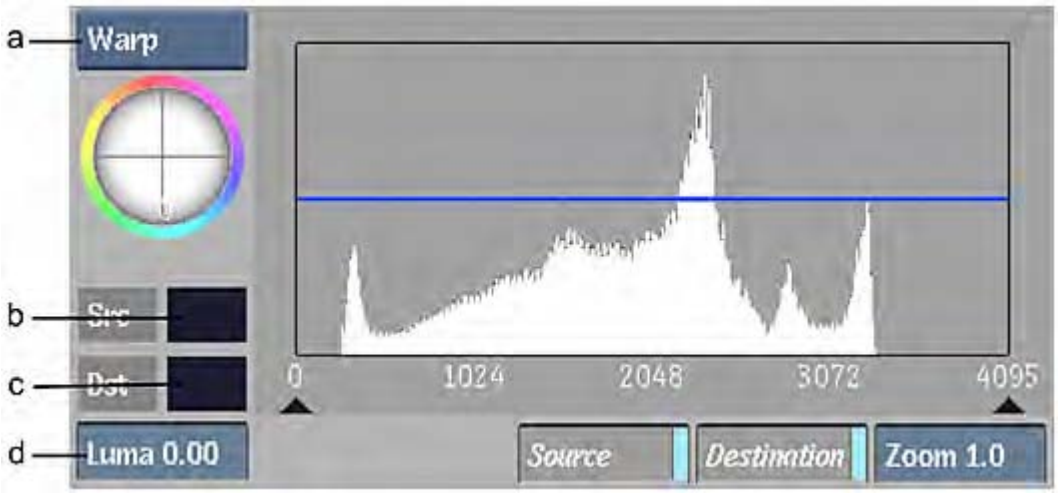
When colour correcting clips, colours that you do not want to alter may be modified in the colour correction process. Use the Warp trackball in the Basics menu to readjust a specific range of colour.

You can also adjust the luminance of the selected colour using the Luma field.

To readjust colours after a colour correction:

1 Click Basics.

The Basics menu appears.



(a) Trackball option box (b) Source Colour pot (c) Destination Colour pot (d) Luma field

2 Select Warp from the Trackball option box.

3 Click Pick and then sample a colour in the clip you want to modify.

The colour you select is outlined in white in the 2D or 3D vectorscope. A cyan bar representing the colour appears in the 2D luma histogram. The Src Colour pot displays the colour.

4 Click the Src Colour pot to set the source colour.

The Dst (destination) Colour pot is updated to match the source colour.

5 Move the Warp trackball towards the destination colour.

The Dst Colour pot dynamically updates to reflect your changes. When you release the mouse, the Src Colour pot is updated to match the destination colour. In the 2D or 3D vectorscope, the selected colour moves to reflect the changes you are making to the image.

NOTE To make the trackballs respond to changes in pressure, use the Pointer preferences in the Preferences menu.

6 To adjust the luminance of the destination colour, adjust the Luma field. When you adjust the luma of the destination colour, the entire luma content of the image is adjusted around the selection. The selected luma value changes the most.

The Dst Colour pot dynamically updates to reflect your changes. When you release the mouse, the Src Colour pot is updated to match the destination colour. In the 2D luma histogram, the cyan bar moves to reflect the changes you are making to the luma content.

7 Release the trackball or click the Luma field.

The Src Colour pot is set to the Dst box colour.

Removing Colour Spill

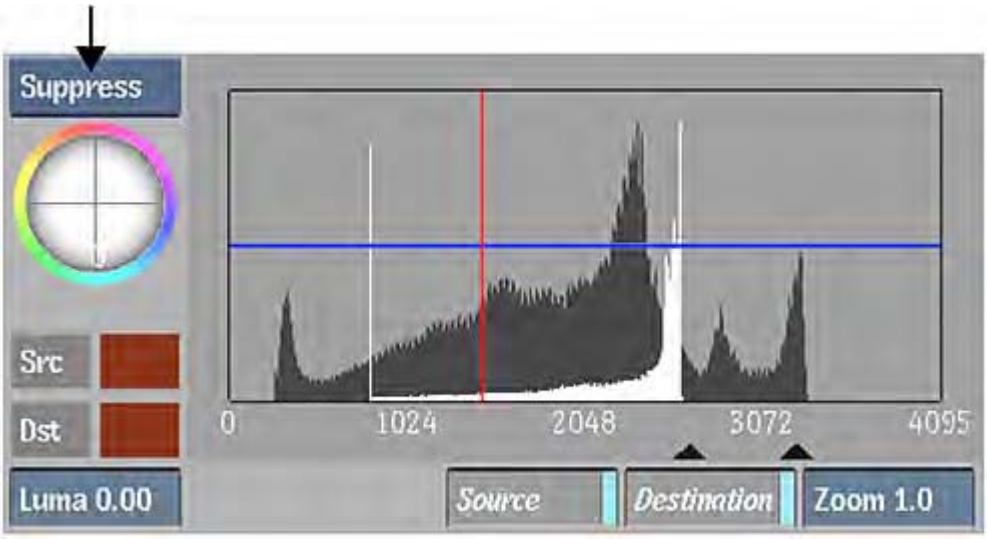
When you key clips in the Modular Keyer, there can be colour spill. You can refine the key by suppressing colour spill in the Colour Warper. Use the Saturate and Suppress trackballs in the Basics menu. Use the Suppress trackball to suppress the colour spill and the Saturate trackball to correct any unwanted colour suppression. When you use these trackballs to modify colour in the image or clip, a range of colour is affected.

In the Modular Keyer, place the Colour Warper node in the CBlend pipe.

To remove a colour spill:

1 Click Basics to display the Basics menu.

2 Select Suppress from the Trackball option box.



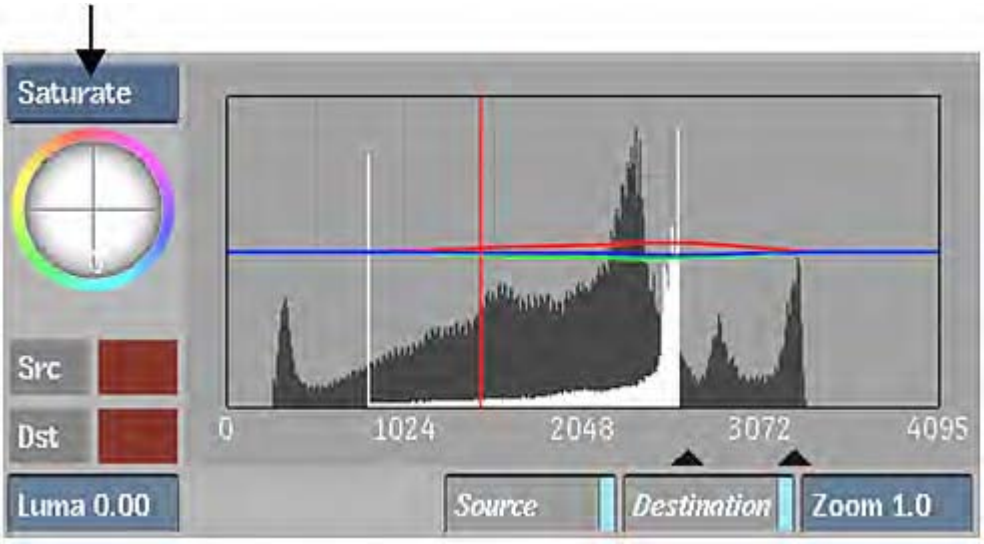
NOTE To make the trackballs respond to changes in pressure, use the Pointer preferences in the Preferences menu.

3 Move the Suppress trackball towards the colour that you want to suppress. For example, to remove a blue background, move the trackball so that the blue colours appear in the centre of the 2D vectorscope.

NOTE The colour you are suppressing appears in the middle of the 2D vectorscope at full suppression.

By comparing the Result clip and the Front clip, you may notice some unwanted colour suppression to the colour that is opposite (on the colour wheel) the one you just suppressed. You can fix such suppression with the Saturate trackball.

4 Select Saturate from the Trackball option box.



5 Move the Saturate trackball towards the colour that you want to saturate.

The colour that you are saturating moves closer to the edge of the 2D vectorscope.

6 It may be necessary to adjust colours in the image using the Suppress and Saturate trackballs several times to achieve suitable results. Continue adjusting the trackballs until you are satisfied with the result.

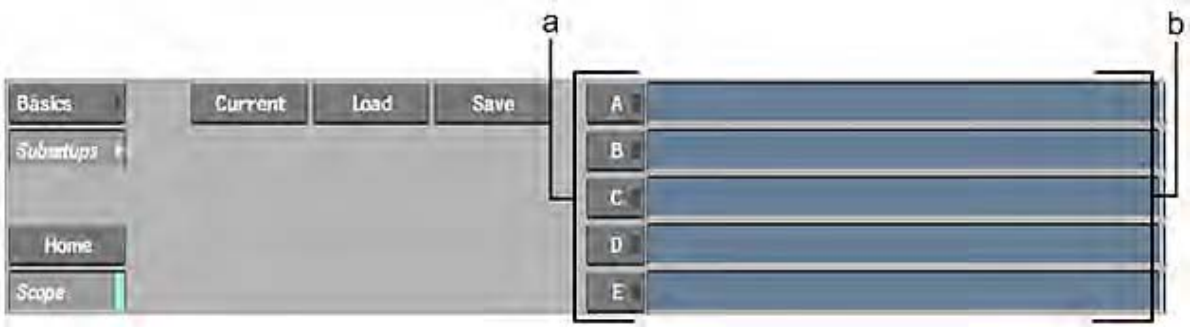
Creating Subsetups

Subsetups are Colour Warper setups selected at a specific moment at a specific frame. Subsetups are for the current frame only and do not include entire animations. You can select up to ten subsetups and compare colour corrections in other subsetups. You can save and load a group of subsetups.

NOTE The group of subsetups loaded affects all Colour Warper nodes in the current setup since subsetups are shared between Colour Warper nodes. Ten subsetups (A-J) constitute a group.

Use the Subsetups menu to:

- Selectively store intermediate setups.
- Compare any subsetup to the current setup.
- Share subsetups between Colour Warper nodes.
- Save and load a group of subsetups.



(a) Subsetup buttons (b) Subsetup fields

Storing and Retrieving Subsetups

You can store and retrieve subsetups using the Subsetup buttons.

To store and retrieve the current setup:

1 Press and hold one of the subsetups buttons until a message appears indicating "Storing subsetup <subsetup letter>" and release.

The LED indicator shows that the subsetup is stored. All active subsetups have a LED indicator. If you have not stored anything into a subsetup, the active light of the button will be off.

2 Use the setup at another frame or in another node by pressing the corresponding Subsetup button.

Naming Subsetups

You can name your subsetups. This allows you to store setups and later compare them to your current setup, or to apply the same correction to different frames in a clip.

To name a subsetup:

1. Click in a subsetup field, type in a name or description, and press `Enter` to store the setup.

Replacing Subsetups

You can replace or override a setup.

To replace an existing setup:

1. Click and hold a subsetup button to overwrite and replace an existing subsetup. A keyframe is set and the new subsetup is saved.

Comparing Subsetups

The current setup is accessed with the Current button and contains the most recently committed setup. Compare any of your subsetups to each other or to the contents of the current setup.

To commit the contents of the current setup:

1. Perform an operation such as advancing a frame or changing a value in the Basics menu. Your present settings will be displayed the next time you click Current.

To compare a subsetup:

1. Click a Subsetup button of the subsetup you want to compare.
2. Click Current to view the current result.

Saving and Loading Subsetups

Use the Save button in the Subsetups menu to save a group of subsetups. Then use the Load button in a later session to reload the same group of subsetups.

To save a group of subsetups:

1. In the Subsetups menu, click Save.
   The file browser appears.
2. Browse to a different directory path if necessary and then type a name in the Name field.
3. Click Save.
   The group of subsetups is saved in the specified directory.

To load a group of subsetups:

1. In the Subsetups menu, click Load.
   The file browser appears.
2. Browse to the appropriate path and select the name of the group of subsetups.
   The group of subsetups is loaded.

Painting 22

Paint Node

Paint Node is a system that provides a scalable matte painting, retouching, or restoration workflow in Batch or Batch FX.

Due to its underlying technology, Paint Node automatically scales strokes when changing the resolution, ratio, or bit depth of input clips, or when switching from Full Resolution to Proxy mode. Paint Node supports "clipless" setups, which can be applied to any image input, while accurately reproducing the sequence of painted strokes.

The Paint Node accepts a front and matte clip as input, and creates a result and output matte clip, respectively. You can paint on the result and output matte, with a selection of brushes in different paint modes.

Paint Node also allows you to connect multiple sources and use them to paint the contents of source images onto the result. This paint operation, applied with the Reveal paint tool, can be used with in-context overlay over the result image. A front and matte clip can be connected as a source by connecting the clips to a source node. The content of source front and source matte input can be used to create brush strokes on the result and output matte.

Accessing Paint Node

Paint Node can be accessed from:

- Batch FX. (page 1001)
- Batch, then select a node from the Node bin. (page 1002)
- Modular Keyer. (page 1002)

Accessing the Paint Node from Batch FX

To access Paint Node through Batch FX:

1. Select Timeline.
2. Select the segment.
3. Click the FX button.

4 Click the Create Batch FX button.

5 Drag a Paint Node into the schematic.

The Paint Node is now in the schematic.

6 Double-click the Paint Node.

You are in the Paint Node editor.

Accessing the Paint Node from Batch

Paint Node is accessed from Batch.

To access the Paint Node from Batch:

1 Select Batch.

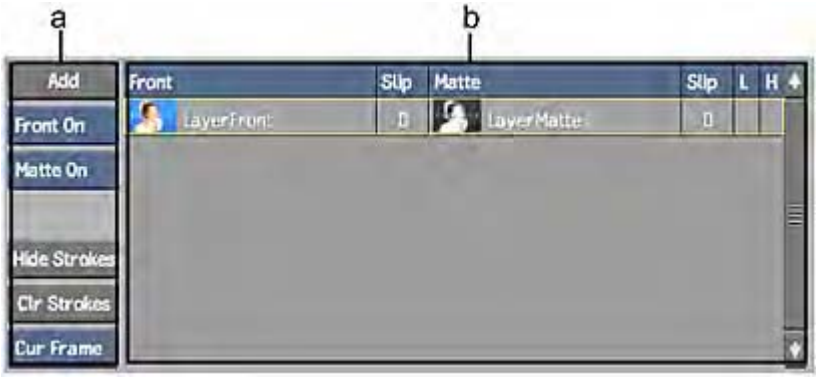
2 In the Batch node bin, double-click the Paint node or drag the node to the Batch schematic.

3 Attach a clip or node output to the front tab.

4 Attach a clip or node output to the matte tab. If a matte clip is not attached, the node interprets the missing input as a white clip.

5 In the Batch schematic, double-click the Paint Node.

The Paint Node menu appears.



(a) Source controls (b) Sources List

(a) Current Colour Pot (b) Current Brush

NOTE The Parent and Delete options in the Paint Tools box are standard Batch features and are not used in Paint Node.

Accessing the Paint Node from the Modular Keyer

To add a Paint Node in the Modular Keyer:

1 Double-click on, or drag, a Paint Node from the node bar and add it to the schematic.

2 Click the Paint Node.

You are in the Paint editor.

Using Sources

Sources are additional image data that can be used in Reveal paint operations and displayed as an overlay on the result or output matte. You can scale, rotate, and offset a source. Any transformations applied to a source will be displayed in Reveal paint strokes.

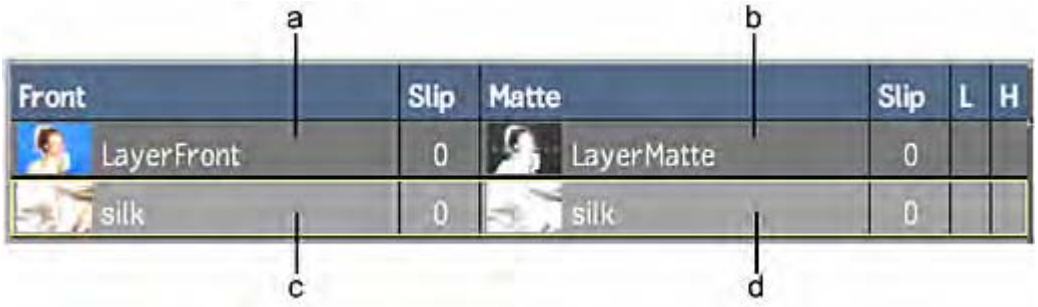
In the Paint Node menu, all sources appear in the Sources list. The Sources list allows you to select sources to be hidden, cleared, or displayed as an overlay. The first entry in the list is the front clip and matte clip.

In the schematic, source clips are connected to a Paint Node through a source node, which accepts a source front and source matte input.

A source can be a clip or the result of any process, and can include a front and/or matte clip. You can add a source node from the menu or the schematic.

If a clip with frames that are missing media is attached to a source node, the Paint Node processes incoming frames set to No Media as transparent. Strokes created with the source are stored, although they are not visible on the canvas at frames with No Media input.

When adding a source input, it is recommended that you use clips as input, or cache the node closest to the source node instead of parenting a large tree to a source node. This speeds up processing when changing frames, making for a more interactive experience.



(a) Front (b) Matte (c) Source front (d) Source matte

When you delete a source, it is removed from the Sources list, along with its associated strokes. When you clear a source, it remains in the Sources list, but all strokes are removed.

Adding a Source

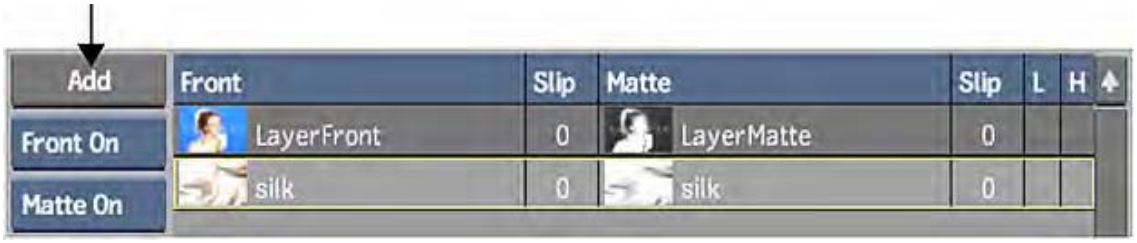
When you add source, it includes both source front and source matte input. When you add a source front only, a black matte will automatically be created.

To add source node, front source and matte source:

1. In the Paint Node menu, click Add.

   The Viewing Panel appears.
2. Select a source front clip, and then a source matte clip.
3. Click Exit Clip Select to return to the Paint Node menu.

   In the Sources list, the source is added. In the schematic, the process tree is updated: a source node with the selected clips is connected to the Paint node.

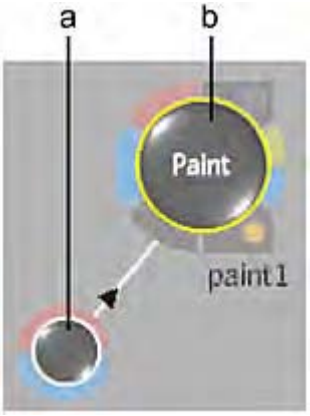


4 To help identify source nodes in the schematic, select a source in the Sources list to automatically highlight in orange the corresponding source node and link to the Paint Node. Conversely, selecting a source node also highlights the corresponding source in the Sources list.

To add a source node only:

1 Hold `Ctrl` and click Add to add a source node.

The source node is added and connected to the Paint Node.



(a) Source node (b) Paint Node

2 Connect clips or process tree input to the source node's front and matte inputs.

Removing a Source

You can remove a source clip from the Sources list. If you disconnected a source clip from the source node, the entry exists in the Sources list, but the clip name and proxy no longer appear. Because the stroke information is stored in the node, you can reconnect the same clip or a different clip to the source node, and the stroke information is preserved. All strokes in the canvas using the source are repainted using the new source clip.

When you delete a source node, the entry is removed from the Sources list, and the strokes stored in the node are deleted permanently.

To remove a source clip from the Sources list:

1 In the schematic, disconnect the source clip from the source node.

To delete a source node:

1 From the Edit Mode box, select Delete (or use the keyboard shortcut), and click the source node in the schematic.

Clearing Strokes on a Source

You can clear strokes from the canvas that were created using a source front or source matte.

WARNING Clearing strokes for an entire sequence cannot be undone.

To clear the strokes associated with a source:

1. In the Sources list, select the source used to create the strokes you want to clear.
2. From the Selection Mode box, select Current Frame or Sequence.



3. Click Clear Strokes.



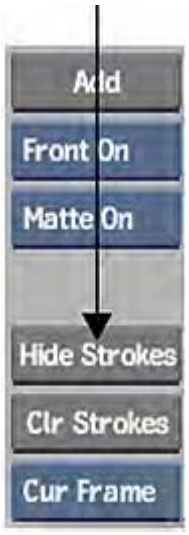
Hiding Strokes

You can hide a group of strokes that were created from the same source clip.

Hiding strokes is useful when you need to focus on a particular component of your image: hide strokes on the canvas that do not require attention. This feature allows you to concentrate on the current stroke. It also improves the system's interactive performance.

To hide strokes on a source:

1. Select the source used to create the strokes that you want to hide.
2. In the Sources list, click Hide Strokes (H).



   A check mark appears in the H column of each source.
3. Click Hides Strokes again to make the strokes visible.

Restricting Strokes with the Source Matte

You can limit the sections of the source that are painted onto the canvas using the source's matte. To limit painting on the canvas using the matte. See Restricting Brush Strokes (page 1014).

To limit source brush strokes using a source matte:

1. In the Paint Modes box, select Reveal (E).
2. In the Sources list, select a source.
3. From the source controls, select an option in the Matte Source box.

Select:	To:
Matte Off	Paint anywhere on the canvas.
Matte On	Limit painting to areas in the source matte.
Matte Invert	Invert the source matte and limit painting to areas outside the source matte.

4. Draw strokes.

   Brush strokes only use areas of the selected source that are delimited by its source matte.

Locking Sources

Lock a source to a specific frame. For the duration of the clip, only the source front and source matte at the selected frame is used.

To lock a source:

1. Select the source you want to lock.
2. Using the playback controls, choose the frame at which you want to lock the clip.
3. In the Source Front option box, select Front Lock.



   A check mark appears in the L column of each selected source.
4. In the Source Front option box, select Front On to unlock the source.

Displaying Sources

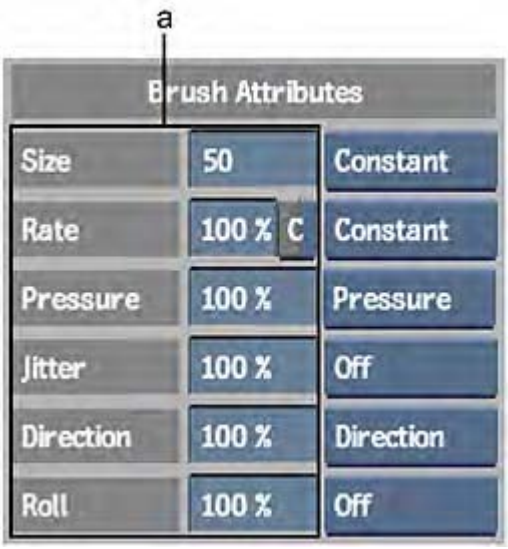
Paint Node offers an intuitive way of previewing the effects of a Reveal operation. In this view, the selected source is superimposed over the result, and the Transparency field is activated to allow for transparency adjustments. This feature allows you to see exactly what your brush strokes will reveal. See Previewing a Reveal Operation Using a Reference Image (page 1024).

Brush Attributes and Attribute Modes

You can set various brush attributes and attribute modes to determine how paint is applied to the image.

Setting Brush Attributes

The Brush Attribute fields set the size of the brush, the distribution of the paint, and the rate and direction of the paint application.



(a) Brush Attribute fields

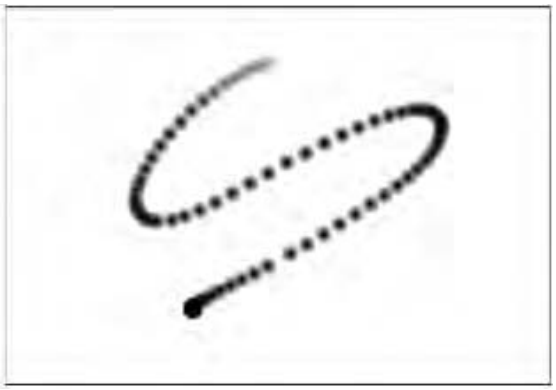
Opacity Affects the transparency of the brush. A value of 100% applies a fully opaque colour. Use a lower value to apply a more transparent colour.

Size Determines the size of the brush, which is indicated by the diameter of the green dashed circle surrounding the cursor brush. To increase the brush size, press S and drag the brush to the right on the canvas. To decrease it, press S and drag to the left.

Rate Determines the rate at which brush strokes are applied to the canvas. Use a high value to produce a smooth continuous stroke, or a low value to produce a less continuous stroke with larger gaps between brush images. The C (Constant) button in the Rate field determines whether the consistency of the stroke is dependent on the speed at which you move the pen or mouse. For example, to create a stroke of evenly spaced dots, you could set the Rate field to 20 percent and enable Constant.



Rate attribute value = 100



Rate attribute value = 25

Pressure Affects the transparency of the paint applied to the image. To apply opaque paint, use a high percentage value. For more transparent paint, use a low value. The Pressure attribute differs from the Opacity

attribute in that you can set the Pressure attribute mode so that the paint transparency varies according to the pressure applied to the pen or the direction of the brush.

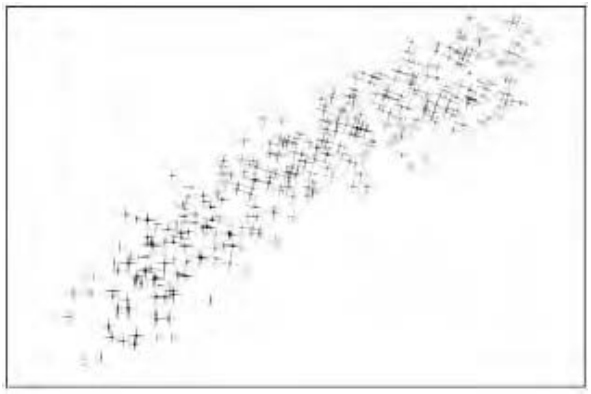


Pressure attribute value at 100%

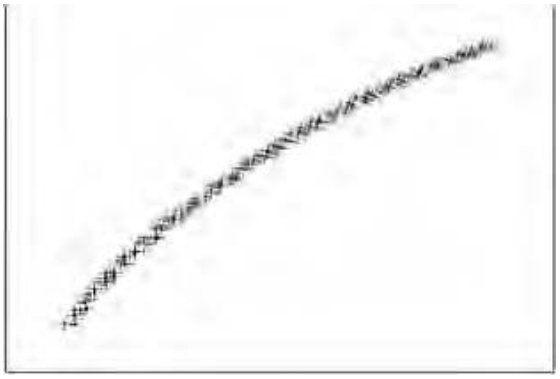


Pressure attribute value at 50%

Jitter Randomizes the brush strokes applied to the image. A high value produces a greater dispersion of paint, while a low value produces a greater concentration.



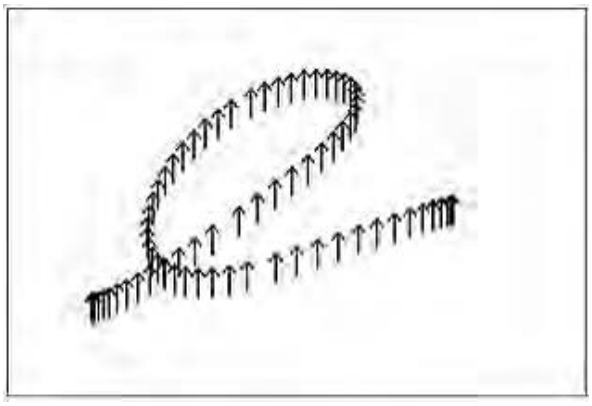
Jitter attribute value at 100%



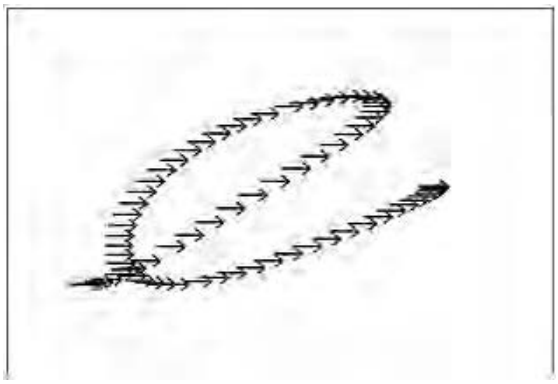
Jitter attribute value at 10%

Direction Causes the brush to rotate around the Z-axis and can be used to produce a calligraphy effect. The effect of the Direction attribute is most noticeable when used with one of the elliptical or star brushes. The

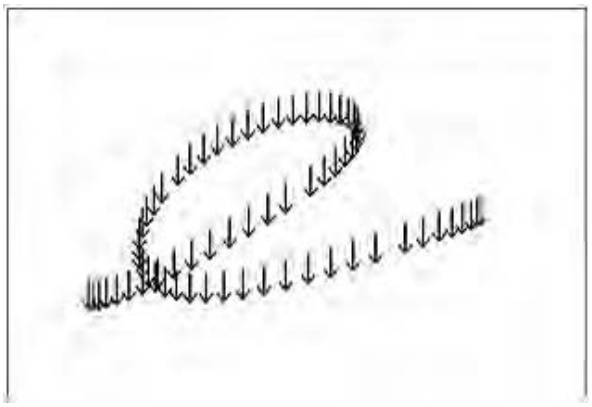
value of the Direction attribute causes the brush strokes to rotate by 90 degrees for each increment of 25 percent.



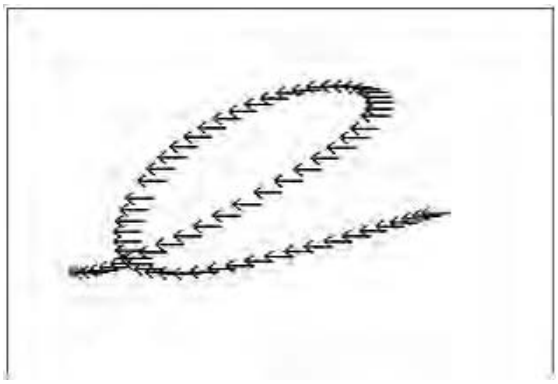
Direction attribute value at 100%



Direction attribute value at 75%



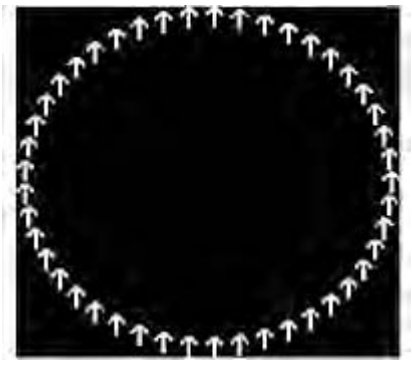
Direction attribute value at 50%



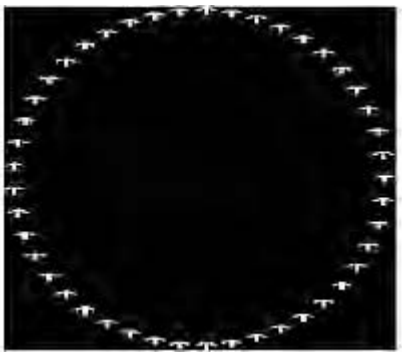
Direction attribute value at 25%

Roll Rolls the brush around the X-axis. The effect of the roll is most noticeable when used with one of the non-symmetrical brushes. For each increment of 25 percent, the Roll attribute value creates a rolled brush stroke of 90 degrees.

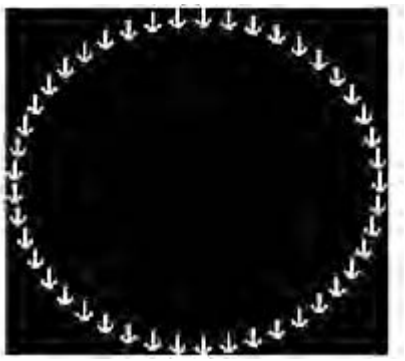
NOTE You can use any attribute mode with the Roll attribute.



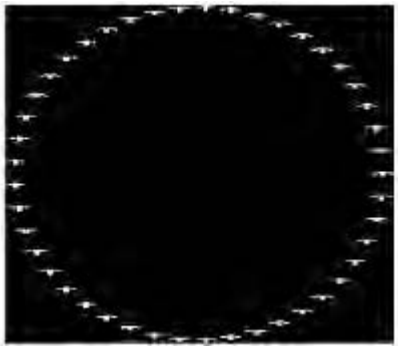
Roll attribute value at 100%



Roll attribute value at 75%



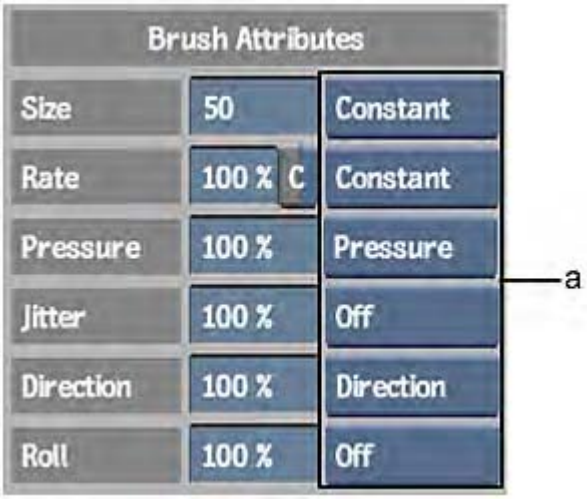
Roll attribute value at 50%



Roll attribute value at 25%

Setting Brush Attribute Modes

The value of a brush attribute depends on the selected attribute.



(a) Brush Attribute Mode boxes

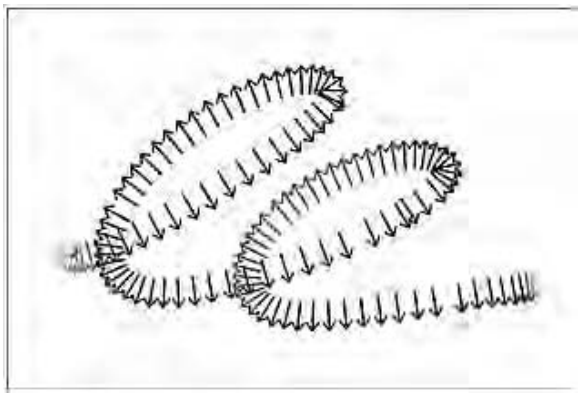
Off Disables the brush attribute.

Constant Keeps the brush attribute values constant.

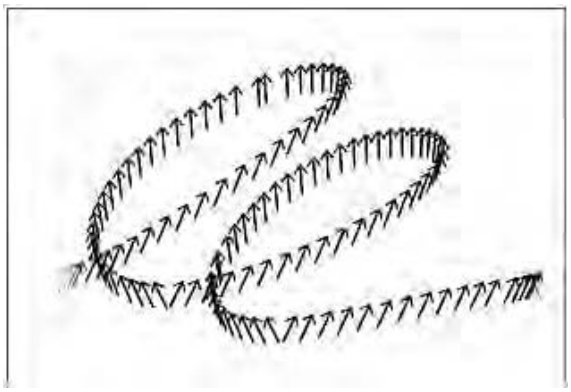
Pressure Uses the pressure exerted on the pen as the reference value. The harder you press on the pen, the greater the brush attribute value. The softer you press, the lower the brush attribute value.

Direction Uses the direction of the brush stroke as the reference. When used with the Direction mode, the Direction attribute causes the brush strokes to follow the trajectory of the brush. Increase the value to enhance

the effect on brush direction. Drag right to use 100% of the brush attribute value, left to use 0%, up to use 25%, or down to use 75%.



Direction attribute value at 100%, Direction attribute mode



Direction attribute value at 15%, Direction attribute mode

Fade Uses the percentage value entered in the Fade field to determine the length of time that elapses before the brush stroke fades. The greater the percentage, the faster the stroke fades. A value of 100 in the Fade field causes the stroke to fade quickly, whereas a value of 1 causes the stroke to last longer. Also, strokes applied quickly will be longer than those applied more slowly.



Pressure attribute value at 100%, Fade attribute mode at 35%



Pressure attribute value at 100%, Fade attribute mode at 85%

Front, **Matte**, **Source Front and Source Matte** Allows you to use the red channel in a reference image to set brush attributes. Source Front and Source Matte mode use the red channel of the selected source in the Sources list.

Painting Brush Strokes

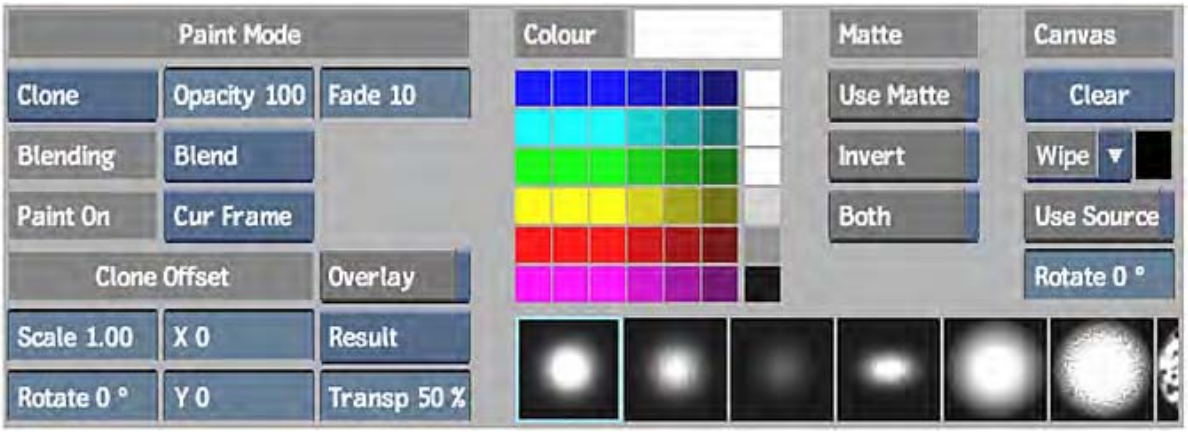
You can paint on the canvas using a variety of brush types and colours. Once you select a brush type, you can modify brush attributes to further refine your strokes. A stroke can be applied to a single frame, or simultaneously to multiple frames. You can paint on the result or output matte, or you can use matte paint mode controls to paint on the result and output matte at the same time.

You can restrict brush strokes to the area delimited by the matte. For example, assume that you have a front input that includes a person shot against a blue screen and a corresponding matte clip. If you opt to limit brush strokes to the matte, your brush strokes on the result or output matte appear only inside the matte

area and, therefore, over the person. If you then limit brush strokes to the existing strokes, new paint strokes are applied only over strokes that have already been applied.

Applying Brush Strokes

You can paint with either a pen or a mouse, and access various pressure settings.



To apply a brush stroke:

1. Select an option from the Paint Tools box.
2. Select an option from the Paint On box to apply the stroke to the current frame, all frames in the sequence, or the current frame and subsequent frames.
3. Select the Result (`F4`) or Output Matte (`F4 F4`) to set the view where you are painting.
4. If you want to paint on the result and output matte simultaneously, enable Both.
5. Click a brush to select it.
6. Set brush attributes. See Brush Attributes and Attribute Modes (page 1007).
7. If you will reveal a source on the image, select the source in the Sources list. See Revealing Sources (page 1019).
8. To change the colour, click a colour pot. If you are painting on the output matte, the brush colour is grey with an equivalent luminance value.

   TIP As an alternative to using the colour pots, you can designate the colour under the pointer as the current brush colour by pressing the left `Ctrl` key and clicking on the canvas.

9. Click and drag on the canvas to draw a stroke.

   NOTE Downstream context views are not automatically updated as you add strokes, since this would cause performance degradation. To force an update, click the Update button.

10. Use the eraser end of the pen if you are using the stylus to erase strokes that you have created.

Removing Brush Strokes

You can undo multiple brush strokes in Paint Node. You can use the Undo button to remove brush stroke operations, beginning with the most recent strokes. Set Undo level operations in the Preferences menu.

If you are using a stylus, remove brush strokes manually by using the back of the stylus to “erase” strokes. If the result is displayed, erasing reveals the front clip on the canvas. If the output matte is displayed, erasing reveals the matte.

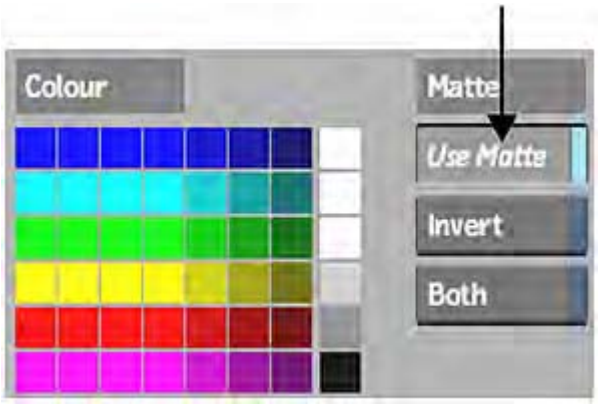
Restricting Brush Strokes

You can limit the area where brush strokes are applied on the canvas. Use the matte paint mode controls to limit painting on the canvas to the matte. You can also invert the matte to limit paint to only the areas outside the original matte.

To limit which regions of a source front can be used by its source matte, use the Matte Clip option box in the source controls. See Restricting Strokes with the Source Matte (page 1006).

To limit where brush strokes are applied using the matte:

1. Enable Use Matte.



2. If you want to paint outside of your selection, enable Invert.
3. Draw strokes.

   Brush strokes are applied only to the areas included in your selection.
4. Disable Use Matte to turn off matte restrictions.

Scaling Brush Strokes

You can set scaling options for brush strokes associated with the Paint Node. By specifying the previous resolution of an input clip before it was resized, the brush strokes applied to the clip can also be scaled based on these settings.

To display the stroke scaling options:

1. Double-click the Paint node.
2. Click Node Setup.



(a) Aspect Ratio Presets box (b) Project Resolution Presets box

Project Resolution Presets box Provides preset aspect ratio options and an option to use a custom resolution. Set this option to indicate the previous resolution of the clip.

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

Aspect Ratio Presets box Provides standard frame aspect ratio options and a w:h option to use a ratio based on the values entered in the Width and Height fields. Also provides a Custom option so you can enter a frame aspect ratio in the Ratio field.

Ratio field Displays the original aspect ratio of the clip. When Ratio is set to Custom, this field becomes active so that you can enter a custom frame aspect ratio.

Stroke Resize box Select a fit method option to be applied to the clip.

Select:	**To:**
Centre/Crop	Center the strokes over the destination frame. If the clip at the original resolution is larger than the destination, the strokes are cropped.
Crop Edges	Fit one edge of the original clip input into the destination frame without stretching or squashing the frame. Excess parts of the original clip after resizing are cropped.
Fill	Fit the original stroke width and height into the destination frame. If the clip at its original resolution and destination frames do not have the same aspect ratio, the brush strokes can become distorted.
Letterbox	Fit the original stroke to the destination frame without squashing or stretching it, and without cropping the source.

Keep Aspect button Enable to preserve the aspect ratio of non-square pixels. This button only appears if you selected Crop Edges or Letterbox in the Fit Method box.

Stroke Pan fields Enter an X and Y value to offset existing strokes on the result. You can also reposition strokes by holding down `Ctrl+Shift` and panning the image.

Current Resolution fields Displays details of the current resolution of the clip.

Active button Enable to activate smooth filtering of pixels for enhanced display

Using Paint Modes

You can select the type of special effects you want to apply to the brush from the Paint Modes box. Strokes applied with these brushes are processed by the graphics hardware, improving interactive performance. To paint sources onto images, see Revealing Sources (page 1019).

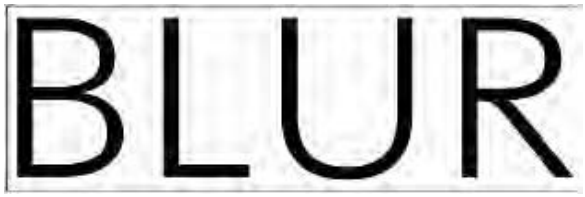
To select a brush:

1. Switch to either Result (F4) or Output Matte (F4 F4) view.
2. From the Paint Modes box, select a brush. Refer to the sections that follow for instructions on using each brush.

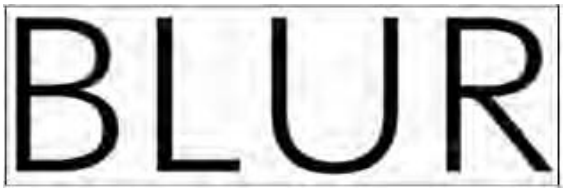
   TIP When a brush is selected, brush opacity and blend modes are disabled. To achieve an effect similar to modified brush opacity, use the Pressure attribute mode.

Blur

Apply a blur filter to the image with each stroke.



Original image



After using the Blur brush

To use the Blur brush:

1. From the Paint Modes box, select Blur (R).
2. Set the brush size according to the size of the area you want to blur.
3. Paint on the image.

Impression

Paint on colours from a reference clip. When you click on the canvas, the colour at the center of the brush is used to fill the entire brush stroke.



Original image



After using the Impression brush

Image courtesy of Buzz Image Group, Inc., and Les Films Stupefiants

Image courtesy of Buzz Image Group, Inc., and Les Films Stupefiants

To use the Impression brush:

1. From the Paint Modes box, select Impression.
2. Set the brush size.
3. Paint on the image.

Smear

The Smear brush smudges areas of the image, creating a fingerpainting effect.



Original image



After using the Smear brush

To use the Smear brush:

1. From the Paint Modes box, select Smear (T).
2. Set the brush size according to the size of the area you want to smear.
3. Paint on the image.

Clone

Use the Clone tool to paint a copy of the result image to any position on the result image. The Clone tool can also be applied to an output matte image. For example, if the result image has a tree that you would like to copy to another position on the image, you can offset a copy of it to a different position and then paint it onto the result.

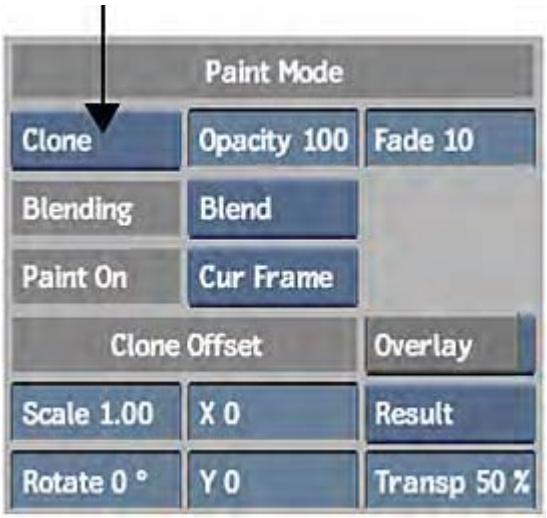
To paint a source onto the canvas, use the Reveal tool. See Revealing Sources (page 1019).

To clone an image and include cloned data in the brush strokes, use the Recursive Clone tool. See Recursive Clone (page 1019).

NOTE If changes are made upstream of the Paint Node, cloned strokes are preserved. However, the strokes will reflect the "old" input unless you force an update by clicking the Update button or by pressing U. Updates are not performed automatically because they can affect system performance.

To clone an image:

1 Select Clone from the Paint Modes box.



The Clone Offset parameters appear.

2 Set brush attributes. See Brush Attributes and Attribute Modes (page 1007).

3 To limit the cloned regions to the areas delimited by the matte, select an option in the Matte Source box.

4 To transform the reference image, do one of the following:

- To manually offset the image, enable the Overlay button (`Tab`), then hold down `Ctrl+Shift` and drag the overlaid reference image.
- To scale the image, enter a value in the Scale field.
- To rotate the image, enter the angle of rotation in the Rotate field.

NOTE To select an area for cloning, Overlay must be turned off. If you offset an image, verify that the Overlay button is disabled after use.

5 To select the area you want to clone, press `Ctrl` and click the canvas.

The cursor turns red when you press `Ctrl`. When you click the canvas, the red cursor is anchored and a green cursor appears.

6 Position the green cursor over the destination area and click the canvas.

The green and red cursors are now locked into positions relative to each other and move in tandem.

7 To set precise coordinates for the clone offset, use the X and Y fields.

8 Click and drag on the canvas to clone to the destination area.

Recursive Clone

Use the Recursive Clone brush to paint the contents of the result image to another position on the result. Unlike the Clone tool, when brush strokes created while using the Recursive Clone tool are used as a reference when the tool is applied elsewhere, the updated image data will be used.



Original image

Image courtesy of Buzz Image Group, Inc., and Les Films Stupefiants



After using the Recursive Clone medium

Image courtesy of Buzz Image Group, Inc., and Les Films Stupefiants

To use the Recursive Clone brush:

1. From the Paint Modes box, select Recur Clone (`Y`).
2. Set the brush size according to the size of the area you want to clone.
3. To select the area you want t clone, press `Ctrl` and click the canvas.

   The cursor turns red when you press `Ctrl`. When you click the canvas, the red cursor is anchored and a green cursor appears.
4. Position the green cursor over the destination area and click the canvas.

   The green and red cursors are now locked into positions relative to each other and move in tandem.
5. To set precise coordinates for the offset, use the X and Y fields.
6. Paint on the image.

   The image contained within the red circle brush is copied to the region defined by the green circle.

Revealing Sources

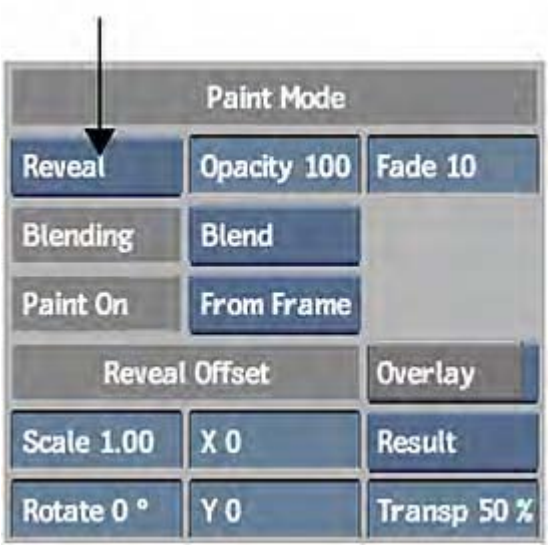
The Reveal tool allows you to paint the contents of one or more source front images directly onto the result. You can also use the Reveal tool to copy source matte images onto the output matte. For example, if one source has a tree that you would like to include on the canvas, you can select the source with the tree and then paint it onto the result.

To paint with a copy of the result or output matte image, use the Clone tool. See Clone (page 1017).

NOTE If changes are made upstream of the Paint Node, revealed strokes are preserved. However, the strokes will reflect the "old" input unless you force an update by clicking the Update button or by pressing `U`. Updates are not performed automatically because they affect system performance.

To reveal a front or matte input:

1 Select Reveal from the Paint Modes box.



The Reveal Offset parameters appear.

2 Set the brush attributes. See Brush Attributes and Attribute Modes (page 1007).

3 In the Sources list, select the source you want to reveal.

NOTE You can reveal sources that are marked as hidden in the Sources list. You will only be able to view the strokes created with a source when you disable Hide Strokes.

4 To limit the revealed sources to the areas delimited by their mattes, select an option in the Matte Source box.

5 To display the source as an overlay of the result image, enable Overlay (`Tab`). You can apply transformations to the source before you apply strokes:

- To interactively offset a source image, press `Ctrl+Shift` and drag the source. See Previewing a Reveal Operation Using a Reference Image (page 1024).
- To scale the source, enter a value in the Scale field.
- To rotate the source, enter the angle of rotation in the Rotate field.

6 Click and drag on the canvas to apply strokes that reveal the transformed contents of the selected source.

Using Blending Modes

Blending modes are Boolean operations that can be applied to the brush's colour components. A mode is applied separately to each of the red, green, and blue components of images. You can apply blending modes to combine the RGB channels of corresponding pixels from a stroke, described as follows.

Lighten Increases the RGB channel values of each pixel of the brush stroke.

Darken Reduces the RGB channel values of each pixel of the brush stroke.

Exclusion Adds the RGB channel values of the brush stroke, then subtracts twice the product of these channels.

Hard Light Multiplies or screens the selected colour of the brush stroke onto the image, depending on the colour. The effect is similar to shining a harsh spotlight on the image, and greatly reduces the contrast levels in the image.

If the blend colour (light source) is lighter than 50% grey, the image is lightened as if it were screened—this is useful for adding highlights to an image. If the blend colour is darker than 50% grey, the image is darkened, as if it were multiplied—this is useful for adding shadows.

Multiply Multiplies the RGB channel values of corresponding pixels of the stroke and the current image and normalizes the result by dividing by 255 in 8-bit mode, or 4095 in 12-bit mode. The resulting RGB channel values are assigned to the corresponding pixels in the generated clip.

Divide Divides the RGB channel values of pixels of the stroke and the current image and normalizes the result by multiplying by 255 in 8-bit mode, or 4095 in 12-bit mode. The resulting RGB channel values are assigned to the corresponding pixels in the generated clip.

Overlay Multiplies or screens the colours, depending on the RGB channel values of the colour of the stroke and the current image. Patterns or colours overlay the existing RGB channel values while preserving the highlights and shadows of the stroke's colour. The colour of the stroke is not replaced, but is mixed with the colour of the current image to reflect the lightness or darkness of the original colour.

Screen Multiplies the inverse of the current image with the colour of the brush stroke. The resulting colour is always lighter. The colour remains unchanged when you screen with black. Screening with white produces white. The effect is similar to projecting multiple photographic slides on top of each other.

Soft Light Shines a soft, diffuse light on the image. If the blend colour (light source) is lighter than 50% grey, the image is lightened. If the blend colour is darker than 50% grey, the image is darkened.

Using this mode with a black brush stroke results in a very dark effect; with white, a very bright one.

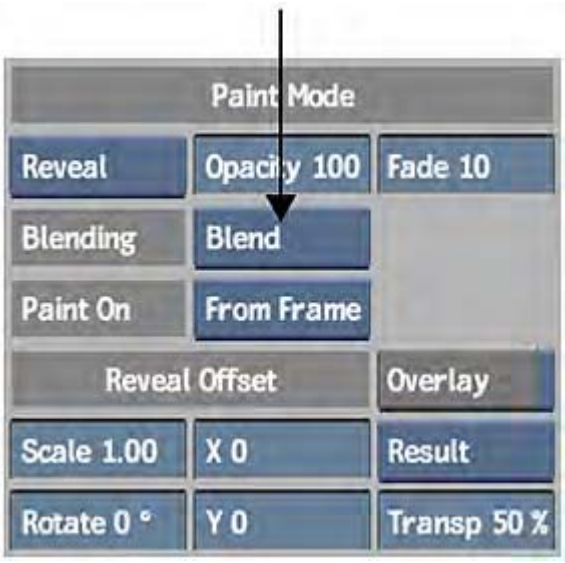
Add Adds the luma values of corresponding pixels of the brush stroke and the current image. The resulting value is assigned to the corresponding pixels in the generated clip. If the result is greater than 255 (in 8-bit mode), the pixel in the destination clip is clamped at a value of 255 (white). In 12-bit mode, the maximum colour value is 4095. The resulting clip is always brighter.

Subtract Subtracts the RGB channel values of the pixels of the current image from the RGB channel values of the pixels of the brush stroke and assigns the result to the RGB channel values of the pixel.

If an RGB channel value of the current image is larger than the corresponding channel value of the brush stroke, yielding a negative result, that result is clamped at 0 (black). The resulting clip is always darker.

To blend a stroke with the current image:

1. Select a Blend mode from the Blend option box.



2 Select a brush and colour.

3 Draw a stroke.

The stroke with the selected blend is applied. The current blend mode only affects new strokes. However, if you draw over existing strokes, the blend is created using the pixel values for the two different strokes.

Using the Canvas

Use the canvas controls to control clearing and wiping of the image.

Clearing the Canvas

You can clear all strokes from the result image and output matte at the current frame or all frames. To limit the operation to a source, see Clearing Strokes on a Source (page 1005).

WARNING Any strokes that have been applied will not be maintained when you clear strokes.

To clear strokes from the canvas:

1 Select one of the following from the Clear Canvas option box:

- Clear to remove strokes from the current frame.
- Clear All to remove strokes from all frames.



(a) Clear Canvas option box

The strokes are removed from the result image.

Wiping the Canvas

You can wipe the result image when you want to apply a uniform colour at the current frame in a single brush stroke. Alternatively, you can use a source, to wipe over an image.

To wipe the result image with a colour or a source:

1. From the View box, select Result (`F4`).
2. Do one of the following:
   - Pick the wipe colour by clicking the colour pot to the right of the Wipe button.
   - Select the source in the Sources list and enable Use Source.



3. To display the source as an overlay of the result image, enable Overlay (`Tab`). You can apply transformations to the source before you apply the wipe:
   - To interactively offset a source image, press `Ctrl+Shift` and drag the source. See Previewing a Reveal Operation Using a Reference Image (page 1024).
   - To scale the source, enter a value in the Scale field.
   - To rotate the source, enter the angle of rotation in the Rotate field.
4. Do one of the following:
   - Click Wipe to wipe the front and matte.
   - Click the Wipe dropdown list and select Wipe F to wipe the front only.
   - Click Wipe dropdown list and select Wipe M to wipe the matte only.

Rotating the Canvas

During the painting process, you can rotate the canvas, making it easier to paint on any part of your image.

To rotate the canvas:

1. Do one of the following:
   - Set the angle of rotation in the Rotation field.
   - Press `Alt+spacebar` and drag in your image.



(a) Rotation field

Previewing a Reveal Operation Using a Reference Image

Overlaying a reference image on the result offers an intuitive way of previewing paint operations.

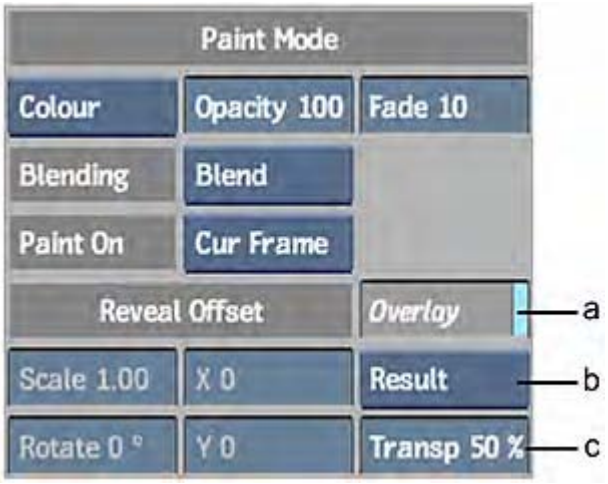
When you overlay a source onto the result, you can preview the effects of a Reveal operation. The selected image is superimposed over the result, and the transparency can be adjusted. This allows you to see exactly what your brush strokes will reveal.

When you overlay a result reference clip onto the result, you can slip the overlaid frame to display previous or next frames, creating an onion skin effect.

The overlay of the front and the matte can also be shown on the result, allowing you to rotoscope more easily, for example.

To display the reference image:

1. Enable the Overlay button (`Tab`).



(a) Overlay button (b) Reference button (c) Transparency field

The reference image appears by default at 50% transparency.

2 Select the reference image you want to display from the Reference box.

3 If you selected Source Matte or Source Front, select an image from the Sources list.

4 With the reference image, you can:

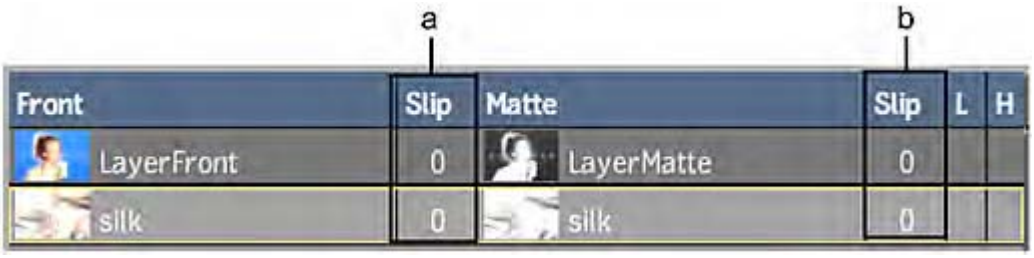
- Hold down `Ctrl+Shift` and drag the source front or source matte to offset it to preview a Reveal operation.
- Scale the image, by entering a value in the Scale field.
- Rotate the image, by entering the angle of rotation in the Rotate field.
- Set the Slip value in the Sources list to show a reference image at a different frame relative to the actual frame position. See Slipping a Reference Image (page 1025).
- Set the transparency for the reference image in the Transparency field.

NOTE You can also drag the result image to offset it for a Clone operation by holding down `Ctrl+Shift`.

Slipping a Reference Image

A reference image that is offset in time can be overlaid on the canvas. You can slip a front, matte, result, output matte, source front, and source matte clips. In the Sources list, use the Slip field to indicate the offset amount.

Each slip value can be set independently. To set the same front and matte values for a source, hold down the `Alt` key and edit either value.



(a) Front and Source Front Slip fields (b) Matte and Source Matte Slip fields

Saving Setups and Preferences

Paint Node setups are saved as XML files with the *.bpaint* extension. The procedures for saving and loading Paint Node setups are the same as for other setups.

Paint Tool

Use the Paint tools to create graphics, paint on images, and retouch clips. Use a matte to protect areas of the front clip during painting. Record, animate, and apply a series of brush strokes to a clip. Use the Graphic and Cut/Paste tools to create rotoscoped sequences and shape animations.

Loading Clips into Paint

When accessing the Paint Tool, you first select the clips that you want to use. Any of the following combinations of clips can be used in Paint:

- A front clip only
- A front clip and a back clip
- A front clip and a matte clip
- A front clip, a back clip, and a matte clip
- None (a blank canvas)

The front clip appears on the Paint canvas, which you can use to apply colours and effects to the clip. The matte clip delimits the area of the front clip affected by painting. The back clip can be revealed or brushed through onto the front clip. The colour of the blank canvas is defined by the wipe colour.

NOTE The image window in Paint is referred to as the *canvas* in the following sections.

To load clips into the Paint Tool:

1. Select **Tools ➤ Composite ➤ Paint**.
   The cursor changes to Pick Front.
2. Select the front clip.
   The cursor changes to Render Here.
3. Click on any free (or a grey) area on the workspace.
   You are now in the Paint Tool.
   The Paint menu appears.



4 Access one of the following Paint menus.

Click:	To:
Paint	Use the brushes or geometric shapes to paint on the canvas, record and apply strokes to the canvas, fill areas of the image with a reference colour or image, and move the image on the canvas.
Graphics	Create, edit, and animate shapes and apply them to the canvas.
CutOut	Create and add effects to cutouts and apply them to the canvas.
Setup	Set preferences, rendering options, grid guides, and colour correction options.

Accessing the Paint menu with a blank canvas

To access the Paint menu with a blank canvas:

1 Select **Tools ➤ Composite ➤ Paint**.
The cursor changes to Pick Front.

2 From the Input Mode box, select None.
The Resolution Parameters controls appear.
The cursor changes to Render Here.

3 Choose a resolution, width, height, pixel aspect ratio, bit depth, and scan mode to apply to the background in Paint.

NOTE The default parameters are the project resolution parameters.

4 Set the frames per second and drop-frame values for the clip from the Frame Code Mode box.

5 Click on any free (or a grey) area on the workspace.
You are now in the Paint Tool.
The Paint menu appears.



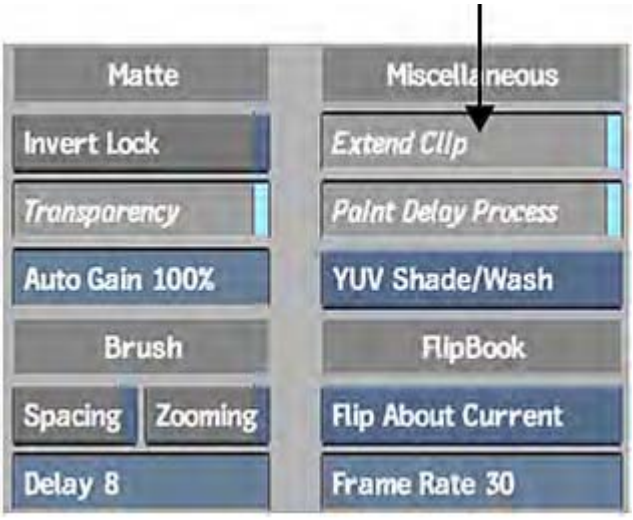
Locking and Unlocking the Duration of a Clip

You can determine whether or not frames are added to the end of a clip.

To add frames to the end of the clip:

1 Click Setup.

2 Enable Extend Clip.



3 Click Next Frame.

A frame is added. You can continue to add frames in the same manner with Extend Clip enabled.

TIP To maintain the length of the original clip, disable Extend Clip. No frames are added to the clip when you click Next Frame.

Using the Mouse

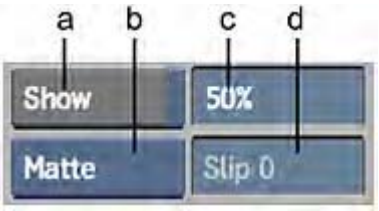
You can use either the mouse or the tablet and pen to paint. Press and hold the right mouse button to apply paint at 80% pressure, the middle button to paint at 40% pressure, and the left button to paint at 20% pressure.

Displaying a Reference Image

You can display a reference image in the background of the result image to use as a guide for rotoscoping.

To display the reference image:

1 Enable the Show button.



(a) Show button (b) Reference box (c) Transparency field (d) Slip field

The reference image appears by default at 50% transparency.

2 Select the reference image you want to display from the Reference box.

3 Set the transparency for the reference image in the Transparency field.

4 Set the Slip field value to show different images from the reference clip.

This option only works if the reference image you selected is from a clip with more than one frame.

NOTE The Slip field is disabled if you select Matte or Saved from the Reference box.

Using Grids and Guides

Use the Grid/Guides button to set up reference points when painting and to accurately place strokes on the image.

Use the field guides to provide reference points when painting, and the grid to help you accurately place strokes on the image. Neither the field guides nor the grids appear on the processed clip.

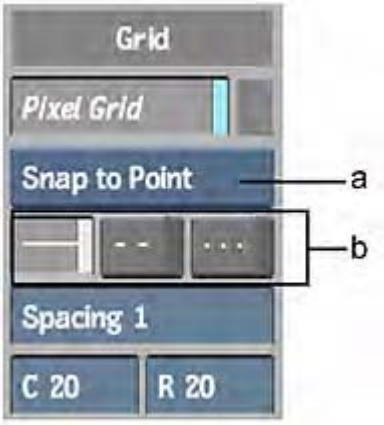
The Grid and Guides menu contains the Safe area guide controls, the Grid controls, and the Centre controls.



The Safe area guide controls and Centre controls are subsets of their corresponding controls in the Grid and Guides menu in the other tools.

Grid Controls

You can set the behaviour of the grid so that paint strokes snap or lock to points on the grid. Use the Grid controls to configure the appearance and behaviour of the grid.



(a) Grid Behaviour box (b) Style buttons

To toggle the grid on and off, click Pixel Grid. When Pixel Grid is on, use the following controls to configure the appearance and behaviour of the grid.

Grid Behaviour box Displays the behaviour of the grid with respect to paint strokes:

- Snap to Point snaps each point of a stroke to the nearest intersection of a horizontal and a vertical grid line.
- Snap to Line snaps the current point of a paint stroke to the nearest point on a horizontal or vertical grid line.
- Lock to Point locks each point of a paint stroke to the nearest intersection of a horizontal and a vertical grid line.
- View uses the grid without the snap to or lock to options.

Style buttons Set a style for the lines of the grid. The style is either a solid line, a dashed line, or a dotted line.

Spacing field Sets the number of pixels between the dashes in a dashed line style, or between the dots in a dotted line style.

C, R fields Set the number of columns and rows respectively in the grid.

Resetting the Grid and Guides

Use the Reset All button in the lower-right corner to reset the grid and all guides to their default values.

Using the Player in the Paint Tool

You can access the Player from the Paint Tool. To do so, click Process in the Paint Tool and then click Play once the clip is processed. Each time you access the Player, a clip is created in the Viewing Panel. This clip cannot be removed from the Paint tool. You must delete it from the Viewing Panel instead.

Selecting Colours

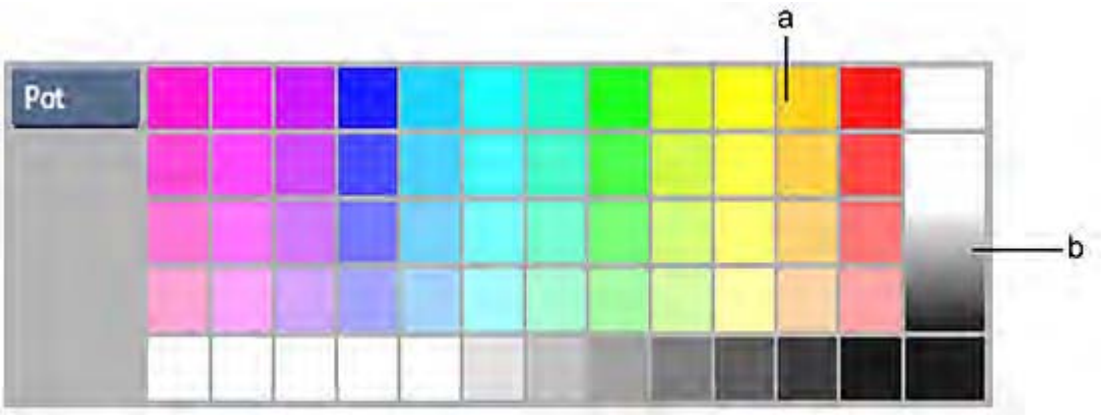
In Paint, you select colours using the colour picker. You store colours in the colour palette, mix or try out colours on the scratch pad, and create colour or greyscale gradients using the colour gradient.

Using the Current Colour

The current colour is used when you paint. It is also used to set the colour for the wipe function and the matte colour. Select the current colour from the colour palette, scratch pad, or colour gradient. Or, click the Current Colour pot to use the colour picker.

Using the Colour Palette

The colour palette appears in the Paint, Graphics, and CutOut menus. It contains the scratch pad and colour gradient. A series of colours are stored in the colour palette in colour pots. To select the current colour in the colour palette, click a colour pot. To set the current colour, press and hold a colour pot.



(a) Colour pot (b) Colour gradient bar

Build your own colour palette by storing the current colour in the colour pots. You can also save and load colour palettes.

To store a colour in the colour palette:

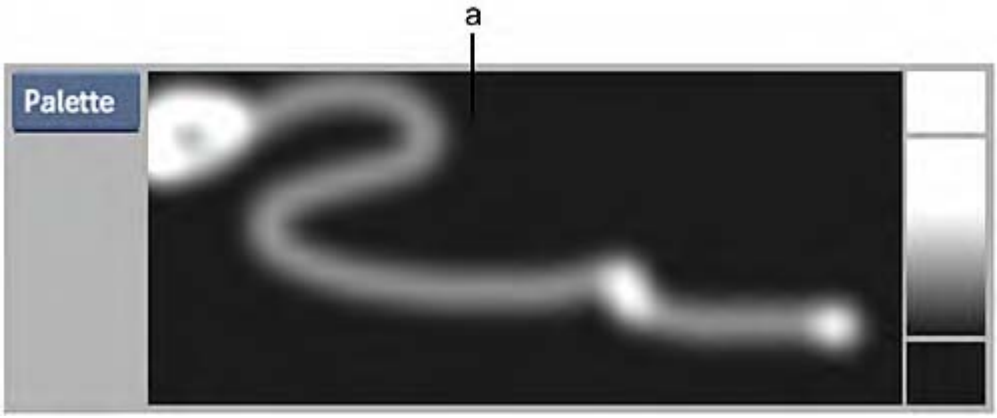
1. To display the colour palette, swipe the bar at the bottom of the screen.
   The colour palette appears.

   TIP If you do not see the palette, click the Palette button. To hide it, swipe the bar a second time.

2. Set the current colour.
3. Click a colour pot in the colour palette and hold the cursor down momentarily.
   The colour is saved in the colour pots.

Using the Scratch Pad

Use the scratch pad to mix colours selected from the image or colour pots and to test the selected brush type. You can also paste cutouts into the scratch pad to use when mixing colours.



(a) Scratch pad

To mix colours on the scratch pad:

1. Click the Pot button.
   The scratch pad appears.
2. Paint on the scratch pad.
3. Change the current colour and paint over the previous brush strokes.

The colours are mixed.

You can also use the Wash, Shade, Smear, Drag, Warp, Impressionist, Recursive Clone, Stamp, and Blur Special Effects media in the scratch pad.

To use one of the media types in the scratch pad:

1. Select the Special Effects medium you want to use.
2. Enable Medium.
3. Use the brush cursor to apply the medium in the scratch pad.

To set the current colour using the scratch pad:

1. Click the Current Colour pot.
2. Use the colour picker to select the mixed colour in the scratch pad.
3. Click the Current Colour pot to use the selected colour.

Using the Colour Gradient Bar

The colour gradient bar is used to set the gradients for graphics you create in the Graphics menu. You can also set the current colour by selecting a colour on the gradient using the colour picker.

To set the gradient:

1. Set the current colour.
2. Click either the upper or lower colour pot on the gradient bar.

   NOTE You must hold the cursor down for a moment to set the Gradient colour pot.

To set the current colour using the colour gradient bar:

1. Click and drag the cursor across the gradient bar.
   The selected colour appears in the Current Colour pot.
2. Select a colour.
   The selected colour becomes the current colour.

Previewing an Animation

Use the Flipbook command to preview an animation by playing a sequence of either five or nine frames.

To set up the Flipbook command:

1. Click Setup.
   The Setup menu appears.
2. In the Flipbook area, select an animation mode from the Flipbook box.



(a) Flipbook box (b) Frame Rate field

Select:	To:
Flip About Current	Play two (or four) frames before the current frame and two (or four) after the current frame. This is the default setting.
Flip From Current	Play five (or nine) frames beginning at the current frame.
Flip To Current	Play five (or nine) frames ending at the current frame.

3 Enter the frame rate in the Frame Rate field.

NOTE If the system is running at high resolution, it may not be able to achieve a flip rate of 30 frames per second.

4 Press `F` to play five frames, or press `Shift+F` to play nine frames.

If you selected Flip About Current, pressing `F` plays two frames before and after the current frame. Pressing `Shift+F` plays four frames before and after the current frame.

Painting on Full-Resolution Film Images

Use the Zoom Mode box to paint on a film resolution image at full resolution from a zoomed out perspective. With this feature, you can perform complex operations like full-frame rotoscoping without having to constantly zoom in and out from the image on which you are painting.

In the Paint menu, the Zoom Mode box shows the zoom mode in which you are working. This mode is set automatically. If you zoom out from the image, the Zoom mode automatically switches from Raster to Tiled. This allows you to paint on the image from a zoomed out perspective.



(a) Zoom Mode box

NOTE If you manually switch the zoom mode to Raster while zoomed out from the image, you will not be able to paint on it.

You can also manually select either Raster or Tiled mode if you want to override the default mode. In Raster mode, if you apply paint to the edge of the image while zoomed in, the paint is applied in an abruptly sharp straight edge along the border of the canvas; the brush stroke is not completed on the image. In Tiled mode, if you attempt the same operation, the paint is applied from the full diameter of the brush you are using to the area of the image that is not visible on the canvas.

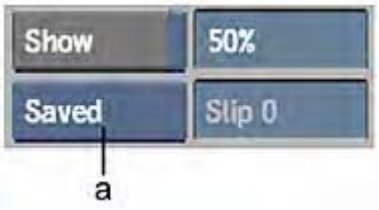
Saving and Restoring the Image

You can save the current frame on the canvas. If you want to reverse a paint application, you can restore the image without having to recreate it.

Save (Image Buffer) Saves the image that currently appears on the canvas. A single frame is stored in the Save buffer. The next time you click Save, the current frame replaces the frame stored in the Save buffer.

Restore Replaces the current image on the canvas with the contents of the Save buffer.

Preview Displays the contents of the Save buffer. Select the Saved option in the Reference box and click and hold Preview to view the contents of the Save buffer.



(a) Reference box

Exiting Paint

When exiting Paint, you may keep or discard the changes you have made to the front clip. Click Exit and select an option.

Select:	To:
Exit	Keep the changes you made to the front clip. The modified front clip appears on the destination reel.
Keep One	Keep only the current frame of the front clip. When this option is selected, a Confirm button appears to the right of the canvas. Click Confirm to keep only the current frame of the front clip. The current frame appears on the destination reel.
Cancel	Quit Paint without saving changes to the front clip. When this option is selected, a Confirm button appears to the right of the canvas. Click Confirm to quit without saving your changes, or click elsewhere to cancel.

About Canvas Mode

When you first open Paint, you are in Canvas mode. In Canvas mode, you can use brushes to modify your images. Brushes apply colour, filters, and Special Effects media to the image on the canvas. The brush cursor appears as a green cross surrounded by a circle when placed over the canvas. Canvas mode features are not available from multiple menus, such Canvas, Geometry, Fill, and Roll.

You can also paint on the canvas using geometric shapes to define the path of the brush.

To ease the painting of canvas edges, you can roll the canvas in the image window.

To paint on the canvas:

1 In the Paint menu, click Canvas.

If the scratch pad is covering this button, swipe down to hide the scratch pad.

2 From the Paint Mode controls, click Paint.

3 Set a colour in the Current Colour pot. See Selecting Colours (page 1030).

4 Select a brush from the Brushes window. See Selecting a Brush (page 1035).

5 Set the brush attributes and modes in the Brush Attributes fields. See Brush Attributes (page 1040) and Brush Attribute Modes (page 1044).

6 Set the brush opacity in the Opacity field. Set the opacity to 100% to apply a fully opaque colour.

7 Stroke the brush over the canvas. To paint straight horizontal and vertical lines, press `Shift` and drag the brush up and down or left and right.

8 Click Undo to erase the strokes applied to the canvas since the last time you zoomed, panned, or changed a brush attribute.

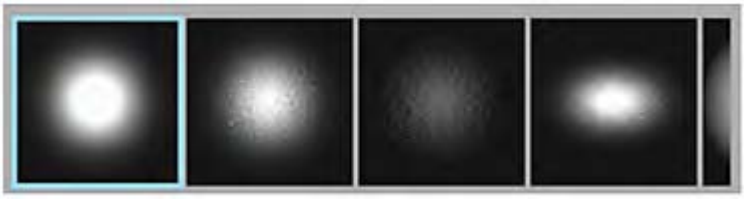
Selecting a Brush

A number of predefined brush types are available in Paint including round, square, and elliptical brushes of various sizes and edge softness. The cursor has the same appearance regardless of which brush type you use. Each brush has its own icon in the Brushes window. The white portions of the brush icon indicate where the paint will be applied on the image.

See Creating a Custom Brush (page 1114).

To select a brush:

1 Scroll through the Brushes window.



To scroll the selections, click the Brushes window and drag left or right.

2 Click the brush icon you want to use.

The selected brush is highlighted by a blue outline.

NOTE Only one brush can be active at a time.

Using the Large Canvas

In Large Canvas mode, almost the entire image window is available for painting, but not all Paint options are available. You can still change the brush characteristics, medium, and show options. You can also save, restore, and wipe the canvas.

To display the large canvas:

1 Click Paint.

2 Click Canvas.

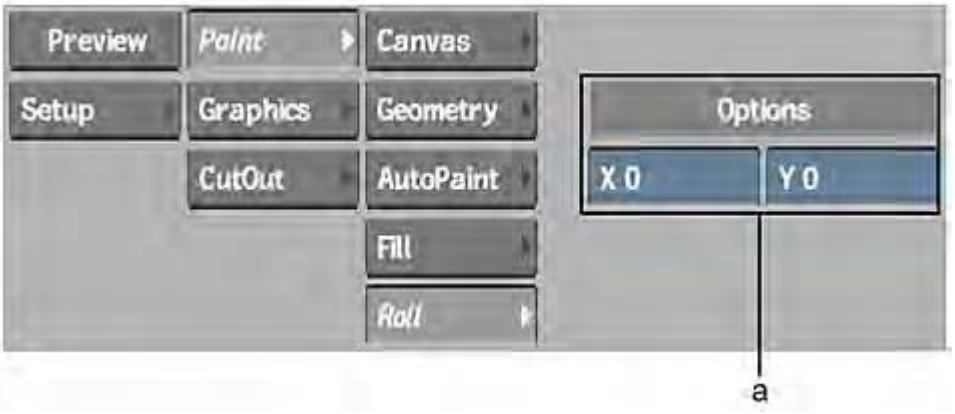
3 Swipe your cursor anywhere on the right edge of the screen, or press `Esc` to toggle between the large canvas and Paint menu.

Rolling the Image

Use the Roll menu to reposition the current frame on the canvas to paint its edges.

To roll the image:

1. Click Paint.
2. Click Roll.
   The Roll menu appears.



(a) Coordinate fields

3. Zoom in on the image if necessary.
4. Drag the image using the pan cursor.
   You can also enter the roll values in the Coordinate fields.
5. To recentre the frame, click Reset.

   NOTE The frame is automatically recentred when you go to another frame or exit Paint.

Painting with Geometry

Use the Geometry feature to draw lines, rectangles, circles, and triangles on the canvas.

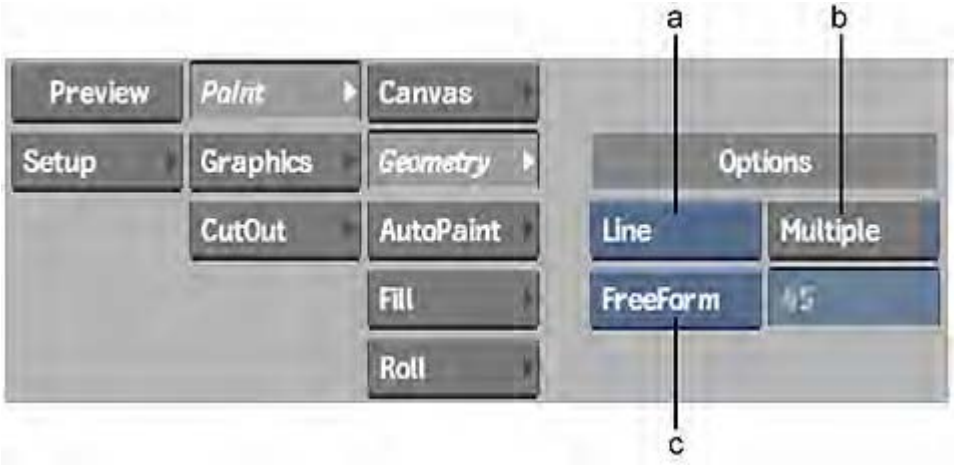
Painting with geometry is different from drawing objects in Graphics mode. In Graphics mode, you create objects that can be edited. When you paint with geometry, you define paths for the brush to follow. Paint uses the current brush setup to apply the stroke to the canvas.

Painting Lines

Use the Line option to paint straight lines. Paint single lines one at a time, or draw connected multiple lines. You can constrain the lines to vertical or horizontal paths or specific angles.

To paint a line or multiple lines:

1. Click Geometry to display the Geometry menu.
2. Select Line from the Geometry box.
   The Multiple Line button and the Line Type box appear.



(a) Geometry box (b) Multiple button (c) Line Type box

3 Select the type of line you want to draw from the Line Type box.

Select:	To:
Angle	Paint a line at a specific angle. This option displays a field in which you enter the angle in degrees.
Vertical	Paint vertical lines.
Horizontal	Paint horizontal lines.
FreeForm	Paint lines at any angle with no constraint.

4 If you selected Angle, enter a degree in the field.

5 To draw single lines, move to the canvas and click, drag, and release.

A brush stroke is painted along the line.

6 To draw multiple lines, enable Multiple, move to the canvas, and click to place the start point of the first line. Click again to draw the end point. Continue clicking to place additional points and draw more lines.

7 To end multiple lines, click below the timebar or on the menu panel to turn the option off.

The brush strokes are painted along the lines.

Painting Rectangles

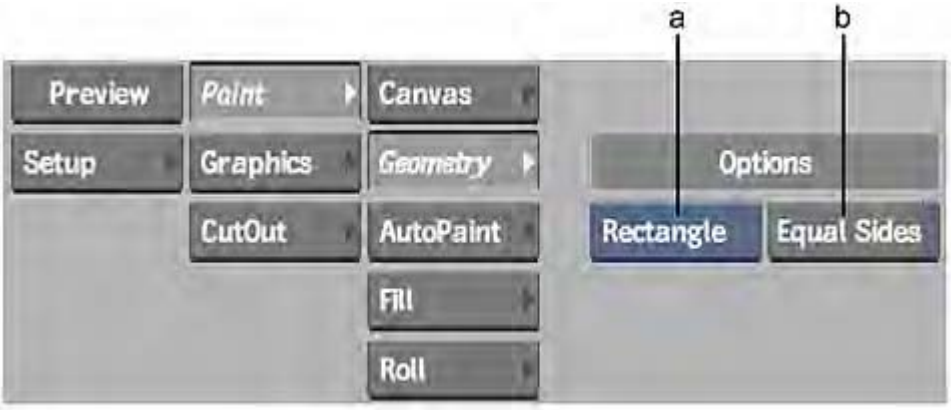
Use the Rectangle option to draw rectangles or squares.

To paint a rectangle or square:

1 Click Geometry to display the Geometry menu.

2 Select Rectangle from the Geometry box.

The Equal Sides button appears.



(a) Geometry box (b) Equal Sides button

3 To draw a square, enable Equal Sides or press and hold `P`.

4 Press the cursor on the canvas to anchor the first corner of the rectangle. Drag the cursor to size the rectangle.

5 When the rectangle is the required size, release the cursor.
A brush stroke is painted along the sides of the rectangle.

Painting Circles

Use the Circle option to draw circles of any size.

To paint a circle:

1 Click Geometry to display the Geometry menu.

2 Select Circle from the Geometry box.

3 Press the cursor on the canvas to anchor the centre point of the circle. Drag the cursor to size the circle.

4 When the circle is the required size, release the cursor.
A circular brush stroke is painted.

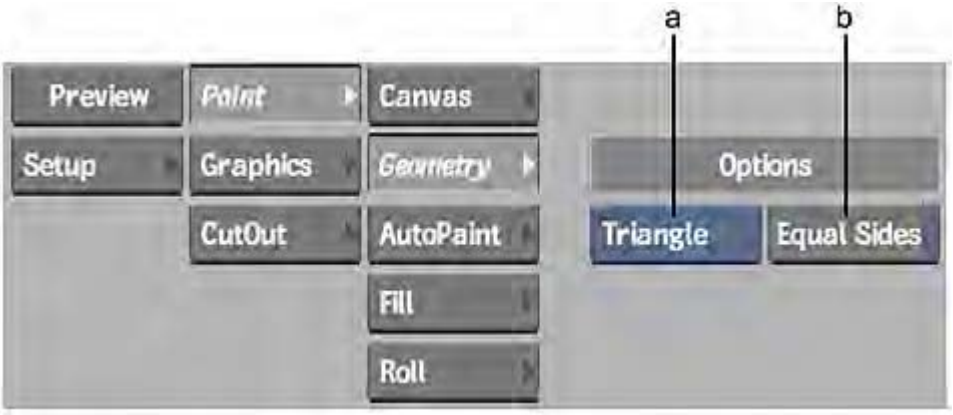
Painting Triangles

Use the Triangle option to draw equilateral or asymmetric triangles.

To paint a triangle:

1 Click Geometry to display the Geometry menu.

2 Select Triangle from the Geometry box.
The Equal Sides button appears.



(a) Geometry box (b) Equal Sides button

3 To draw an equilateral triangle, enable Equal Sides or press and hold P.

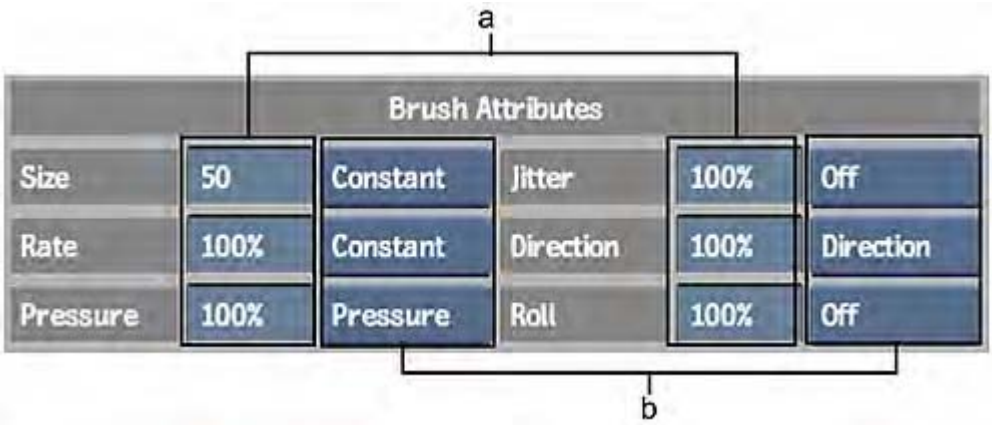
4 Move to the canvas and click to anchor the first vertex of the triangle.

5 Click to place the second vertex and then again to place the third vertex. If you are drawing an equilateral triangle, drag the cursor until the triangle is the required size.

A brush stroke is painted along the sides of the triangle.

About Brush Attributes and Modes

You can set various brush attributes and attribute modes that determine how paint is applied to the image. Each brush attribute has a corresponding Attribute Mode control. You use the Preferences menu to affect the way paint is applied to the image.



(a) Brush Attribute fields (b) Attribute Mode controls

In the Paint menu, the Brush Attributes and Attribute Mode controls share the same space as the colour palette. To display the brush attributes and modes in the Paint menu, swipe the cursor at the bottom of the screen.

In the Graphics Edit menu, the Brush Attributes and Mode controls appear only when the object attribute is set to Outline or Fuzzy. To display the brush attributes and modes in the Graphics menu, swipe the cursor twice at the bottom of the screen.

NOTE The brush attribute and attribute mode values set in the Paint menu are independent from the values set in the Graphics menu.

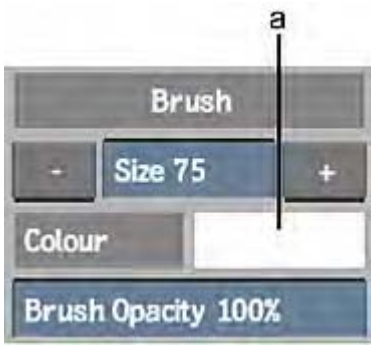
Brush Attributes

The Brush Attribute fields set the size of the brush, the distribution of the paint, and the rate and direction of the paint application. Brush attributes are listed as follows:

- Size
- Rate
- Pressure
- Opacity
- Jitter
- Direction
- Roll

Brush Opacity

The brush opacity affects the transparency of the brush. A value of 100% applies a fully opaque colour. Use a lower value to apply a more transparent colour. Enter a value in the Opacity field.



(a) Current Colour field

NOTE You can only use the Front, Back, Result, and Saved attribute modes with the Opacity attribute.

Brush Size

The brush size is indicated by the diameter of the green dashed circle surrounding the cursor brush. To increase the brush size, set a value in the Size field and drag the brush to the right on the canvas. To decrease it, set a value in the Size field and drag to the left.

You can also use the Size buttons to increase or decrease the brush size. Click the + button to increase the brush size. Click the - button to decrease the brush size. You can also click and drag in the Size field or click and enter a value.

NOTE You can use any attribute mode with the Size attribute.

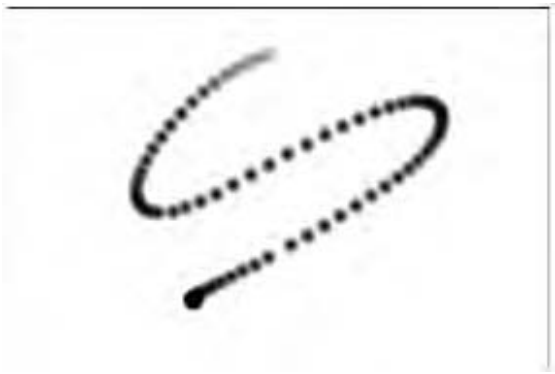
Brush Rate

The brush rate is the rate at which brush strokes are applied to the canvas. Use a high value to produce a smooth continuous stroke, or a low value to produce a less continuous stroke with larger gaps between brush images.

NOTE You can use any attribute mode with the Rate attribute.



Rate attribute value = 100



Rate attribute value = 25

Brush Pressure

The brush pressure affects the transparency of the paint applied to the image. To apply opaque paint, use a high percentage value. For more transparent paint, use a low value.

The Pressure attribute differs from the Opacity attribute in that you can set the Pressure attribute mode so that the paint transparency varies according to the pressure applied to the pen or the direction of the brush.

NOTE You can use any attribute mode with the Pressure attribute.



Pressure attribute value at 100%

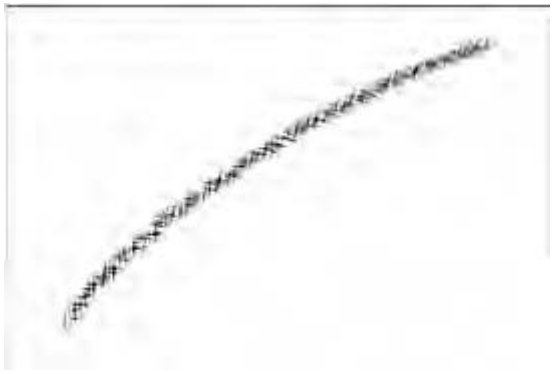


Pressure attribute value at 50%

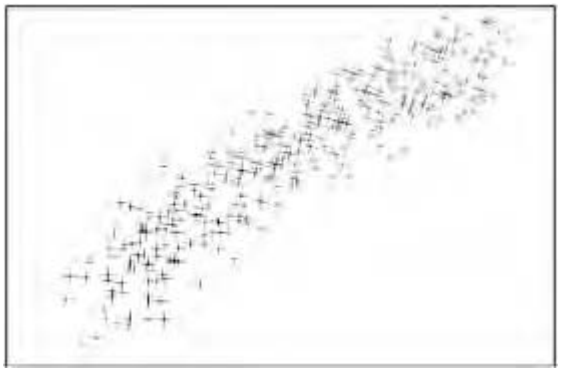
Brush Jitter

The Jitter attribute randomizes the brush strokes applied to the image. A high value produces a greater dispersion of paint, while a low value produces a greater concentration.

NOTE You can use any attribute mode with the Jitter attribute.



Jitter attribute value at 10

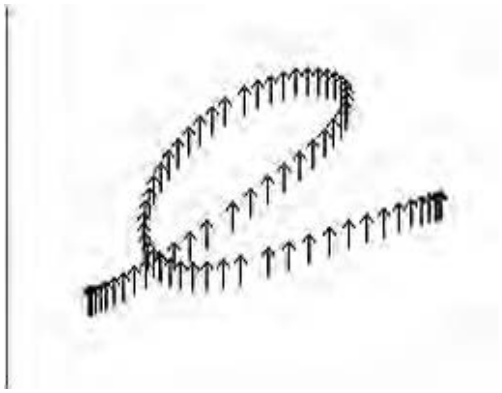


Jitter attribute value at 100

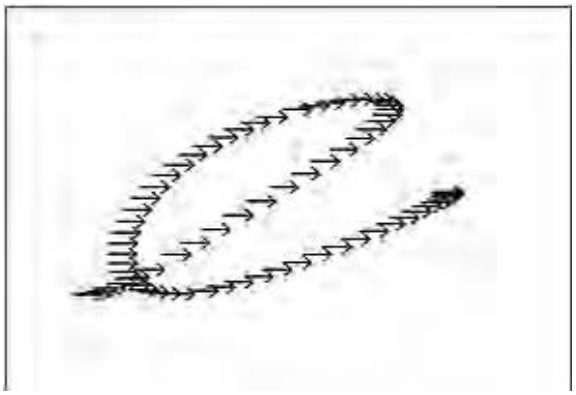
Brush Direction

The Direction attribute causes the brush to rotate around the Z-axis and can be used to produce a calligraphy effect. The effect of the Direction attribute is most noticeable when used with one of the elliptical or star brushes. The value of the Direction attribute causes the brush strokes to rotate by 90 degrees for each increment of 25 percent.

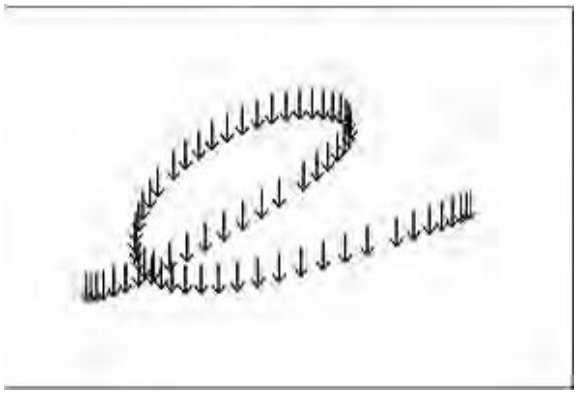
NOTE You can use any attribute mode with the Direction attribute.



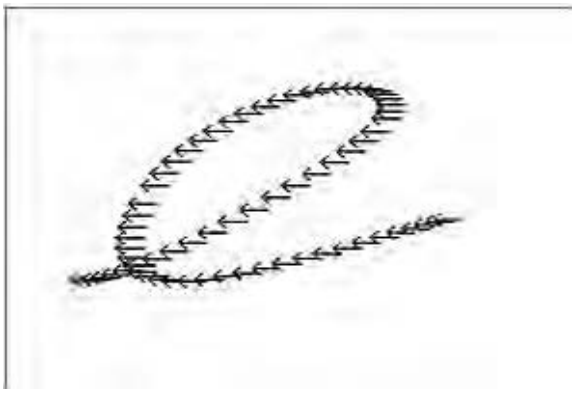
Direction attribute at 100%



Direction attribute at 75%



Direction attribute at 50%

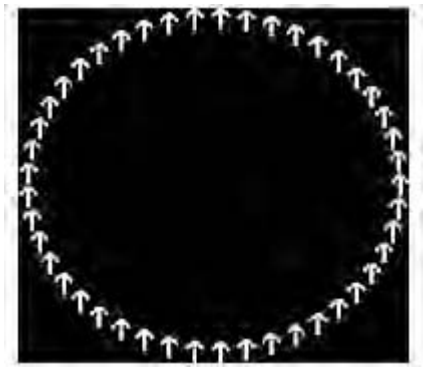


Direction attribute at 25%

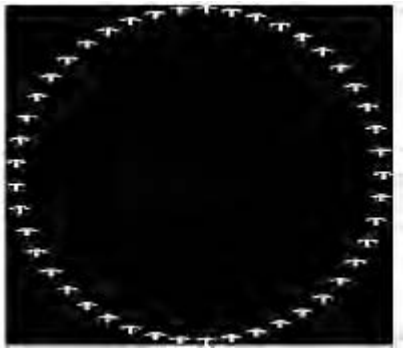
Brush Roll

The Roll attribute rolls the brush around the X-axis. The effect of the roll is most noticeable when used with one of the non-symmetrical brushes. For each increment of 25 percent, the Roll attribute value creates a rolled brush stroke of 90 degrees.

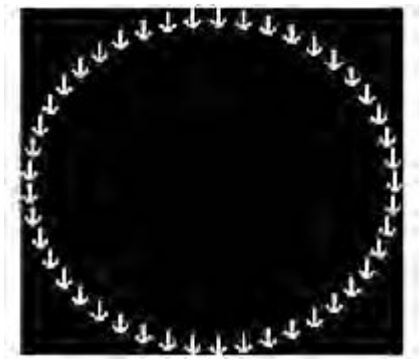
NOTE You can use any attribute mode with the Roll attribute.



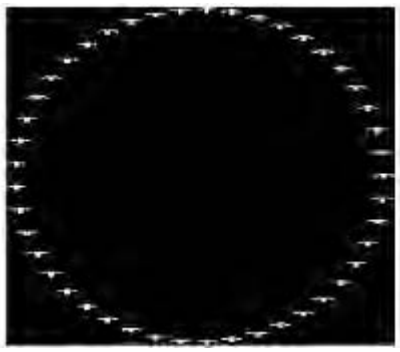
Roll attribute value at 100%



Roll attribute value at 75%



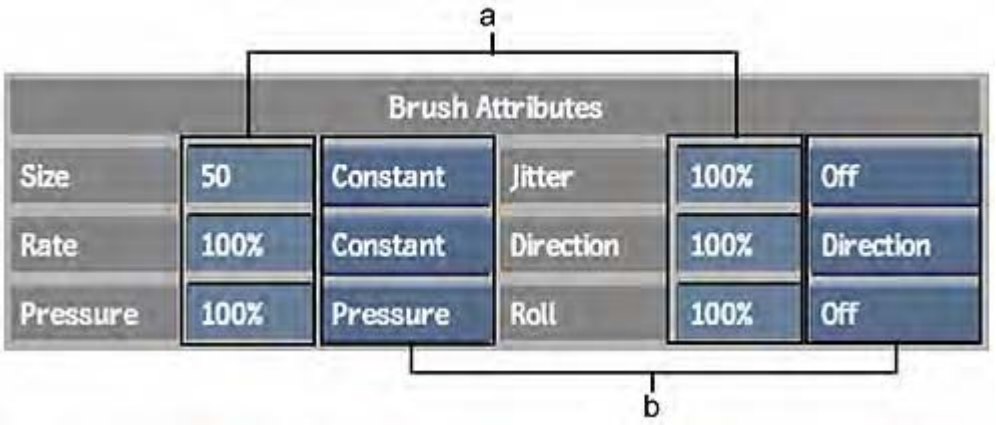
Roll attribute value at 50%



Roll attribute value at 25%

Brush Attribute Modes

The value of a brush attribute depends on the selected attribute mode. You can choose Constant, Off, Front, Back, Result, Saved, Pressure, or Direction.



(a) Brush Attribute fields (b) Attribute Mode controls

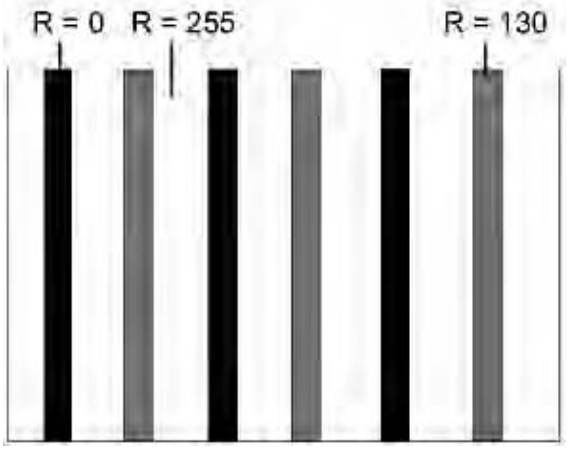
In Constant mode, the brush attribute values remain constant. In Off mode, the attribute is disabled.

The Front, Back, Result, and Saved modes use the colour values in a reference image to vary the brush attribute value. Pressure and Direction modes affect how paint is applied by causing the brush attribute value to vary in relation to the pressure exerted on the pen and the direction of the brush, respectively.

NOTE Do not set the Size, Rate, or Pressure attributes to Off mode.

Using Reference Images

The Front, Back, Result, and Saved modes use the red channel in reference images to set the brush attribute values. Front mode uses the front clip as the reference, Back mode uses the back clip, Result mode uses the result clip, and Saved mode uses the image in the Save buffer.



Back image



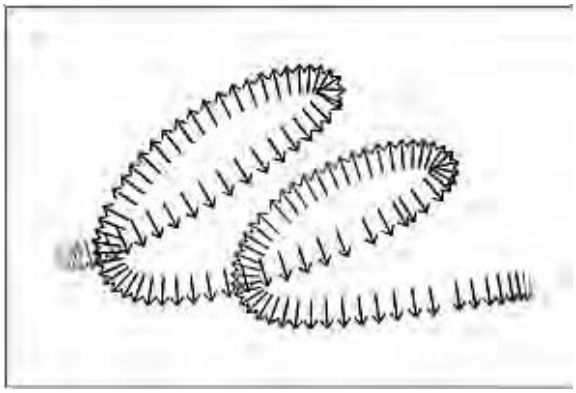
Using Size attribute with mode set to Back

Direction of the Brush

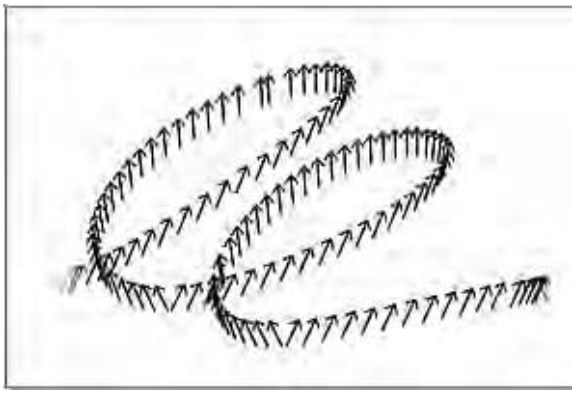
The Direction mode uses the direction of the brush stroke as the reference.

When used with the Direction mode, the Direction attribute causes the brush strokes to follow the trajectory of the brush. Increase the percentage value to enhance the effect on brush direction.

Drag:	To:
Right	Use 100% of the attribute value.
Left	Use 0% of the attribute value.
Up	Use 25% of the attribute value.
Down	Use 75% of the attribute value.



Direction attribute value at 100%, Direction mode



Direction attribute value at 15%, Direction mode

Pressure Exerted on the Pen

The Pressure mode uses the pressure exerted on the pen as the reference value. The harder you press on the pen, the greater the brush attribute value. The softer you press, the lower the brush attribute value.

NOTE The Pressure attribute mode cannot be used in the Graphics menu.

Preferences Affecting Brushes

The Spacing setup preference affects how paint strokes are applied to the canvas.

To enable Brush Spacing:

1. Click Setup in the Paint menu.
2. Enable Spacing.



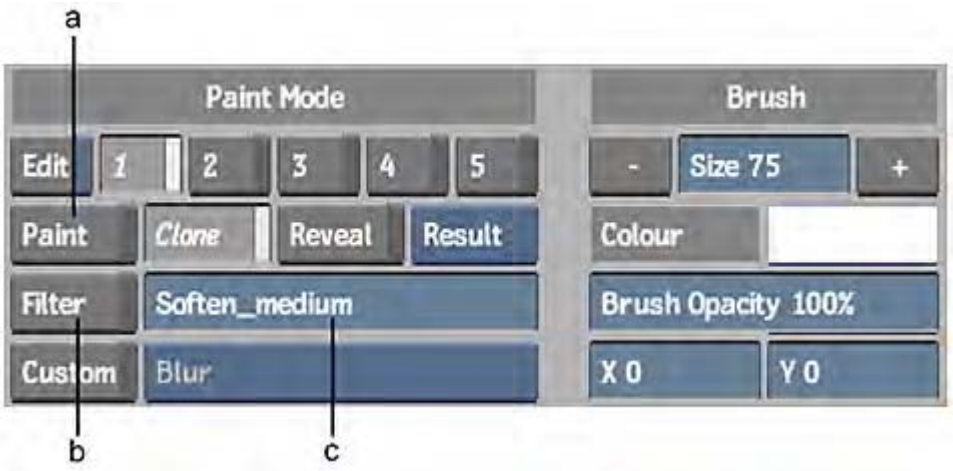
This option sets a uniform distance between paint strokes. No matter how fast you move the brush, the brush spreads the paint evenly.

TIP Use a high brush rate when using stamps with the Spacing option enabled. See Using the Stamp Medium (page 1054).

Applying Filters

When applying filters, it is important to try different brush types and change the brush attributes to create different effects.

Apply filters to the canvas using the brush, Wipe command, or Wash and Shade media. Paint uses the same filter library as the Filter command in the Processing menu.



(a) Paint Media button (b) Filter button (c) Filter field

To brush a filter onto the image:

1. In the Paint menu, click the Filter field.
   The filter library appears.
2. Select the filter you want to use.

You are returned to the Paint menu and the filter name appears in the Filter field.

3 Click Filter to enable the selected filter.

4 Set the brush opacity. The opacity determines the level of filtering. Reduce the opacity value to reduce the level of filtering.

5 Paint on the image.



Original image

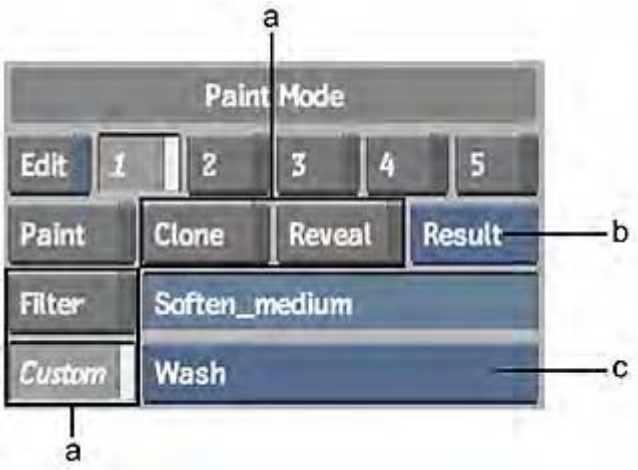


After using the emboss filter

Using Special Effects Media

Special Effects media can be brushed onto the image in Canvas mode. You can also apply the Wash, Shade, and Reveal media to the entire image using the Wipe command. See Wiping Using Special Effects Media and Filters (page 1058). Only one media type can be used at a time.

Use the media buttons to enable some Special Effects media, for example, the Clone and Reveal media. Other media types, such as Drag and Smear, are enabled using the Custom Media box.



(a) Special Effects Media buttons (b) Reference box (c) Custom Media box

The following Special Effects media are available.

Use:	To:
Blur	Apply a blur filter to portions of the image. See Blurring the Image (page 1048).

Use:	To:
Clone	Copy a portion of the image to a new location. See Cloning the Image (page 1049).
Drag	Create an image trail from a selected region of the screen. See Dragging the Image (page 1050).
Impressionist	Paint on colours from a reference clip. See Using the Impressionist Medium (page 1051).
Recursive Clone	Make a number of copies of a selected area of the image. See Using the Recursive Clone Medium (page 1052).
Reveal	Brush a reference image onto the current image. See Revealing a Reference Image (page 1052).
Shade	Darken or lighten the image depending on the luminance value of the current colour. See Washing and Shading the Image (page 1056).
Smear	Smudge areas of the image. See Smearing the Image (page 1053).
Stamp	Apply a captured image to the image. See Using the Stamp Medium (page 1054).
Warp	Stretch and distort a region of the image. See Warping the Image (page 1055).
Wash	Apply a transparent wash of the current colour to the image. See Washing and Shading the Image (page 1056).

Blurring the Image

Use the Blur medium to blur portions of the image.

To blur the image:

1 Click Custom and select Blur from the Custom Media box.
The Blur option boxes appear.



(a) Filter Type box (b) Blur Strength box

2 Click Canvas.

3 Set the brush size according to the size of the area you want to blur.

4 Select either a Box or Gaussian filter from the Filter Type box.

5 Set the density of the Blur brush from the Blur Strength box.
You can choose Light, Medium, or Heavy.

6 Drag the brush on the image.



Original image

After using the Blur medium

Cloning the Image

Use the Clone medium to copy a region of the image and paint it on a destination point. The result image is used as the source for the Clone medium. The offset between the origin point and the destination point is set in the Offset fields.

To paint using the Clone medium:

1 Click Clone.
The Clone controls appear.



(a) Offset fields

2 Click Canvas.

3 Set the brush opacity. The brush opacity determines the transparency of the clone. When the opacity value is set to 100%, the clone is completely opaque.

4 Set the brush size.

5 Set the offset between the origin point and the destination point in the Offset fields.

A tracking circle appears at the offset co-ordinates you specified; this is the destination point for the cloned image.

TIP Press `Ctrl` and drag the cursor to set the destination point.

6 Paint on the image.

The image contained within the red circle brush is copied to the region defined by the green circle.



Original image

After using the Clone medium

Dragging the Image

Use the Drag medium to drag a selected area of the image across the canvas. The selected area is painted on the canvas as you drag the brush, creating an image trail.

NOTE The Direction brush attribute cannot be used with the Drag medium.

To use the Drag medium:

1. Click Custom and then select Drag from the Custom Media box.
2. Click Canvas.
3. Set the brush size.
4. Position the brush over the region of the image that you want to use. Hold down the cursor to select that region.
5. Drag the brush.

The selected region is painted onto the image as you drag. The pressure of the brush determines the density of the image trail. See Brush Pressure (page 1041).



Original image



After using the Drag medium

Using the Impressionist Medium

Use the Impressionist medium to brush on colours from a reference clip. The brush opacity determines how much colour is taken from the reference clip. When the opacity value is set to 100%, all the colour is taken from the reference image. At 50%, the colour applied is a 50/50 blend of the reference colour and the result image.

To use the Impressionist medium:

1. Click Custom and then select Impressionist from the Custom Media box.
2. Click Canvas.
3. Set the brush opacity.
4. Select a reference image from the Reference box.
5. Paint on the image.



Original image



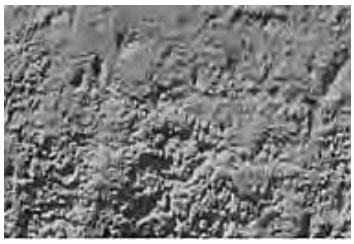
After using the Impressionist medium

Revealing a Reference Image

Use the Reveal medium to reveal portions of an image or an entire reference image. You can reveal specific areas using the brush or the entire image using the Wipe command. The opacity of the brush determines how much of the reference image is revealed on the image. When the opacity is set to 100%, the reference image applied is completely opaque. At 50%, the reference image applied is a 50/50 blend of the reference image and the result clip.

To reveal a reference image on the canvas:

1. Click Reveal.
2. Click Canvas.
3. Set the brush opacity.
4. Select a reference image from the Reference box.
5. Paint on the image.



Back image



Result image



The Back image revealed on the Result image

Using the Recursive Clone Medium

Use the Recursive Clone medium to make multiple copies of an area of an image. The result image and the paint applied to it is used as the source. Each copy is a degraded version of its predecessor. The offset between the origin point and the destination point is set in the Offset fields.

To paint using the Recursive Clone medium:

1. Click Custom and select Recursive Clone from the Custom Media box.
   The Recursive Clone options appear.

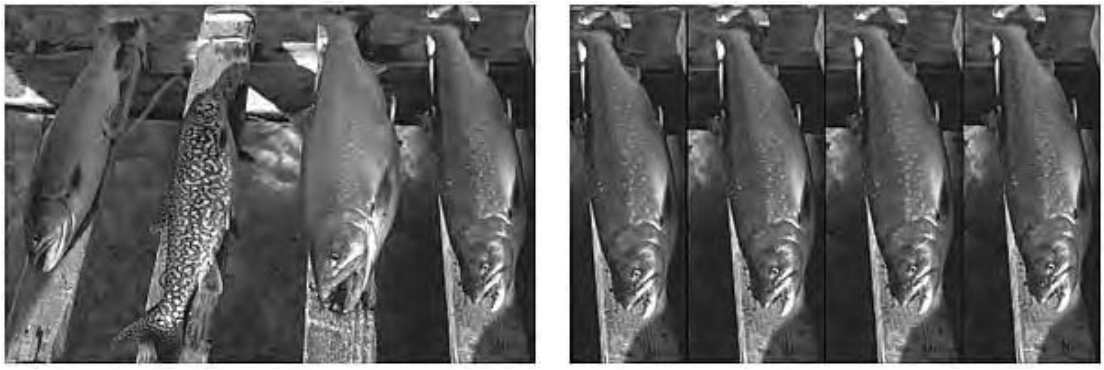


(a) Offset fields

2 Click Canvas.

3 Set the offset between the origin point and the destination point in the Offset fields.
A red tracking circle appears around the destination point.

4 Paint on the image.
The image contained within the green circle brush is copied to the region defined by the red circle.



Original image

After using the Recursive Clone medium

Smearing the Image

Use the Smear medium to smudge portions of the image.

To smear the image:

1 Click Custom and select Smear from the Custom Media box.

2 Click Canvas.

3 Drag the brush over the area you want to smear.



Original image

After using the Smear medium

Using the Stamp Medium

Use the Stamp medium to capture a portion of the image and apply it to the canvas.

To capture and apply a stamp:

1 Click Custom and select Stamp from the Custom Media box.
The Stamp window appears beneath the Brush Size field.



(a) Brush Size field (b) Stamp window

2 Click Canvas.

3 Set the brush size. Use a small brush to isolate a specific detail of the image. Use a large brush to capture a bigger sample.

4 Click and hold the cursor on the Stamp window.

5 Without releasing the cursor, move it over the image.
The Stamp window is updated as you move the cursor across the image.

6 Release the cursor when the Stamp window contains the part of the image you want to capture.
The brush cursor appears.

7 Paint on the image.



(a) The captured stamp



(a) Stamp applied using a circular brush (b) Stamp applied using an air brush (c) Stamp applied using a chalk brush

Saving and Loading Stamps

You can save a stamp and load it in another session to use with a different clip. See Saving Setups (page 1111).

Warping the Image

Use the Warp medium to stretch and distort regions of the image.

To warp the image:

1. Click Custom and select Warp from the Custom Media box.
2. Click Canvas.
3. Set the brush size.
   The area that can be warped is determined by the brush size.

4 Select a portion of the image and hold down and drag the cursor to warp the selection.



Original image



After using the Warp medium

Washing and Shading the Image

Use the Wash medium to apply a transparent wash of the current colour to the image. Use the Shade medium to darken or lighten the image. You can wash or shade specific areas using the brush or the entire image using the Wipe command.

With the Shade medium, you darken images using a colour with a low luminance value, and lighten them using a colour with a high luminance value. The opacity of the brush affects the transparency of the paint applied to the image. When the opacity value is set to 100%, the paint applied is completely opaque. As you decrease the opacity, the paint becomes more transparent.

Selecting a Colour Model

You can apply the colour using either the YUV or HLS.

To select a colour model:

1 Click Setup.
2 Select either YUV Shade/Wash or HLS Shade/Wash.



To use Wash or Shade:

1. Click Custom, and select Wash or Shade from the Custom Media box.
2. Click Canvas.
3. Set the current colour.
4. Set the brush opacity.
5. Paint on the image.



A paint stroke using the Paint medium, current colour red.



A paint stroke using the Wash medium, current colour red.



A paint stroke using the Shade medium, current colour red.

Wiping an Image

Use the Wipe command to apply colours, filters, and Special Effects media to the entire result image in a single stroke.



(a) Wipe colour pot

NOTE You can also wipe an image with the AutoPaint Wipe mode. See Wiping the Canvas in AutoPaint (page 1064).

Wiping Using a Colour

Use the Paint medium to wipe the image with a selected colour. The colour used is set in the Wipe colour pot.

To wipe the image using a colour:

1. Set the current colour you want to use for the wipe. See Selecting Colours (page 1030).
2. Click the Wipe colour pot.
   The current colour is transferred to the wipe colour.
3. Set the brush opacity.
   A value of 100% wipes the image with a completely opaque colour.
4. Click Wipe.

Wiping Using Special Effects Media and Filters

You can use a filter, or the Reveal, Clone, Wash, and Shade Special Effects media.

To wipe with Special Effects media and filters:

1. Set the wipe colour.
2. From the Paint Mode controls, select the Special Effects medium to be applied. See Using Special Effects Media (page 1047).



NOTE If you are using a filter for the wipe, you must enable the Filter button.

3 Set the brush opacity.

A value of 100% wipes the image with the full effect of the Special Effects Media or filter.

4 Enable the Wipe Using Paint Mode button.

5 Click Wipe.

Using the Wipe Command in Graphics

You can also use the Wipe command in the Graphics menu. The Wipe command wipes over any objects tacked down on the image. Objects not tacked down are not part of the image and are not wiped over.

Filling an Image

Use the Fill command to fill areas of an image with similar colour values or areas delimited by a colour. These areas can be filled with either a solid colour or a reference image. You can choose how far the filled area extends by specifying how similar the pixels must be in order to be filled. This enables you to fill only the dark areas of an image, or include slightly lighter areas.

Use the Fill controls to set the colour model, range, and softness of the fill.

To display the Fill controls:

1 Click Fill in the Paint menu. If the Fill button is hidden by the colour palette, swipe the bottom of the screen.

Defining the Range for the Fill

When you use the Fill command, you specify a range for the colour comparison. The Fill command compares the colour values of adjacent pixels to determine if the values are within the specified colour range. Adjacent pixels within the range are filled.

The range is determined using two values: the colour value of the pixel you select to begin the fill, called the *selection point*, and the value you set in the Range field.

Selecting Colour Channels

You can use either the RGB or YUV model. Within each colour space, you can work with any combination of colour channels. For example, if you select the R, G, and B channels in the RGB colour model, the Fill command considers the red, green, and blue values. Adjacent pixels with red, green, and blue values within the specified range are filled.

Adjusting the Softness

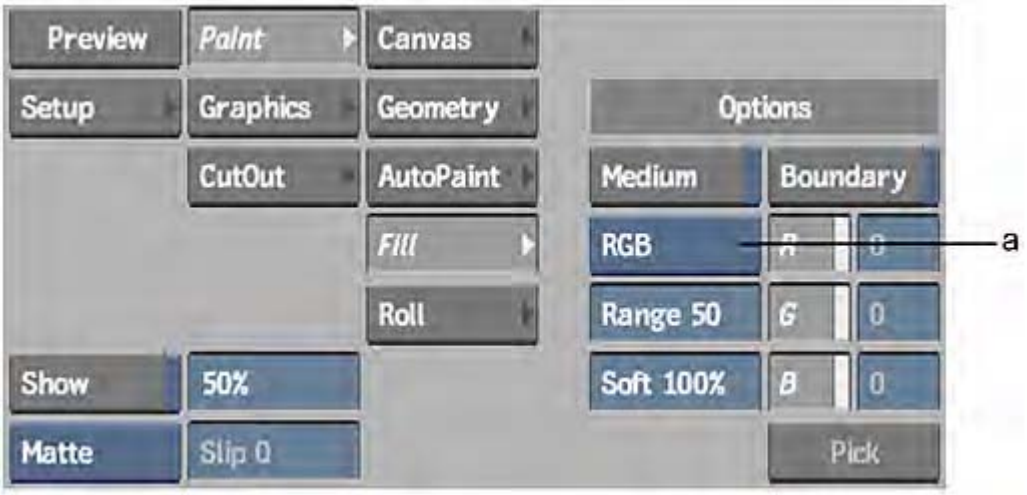
The softness value for the fill determines the amount of colour diffusion applied at the edges of the fill. This value can be adjusted to modify the transition between the filled and unfilled areas. A softness value of 100% produces the greatest amount of diffusion. A softness value of 0% produces a filled area with well-defined edges.

Filling a Region with a Colour

You can choose to fill an image with the current colour.

To fill a region of the image with a colour:

1. Click Fill.
   The Fill controls appear.



(a) Colour Model box

NOTE To fill a matte, click From Matte. When this button is enabled, the pixels in the matte are used for the colour comparison. If this button does not appear at first, click Matte on the right side of the menu panel.

2. Select a colour model from the Colour Model box.

Select:	To:
RGB	Display the R, G, and B channel buttons. Enable each channel you want to use.
YUV	Display the Y, U, and V channel buttons. Enable each channel you want to use.

3. Set the colour range in the Range field.
   You can also set a tolerance by enabling the Boundary button and selecting a distinct boundary colour from the image.
4. Set the softness in the Soft field.
5. Set the current colour.
6. Select a point in the area you want to fill. This is the selection point.

The pixels that fall within the specified range are filled with the current colour.



Original image



Fill with range 75 and softness 100



Fill with range 75 and softness 50



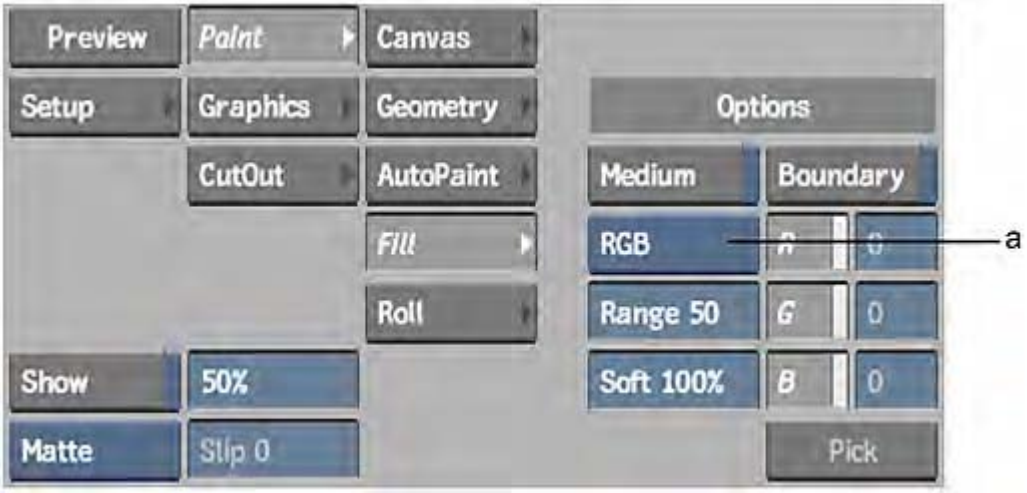
Fill with range 50 and softness 100

Filling a Region with an Image

You can fill a region of an image with a reference image instead of a colour. The front, back, or result image can be used as the reference image. The Wash, Shade, Clone, or Reveal media can also be used to perform the fill.

To fill a region with a reference image:

1 Click Fill.
The Fill controls appear.



(a) Colour Model box

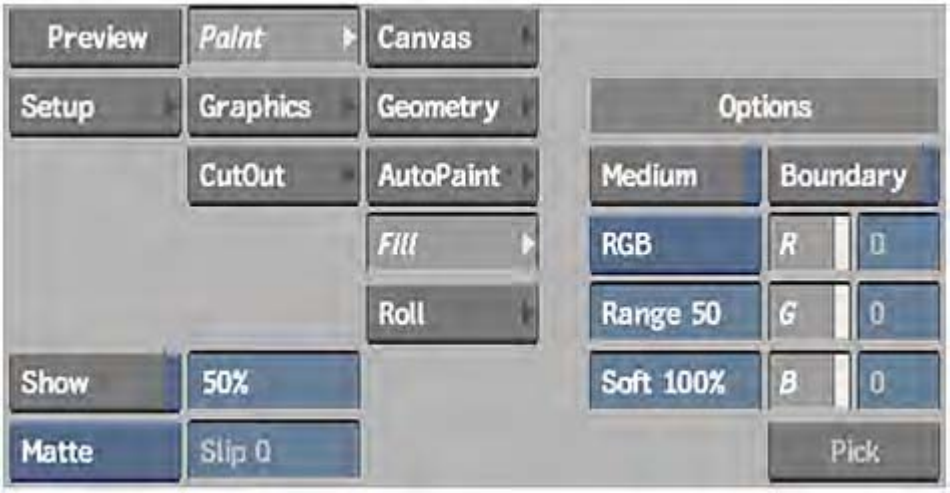
2. Select the colour model and channels you want to work with.
3. Set the range and softness in the Range and Softness fields.
4. Select the Special Effects medium you want to use (Clone, Reveal, Wash, or Shade). See Using Special Effects Media (page 1047).
5. Set the Reference box to Front, Back, Result, or Saved, and set the brush opacity.
   A value of 100% fills the area completely with the reference image.
6. Enable Medium.
7. Select a point in the area you want to fill.

Filling a Boundary

Use the Boundary option to fill an area of the canvas delimited by another colour.

To use boundary fill:

1. Click Fill.
   The Fill controls appear.



2 Select the colour model and channels you want to work with.

3 Set the range for the fill in the Range field.

4 Enable Boundary.

The Pick button is no longer greyed out.

5 Click Pick.

The cursor changes to a colour picker when dragged over the image.

6 Drag the colour picker over the canvas without clicking.

The various colour channel values in areas of the image are displayed as you move the colour picker around the canvas.

7 Click a point to select the colour for the boundary. You can also select a colour by entering the RGB values directly in the colour channel fields.

The cursor changes to a paint bucket.

8 Click the area inside the boundary to fill that region.

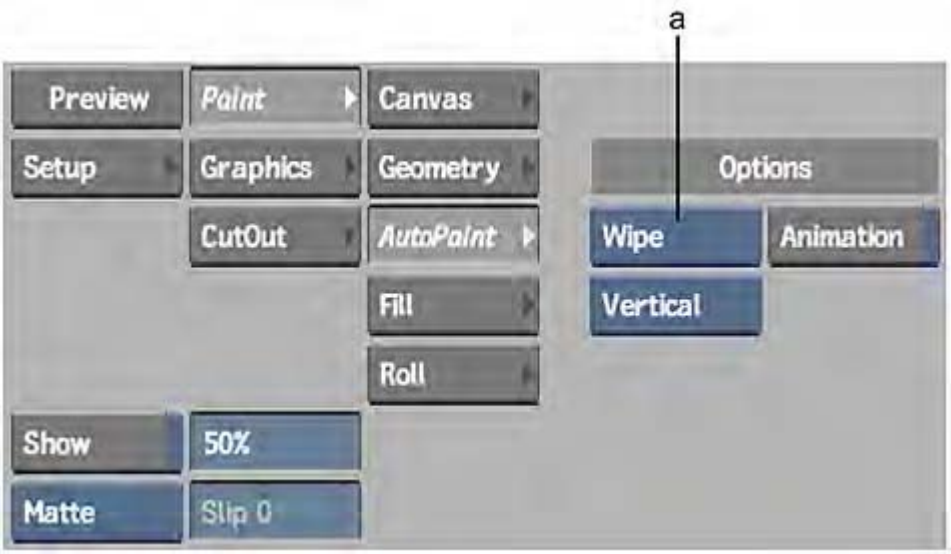
NOTE To fill a boundary with a reference image, set the Reference box to Front, Back, or Saved, and enable Medium.

Using AutoPaint

Use the AutoPaint controls to apply paint strokes to each frame in the result clip, a range of frames, or the current frame.

To display the AutoPaint controls:

1 Click AutoPaint in the Paint menu. If the controls are hidden by the colour palette, swipe the bottom of the screen.



(a) Autopaint Mode box

The AutoPaint Mode options are described as follows.

User Records and plays back a series of manual paint strokes.

Random Applies a number of random strokes to the image.

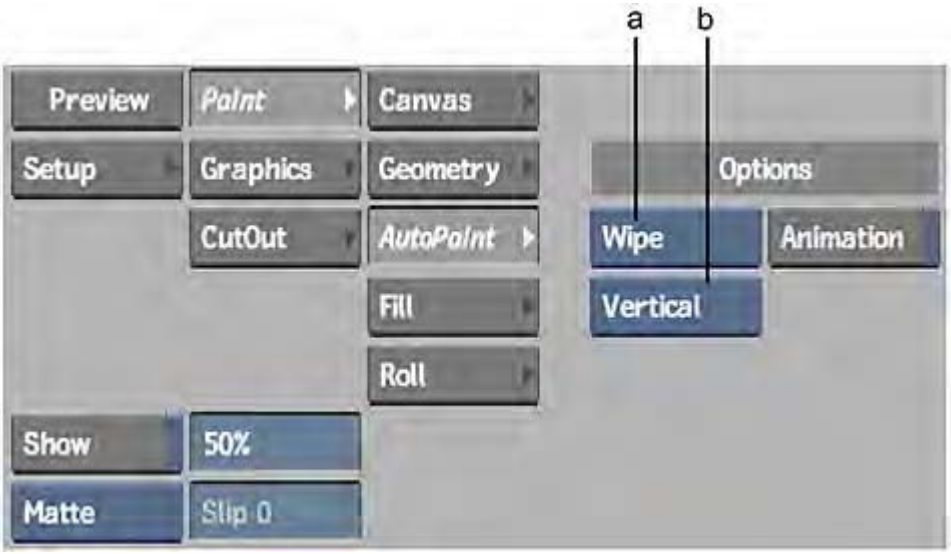
Wipe Wipes the entire canvas with the current colour or medium.

Wiping the Canvas in AutoPaint

Use Wipe mode to automatically apply paint strokes from left to right, top to bottom, or diagonally across the entire canvas. You can use a filter or any of the Special Effects media except Warp, Drag, and Smear. The size of the brush determines the number of strokes applied. Use a small brush to apply several strokes or a large brush to apply fewer strokes.

To wipe the canvas in AutoPaint:

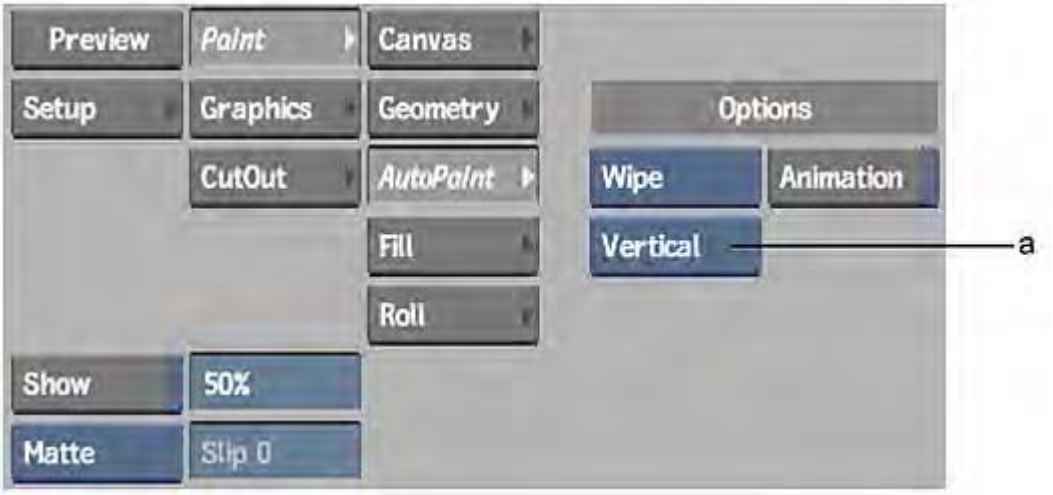
1. Click AutoPaint, and select Wipe from the AutoPaint Mode box.
   The Wipe menu appears.



(a) AutoPaint Mode box (b) Wipe Mode box

2. Set the current colour.
3. Select the Special Effects medium or filter, and set the brush attributes. See Using Special Effects Media (page 1047), and Brush Attributes (page 1040).
4. Select the Wipe mode in the Wipe Mode box. If this box is not visible, make sure that the attribute mode for the Direction attribute is set to Direction.

Select:	To:
Diagonal	Apply diagonal strokes.
Vertical	Apply strokes from top to bottom.
Horizontal	Apply strokes from left to right.



(a) Wipe Mode box

5 Enable Animation to display the Channel Editor.

If the Channel Editor is not visible, swipe the bar below the menu.

You can animate all brush attributes as well as the current colour.

6 Enable Wipe if you are using a filter or colour.

7 Click the image to apply the paint strokes only to the current frame. Click Process to apply the paint strokes to each frame in the front clip.



Original image



After using the Wipe command with Jitter attribute at 35% and Colour attribute mode set to Front

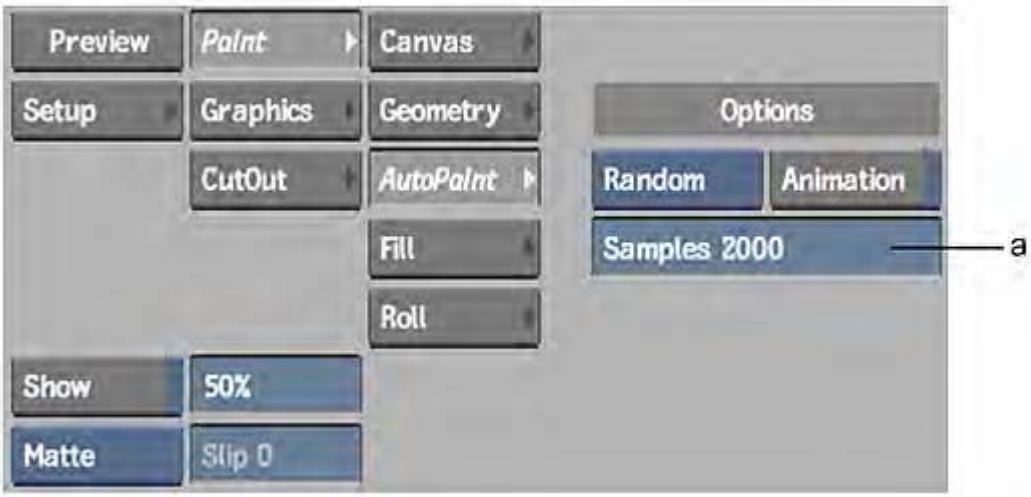
Applying Random Strokes

Use Random mode to automatically generate a number of paint strokes with random orientation on the image. Use the Samples field to specify the number of random strokes you want to generate. You can use any Special Effects media in Random mode except Warp and Drag.

To apply random paint strokes to the image:

1 Click AutoPaint and select Random from the AutoPaint Mode box.

The Samples field appears.



(a) Samples field

2 Enter the number of strokes you want to apply in the Samples field.

3 Set the current colour.

4 Select the Special Effects medium, and set the brush attributes. See Using Special Effects Media (page 1047), and Brush Attributes (page 1040).

5 Enable Animation to display the Channel Editor. If the Channel Editor is not visible, swipe the bar below the menu.

You can animate the following parameters:

- Number of strokes
- All the brush attributes
- Colour

6 Click the image to apply the paint strokes to the current frame only. Click Process to apply the paint strokes to each frame in the front clip.



Original image



After using the Random command with Jitter attribute at 30% and Colour attribute mode set to Front

Recording Brush Strokes

Use User mode to record and play back a series of paint strokes. Only the positions of the brush strokes are recorded. This means that you cannot change the brush type, current colour, or brush attributes while

recording the strokes. You can, however, change or animate these parameters after you finish recording and before you play back the strokes.

TIP You can also play paint strokes created by converting objects (write-ons).

To record a series of brush strokes:

1. Click AutoPaint and select User from the AutoPaint Mode box.
2. Set the brush characteristics so that you can see the recorded brush strokes on your image.

   NOTE The strokes you paint are recorded as a series of points or stamps of the brush. The number of strokes is determined by dividing the number of points by the number of frames. The quality of the rendered strokes depends on the number of recorded points. To record a larger number of points, select a small brush size.

3. Enable Record.
4. Begin painting on the image.

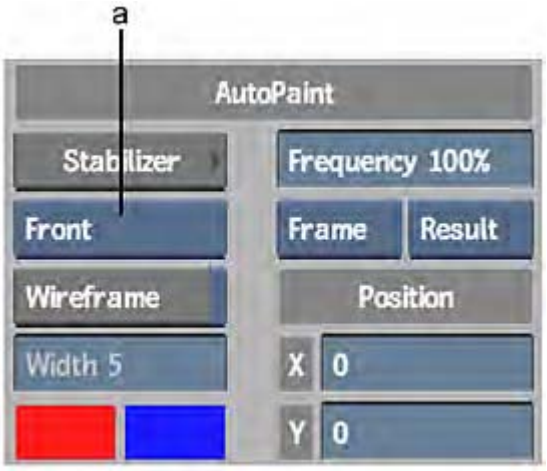
   The position of each brush stroke is recorded, and the stroke count appears in the message bar.
5. To stop recording, click below the timebar.

   All paint strokes applied to the image while recording are removed from the image and the number of strokes is recorded.

   NOTE AutoPaint stores only one set of recorded paint strokes at a time. If you record another set of paint strokes, you lose your previously recorded strokes.

Tracking with AutoPaint

You can apply tracking data to the painted strokes.



(a) Tracking box

To apply tracking data to recorded strokes:

1. Enable Animation.
2. Select Front, Back, or Result from the Tracking box and click Stabilizer to track a point and apply the offset information to the recorded strokes.

   NOTE You can only track after you record paint strokes.

3. Use the X and Y field to offset the painted strokes.

Playing Recorded Brush Strokes

After recording brush strokes, you can apply them to the front clip using the Play option, which is activated automatically when you finish recording the paint strokes.

To play recorded strokes:

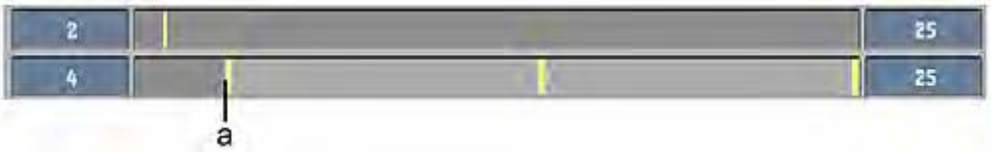
1 Enable Play.

The Play options appear.



2 Define the brush characteristics using the Brush Attribute fields.

3 Define the duration of the stroke sequence using the second timebar. Drag the indicator to modify the duration.



(a) Indicator

The number of strokes is divided by the length of the stroke, defined by the second timebar.

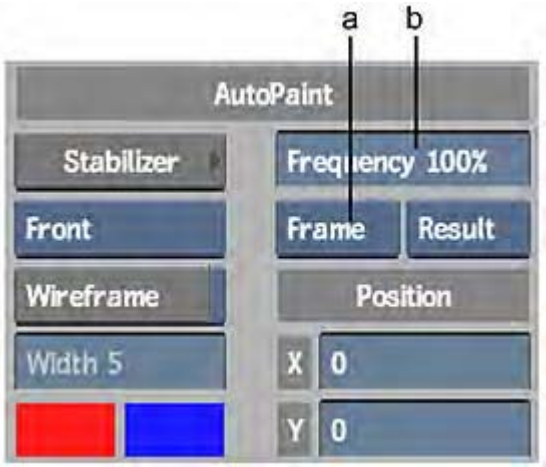
The indicator in the second timebar is identical to a track in the Channel Editor.

4 Enable the Play options.

Enable:	To:
Part	Play only a part of the recorded strokes. AutoPaint applies the strokes to a frame, erases those strokes, and moves to the next frame. This has the effect of creating streaks on your rendered clip.
Backward	Play the paint strokes backward.
Distance	Play paint strokes based on distance. AutoPaint divides the length of paint strokes by the number of frames to determine what to render in each frame.
Stroke	Play the paint strokes simultaneously.

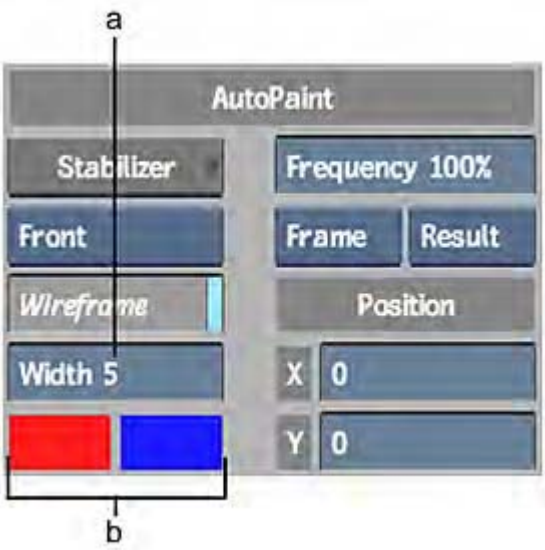
5 Enable Animation.

The Animation menu appears.



(a) Animation Type box (b) Frequency field

6 Enable Wireframe to preview the path(s) of the painted strokes.



(a) Width field (b) Wireframe colour pots

The wireframe preview shows the complete path(s) of the painted strokes. It also shows what will be painted in each frame as set using the second timebar.

You can set the width of the wireframe using the Width field. You can also change the colour of the path and progression wireframes using the colour pots beneath the Width field.

For Path animation, you define how often AutoPaint reads the Channel Editor values when rendering the points or stamps that make up a given stroke. At a frequency of 100%, AutoPaint reads the Channel Editor values the most frequently.

For example, assume you set your AutoPaint sequence of 100 stamps to run over 10 frames using Path animation. If you set the Frequency field to 100%, AutoPaint reads the Channel Editor values 10 times every frame. If you set the Frequency field to 50%, AutoPaint reads the Channel Editor values 5 times every frame.

TIP Use the Frequency field to lower the processing time for your AutoPaint sequence. Use a low frequency to render a quick preview of the result.

7 Swipe the bar at the bottom of the menu to display the Channel Editor.

You can animate the following parameters:

- Sampling amount
- X and Y values
- Brush size, rate, pressure, jitter, direction, roll, opacity, and colour
- Tracker translation

NOTE The Channel Editor only appears if Animation is enabled. Also, animation data in the Channel Editor is not erased when you record new strokes.

8 To apply the paint strokes, click Process.

NOTE If you are zoomed in on the image and are in the Raster zoom mode, only the visible portion of the image will be processed. This is much faster than using Tiled mode, but will not apply your modifications to the entire frame. For more information on zoom modes, see Painting on Full-Resolution Film Images (page 1033).

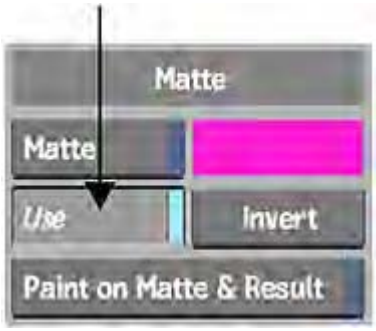
About Mattes

Mattes protect specific areas of the image when you apply paint, filters, or Special Effects media to the canvas. You can also use mattes to limit the area of a cutout.

For instructions on loading mattes into Paint, see Loading Clips into Paint (page 1026), and Loading Setups (page 1112).

To use the matte:

1 Click Use to enable the matte.



2 Paint on the image.

3 Click Use again to disable the matte.

Using Mattes with Cutouts

If you enable your matte when creating a cutout, the cutout is limited by the matte. Only objects outside the matte will show up in the cutout. See Creating a Cutout (page 1102).

Viewing Mattes

You can view the matte as you use it.

To view the matte:

1. Load the matte into Paint.
2. Enable Show.
3. Select Matte in the Reference box.



The matte appears.

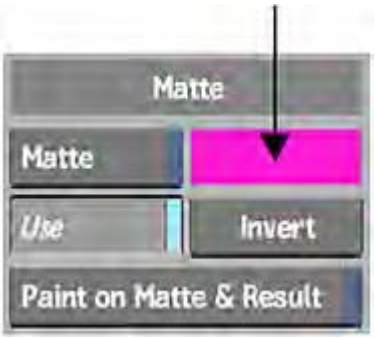
Changing the Matte Colour

If the colour used to display the matte blends with the image, you can change its colour.

NOTE The matte is always created using shades of grey.

To change the matte display colour:

1. Set the current colour.
2. Click the Matte colour pot.



The matte colour display changes to the current colour.

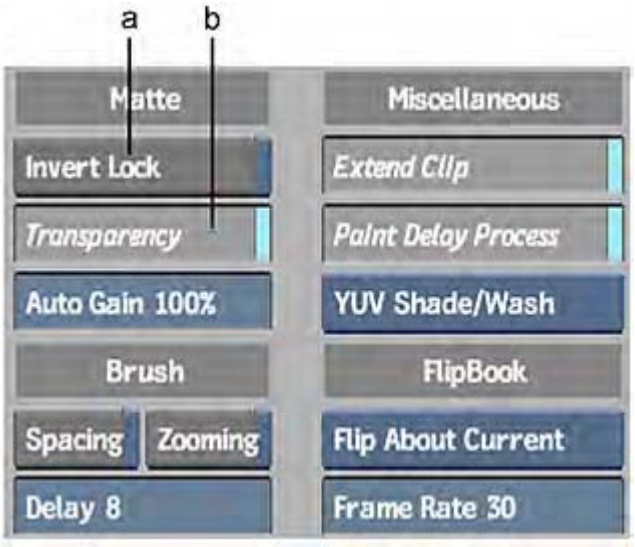
Changing the Matte Display Transparency

When viewing the matte, use the Transparency field to vary the matte display transparency from 0% (completely transparent) to 100% (fully opaque). Press `Alt+T` or click Transparency in the Setup menu to toggle between the last set value and 100%.



(a) Transparency field

NOTE The transparency value only affects the display of the matte. It does not affect how the matte is used.



(a) Invert Lock button (b) Transparency button

Inverting the Matte Clip

Click Invert to invert the matte in the current frame. Enable Invert Lock in the Setup menu to invert the matte in every frame of the clip.

Creating or Modifying Mattes

You can create or modify a matte by painting directly on the matte using any brush, Special Effects media, or graphic tool. For information on using graphic and cutout tools to create mattes, see Using the Cutout Commands (page 1103).

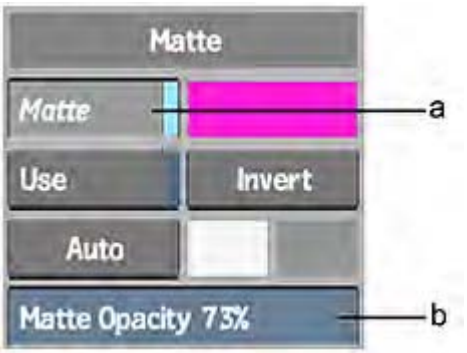
To create or modify a matte:

1. Load the matte into Paint.

   NOTE You must load a matte clip in order to process a matte in Paint. If you want to create a matte from scratch, load a black source as the matte clip.

2. Click Matte.

   The Matte menu appears.



(a) Matte button (b) Matte Opacity field

3 Set the paint transparency in the Matte Opacity field.

The value in the Matte Opacity field determines the transparency of the paint applied to the matte. Use a value of 100 to apply fully opaque paint or objects to the matte. Reduce the Matte Opacity value to increase the transparency of the paint.

4 Paint or place graphics or cutouts on the matte.

The paint and graphics are applied to the matte. If you use graphic tools, use Tack to tack the graphic to the matte. See Tacking Down Selected Objects (page 1101).

5 You can save the matte in the matte library. See Saving Setups (page 1111).

6 To process the matte clip, click Exit.

The new matte clip is saved to the rendered destination reel.

Painting on the Matte and Image

When applying paint strokes or graphics to the image, enable Both to paint the image and the matte simultaneously.

Resetting the Matte

If you do not load a matte, Paint displays the last matte that was loaded.

To erase the matte:

1 Click Matte.

2 Set the Matte Opacity to 0%.

3 Click Wipe.

The matte is erased.

Creating a Gradient Matte

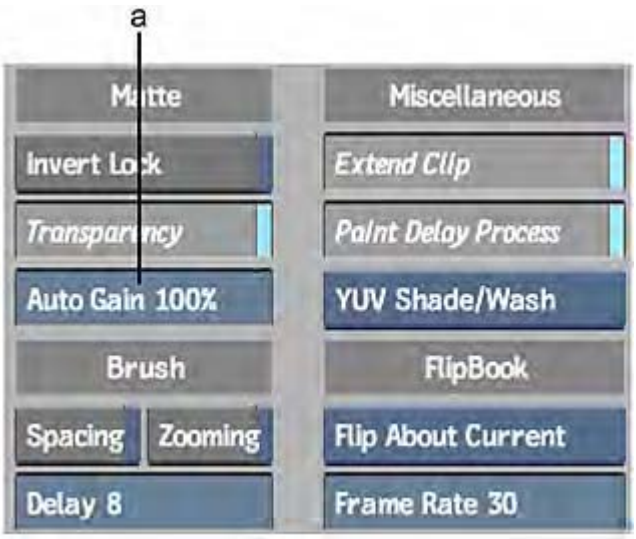
You can use graphics that contain gradients to create a matte that blends from one level of the matte colour to another level. A gradient matte is useful for blending paint strokes or creating soft edges.

To create a gradient matte, draw a graphic with a gradient on the matte, and then use the Channel Editor to change the "a" channel of the start or end colour of the gradient. See Creating Graphics (page 1075), and Changing the Object Gradient (page 1089).

Creating High-Contrast Mattes

Use the Auto command to create a high-contrast matte. This command works like the Auto Matte command in the Processing menu. The minimum and maximum luminance values for the matte are set using the two colour pots in the AutoMatte menu. Any pixel with a luminance value below the minimum is set to black, and any pixel with a luminance value above the maximum is set to white.

Unwanted grey areas can be removed from the matte by increasing the percentage in the Auto Gain field in the Setup menu.

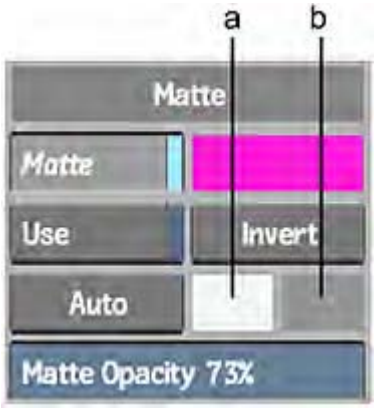


(a) Auto Gain field

Pixel values between the minimum and maximum values are multiplied by the Gain value. The resulting values are clipped at the specified maximum luminance value. The Gain is expressed as a percentage value. The default value of 100% has no effect on the image since the luminance values are multiplied by 1.

To use the AutoMatte command:

1. In the Paint menu, click Matte.
   The AutoMatte menu appears.



(a) Minimum luminance (b) Maximum luminance

2. Set the minimum luminance value for the matte in the colour pot on the left. To set the value, click the field and use the colour picker to select the lightest colour from the image.
3. Set the maximum luminance value for the matte in the colour pot on the right.

4 Set the Auto Gain for the matte. The Auto Gain field is in the Setup menu.

5 Click Auto to generate the matte.

6 Click Use to enable the generated matte.

Using Graphics in Paint

Accessing the Graphics Menu

To access the Graphics menu, click the Graphics button in the Paint menu. Some options may be hidden if the colour palette is open. Swipe the cursor across the bars at the bottom of the screen to hide the colour palette.



Creating Graphics

Graphics can be used to create cutouts or mattes, or to apply geometrical shapes, text, and fills to the result clip.

To create a graphics object:

1 In the Paint menu, click Graphics.

2 Click Add.

3 Select the type of object to add by selecting an icon in the Object Tools window.



4 Draw the object on the image. See Using the Object Tools (page 1076).

5 Set the resolution to be used when adding or editing objects. See Setting the Object Resolution and Display (page 1085). Objects appear at full resolution when they are drawn.

6 Set the object's attributes and gradient. See Setting the Object Attribute (page 1087), and Changing the Object Gradient (page 1089).

7 Set the object's colour in the Current Colour pot. See Selecting Colours (page 1030).

8 Resize and move the object. See Changing the Size or Position of an Object (page 1091).

9 Use the Transformation box or controls to move, rotate, or scale the object. See Changing the Shape of an Object (page 1092).

10 Use the Animation controls to animate the object. See Animating Graphics (page 1096).

11 Click the Tack button to permanently place the object on the result clip.

Once the object has been tacked down, it cannot be moved, rotated, deleted, resized, or copied. See Tacking Down Objects (page 1100).

Auto Edit After Add

After you add an object to the image, Paint automatically switches from Add mode to Edit mode. You can turn off this default setting by disabling the Auto Edit After Add button in the Setup menu. You can then add objects one after another without interruption.



(a) Auto Edit After Add button

Using the Object Tools

A number of predefined graphic tools are available in Paint. The cursor looks the same for all graphic types. When you move the cursor onto an image, it appears as a small green cross with crosshairs that extend the width and height of the canvas. The crosshairs help align objects on the image.

Each tool has its own icon in the Object Tools window.

To select an object tool:

1 Scroll through the Object Tools window.



Click the Object Tools window and drag left or right. Use the left mouse button to scroll slowly, the middle button to scroll faster, and the right button to scroll the fastest.

2 Click the object tool icon you want to use.

The selected object tool is highlighted by a blue outline.

NOTE Only one object tool can be active at a time.

Drawing a Line

Use the Line object tool to draw a single straight line or a multi-line object (a series of lines joined end-to-end).

To draw a line or a series of lines:

1. In the Graphics menu, click Add.
2. Select the Line tool in the Object Tools window.
   The Create Multiple Lines button appears.
3. Click Create Multiple Lines.
4. To draw multiple lines, move to the canvas and click to place the start point of the first line. Click again to draw the end point. Continue clicking to draw more lines.
5. To end a multiple line object, click Create Multiple Lines.
6. To draw single lines, move to the canvas and click, drag, and release.
   Paint draws the line as you drag.

Drawing a Rectangle

Use the Rectangle object tool to draw a rectangle or square in one of two ways:

- Corner-to-corner, with the two vertices located at diagonally opposite corners of the rectangle
- Centre-to-corner, with the first vertex at the centre of the rectangle and the second at one corner

To draw a rectangle from corner-to-corner:

1. In the Graphics menu, click Add.
2. Select the Rectangle tool in the Object Tools window.
3. Press the cursor to anchor one corner of the rectangle. Do not release the cursor.
4. Drag the cursor diagonally. You can adjust the width and height of the rectangle as long as you hold down the cursor.
5. Release the cursor to anchor the second vertex.
   The rectangle is drawn on the image.

To draw a rectangle from centre-to-corner:

1. In the Graphics menu, click Add.
2. Select the Rectangle tool in the Object Tools window.
3. Press `Alt`.
4. Position the cursor where you want to place the centre of the rectangle and press down.
5. Drag diagonally, and release the cursor when the rectangle is the correct size.

Drawing a Square

Press `P` while you draw and repeat the procedure for drawing a rectangle from corner-to-corner.

Drawing a Triangle

Use the Triangle object tool to draw a triangle or equilateral triangle. The vertices are located at the three corners of the triangle.

To draw a triangle:

1. In the Graphics menu, click Add.
2. Select the Triangle tool in the Object Tools window.
3. Position the cursor and press to anchor the first vertex on the image. Do not release the cursor.
4. Drag the cursor horizontally to draw the base of the triangle. You can continue to adjust the length of the base as long as you press down on the cursor.
5. Release the cursor to anchor the second vertex.
6. Move the cursor to where you want to place the third vertex. You can continue to adjust the position of the third vertex until you press down on the cursor. Press to anchor the third vertex.

   The triangle is drawn on the image.

Drawing an Equilateral Triangle

An equilateral triangle has three sides of equal length. To draw an equilateral triangle, repeat the procedure for drawing a triangle and press `P` while drawing the base of the triangle. This also establishes the height of the triangle. Release the cursor to anchor the second and third vertices on the image.

Drawing an Ellipse

Use the Ellipse object tool to draw an ellipse defined by three vertices. The first vertex determines the centre of the ellipse. The second vertex determines the horizontal radius of the ellipse. The third vertex determines the vertical radius.

To draw an ellipse:

1. In the Graphics menu, click Add.
2. Select the Ellipse tool in the Object Tools window.
3. Position the cursor at the centre of the ellipse and press to anchor the point on the image. Do not release the cursor.
4. Drag the cursor horizontally to establish the width of the ellipse. You can continue to adjust the width as long as you press down on the cursor.
5. Release the cursor to anchor the second vertex.
6. Move the cursor along the vertical axis to position the third vertex. You can continue to adjust the height of the ellipse until you press down on the cursor. Press to anchor the third vertex.

   The ellipse is drawn on the image.

To draw a circle with the Ellipse tool:

1. In the Graphics menu, click Add and select the Ellipse tool.
2. Press `P`.
3. Position the cursor at the centre of the circle and press to anchor the point on the image. Do not release the cursor.

4 Drag the cursor horizontally to establish the radius of the circle. Notice that there are two vertices at the edge of the circle, as there are for an ellipse.

5 Release the cursor and the `P` key to anchor the second and third vertices.

The circle is drawn on the image.

Drawing a Circle

Use the Circle object tool to draw a circle in one of two ways:

- Centre-to-edge, with one vertex at the centre of the circle and a second on the circumference
- Edge-to-edge, with two vertices at opposite points on the circumference

To draw a circle from edge-to-edge:

1 In the Graphics menu, click Add.

2 Select the Circle tool in the Object Tools window.

3 Position the cursor at the centre of the circle and press to anchor that point on the image. Do not release the cursor.

4 Drag the cursor to establish the radius of the circle. You can continue to adjust the size of the circle as long as you press down on the cursor.

5 Release the cursor to anchor the second vertex.

The circle is drawn on the image.

To draw a circle from centre-to-edge:

1 In the Graphics menu, click Add and then select the Circle tool.

2 Press and hold `Alt`.

3 Drag the cursor to establish the radius of the circle.

When the circle is the required size, release the cursor.

Drawing a Polygon

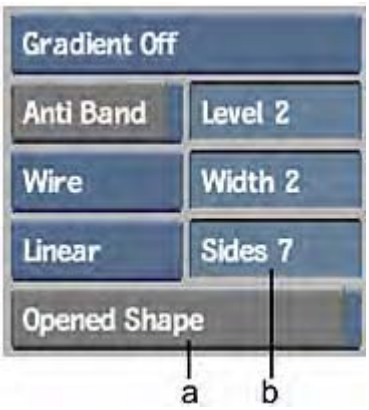
Use the Polygon object tool to create a series of connected lines that form a closed or open object. You can specify how the vertices of the polygon should be connected by selecting one of the options from the Polygon Mode box.

To draw a polygon:

1 In the Graphics menu, click Add.

2 Select the Polygon tool in the Object Tools window.

The Polygon menu appears.



(a) Polygon Mode box (b) Sides field

3 Select an option from the Polygon Mode box to set the type of curve used to join the vertices of the polygon.

Select:		To:
Linear		Use straight lines to join the vertices of the polygon.
Cardinal		Use a smooth curve that passes through the vertices of the polygon.
Bspline		Use a very smooth curve that passes on the inner side of the vertices of the polygon.
Bezier		Use Bezier curves. Each vertex of the polygon has a tangent with two tangent handles. In Edit mode, you can move the tangent handles to adjust the slope of the polygon. See Editing a Bezier Curve (page 1093).

The minimum number of vertices for a polygon depends on the type of interpolation used. A polygon should have at least three vertices when using Linear interpolation, or four when using Cardinal or Bspline interpolation. The maximum number of vertices is 255. A polygon can be concave.

4 To create an open-ended polygon, click Opened Shape.

5 Position the cursor at one vertex of the polygon, and click to anchor that point on the image.

6 Release the cursor and move it to where you want the next vertex of the polygon. You can continue to adjust the positioning of the next vertex until you press down on the cursor. Press to anchor the point on the image.

7 Repeat steps 4 and 5 for each vertex.

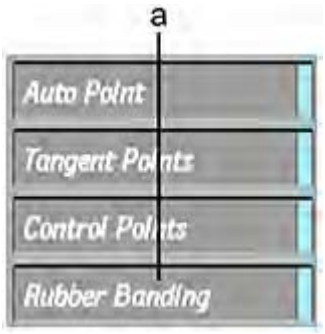
8 To close the polygon, press anywhere outside the canvas.

The polygon is drawn on the image.

Rubber Banding

Rubber banding is the default setting for drawing polygons and lines. This means that a vertex is added to the polygon only when you press and release the cursor.

To draw a polygon with multiple vertices, disable the Rubber Banding button in the Setup menu. When you drag the cursor while drawing a polygon or line, vertices are drawn on the image.



(a) Rubber Banding button

Drawing Regular Polygons

Draw regular polygons using the Sides field and the P key. A polygon can have three or more sides. This enables you to draw a triangle or a square based on the centre point of the object.

To draw a regular polygon:

1 In the Graphics menu, click Add.

2 Select the Polygon tool in the Object Tools window.

3 Enter the number of sides for the polygon in the Sides field.

4 Press P.

5 Position the cursor over the image and press down.

This is the centre of the polygon.

6 Drag the cursor away from the centre point. The polygon is drawn on the image. You can continue to adjust the size of the polygon as long as you press down on the cursor.

7 When the polygon is the correct size, release the cursor.

The polygon is drawn on the image.

Drawing a Fill Object

Use the Fill object tool to fill areas of an image with colour. While this is similar to using the Fill command in the Paint menu, you can only use this tool with colour, Shade, and Wash, and not to fill areas with a reference image. However, you can edit fill objects after you draw them.

NOTE A fill object cannot have a gradient.

To draw a fill object:

1 In the Graphics menu, click Add.

2 Select the Fill object in the Object Tools window.

The Fill object controls appear.



3 Select a colour model from the Colour Model box (RGB or YUV).

4 Select the colour channels you want to use (R, G, and B, or Y, U, and V).

The choice of colour channels determines how Paint evaluates the area you want to fill. For example, if you select R, G, and B, Paint fills areas whose adjacent pixels have red, green, and blue values in the specified range. If you select only R, Paint fills only areas whose adjacent pixels have red values in the specified range.

5 Enter the Range and Softness values for the fill.

The range determines how far the fill spreads from the point you click on the image; if you increase the range, you increase the fill area. The softness determines how much diffusion occurs at the edges of the fill; if you increase the softness, you increase the diffusion at the edges of the fill.

6 Move the cursor to the canvas and click inside the area to fill.

Paint fills the area with the current colour starting at the point you click.

NOTE The colour of the image and the channel, range, and softness settings determine the extent of the fill. As you move the fill object, it changes size and shape in response to the different colours in the image.

Drawing a Text Object

Use the Text object tool to add text to the image. You can change the size, kerning, and inclination of the text string.

To add a text object:

1 In the Graphics menu, click Add.

2 Select the Text tool in the Object Tools window.

The Text object controls appear.



(a) Text field (b) Font field

3 Click the Font field and select a new font from the font library.

4 Click the Text field, type the text string, and click Enter.

5 Click the image to place the text object.

You can adjust the size, kerning, and italics of the text in Edit mode.

Use:	To:
Size	Adjust the size (in pixels) of the text.
Italic	Incline the text. Enter a positive value to slant the text to the right, and a negative value to slant it to the left.
Kern	Adjust the space (in pixels) between all letters in the text.

Selecting Objects

You must select an object before you can edit it. In Edit mode, you can select objects in three ways:

- Click an object to select it.
- Use the Selected field to select individual objects.
- Use the All command to select all objects at once.

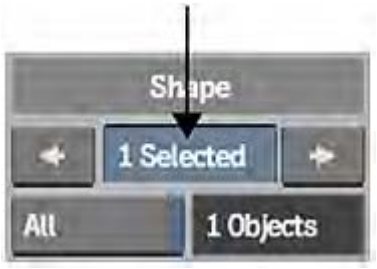
Using the Selected Field

As you add objects to the image, they are numbered sequentially, starting at 1. You can select an object by specifying the number of the object in the Selected field.

As you change the number in the Selected field, the corresponding object is highlighted by a selection box.

To select a single object using the Selected field:

1. Set the number in the Selected field to the number of the object you want to edit.



   You can also click the < and > buttons beside the Selected field to move through the sequence.
2. When the selection box outlines the object, release the cursor. You can now edit it.

Selecting More than One Object

To select an additional object, hold down `Shift` and click another object. Repeat this step for each object you want to select.

Deselecting Objects

To deselect an object or objects, select a different object or click the Add button.

Selecting All Objects

To select all the objects, click the All button. All is shown in the Selected field. Click it again to deselect them.

Editing Objects

Use the object controls to copy, delete, hide, or layer objects.



Copying Objects

Use the Copy command to create multiple copies of an object. Once a copy is created, it can be selected and edited like any other object.

To copy an object:

1. In the Graphics menu, click Copy.
2. Click the object you want to copy. Do not release the cursor.

   A copy of the object is superimposed on the original image. You cannot see the copy until you drag it to a new location.
3. Drag the copy to a new location.
4. When the copy is positioned correctly on the image, release the cursor.

Deleting Objects

Use the Delete command to remove one or more objects from the image.

To delete one or more objects:

1. Select the objects you want to delete. Click an individual object or use the Selected box to select an object. To select multiple objects, click an object, press `Shift`, and click any other objects you want to delete.
2. Click Delete.

   All selected objects are removed from the image.

To delete all objects from the image:

1. Enable All.
2. Click Delete.

All objects are deleted from the image.

Hiding Objects

Use the Hide/Unhide commands to hide or unhide objects before they are tacked onto the image.

To hide or unhide one or more objects:

1. Select the objects you want to hide. Click an individual object or use the Selected box to select an object. To select multiple objects, click an object, press `Shift`, and click any other objects you want to hide.
2. In the Graphics menu, click Edit and then click Hiding.
   The Hide options appear.

Select:	To:
Hide Selected	Hide all selected objects.
Hide Unselected	Hide all objects that are not selected.
Unhide All	Show all hidden objects.

Changing the Order of Overlapping Objects

Objects can be drawn so they overlap other objects to produce a stack of objects.

To send an object to the bottom of the stack:

1. Select the object.
2. In the Graphics menu, click Push and then click the object until it is at the bottom of the stack.

   NOTE To bring the object back to the front, click the object until it moves to the front.

Setting the Object Resolution and Display

Use the Object Resolution box to set the resolution while drawing or editing objects. The objects are regenerated at high resolution once you release the cursor at the end of each editing operation. Use the options in the Setup menu to affect the display of the graphics before they are tacked down onto the image.

To set the object resolution:

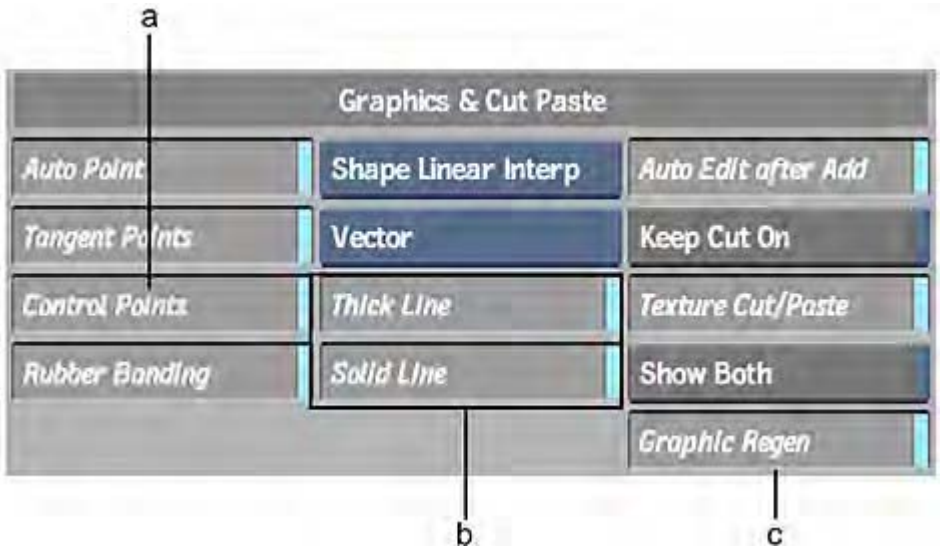
1. Select an option from the Object Resolution box.



Select:	To:
Full Res	Display the object at full resolution when updating. This is helpful for displaying gradients, but slows down the rate at which image display is refreshed.
Low Res	Display the object at low resolution when updating.
Wireframe	Display the object as a wireframe when updating.

Wireframe Display Options

The Thick Line and Solid Line buttons in the Setup menu determine the thickness and continuity of the wireframe.



(a) Control Points button (b) Thick Line and Solid Line buttons (c) Graphic Regen button

Enabling and disabling the Thick Line and Solid Line buttons affects the wireframe as follows:

- When Thick Line is enabled, the wireframe has a thickness of 2 pixels.
- When Thick Line is disabled, the wireframe has a thickness of 1 pixel.
- When Solid Line is enabled, the wireframe is unbroken.
- When Solid Line is disabled, the wireframe is dashed.

Increasing Editing Speed

To speed up editing, disable the Graphic Regen button in the Setup menu. Instead of waiting for the graphics objects to be regenerated at the end of an editing operation, the objects appear at the display resolution. This can be a useful time-saving feature.

Hiding the Object Vertices

Each object is defined by two or more vertices. These vertices, or tangent handles, appear while you are drawing or editing the object.

Disable the Control Points button in the Setup menu to turn off the display of the vertices while you are drawing or editing the object. This feature is useful if you want to trace the outline of a figure.

Displaying Anamorphic Geometry

Select Use Aspect Ratio in the Rendering area of the Setup menu to draw and display anamorphic geometry. For normal display, click Use Aspect Ratio and switch to Use Square Pixels.



(a) Anti-alias field (b) Use Aspect Ratio option

Anti-Aliasing for Objects

The jagged edges that can occur along diagonal and curved lines in geometry are caused by aliasing, or insufficient spatial sampling of the image. The process of minimizing jagged edges by increasing the sampling rate is called *anti-aliasing*.

Use the Anti-alias field in the Rendering Setup menu to specify an anti-aliasing factor for geometry. When the value is set to 1, no anti-aliasing occurs. Best results are obtained with a value of 4.

Setting the Object Attribute

Use the Object Attribute box to set the display mode for the object you add to the image.

To change the object attribute:

1 In the Graphics menu, click Edit.

2 Select the object you want to edit.

3 Click Attributes.

4 Select an option from the Object Attribute box.



Select:	To:
Solid	Draw a filled object with a well-defined edge. Use the Brush Opacity field to set the object's transparency. No other brush attributes or types are available. Line objects cannot be set to Solid.
Outline	Draw an outline of the object. Use the Brush Attribute buttons to set the appearance of the outline. See Changing the Object's Appearance (page 1088).
Fuzzy	Draw a solid object with a soft edge. Use the Brush Attribute fields to set the appearance of the outline.
Wire	Draw a wireframe object. Use the Brush Opacity field to set the object's transparency. No other brush attributes or types are available.

Changing the Object's Appearance

You can change the appearance of an object by changing any of the following display attributes:

- Current colour
- Brush attribute values or modes
- Special Effects media or filters
- Colour gradient

Changing display attributes affects all currently selected objects. See Selecting Objects (page 1083).

Setting Brush Attributes and Modes

In the Graphics menu, you can only change brush attributes and modes when using the Outline or Fuzzy attribute. See Brush Attributes (page 1040).

If the Brush Attribute controls are not visible, swipe the bar at the bottom of the menu. Use the Current Colour pot to set the graphic's colour, and the Opacity field to set the opacity of the object, or the effect a graphic has on the image.

Using Filters and Special Effects Media with Graphics

Any filter can be used with objects. Only the Paint, Clone, Reveal, Wash, and Shade Special Effects media can be used. See Using Special Effects Media (page 1047).

Changing the Object Gradient

All objects, except the fill object, can have a gradient. You can edit a gradient or apply a gradient to an object that does not have one. A selected object with a gradient displays a gradient control bar that you can use to edit the orientation of the gradient.

To edit a colour gradient:

1. In the Graphics menu, click Edit.
2. Select the object you want to edit.
3. Click Attributes.
4. Select an option from the Gradient box.



(a) Gradient box (b) Noise Level field

Select:	To:
Rectangular Gradient	Use a rectangular gradient in which colour changes from top to bottom.
Circular Gradient	Use a circular gradient in which colour changes from centre to edge.
Gradient Off	Turn off the gradient and use a solid colour.

5. Set the colours for the gradient. See Using the Colour Gradient Bar (page 1032).
6. To eliminate banding, enable Anti Band.

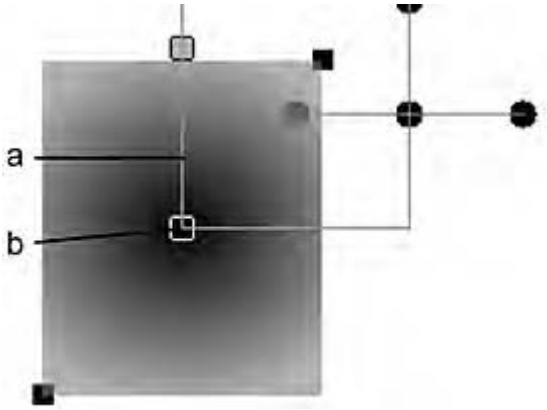
   Banding normally appears only if you are working at 8-bit colour depth and if the gradient that you apply is stretched.

TIP You can also set a noise level to the anti-band function. Noise may help when colour bands are very large. Set the value in the Noise Level field to 7 to apply the maximum noise, or lower it to apply less dither/noise. If you set the value to 0, no noise is applied.

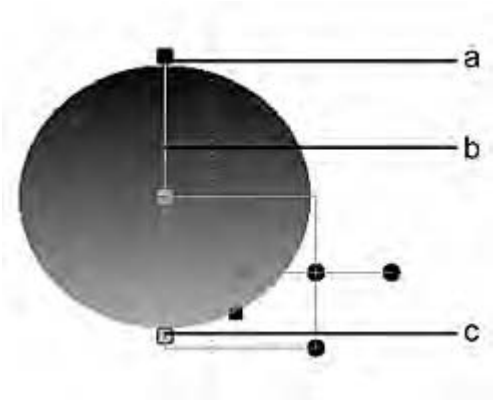
7 Use the gradient control bar to change the object's gradient.

Using the Gradient Control Bar

When you select an object with a gradient, Paint displays a gradient control bar in addition to the object transformation box. The gradient control bar has two handles at each end that show the colours of the gradient. In a rectangular gradient, the bar also indicates the direction of the gradient.



(a) Gradient control bar (b) Centre point of gradient



(a) Start point of the gradient (b) Gradient control bar (c) End point of the gradient

NOTE If you want to move the transformation box, press M and click the location where you want to move the box.

To use the gradient control bar:

1 Select an object with a gradient, or add a gradient to an object.

Paint displays the transformation box and the gradient bar.

2 Move the handles of the gradient control bar to change the orientation and location of the gradient.

In a rectangular gradient, this affects the gradient orientation. For example, the default orientation for a rectangular gradient is top to bottom. If you drag one of the handles left or right, you change the orientation to diagonal. If you drag the handles so the gradient bar is horizontal, you get a rectangular gradient with the colours blended from side to side.

If you select the centre colour handle in a circular gradient, you can move the gradient's centre inside the object or even outside it.

3 To decrease the amount of one colour visible in an object, move the handle to the edge or outside the object.

Changing the Size or Position of an Object

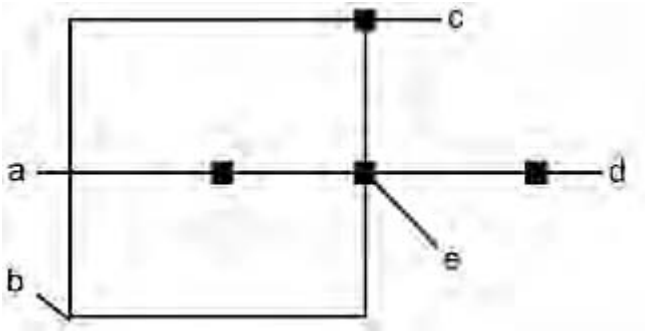
Use the transformation box, or the transformation controls in the Axis Transformation menu, to change the size or position of an object. You can use the transformation box and controls to modify the object in the following ways:

- Move the object in any direction on the image.
- Rotate the object about its centre point.
- Change the size of the object.

You can translate, resize, or rotate more than one object at the same time by selecting all the objects you want to edit.

Using the Transformation Box

To display the transformation box, click the Edit button in the Graphics menu and click the object you want to edit.



(a) Reset handle (b) Centre point (c) Resize handle (d) Rotation handle (e) Translation handle

The position of the transformation box is saved for each object. To move the transformation box, press M and click the destination.

Resize handle To change the size of an object, drag the resize handle in any direction. To maintain an object's proportions, press PE as you drag the resize handle.

Centre point To change the point around which the object rotates, move the centre point.

Rotation handle To rotate the object about its centre point, drag the rotation handle.

Translation handle To move the object in any direction on the image, drag the translation handle. The transformation box moves with the object.

Reset handle To restore the object to its initial size and position on the image, click the reset handle.

Using the Transformation Controls

Use the transformation controls to translate, rotate, and resize a selected object. To display the transformation controls, select an object and click the Transform button.



X (Position) Move the object along the horizontal axis (X-axis).

Y (Position) Move the object along the vertical axis (Y-axis).

X (Scale) Change the size of the object along the horizontal axis.

Y (Scale) Change the size of the object along the vertical axis.

Rotation Set the object's angle of rotation along the Z-axis. The object is always rotated around its centre point.

Reset Use the Reset button to reset the object to its initial size and position.

Start and End Numeric Fields Use these fields to set when the object appears in the clip.

Changing the Shape of an Object

To change the shape of an object, such as a polygon or a line, you must add, delete, or move a vertex on the object.

To change the shape of an object:

1. In the Graphics menu, click Edit.
2. Select the object you want to edit.
   The object vertices and the transformation box appear on the selected object.
3. Press the vertex you want to edit and drag it to a new location. When you are satisfied with the new position, release the cursor.

Adding and Deleting Vertices on a Polygon or Line

Change the shape of a polygon or line by changing its number of vertices.

To add a vertex to a polygon or line:

1. In the Graphics menu, click Edit.

2 Select the polygon or line you want to edit.

   The vertices for that object are selected.

3 Press A.

4 Press on one of the existing vertices and drag outward. A vertex is added counterclockwise to the selected point.

   To add a point clockwise to the selected point, press Ctrl+A.

5 Repeat steps 3 and 4 for each vertex you want to add to the object.

To delete a vertex from a polygon or line:

1 In the Graphics menu, click Edit.

2 Select the polygon or line you want to edit.

   The vertices for that object are selected.

3 Press D.

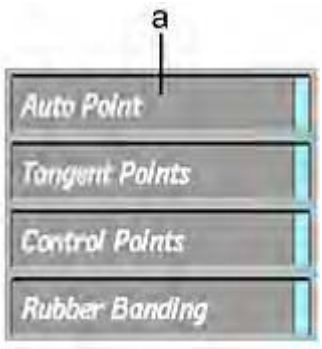
4 Click the vertex you want to delete.

   The selected vertex is deleted from the object.

5 Repeat steps 3 and 4 for each vertex you want to delete on the object.

The Auto Point Command

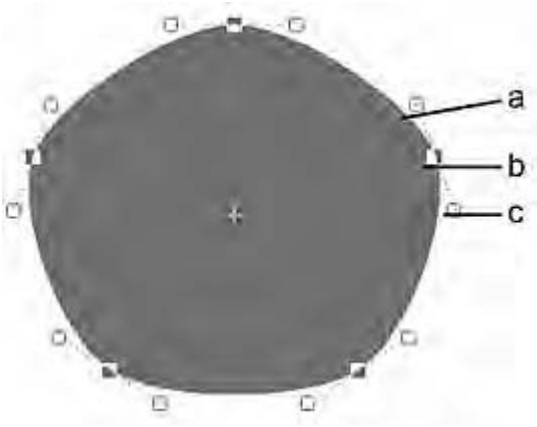
When you add or delete a vertex on a polygon or line, the same point is added or deleted on all shapes for that object. You can disable this option in the Setup menu. When the Auto Point button is disabled, any point you add or delete on a shape is added/deleted in the current shape key only.



(a) Auto Point button

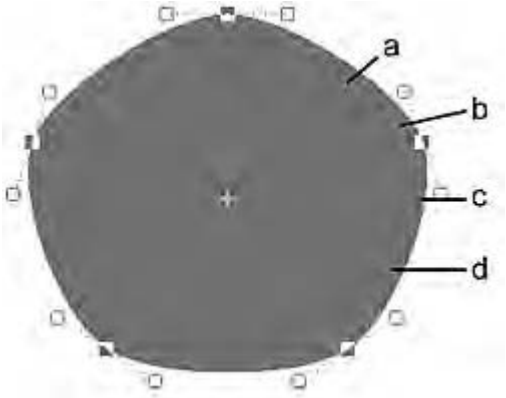
Editing a Bezier Curve

When you use the Bezier option to draw a polygon, each vertex of the polygon has two tangents. Each tangent ends with a handle.

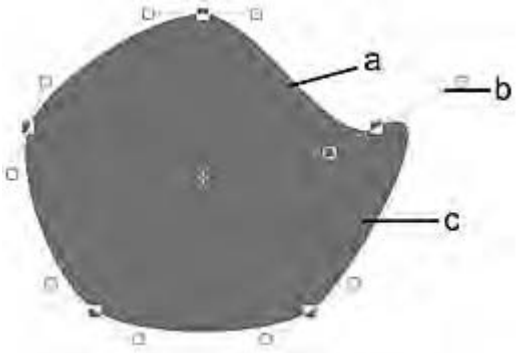


(a) Tangent (b) Vertex (c) Handle

Use Edit mode to move a tangent handle and adjust the slope of the adjacent side of the polygon. When you move one tangent handle, the tangent's reciprocal handle moves in the opposite direction because handles A and B are joined to keep the joint at the intersection of the two edges smooth.

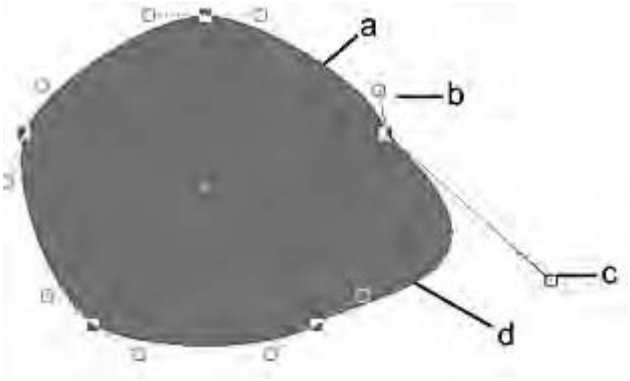


(a) Edge A (b) Tangent A (c) Handle B (d) Edge B



(a) Edge A (b) Handle B (c) Edge B

To adjust the slope on one side of the vertex only, press B and click the vertex. This breaks the tangent so that you can move its handles independently. To reset the tangent, press B and click the vertex again.



(a) Edge A (b) Handle A (c) Handle B (d) Edge B

Converting Graphics to AutoPaint Strokes

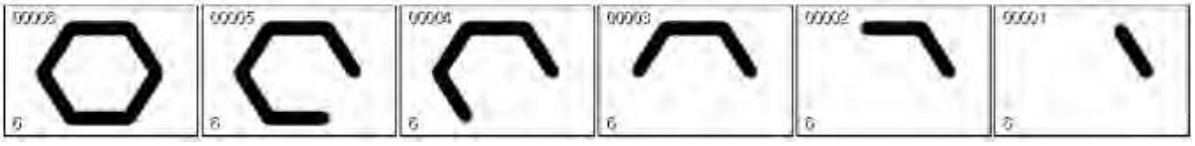
Use the Convert command to convert objects to AutoPaint strokes.



(a) Convert button

You can play the paint strokes using the AutoPaint controls in the Paint menu. See Using AutoPaint (page 1063).

The following shows how a hexagon is drawn over six frames. One edge is drawn in each frame. After the object is converted to AutoPaint strokes, it is always drawn as an outline regardless of its attributes in the Graphics menu.



To determine how much of an object to draw in each frame, the number of edges in the object is divided by the number of frames in the clip. The polygons are drawn one after the other in the order in which they were drawn in the Graphics menu.

To convert an object to AutoPaint strokes:

1. Select the object.
2. Click Convert and Confirm.

Animating Graphics

Use the Channel Editor and Shape Animation controls to animate Paint graphics. You can animate the position, display, and shape of a graphic. To display the Channel Editor, click the Animation button and swipe the cursor across the bottom of the screen.

Animation Parameters

The following parameters can be animated for every object:

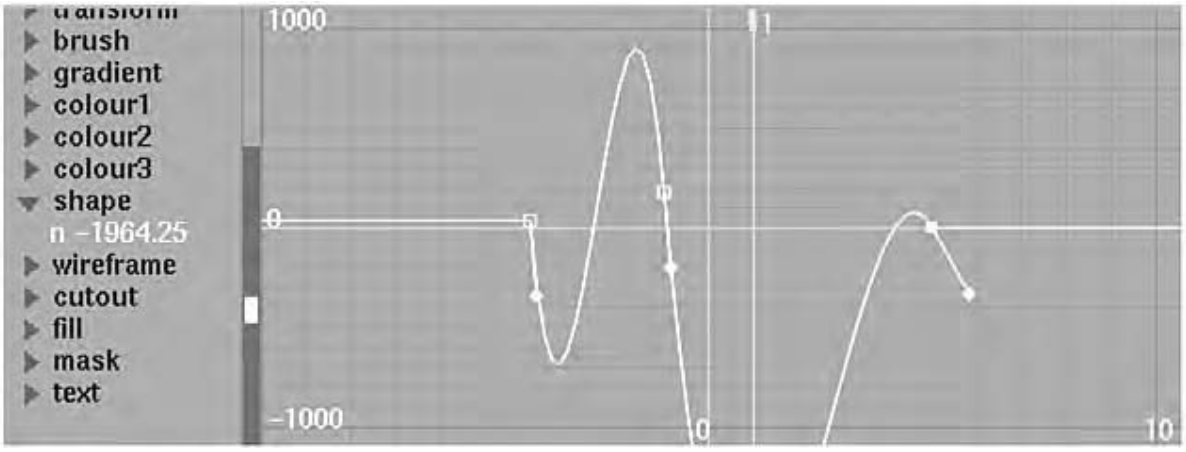
- Position, rotation, and size
- Brush attributes
- Gradient orientation, transparency, and colour
- Colour (red, green, and blue channels)
- Wireframe width
- Range and Softness of a fill object
- Transparency
- Size, Kerning and Italics for a text object

NOTE In Paint, the commands in the Edit Mode box can be performed on animation curves only. To modify a keyframe by editing the object in the image window, you must use the editing commands in the Graphics menu.

Shape Animation

To animate the shape of an object, you must create keyframes with the Shape Animation controls. Each shape you define becomes a keyframe in the Shape channel of the Channel Editor. The difference between the keyframes is interpolated and the shape animation is created.

The Shape channel is used to identify the number and location of shape keyframes in a clip. Use the shape curve to control the rate at which an object changes to a new shape. The following figure shows the Shape channel for an object that changes from shape 1 in frame 1, to shape 2 in frame 8, and shape 1 in frame 15.



To animate the shape of an object:

1 Create an object.

This original shape becomes shape keyframe 1 in the Channel Editor.

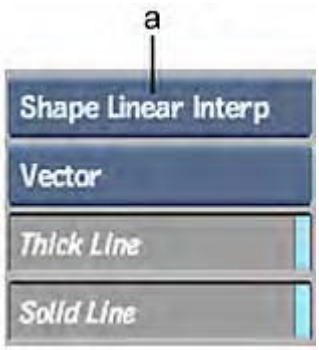
2 Advance to another frame in the clip.

3 Select the object, and change its shape.

To change its shape, you must move, add, or delete a vertex on the object. This new shape is shape keyframe 2 in the Channel Editor.

Shape Interpolation

Use the Shape Interpolation box in the Setup menu to specify the interpolation between shapes in the animation.



(a) Shape Interpolation box

Select:	To:
Shape Linear Interp	Produce sudden transitions between shapes.
Shape Cardinal Interp	Produce smooth transitions between shapes.

NOTE The type of shape interpolation you use is independent of the type of polygon interpolation you use.

Moving between Shape Keyframes

Use Previous and Next to move between shape keyframes in an animation.

To move between keyframes:

1 Select the object you want to modify.

2 Click Edit and then Shapes.

The Shapes menu appears.



3 Click Next to advance to the next shape keyframe.

4 Click Previous to go to the previous shape keyframe.

Deleting Keyframes

Use Delete to delete a shape keyframe from the animation.

To delete a shape keyframe:

1. Select the object.
2. Click Shapes.
   The Shapes menu appears.
3. Select the keyframe you want to delete.
4. Click Delete.
   The selected shape is deleted.

Adding Keyframes

Use Add to create a shape keyframe. This can be useful for creating animations that start and end with the same shape.

To add a shape key using Add:

1. Select the object you want to animate.
2. Go to the frame where you want to add the shape keyframe.
3. Click Shapes.
   The Shapes menu appears.
4. Click Add.
   A new keyframe is added to the Shape channel in the Channel Editor.

Gradient Animation

Use the Channel Editor to animate the colours, orientation, and direction of a colour gradient. The Gradient folder includes Start and End folders for the two colours in the gradient. The Start and End folders each contain channels for the X and Y position of the colour, as well as the R, G, B, and alpha values.

Although you can animate the gradient independently, by default, the gradient will follow any transformations of the object.

To animate a gradient:

1. Select the object with the gradient you want to animate.
2. Click Animation. If necessary, swipe the cursor across the bottom of the screen to display the Channel Editor.
3. Click the Geometry and Stroke folders to expand them.
   If you selected more than one object, a Stroke folder appears for each selected object. Click the Stroke folder for the object you want to animate.
4. Click the Gradient folder to expand it, and click the Start and End folders:
   - The Start folder contains the X and Y channels and a Colour folder for the gradient's colour.
   - The End folder contains the X and Y channels and a Colour folder for the gradient's colour.

5 Click to expand each Colour folder.

Use:	To:
x	Move colour origin left or right. This is the same as moving the gradient bar handle.
y	Move colour origin up or down. This is the same as moving the gradient bar handle.
r	Change percentage of red in the colour.
g	Change percentage of green in the colour.
b	Change percentage of blue in the colour.
a	Change percentage of the colour's alpha channel. Use this channel to create a gradient matte or to change the opacity of the colour.

6 Change the values for the channels in different frames to create a gradient animation.

Processing the Animation

To process the animation, click Process in the Graphics menu. Make sure you are at the first frame of the animation since the clip will be processed from the currently displayed frame until the end of the clip.

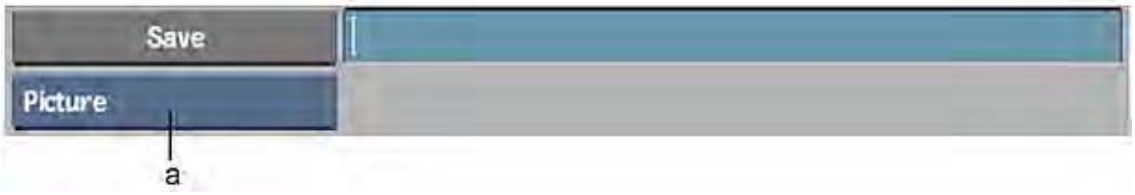
Saving and Restoring Objects

Use the Store and Recall commands to save and restore objects to the image. Use the Library menu to save or load graphics from the graphics library.

To save or load a graphic:

1 In the Paint menu, click Load or Save.

2 Select Geometry from the Load or Save option box.



(a) Save option box

3 Select a file or type in a name for the graphic.

NOTE If you are using Save and have typed in a name for the file, you must press `Enter` to finish saving the file.

The Store and Recall Commands

To save the selected graphic in the buffer, click Store in the Graphics menu. The current object replaces the graphic saved in the buffer.

The Store command does not save the result image. This means that if you tacked the objects onto the image, they are not saved using the Store command. To restore the geometry to the image, click Recall.



(a) Store button (b) Recall button

TIP Use Store and Recall to add graphics to clips loaded at a later time.

Tacking Down Objects

An object can be selected, deleted, edited, and transformed until it is tacked down on the image. As soon as it is tacked down, the object becomes part of the image and can no longer be manipulated.

There are several ways to tack down an object:

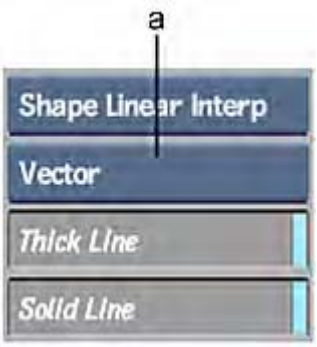
- Select a Tack mode so that an object is either tacked down as soon as it is drawn or as soon as the next object is drawn.
- Tack down selected objects only.
- Tack down all objects simultaneously using the Tack All command.

Tack Modes

The Tack mode is selected with the Tack Mode box in the Rendering Setup menu.

To set the Tack mode:

1 In the Paint tool, click Setup.
The Setup menu appears.



(a) Tack Mode box

2 In the Rendering area, select an option from the Tack Mode box.

Select:	To:
Vector	Explicitly tack down the objects using the Tack command.
Edit Last	Edit the last object drawn until the next object is added to the image. As soon as the next object is added, the last object is tacked down.
Raster	Tack down an object as soon as it is added to the image.

Tacking Down Selected Objects

When the Tack mode is set to Vector, objects must be explicitly tacked down on the image using the Tack button.



(a) Tack button

To tack down one or more objects:

1 Select the objects you want to tack down.

2 Click Tack.

All selected objects are tacked down on the image.

To tack down all objects on the image:

1. In the Graphics menu, Enable All.
2. Click Tack.
   All objects are tacked down on the image.

Cutting and Pasting in Paint

Accessing the CutOut Menu

To access the CutOut menu, click the CutOut button in the Paint menu. Some options may be hidden if the colour palette is open. Swipe the cursor across the bar at the bottom of the screen to hide the colour palette.

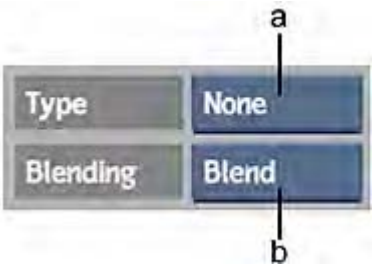


Creating a Cutout

Creating a cutout is similar to creating a graphic. In the CutOut menu, you use the same controls as you would in the Graphics menu to select, copy, hide, move, rotate, resize, delete, store, recall, push, and tack cutouts.

To create a cutout:

1. Click CutOut.
2. Click Add.
   The CutOut Shadow options appear.



(a) Shadow Type box (b) Blending Mode box

3. To limit the area of the cutout with a matte, enable Use in the Matte controls.
4. Select an option from the Shadow Type box.

The Shadow Type box determines how the cutout will be pasted onto the result clip. See Pasting Cutouts (page 1106).

5 Click Attributes.

6 To create a cutout that spans the entire clip, enable Sequence.

7 Click one of the four Cutout commands. See Using the Cutout Commands (page 1103).

NOTE To use the GeoCut option, select all the objects you want to use in the cutout in the Graphics menu before clicking the GeoCut button.

8 Draw the cutout on the image.

9 Set the resolution to be used when updating the object.

The object appears at full resolution when it is drawn. See Setting the Object Resolution and Display (page 1085).

10 Set the cutout's position using the transformation box or controls.

11 Set the cutout's colour using the Current Colour pot.

12 Click Tack to permanently place the object on the result clip.

Once the object is tacked down, it cannot be moved, rotated, deleted, resized, or copied.

Keep Cut On

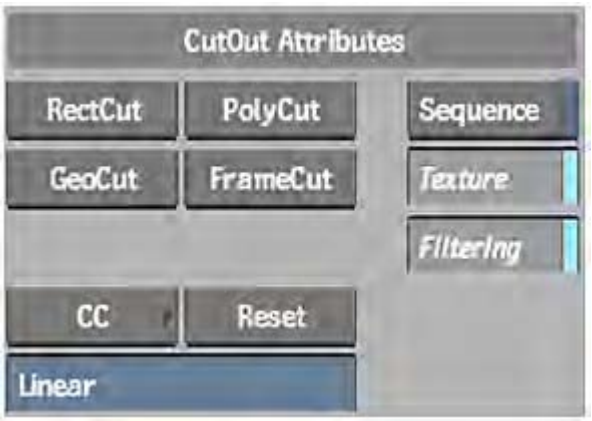
Once you add a cutout to the image, the default setting automatically switches from Cut mode to Edit mode. To prevent the switch to Edit mode, enable Keep Cut On in the Setup menu.



(a) Keep Cut On button

Using the Cutout Commands

Four commands are used to create cutouts: FrameCut, RectCut, PolyCut, and GeoCut. FrameCut, RectCut, and PolyCut create cutouts using shapes you define in the CutOut menu. Geocut creates cutouts using objects you select in the Graphics menu.



The FrameCut Command

Use the FrameCut command to make a cutout of the entire frame.

To create a cutout of an entire frame:

1. Click Add in the CutOut menu.
2. Click FrameCut.
   The entire frame is cut out and pasted onto the image.

The RectCut Command

Use the RectCut command to cut and paste a single rectangular area of the image.

To use the RectCut command:

1. Click Add in the CutOut menu.
2. Click RectCut.
3. Draw a rectangle over the area of the image that you want to cut out.
   The image area defined by the rectangle is copied and pasted onto the image.

The PolyCut Command

Use the PolyCut command to cut and paste a single polygonal area of the image.

To use the PolyCut command:

1. Click Add in the CutOut menu.
2. Click PolyCut.
3. Draw a polygon over the area of the image that you want to cut out.
   The image area defined by the polygon is copied and pasted onto the image.

The GeoCut Command

Use the GeoCut command to create a cutout with objects you selected in the Graphics menu.

To create a cutout using GeoCut:

1. Click Graphics in the Paint menu.
2. Create any number of objects.
3. Select all the objects you want to use to make the cutout.
4. Set the opacity of the object(s) using the Brush Opacity field.

   The opacity of the object(s) used in the cutout determines the transparency of the cutout. Use objects with a low opacity value to create an opaque cutout. Use objects with a high opacity value to create a transparent cutout.
5. Click CutOut in the Paint menu.
6. Click GeoCut.

   The image area defined by the selected objects is copied and pasted onto the image.

Displaying Graphics and Cutouts

To display both the cutouts and the graphics objects in the CutOut menu, enable Show Both in the Setup menu. Although the graphics objects appear, they cannot be edited. Enabling Show Both also displays both the graphics objects and the cutouts in the Graphics menu. Although the cutouts appear, they cannot be edited.



(a) Show Both button

Increasing Editing Speed

To streamline cut and paste functionality, use Graphic Regen and Texture Cut/Paste in the Setup menu, and Texture in the CutOut menu.

The Graphic Regen Button

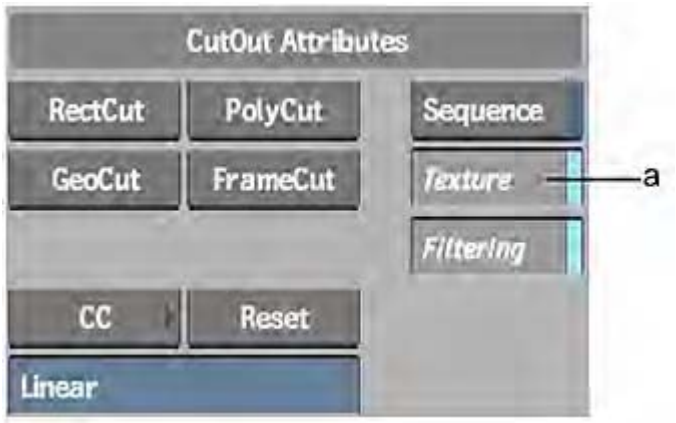
Disable Graphic Regen (regeneration) in the Setup menu to ensure the cutout always appears at the display resolution.



(a) Texture Cut/Paste button (b) Graphic Regen button

Texture Cut/Paste and Texture Button

Enable Texture Cut/Paste in the Setup menu or Texture in the CutOut menu to speed up cut and paste operations. These preferences are designed for platforms that support fast texture mapping.



(a) Texture button in CutOut menu

The Filtering button can be used when Texture is enabled. Use Filtering to avoid image degradation as a result of recursive cutting and pasting. When Filtering is enabled, the image should not degrade.

Pasting Cutouts

Four options in the Shadow Type box are available for pasting a cutout onto the image.

Select:	To:
Emboss	Create an embossed cutout. See The Emboss Option (page 1107).
Extrude	Create cutout extrusions. See The Extrude Option (page 1108).
None	Use the blend functions available for pasting cutouts. See The Blend Options (page 1108).

Select:	To:
Shadow	Add a drop shadow to the cutout. See The Drop Shadow Option (page 1109).

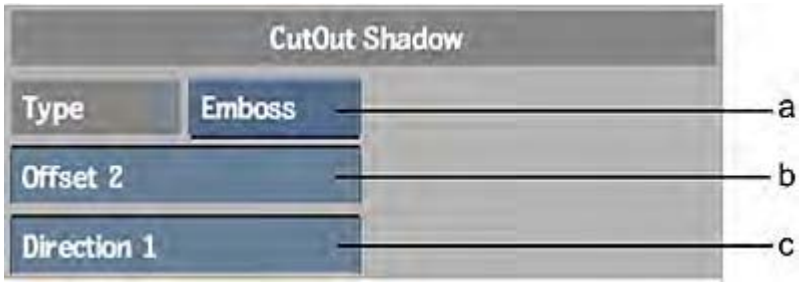
The Emboss Option

The Emboss option makes the cutout appear etched into the image. Shadows and highlights are added to simulate the effect of light shining on a raised object. Three layers are used to produce this effect:

- The top layer is a copy of the cutout in which the luminance values are increased by 50%. The top layer adds the highlights to the cutout.
- The middle layer is the cutout.
- The bottom layer is a copy of the cutout in which the luminance values are decreased by 50%. The bottom layer adds the shadow to the cutout.

Offset Field

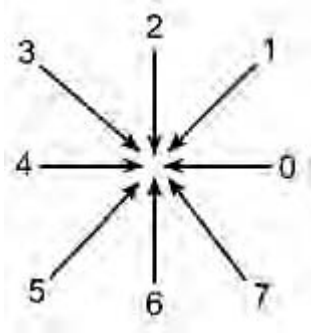
Use the Offset field to specify the offset value in pixels for the three layers. The best results are obtained with values of 1 or 2.



(a) Paste Mode box (b) Offset field (c) Direction field

Direction Field

Use the Direction field to specify the angle of the simulated incident light. The values range from 0 to 7 and specify the following angles of incident light.



The Extrude Option

The Extrude option makes the cutout appear to pop out of the image by redrawing the cutout one layer on top of another. Each layer in the stack is offset slightly from the one below so only the edges of the layer are visible.



Editing the Layers

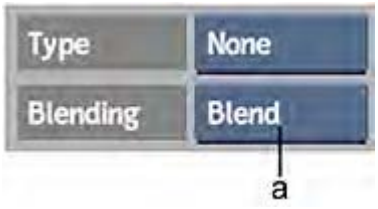
When you create a cutout using the Extrude option, two layers are created to produce the effect. There are vertices at the centre of each layer for manipulating the layer. You can change the depth and direction of the extrusion by moving either layer.

Changing the Colour of the Layers

Enable Use Colour to apply the current colour to the extrusion layers. The cutout image is used for the top layer only. When Use Colour is enabled, use the Brush Opacity field to set the transparency of the extrusion layers.

The Blend Options

The Blend options are similar to the commands in the Processing menu. They use the RGB values of the cutout and those of the result image. Select a blend option from the Blend Mode box.



(a) Blend Mode box

Blend Pastes the cutout onto the result image.

Add Adds the RGB values of the corresponding pixels in the cutout and the result image. Values greater than 255 are clipped.

Subtract Subtracts the RGB values of pixels in the image from those of the cutout. Values less than 0 are clipped.

Black Pastes a black object in the shape of the cutout on the result image.

White Pastes a white object in the shape of the cutout on the result image.

Colour Pastes a coloured object in the shape of the cutout on the result image. The current colour is used for the object.

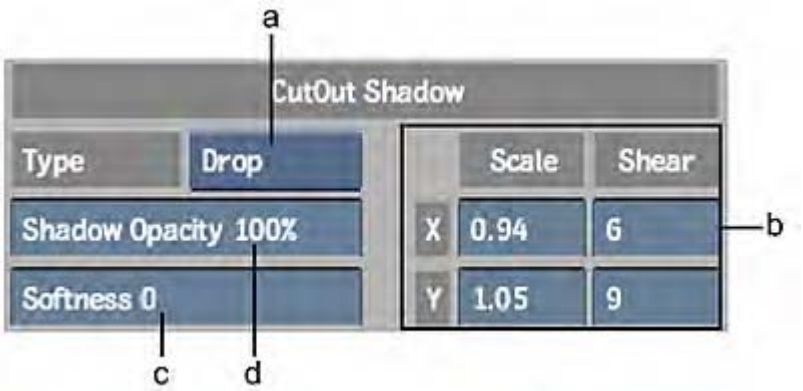
NAdd Min Compares the brightness values of corresponding pixels in the cutout and the result image, and uses the pixel with the smaller value in the tacked down cutout.

NAdd Max Compares the brightness values of corresponding pixels in the cutout and the result image, and uses the pixel with the greater value in the tacked down cutout.

Multiply Multiplies the RGB values of corresponding pixels in the cutout and the result image. The value is normalized by dividing the result by 255.

The Drop Shadow Option

You can add a drop shadow to a cutout by selecting the Drop option from the Paste Mode box. The drop shadow's colour is taken from the Current Colour pot.



(a) Paste Mode box (b) Scale and Shear fields (c) Softness field (d) Shadow Opacity field

Shadow Opacity field Adjusts the transparency of the drop shadow. Decrease the value to increase the transparency.

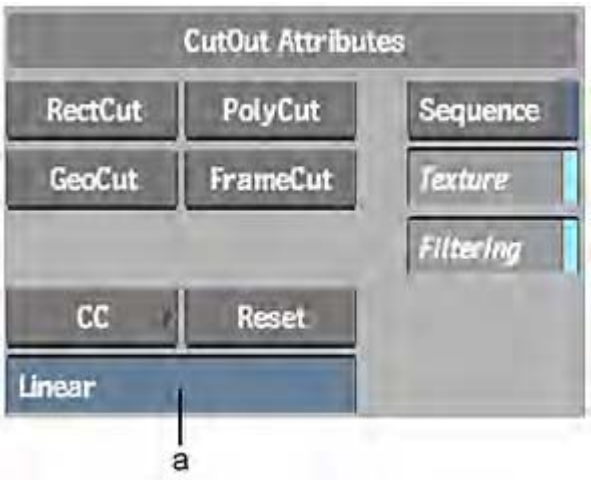
Softness field Adjusts the edge softness of the drop shadow. Increase the value to increase the softness of the shadow.

Scale and Shear fields Shears and scales the drop shadow along the X- or Y-axis. These fields can be used to add lighting perspective to the image.

Colour Correcting Cutouts

Apply colour correction to cutouts using the Colour Correction field and CC in the CutOut menu. Click the Colour Correction field to load a setup directly from the colour correction library. Click CC to load the cutout into the Colour Corrector.

The default colour correction setup is Linear. When you load a different setup, the name of the setup appears in the Colour Correction field. To reset the colour correction setup, click Reset.



(a) Colour Correction field

Animating Cutouts

Use the Channel Editor to animate cutouts. To display the Channel Editor, click Animation and swipe the cursor across the bottom of the screen.

The following parameters can be animated for every cutout:

- Position, rotation, and shearing
- Brush attributes
- Colour (red, green, and blue channels)
- Emboss options
- Shadow options

Loading and Saving Cutouts

Use the Cutout Library to load and save cutouts.

To save or load a cutout:

1. In the Paint menu, click Save or Load.
2. Select CutOut from the Save or Load option box.
3. Select a file or type in a name for the cutout.

   NOTE If you are using Save and have typed in a name for the file, you must press `Enter` to finish saving the file.

Preserving Paint Cutouts

Paint cutouts are saved in the Lost_+_Found library on the Paint reel rather than in memory in order to conserve memory. During a session, you must not delete the Paint reel or its cutouts because they will be used when you return to Paint. However, you can safely delete the cutouts once you exit and restart the application because the Paint reel in the Lost_+_Found library is not searched in subsequent sessions.

About Paint Setups

A setup is a file that contains a record of all changes you make to a clip in a particular tool. This record includes references to clips used. Setups let you save your work separately from the clips, so you can load and work on them any time, or apply the setup to other clips. Both the procedure and the interface involved in saving Paint setups and preferences differ slightly from those involved in saving setups in most other tools

NOTE Custom brushes, brush sets, colour pots, and palette setups that you create in Paint are stored in your user directory. These resources are loaded when you specify a user in the Project Management menu.

Saving Setups

You can save pictures, preferences, mattes, palettes, brush groups, brushes, and stamps. You can also save cutouts, cut sequences, recorded strokes in AutoPaint, as well as geometry created in the Graphics menu.

To save a setup:

1. In the Paint menu, click Save.
2. Select the item you want to save from the option box.

Select:	To:
Picture	Save an image created in Paint.
Matte	Save the matte.
Palette	Save the current colour palette.
BrushSet	Save the current set of brushes.
Brush	Save the brush that is currently active.
Stamp	Save the Stamp currently in the Stamp window.
Geometry	Save the geometry created in the Graphics menu. This also includes all cutouts.
CutOut	Save the cutouts created in the CutOut menu.
CutSequence	Save the cutout sequence created in the CutOut menu.
AutoPaint	Save the last series of recorded strokes.
Preferences	Save current preferences.
Defaults	Save current default values.

   The file browser appears, listing any existing setups for that item.
3. Type the name of the setup in the Save field.

4 Press Save (Enter) to save the setup.

Cropping a Setup

Use the crop box to limit the area of the picture, matte, cutout, or paint graphics to be saved in the setups directory.

To save a cropped matte, image, or cutout:

1 In the Paint menu, click Save.



2 Click Picture, Matte, or CutOut to specify the element from the image you want to save.

3 Draw the crop box on the canvas by pressing and dragging the cursor diagonally across the screen. Alternatively, use the left, right, top, and bottom Crop fields to set the boundaries of the crop box.

NOTE Click Reset to reset the crop box values.

4 Click Save.

5 Select the item you are saving from the option box and name the setup in the keyboard display.

If you want to overwrite an existing setup, select the filename from the file browser.

6 Press Save (Enter) to save the setup.

Loading Setups

Use the Load button to load any type of Paint setup.

To load a setup:

1 Click Load.

The Load Paint menu appears.

2 Select the type of setup you want to load from the Load option box.

A list of existing setups appears in the file browser.

3 Click the title or proxy of the setup you want to load.

Deleting Setups

Use the Delete button to remove setups from the setup directory.

To delete a setup:

1. Click Load.
   The Load Paint menu appears.
2. Enable Delete.
3. Select the type of item you want to remove from the Load option box.
   A list of existing setups appears in the file browser.
4. Click the title or proxy of the setup you want to remove and click Confirm.

Creating and Saving Brush Setups

Within a given brush set, there are five individual brushes. You can create and save five brush setups using the buttons numbered 1 to 5 in the Paint menu. Each brush setup saves the brush type, attribute values and modes, Special Effects medium, filter, fill parameters, and colour correction setups.



(a) Brush Setup buttons

To create and save a brush setup:

1. In the Paint menu, click one of the five Brush Setup buttons.
2. Enable Edit next to the Brush Setup buttons.
3. Set the brush attributes. See Brush Attributes (page 1040).
4. Disable Edit to save the changes to the selected brush setup.

NOTE To save a set of five brush setups, use BrushSet in the Save menu.

Copying Brush Setups

You can copy brush setups using the setup buttons.

To copy brush setups:

1. Select a setup using the Brush Setup buttons and enable Edit.
2. Press `Ctrl` and click another Brush Setup button to copy the setup.
3. Disable Edit to save the setup.

Creating a Custom Brush

With Paint, you can create your own custom brushes for specialized applications. You can modify an existing brush, draw your own brush shape, or grab a part of the image to use as a custom brush.

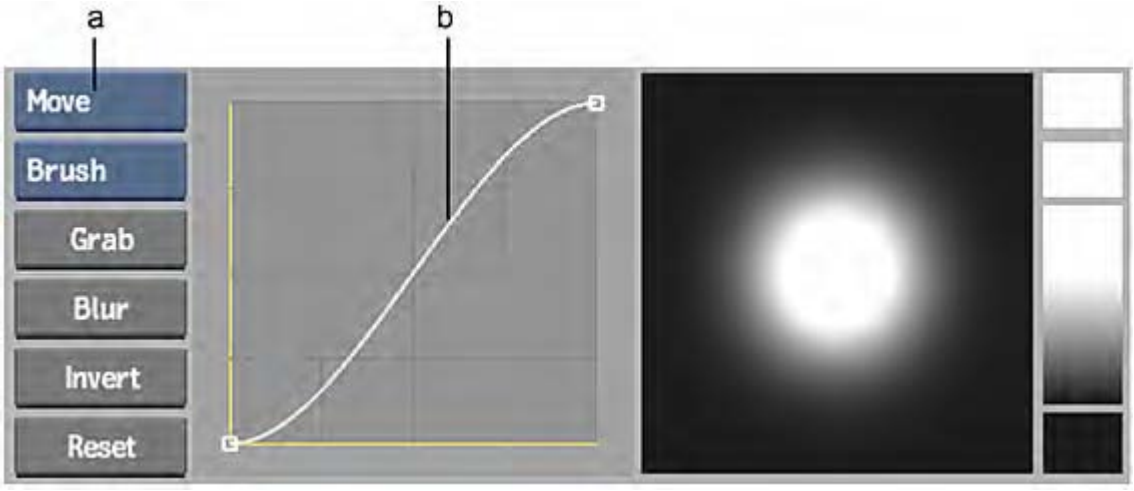
Use the Custom Brush menu to create your own brush. To display the Custom Brush menu, double-click one of the brush icons in the Brushes window. An enlarged view of the selected brush appears in the editing panel.

There are three ways to create a custom brush:

- Using the brush profile
- Drawing a brush shape in the editing panel
- Grabbing an area of the image

Changing the Brush Profile

The brush profile describes the size and edge softness of the custom brush. The default S-curve defines a circular airbrush. You can change the curve by manipulating the two points that define the curve, or you can add points to the curve.



(a) Profile box (b) Brush profile

The profile does not affect the brush in the editing panel until you modify the curve or click the Update button.

Changing the Curve

By default, there are two points that define the brush profile. For the following table and examples, these points will be named A and B as shown in the preceding illustration. To move the points that define the curve, select the Move option from the Profile box.

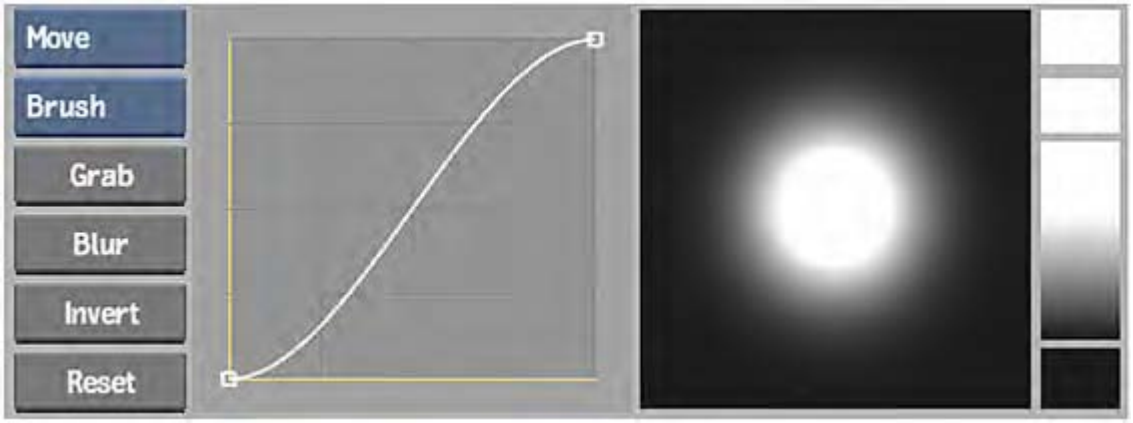
Move:	To:
Point A up	Lighten the brush.
Point B down	Darken the brush.

Move:	To:
Point A right	Harden the outer edge of the brush.
Point B left	Soften the middle of the brush.

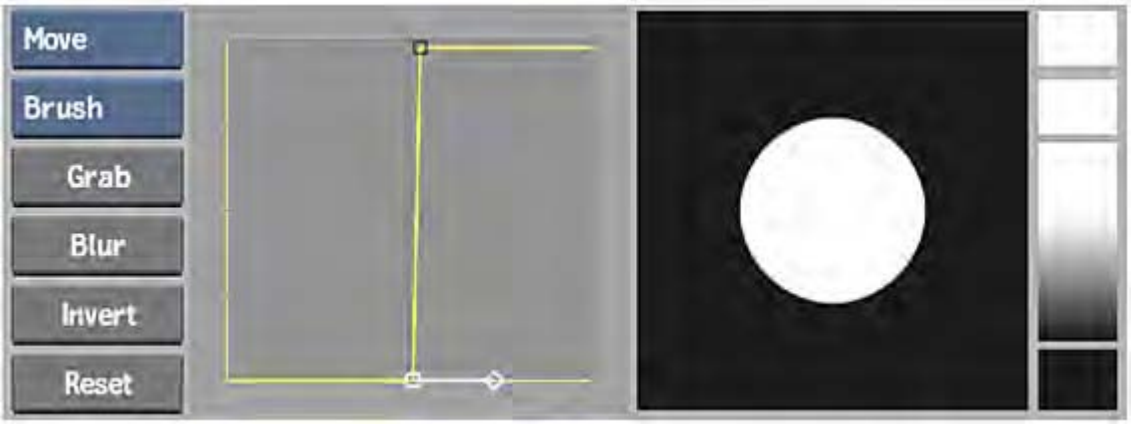
To move the points only on the X-axis, select the Xscale option from the Profile box. To move the points only on the Y-axis, select the Yscale option. To break a point and manipulate its tangent, select the Break option and click on a point.

Adding Points to the Curve

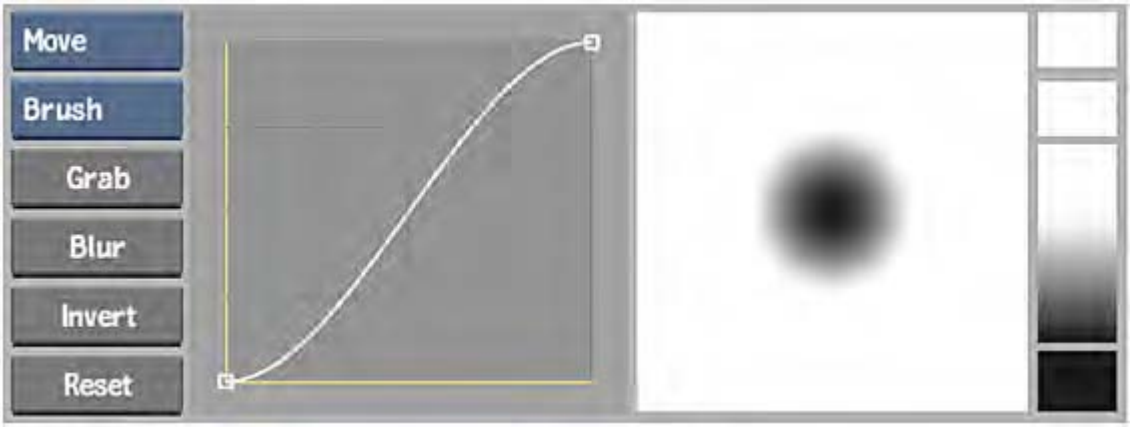
To add points to the curve, select the Add option from the Profile box and click on the curve. To delete points from the curve, select the Delete option and click on one of the points on the curve.



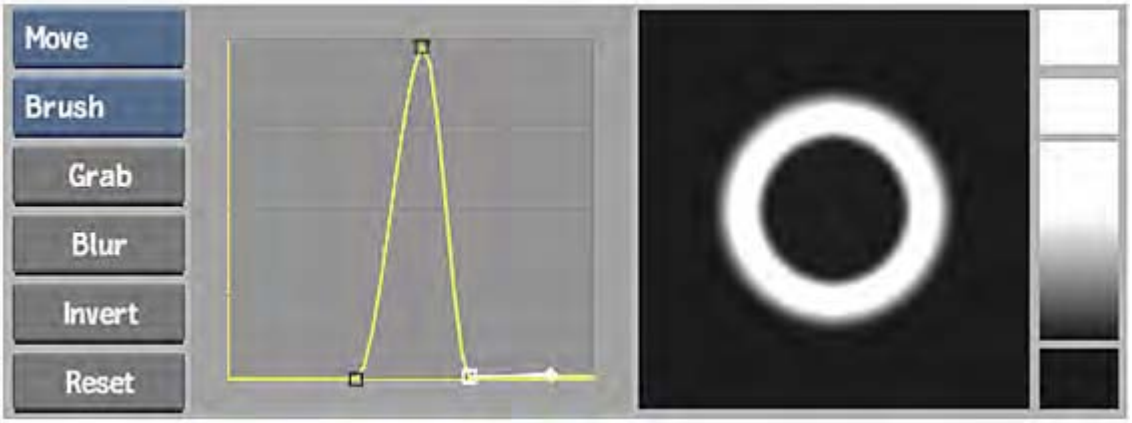
Default S-shaped curve



Hard edge curve



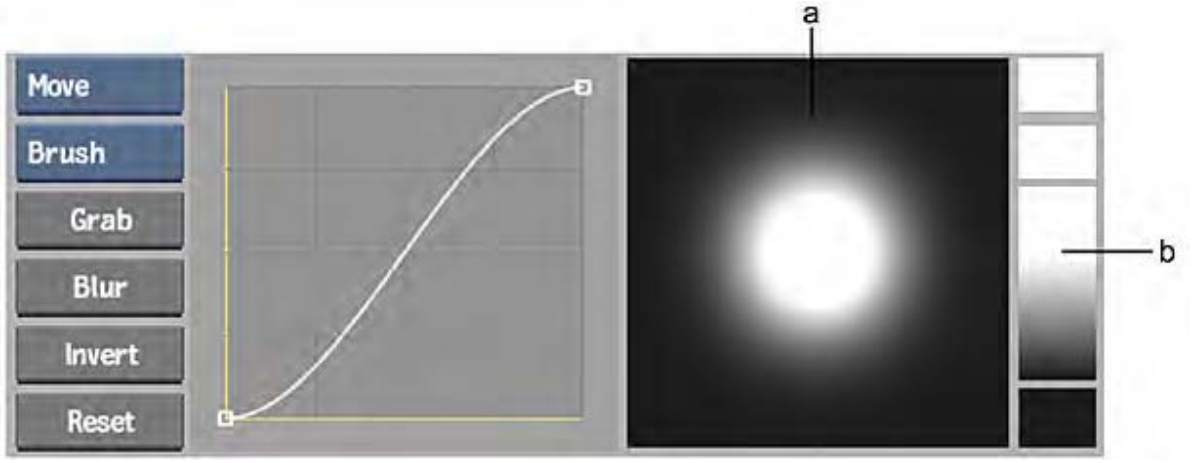
Invert brush curve



Ring-shaped brush curve

Drawing and Updating a Brush Shape

You can draw a new brush shape or update the current brush shape in the editing panel. The new shape is drawn using the brush that you selected to open the Custom Brush menu.



(a) Editing panel (b) Gradient bar

NOTE You cannot use the Undo command when drawing or updating a custom brush.

To draw a brush shape:

1. To create a new shape, erase the existing brush shape by painting over the editing panel.
2. Draw the new shape.
3. To invert the brush shape, click Invert.
4. To blur the brush shape, click Blur. Each time you click Blur, a 3 x 3 filter is applied to the brush image.

Selecting an Area of the Image

You can select an area of the image to use as the custom brush. The brush created from the selected area will be a square monochrome brush.

To grab an area of the image:

1. Click Grab in the Custom Brush menu.
2. Click on the image and drag across the screen to define the selection box.

   To select a square area, press `P` as you sweep out the selection box.

   The selected area appears as a monochrome image in the editing panel. If the area that you select is not square, then the selected area is resized to fit in the editing panel.

Updating an Existing Brush

To update an existing brush, click the Update button. The changes are applied to the brush icon that you selected to open the Custom Brush menu.

Creating a New Brush

To create a new brush, click the New button. This creates an icon for the new brush shape and adds it to the Brushes window.

Saving the Custom Brush

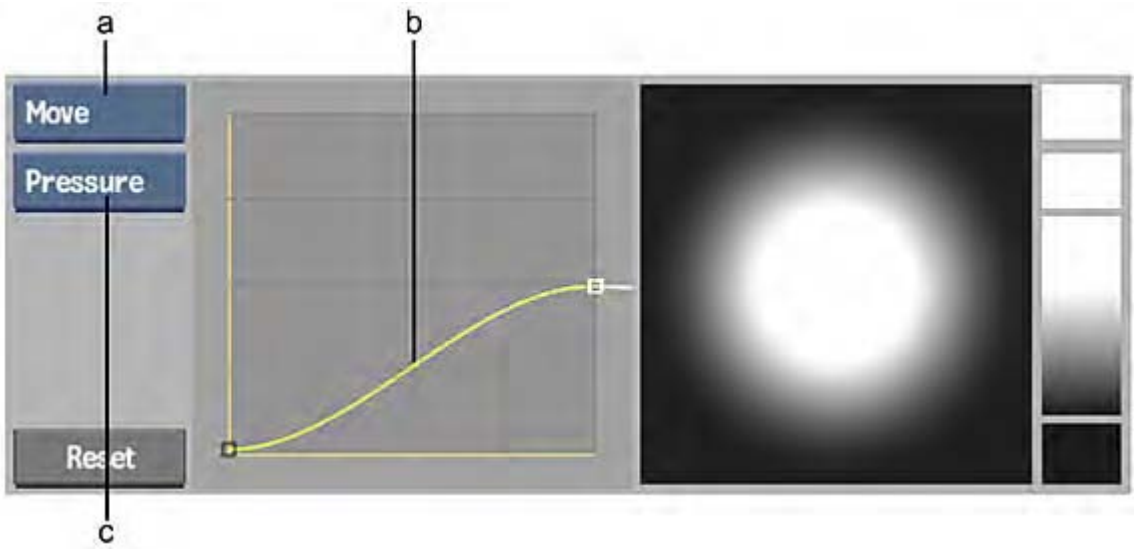
Custom brushes are not saved automatically when you exit Paint. To save the updated or new brush for use in another work session, click the Save button. This opens the Brush setups directory. Use the keyboard to enter the name of the brush, and click the Enter button. The new brush is saved in the Brush setups directory.

Loading a Custom Brush

Click the Load button to load a custom brush from the Brush setups directory.

Customizing the Pressure Profile

Adjust the pressure profile to change the amount of pressure needed to paint with a given brush. The slope of the curve indicates how quickly paint is applied as you press on the tablet. If the curve is steep, a small amount of pressure applies full paint to the canvas. If the curve is soft and rounded, you must press harder and longer to apply full paint.



(a) Profile box (b) Pressure curve (c) Brush/Pressure box

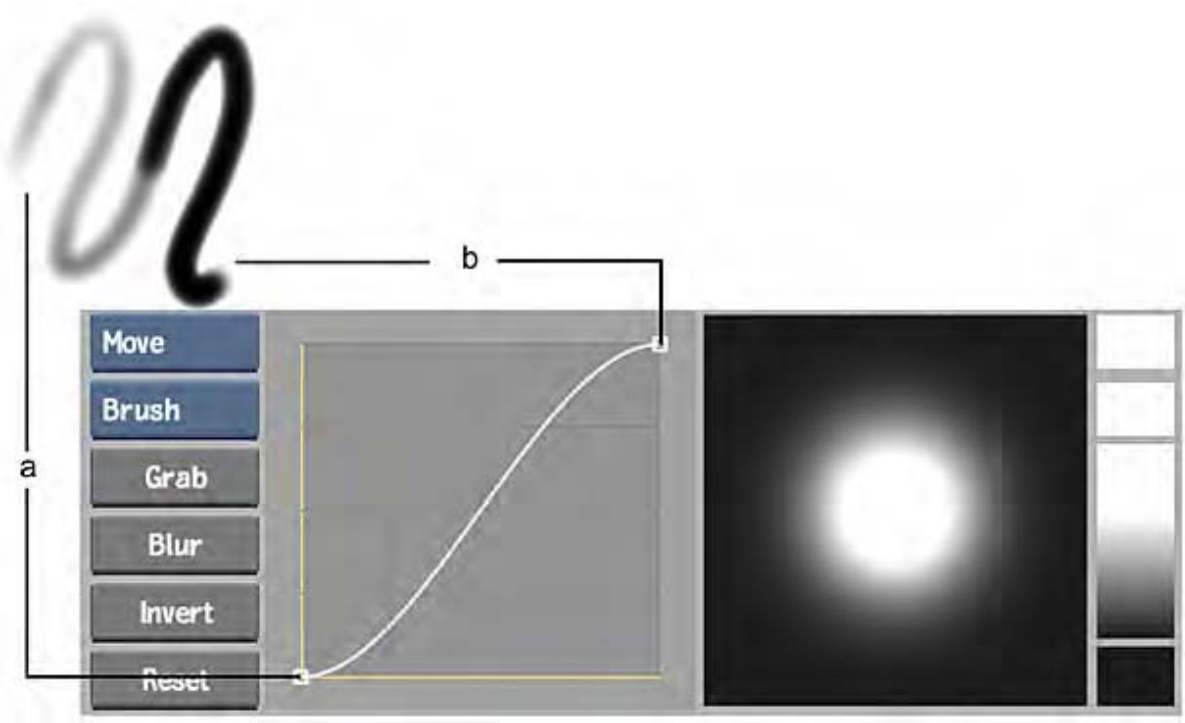
You use the options in the Profile box to change and add points to the pressure curve. Click Load to load a Pressure profile. Click Save to save a custom pressure profile.

To edit the pressure profile:

1. Toggle the Brush/Pressure box to Pressure.
2. Double-click a brush icon in the Brushes window to display the Custom Brush menu.
3. Click the Brush/Pressure box to display the pressure profile.
4. Select Move from the Edit Mode box.
5. Press the left point (A) and drag it to its new position.
   This changes the amount of paint the brush applies when you press lightly on the tablet.
6. Press the right point (B) and drag it to its new position.
   This changes the amount of paint the brush applies when you press down firmly on the tablet.
7. Click Update to update the pressure profile for the brush.
   The pressure settings will be used when you paint with the brush during the current work session.

Default Pressure Profile

Pressure is mapped along the horizontal axis (X-axis) of the curve, and the amount of paint applied is mapped along the vertical axis (Y-axis). If you are using the default S-shaped curve and you press lightly on the pen, very little paint is applied. As you press harder, more paint is applied.



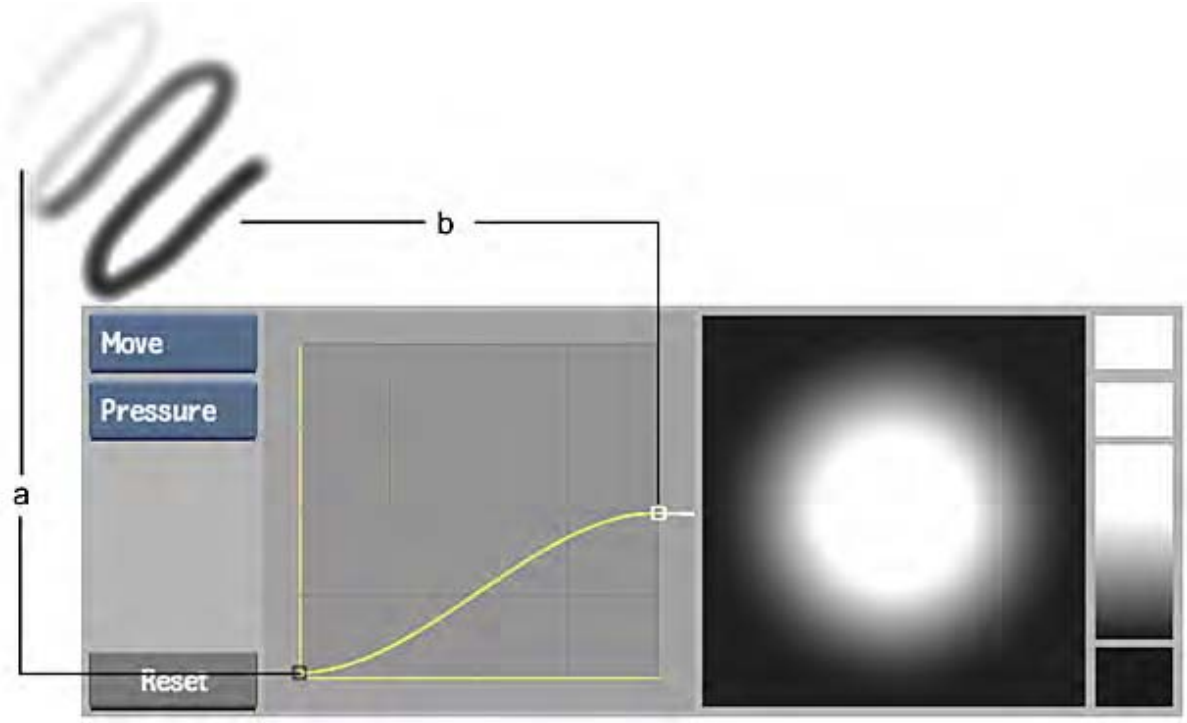
(a) Point A: Low paint at low pressure (b) Point B: Full paint at full pressure

Maximum Pressure Curve

If you move point A in the previous figure to the maximum position on the Y-axis, the curve results in full paint applied all the time, no matter how much or how little pressure you use.

Soft Airbrush

If you move Point B in the previous figure to a point midway on the vertical axis, the curve gives you a soft airbrush, even at full pressure.



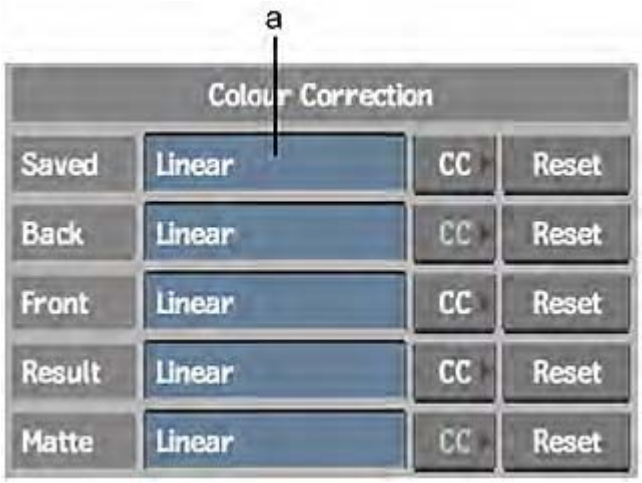
(a) Point A: Low paint at low pressure (b) Point B: Soft paint at full pressure

Using Colour Correction Setups

Use the Colour Correction Setup menu to apply colour correction setups to the clip loaded into Paint. You can bring the clip into the Colour Corrector or you can load a setup from the Colour Correction setups directory into Paint.

To load a clip into the Colour Corrector:

1 In the Paint menu, click Setup.
The Colour Correction menu appears.



(a) Setup Name field

2 Click CC next to the clip you want to load into the Colour Corrector.

The Colour Corrector appears.

3 Use the Colour Corrector as described in About Colour Correcting (page 933).

4 Save the setup to the Colour Correction setups directory.

5 Click Exit.

The name of the Colour Correction setup appears in the Setup Name field beside the clip label for the selected clip.

To load a setup directly from the Colour Correction setups directory:

1 Click the Setup Name field for the clip you want to colour correct.

The file browser appears, displaying the contents of the Colour Correction setups directory.

2 Select the Colour Correction setup you want to use.

To reset the colour correction for a clip:

1 Click Reset beside the clip you want to reset.

2 Click Confirm.

Text and Titling

23

Text is comprised of layers, paragraphs, and characters. You can make text spin, dance, and change colour over time. You can also create effects such as a text roll of credits, text crawls, bumpers, and text that moves on a motion path. You can use logo images in a text roll so that the logo of a sponsor appears in the credits list.

You use the Text Tool to add text and text effects to a clip. Once text effects are applied to a clip, you can then use the clip in Action or Batch or Batch FX depending on the final effect you want to achieve. Use Action to create 3D text deformation effects, including sliding, magnifying, and oscillating effects. For example, to create an effect of text rolling on a cylinder in a composite, you first render a text roll in Text and then, in Action, project it as a texture onto a cylinder. See Diffuse Mapping (page 615).

Text Workflow

The following table shows the recommended workflow for creating text in the Text tool.

Step:	Refer to:
1. Determine whether the text will appear over a clip, a black background, or a coloured frame.	Accessing the Text Tool from Tools (page 1124).
2. Set text processing options.	Setup and Processing Options (page 1126).
3. Determine whether you want to create a text roll, a text crawl, or text on a path or standard layer.	Creating Text Rolls and Text Crawls (page 1141).
4. Create a layer.	Adding Text to Clips (page 1130).
5. Set the position, size, and offset of the layer.	Modifying Layer and Character Properties (page 1135).
6. Enter text or load an existing text file.	Entering Text (page 1131) and Loading Text Files (page 1129).
7. Set text attributes.	Creating Text Effects (page 1135).
8. Preview the result and process the final clip.	Setup and Processing Options (page 1126).

Accessing the Text Tool

The Text Tool can be accessed from the following locations:

- The Timeline, using a Text timeline effect. (page 1124)
- Accessing the Text Tool from Tools (page 1124)
- Accessing the Text Tool from Batch (page 1125).
- Batch FX. (page 1125)

Accessing the Text Tool from the Timeline

To access the Text Tool from the Timeline:

1. Select Timeline.
2. Right-click on the segment that will be colour corrected.
3. Select Add Effect.
   The Video Effects ribbon appears.
4. Enable TX (Text).
5. Click Enter Editor...
   You are in the Text Tool.

Accessing the Text Tool from Tools

When accessing the Text Tool, you first select whether to add text to a source clip, a coloured frame, or a black background. The first time you open the Text Tool, the frame contains a clip image, but no text. The next time you open it, the text and settings from the previous Text session appear.

To access the Text Tool from Tools:

1. Select **Tools ➤ Composite ➤ Text**.
   If this is the first time accessing the Text Tool, the cursor changes to Pick Back.
   To change to None, from the Input Mode box, select None.

   NOTE You can also `Alt`-click the Text button to enable the Text button and automatically set the Input Mode box to None.

2. Do one of the following:
   - If you selected None, choose a resolution, width, height, pixel aspect ratio, bit depth, scan mode, and frame depth to apply to the background in the Text tool.



- If you selected Back, select the clip to which you want to add text.

  TIP If you want to change the Input mode to None once in the Text Tool, you can do so from the Text Setup menu. See Setup and Processing Options (page 1126).

The cursor changes to Render Here.

3. Click on any free (or a grey) area on the workspace.
   You are now in Text.
4. If text from a previous Text session appears, click Reset All.
   The Text Tool is cleared and settings are reset to their default settings.
5. To delete any layers from a previous session, click Delete All.

Accessing the Text Tool from Batch

To access the Text Tool from Batch:

1. Select Batch.
2. In the Batch node bin, double-click the Text node or drag the node to the Batch schematic.
   Make sure you connect to other nodes or clips in the process tree.
3. Click the Text node.
4. Click the Edit button.
   The Text node editor appears.

Accessing the Text Tool through Batch FX

To access Text through Batch FX:

1. Select Timeline.
2. Select the segment.
3. Click the FX button.
4. Click the Create Batch FX button.
5. Drag a Text node into the schematic.
   The Text node is now in the schematic.
6. Double-click the Text node.
7. Click the Edit button.

You are in the Text node editor.

Setup and Processing Options

Before processing a text clip and rendering text effects, you must set crop settings for the text layer, blur settings, text animation, softness, and whether or not you want to use prerendering. The prerendering feature accelerates the rendering process for a text layer.

Set text processing options in the Text Setup menu. To access the Setup menu, click Setup. The Setup controls are described as follows.



Software anti-aliasing values can range from 1 to 64. A higher value increases the sampling rate, but also increases processing time. An anti-aliasing of 1 means no anti-aliasing is performed and the result is a hard edge.

TIP A higher anti-aliasing value is usually needed with outlined text.

Softness box Defines the softness of the anti-aliasing samples.

Blur box Sets Gaussian Blur or Box Blur. Gaussian Blur uses subpixel resolution and creates a subtle effect with the Blur Shadow attribute. Gaussian has rounded, smoother edges and is better for animation; however, processing time is increased.

Box Blur has rectangular, rougher edges, but is more economical with processing time, especially if you are working on a rough draft.

Processing Text Clips

You can process clips in progressive or interlaced rendering. Interlaced rendering produces a better result, especially if you use keyframes that are far apart, but it takes longer to process. You should preview the quality of the image and the text before you process the final clip.

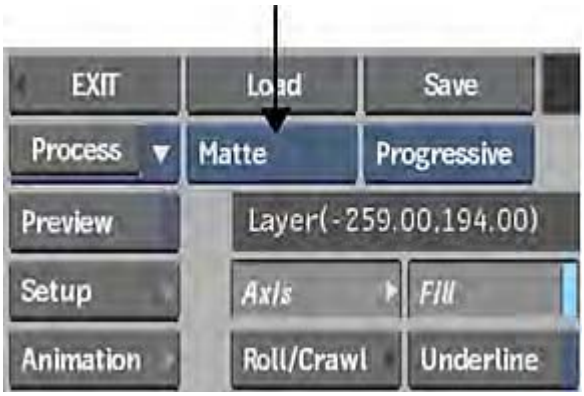
To process a clip with text:

1. From the Rendering box, select Interlaced, Progressive, or As Back (to render using the mode of the back clip).



2 From the Process Mode box, select which clip gets processed.

Select:	To:
RGB	Process only the text on the back layer.
Matte	Process only the matte of the text. The text fill transparency attribute is transferred to the matte. By generating a matte, you can easily composite the text onto another clip.
Both	Process the text on the back layer and the matte of the text. The fill transparency is removed from the text and transferred to the matte.



3 Click Setup.

4 From the Setup menu Rendering section, select the sampling level from the Auto-Softness box.

5 Set other options as required.

6 Position the clip at the first frame or the specific frame where you want to start processing.

7 Click Preview.

TIP You can zoom (`Ctrl+spacebar`) or pan (`Ctrl+Shift+spacebar`) the image window while in Preview mode. To disable Preview, make a modification to the text or processing options, or click anywhere in the image window.

8 Make modifications to the text and processing options and preview until you are satisfied with the result.

9 When you are ready to process the text clip, click Render.

The clip is processed from the currently displayed frame until the end of the clip.

Processing Considerations

Processing is contingent upon PreRender, Blur, and Animation settings. Depending on which options you choose, you can increase the processing speed:

- When more than one non-static layer has blur shadows, Global Blur is faster, although all shadows are blurred with the same unique colour and appear under all other layers.
- When there is a combination of animated and non-animated layers, speed depends on the blur shadows. If animated text has blurred shadows, Global Blur is faster. If you want a static blurred shadow, and a non-blurred animated layer, Layer Blur is faster.
- If you have static layers only (no animation), speed depends on the number of layers and whether PreRender can be used. Global Blur is faster since it uses PreRender. The processing speed of Layer Blur is the same with one or two layers. However, it becomes progressively slower with a higher number of layers and if the layers use blur shadows.

Saving and Loading Files

You can save, load, and import text files, text setups, and images for specific use with the Text tool. You can save text, text effects, text preferences, layers and new default text settings.

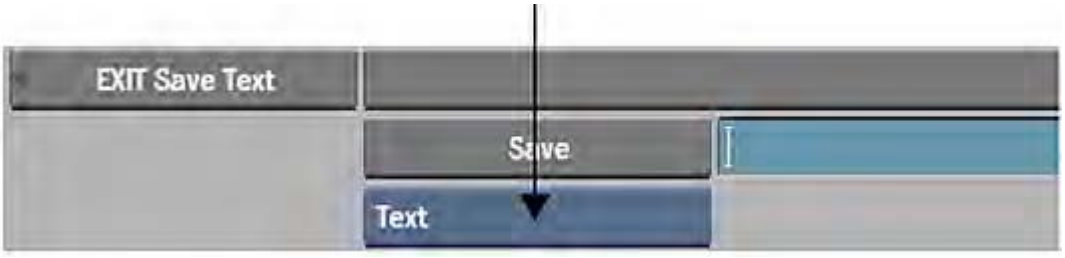
You can load text files from third-party word processing packages, existing Text tool setups, and logos. Loading existing text files is particularly convenient if you are creating a long text roll of credits as it saves time that you would otherwise spend typing and spell checking.

Saving Text Files

When saving files, you can save text setups, preferences, and default settings.

To save a file:

1. Click Save.
2. In the Save menu, select an option from the Save box.



Select:	To:
Defaults	Override the default menu options in the Setup menu and replace them with your specifications.
Preferences	Save Setup menu specifications only and omit text and text attributes.
Selected Path	Save any selected text paths.

Select:	To:
Selected Logos	Save any selected logos.
Selected Layers	Save selected layers, their text, text attributes and Setup menu options.
Text	Save text files with text attributes and Setup menu options.

3 Navigate and choose the directory where you want to save the file.

4 Type a name for the file and press `Enter` or click the Save button.
The file is saved and you return to the Text menu.

Loading Text Files

You can load text files, preferences, and ASCII files. You can also reset the Text menu settings with its factory defaults.

To load a file:

1 Click Load.

2 In the Load menu, select an option from the Load box.



Select:	To load:
Preferences	Setup menu options that you have saved.
Path	Previously saved text path setups. Setups between text paths and garbage masks are interchangeable. Therefore, you can also load a garbage mask shape as a path. **NOTE** Preset paths are available in the *user/discreet/<product home>/path/default* directory.
Logo	Previously saved logos.
Text File	Text files. When you select Text File, you can select the encoding of the loaded file. This applies the correct conversion from the selected file's encoding to the application's internal encoding. For example, select ISO8859-1 to import Latin-1 ASCII files, or UCS-2 to import 2-byte Unicode encoded text files.

Select:	To load:
	The encoding options reflect the character encodings available with your system. The load mechanism supports the same encodings as the iconv utility. Optional iconv converters are available by installing national language support options.
Multiple Text Setup	Several layers or text files at a time with preferences. Text tool preferences that were saved with the selected file are loaded as well.
Text Setup	A layer or text file created in the Text tool. Text tool preferences that were saved with the selected file are loaded as well.
Factory Defaults	The original Text tool settings that were delivered with Flame Premium.

3 Choose a directory in either Proxies or Titles mode.

4 Select the file that you want to load in the file browser. If you selected Multiple Text Setup, `Ctrl`-click the files you want to load.

NOTE If you do not select Multiple Text Setup and you use the `Ctrl` key to load multiple text files, only the last selected file is loaded.

5 If you are loading a text file or text setup, you can click anywhere in the user interface to abort the load before it is completed.

6 To discard unwanted files, enable Remove.

7 If necessary, click Exit Load Text to exit the file browser.

Adding Text to Clips

You add text to a clip by first creating a layer for entering the text, setting the layer attributes, and then typing the text. You can create several layers of text that overlap each other.

Creating a Layer

Text is contained in a layer called a text layer. When you create a text layer, a coloured border called the *crop border* appears in the image window indicating the region the text will occupy.

To create a layer:

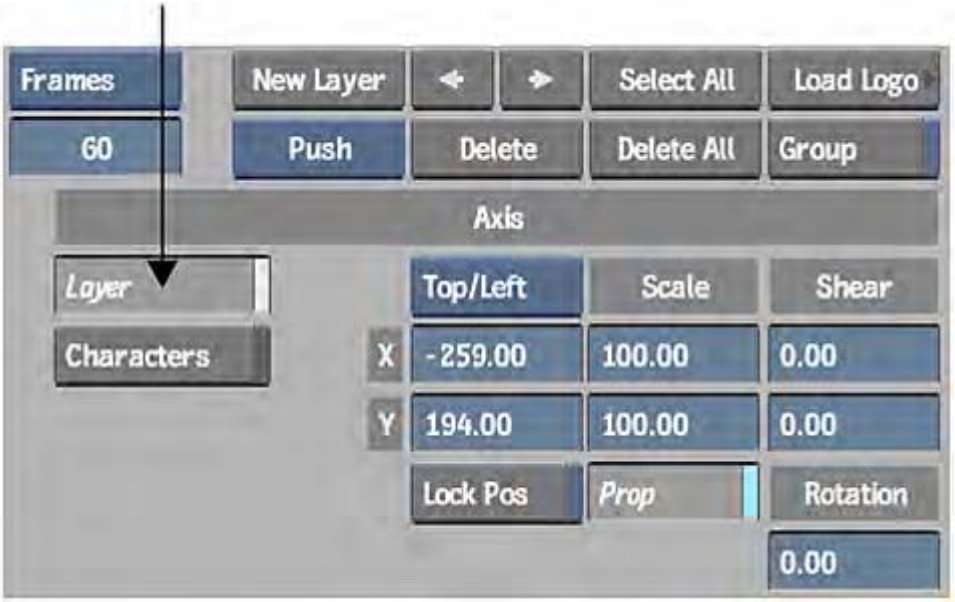
1 Click New Layer. If this is the first layer you are creating, you can click directly in the image window.



The text layer is positioned in the upper-left corner of the safe title area by default. The text in the layer is left justified, while the width is equal to the safe title overlay. You can modify the safe title overlay using the Grid and Guide controls. You can also modify the boundary colour. See Processing Text Clips (page 1126).

2 Click Axis to make sure you are in the Axis menu.

3 To set the text layer dimensions, enable Layer.



Each layer has its own parent axis for translating, rotating, resizing, and shearing. The axis for the layer can be in the upper-left or centre of the layer. See Creating Text Effects (page 1135).

Entering Text

The text you type may be as short as one letter or word or as long as several paragraphs. If the text is long or exists in a word processor, you can load the text file into the current layer.

You enter and edit characters much like in a word processor. You type characters using the keyboard and many known text editing conventions, such as `Shift+arrow keys` to select text lines. You can also click once to insert the cursor and select the adjacent character, double-click to select the whole word, and triple-click to select the entire line. Quadruple-click to select all characters on the layer and use the middle mouse button to paste selected text.

Use either the workstation or on-screen keyboard to enter characters in a layer. You can also paste the current text selection (from a shell or any other application) into any keyboard input field by pressing the middle mouse button or by using the pen button.

You can also input Asian characters using their corresponding ASCII codes with the numeric keypad.

To enter text in a layer:

1 From the Text Mode box, select Edit.



TIP Use Edit mode to type text strings, select text, edit text in layers, and modify text attributes. Press `Esc` to switch between Edit and Move modes. The message bar displays the current mode.

2 In the image window, create a new layer by doing one of the following:

- Click New Layer.
- Press `Alt+N`.
- Click directly in the image window (if this is the first layer you are creating).

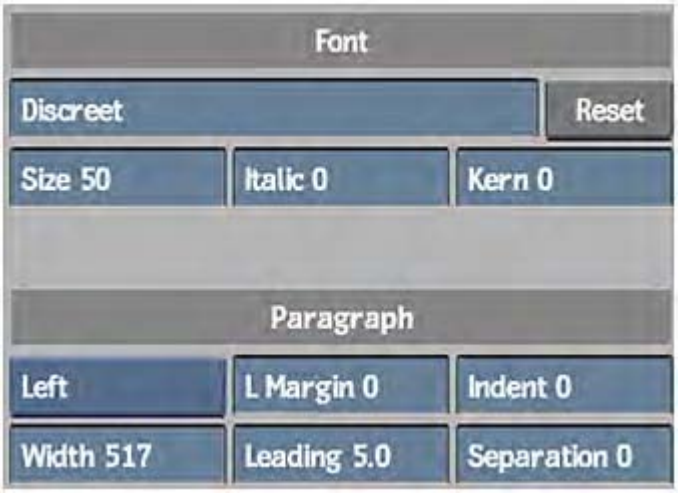
3 To begin a new paragraph within the layer, press `Enter`.

Notice that each paragraph has its own coloured boundary and a paragraph tag that is a coloured rectangle in the upper-left corner of the paragraph. In Edit mode, a selected paragraph's boundary is white.



(a) Blue indicates paragraphs in text layer (b) Red indicates currently selected paragraph, which has a paragraph boundary (c) Layer boundary is shown in red

4 In the Paragraph and Font menus, set properties.



Using the Text Keyboard

When you use the on-screen keyboard, consider the following:

- Special characters use the ISOLatin1 encoding vector. You specify the encoding vector in the *init.cfg* configuration file using the FontMapping keyword.
- The extended keyboard uses the Standard encoding vector.

- Asian character sets may not display all characters on the on-screen and extended keyboards. Use the numeric keypad or load an ASCII file that contains unavailable characters for the selected font. See Loading Text Files (page 1129).

To use the Text keyboard:

1. In the Text menu, click Keyboard.
2. On the on-screen keyboard that appears, enable Up ASCII to display extended characters contained in the selected font set.



NOTE Enabling Up ASCII also affects your workstation keyboard, so make sure you disable the button again before exiting the Text menu.

3. If the selected font has special symbol characters, enable Symbols to see them.
4. Type the characters in the text layer.

   The keyboard characters appear in the current font.

Word Processing Keyboard Shortcuts

Typical word processing tasks include text selection, cutting, copying, and pasting. The Text tool includes several character manipulation keyboard shortcuts for these tasks.

Press:	To:
`Alt+A`	Select all characters in the selected text layer.
`Alt+P`	Select all characters in the selected paragraph within a layer.
`Alt+Shift+A`	Select all characters in all layers. This operation is the same as using the Select All Layers button.
`Shift+up arrow`	Extend the selection one line up.
`Shift+down arrow`	Extend the selection one line down.
`Shift+left arrow`	Extend the selection one character to the left.

Press:	To:
Shift+right arrow	Extend the selection one character to the right.
Shift+Home	Extend the selection to the beginning of the current line.
Shift+End	Extend the selection to the end of the current line.
Ctrl+Shift+PgUp	Extend the character selection to the beginning of the text layer.
Ctrl+Shift+PgDn	Extend the character selection to the end of the text layer.

Creating Text Effects

Use the Attributes controls to change the appearance of individual characters, paragraphs, and layers. You set similar properties for both layers and characters. You can also load a logo in a paragraph and then format it using the Attributes menu. See Loading Logos (page 1139). If you create text formats that you plan to reuse, save them using the Styles menu. See Defining Styles (page 1150).



These text attributes have a cumulative effect on the selected characters. For example, if you enable the Fill and Outline buttons, the text appears as solid characters with a coloured outline.

Modifying Layer and Character Properties

Use the Layer controls in the Axis menu to rotate, translate, scale, or shear a selected layer and to apply an effect uniformly to all characters.



Use the Character controls in the Axis menu to view and position the axis of each character. You can use the position, rotation, scale, and shear settings for each character in a layer to create an effect of scrambled letters.



To edit text characters, you must be in Move or Edit mode and some text must be selected. The Layer and Character controls are described as follows.

Layer Order box (in the Layer Attributes section) Moves a layer in front of or behind another layer.



Select:	To move the layer:
Push	One position down in the stack behind another layer.
Pop	One position up in the stack in front of another layer.
Bottom	To the bottom of the stack behind all other layers.

Select:	To move the layer:
Top	To the top of the stack in front of all other layers.

Aligning Layers

Use Align Sel mode to align multiple layers and specify the direction for the alignment.

To align text layers:

1. In the image window, select two or more layers to be aligned. To select multiple layers, `Ctrl`-click the layers (or click the pen button).
2. In the Text Mode box, select Align Sel.
3. On the numeric keypad, press the number that corresponds with the direction in which you want to align the selected layers.

Press:	To:
4	Align left.
6	Align right.
8	Align to the top.
2	Align to the bottom.
5	Centre align.

The text mode returns to Move when the Align Sel operation is done.

Adjusting Text Leading

Use Leading mode to adjust the spacing between lines of text in a paragraph. The selected lines are adjusted in proportion to the leading values already specified. Use this mode to adjust the leading among paragraphs with various leading values.

Rekerning Text

Kerning refers to the space between characters that you can either increase or decrease. Use Rekern mode to change the kerning of all selected characters in relative proportion to the current kerning values.

Use Rekern to adjust the kerning among paragraphs with various kerning values.

To rekern text:

1. Select the string of characters you want to rekern.
2. In the Text Mode box, select Rekern.

   In the Paragraph menu, the Kern field changes to the Rekern field.
3. Enter a new value in the Rekern field or use the `up` and `down arrow` keys to rekern the text in single increments. Press `Shift+up arrow` or `Shift+down arrow` to rekern in increments of 10 pixels.

Resizing Text

Use Resize mode to change the font size of all the characters in a selection in relative proportion to the current font sizes. Use this mode if you have several font sizes represented in a selection.

The Size field allows you to apply an absolute font size value to selected text.

To resize text:

1. Select the string of characters you want to resize.
2. In the Text Mode box, select Resize.

   The Size field you used to specify the original size of the text changes to the Resize field.
3. Enter a new value in the Resize field or use the `up` and `down arrow` keys to resize the text in single increments. Press `Shift+up arrow` or `Shift+down arrow` to resize in increments of 10.

Offsetting Text Layers

Use Y Offset mode to shift selected layers along the vertical axis—the Y-axis. Use the `up` and `down arrow` keys to offset the selection.

Using Safe Title

Use Safe Title mode to align selected text within the safe title overlay. Use the numeric keypad (0-9) to specify the direction of alignment. Safe title is used to define the boundary for positioning text and how it appears in a rendered clip. By default, new layers appear in the upper-left corner of the safe title area.

To align to the safe title:

1. Select the layer you want to align with the safe title.
2. In the Text Mode box, select Safe Title.
3. Press the number on the numeric keypad that corresponds with the direction in which you want to align the selected layers. Press `1`, `2`, `3`, `4`, `6`, `7`, `8`, or `9` to determine the direction in which to align; press `5` to align the layer in the centre of the safe title area. Press `0` to align the layer horizontally in the safe title area.

Copying and Pasting Character Channels

You can copy character attributes, transformations, or both, from one character and paste them onto a selection of others.

To copy and paste character channels:

1. Select the text from which you want to copy the attributes or transformations.
2. In the Character Channels section of the Text menu, click Copy Attributes, Copy Transformations, or Copy All.



3 Select the text to which you want to paste the attributes or transformations.

4 Click Paste.

Grouping Text Layers

You can group two or more layers together to create a montage of text and move text layers together. Grouped layers can be saved and loaded as one unit. You can add more text layers to an existing group, resulting in a new single group.

NOTE When a group of layers is selected, you cannot switch to Edit mode.

To create a text layer group:

1 In the Text menu, select Move from the Edit Mode box.

2 `Ctrl`-click (or click the pen button) to select two or more text layers, or click Select All to select all text layers in the scene.

3 In the Layer Attributes section, enable Group.



Loading Logos

You can use a logo so that it appears on every frame in a clip. Also, you can use logos in text rolls to credit sponsors, contributors, and products. Place a logo directly in a paragraph and then set text formatting properties for the logo.

To insert a logo in a paragraph:

1 In the layer Attributes section, click Load Logo.



The Desktop appears.

2 Select a front clip, and optionally, a matte clip for the logo.

The logo is inserted at the cursor position in the text layer.

3 In the Font and Paragraph menus, increase or decrease the size of the logo in the text layer by adjusting the font size, leading, and kerning. See Entering Text (page 1131).

To save a logo:

1 Select the logos you want to save.

You can select and save multiple logos.

2. Click Save.
3. From the Save box, select Selected Logos.
4. Use the File field to name the logo.
5. Click Save.

   The logo is saved as a *.tif* file.

Tabulating Text

You can organize text into columns by setting tabs in the Tabulation menu. In a text layer, you can also set tab stops to align text at specific locations in a paragraph.

To access the Tabulation menu:

1. Create a text layer.
2. In the Text menu, click Tabulation.

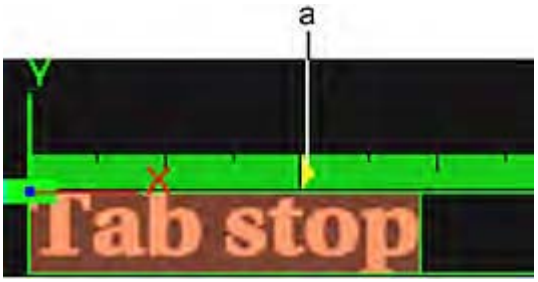
   The Tabulation menu appears.



The Tabulation controls are described as follows.

To set a tab stop:

1. In the Tabulation menu, click Add to add the tab stop in a text layer.



(a) Tab stop shown in the ruler

2. Click the tab stop and drag it to its new location on the ruler. Alternatively, enter a pixel value in the Position field.

Once you set tab stops for a paragraph, press `Tab` in subsequent paragraphs to type text in the location of the next tab stop. In a text layer, each time you press `Enter`, the tab stops you set in the previous paragraph are carried to the next paragraph.

Spell Checking

In the Spell Check menu, you can check the spelling of the text in a text layer. When spelling errors are encountered, a red line is drawn through words not in its dictionary. By default, the spell checker uses the language and dictionary installed with the operating system to verify the spelling of words. You can also create your own custom dictionary.

NOTE If the spell checker is not installed on your system, the Spell Check menu controls are disabled.

To access the Spell Check menu:

1. In the Text menu, click Spell Check.
   The Spell Check menu appears.



To spell check a text layer:

1. In the Text menu, click Spell Check.
2. Enable Check All Layers.
   Lines appear through misspelled words.
3. To correct misspellings, in Edit mode, click Next or Previous to navigate to each misspelled word.

Creating Text Rolls and Text Crawls

You can create text rolls, which are commonly used for credit rolls. A text roll is a layer of text that rolls from the bottom to the top of a clip for a series of frames. In a text roll, you can use special fonts as well as coloured, animated, and tabulated text.

You can include logos of sponsors, contributors, and products in text rolls. For example, you can create a list of credits that uses a green, sheared font for all contributors' names and includes a single, larger capitalized letter spinning on an axis at the beginning of each title line. The contributors' names can appear in a white, semi-transparent, Courier-type font.

A text crawl scrolls text horizontally across the frame. For example, you can create a list of phone numbers that scroll from left to right across the bottom of the screen during a telethon broadcast.

To access the Roll/Crawl menu:

1 In the Text menu, click Roll/Crawl.
The Roll/Crawl menu appears.



The Roll/Crawl controls are described as follows.

To create a text roll or crawl:

1 In the Text Setup menu Rendering section, enable PreRender, and then select Interlaced from the Render box.

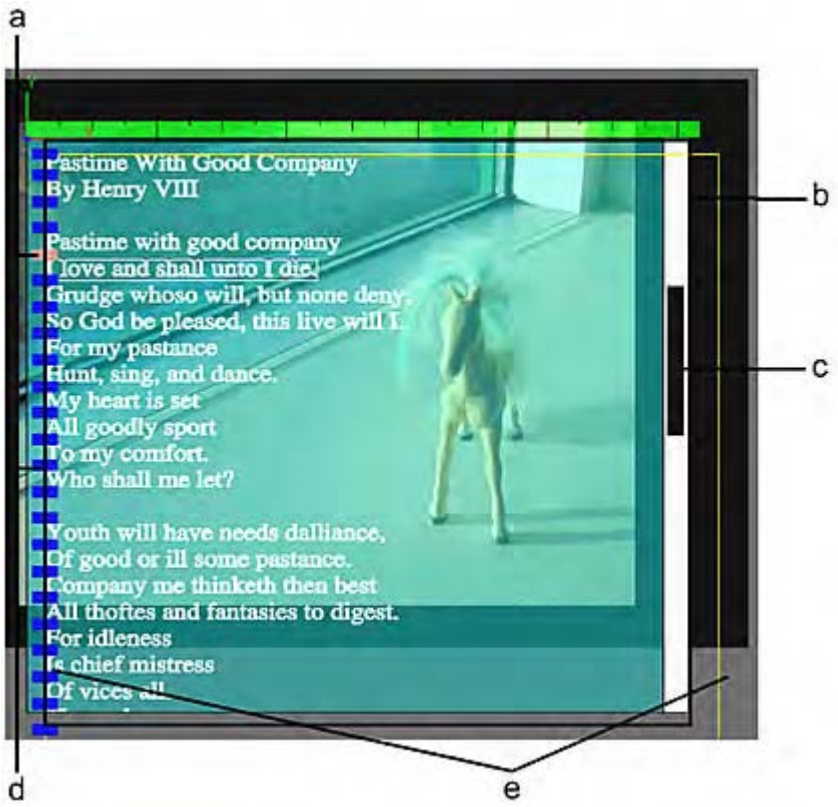


2 In the Text menu, select Interlaced from the Render box.
3 Enable Auto Key.
4 Click Roll/Crawl to enter the Roll/Crawl menu.
5 Go to the first frame of the clip, and enable Roll or Crawl.



6 Click New Layer and add the text you want to use for the text roll or crawl.

A shaded layer appears in the image window with a vertical scrollbar for a roll or a horizontal scrollbar for a crawl. The shaded area represents the crop box area. The following is an example of a text roll.



(a) Red indicates currently selected paragraph (b) Crop box (c) Scrollbar (d) Blue indicates paragraphs in text roll layer (e) Yellow border delimits text

7 To set the start position of where the text appears in the text roll or crawl, use the Scrollbar X and Y fields or drag the scrollbar up or down (roll), or left or right (crawl).

8 Go to the frame where you want the text roll or crawl to finish and set the end position by dragging the scrollbar or using the Scrollbar X and Y fields.

9 If you want to calculate the ideal number of frames for the text roll, choose a roll speed: 1X, 2X, 3X, or 4X.

The suggested duration appears in the Best Roll Duration field. You can either change the number of frames in your clip to match the one in the Best Roll Duration field, or you can leave the clip as is.

10 If the suggested duration is acceptable, enter it in the Duration field and then click Fit Best Speed.

The roll is created and the Scrollbar position and Leading are changed (if required) and locked.

11 If the suggested duration is not acceptable, click Fit Best Speed.

The roll is created and the Scrollbar position and Leading are changed (if required) and locked.

12 Click Process to render the text roll and view the results.

When you play the clip, the text rolls or crawls through the image area from the first position to the final position.

NOTE When processing text rolls and crawls, use PreRender in the Text Setup menu to render the roll or crawl relatively fast while bypassing the animation. Check the message bar to see whether PreRender is enabled.

TIP When creating credit rolls, the anti-aliasing on the text should be turned on.

Animating Text

You can animate text properties including colour, softness, size, shadow, and transformation effects such as rotation, scale, and shear. For example, you can animate text channels to create an effect where the text appears gradually over time and then disappears from view.

Use Auto Key to create keyframes automatically as you make changes to text attributes. The movement between keyframes is interpolated according to the interpolation mode you set.

NOTE PreRender must be disabled when creating text animations. See Processing Considerations (page 1128). Check the message bar to verify the PreRender status.

To animate text automatically:

1 In the Text Animation menu, enable Auto Key.



2 Go to the first frame in the clip.

3 With the Attribute controls, set the attributes and transformation properties.

4 Navigate to another frame in the clip and add more attributes and transformation properties according to the effect you want to produce.

5 Click Preview.

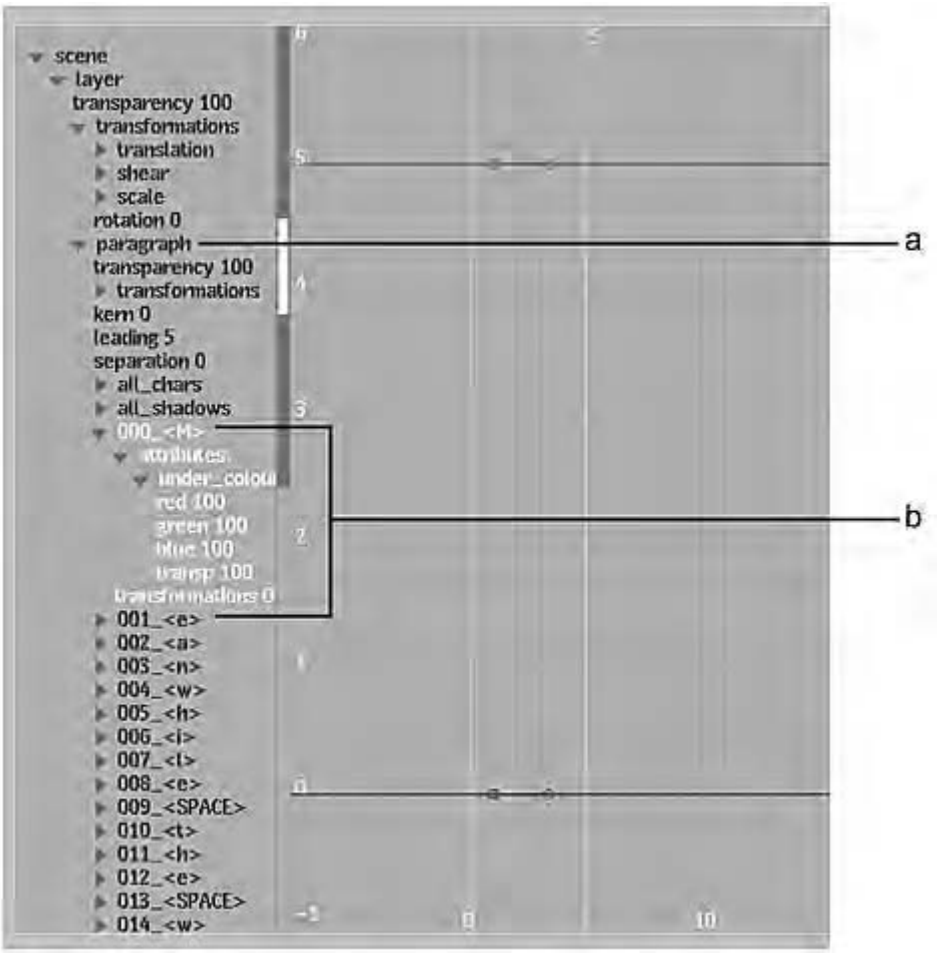
6 Make modifications as required and when finished, click Process.

7 Click Play to view the result.

When you play the clip, the animation moves from the first frame to the last.

Animating Paragraph Channels

In the Text Tool, the Animation controls include a Paragraph Channel View for viewing text channels in the Channel Editor. Use the attributes and transformation channels to animate paragraphs and characters and fine-tune animations created using the Attributes menu.



(a) Paragraph folder contains all the characters, including spaces between words (b) You can animate specific channels for each character

To animate paragraph channels:

1. In the Text menu, click Animation to display the Channel Editor.
2. From the Paragraph Channel View box, select the paragraph channels you want to view in the Channel Editor.

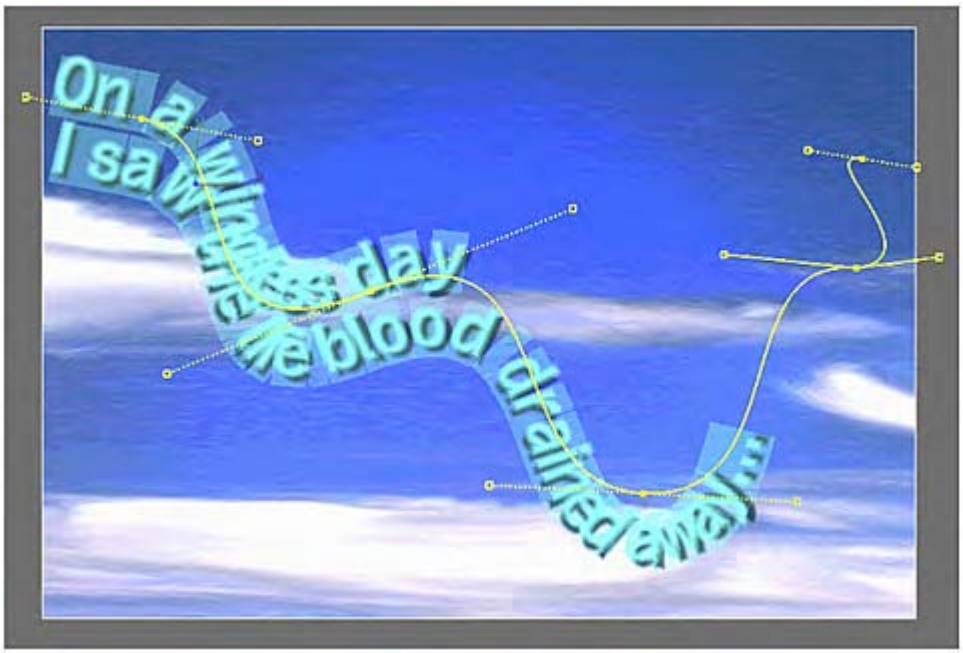


Select:	To view:
Current	The channels for the selected paragraph—the current cursor location.
Layer	The channels for all paragraphs in the selected layer.
All	All channels for all paragraphs in all layers.

3 Animate the selected channels as required.

Animating Text on a Motion Path

You can create a text layer on a spline and animate it over time. In this way, you create marquee text that twists and turns over an image or in time with other objects in the scene. The spline is fully editable; you can add control points and manipulate their tangent handles. In the Channel Editor, the Path group contains a Shape channel for the path and an Offset channel for the offset value.



You can set custom colours for the text path wireframe and vertices in the Text Setup menu. See Setup and Processing Options (page 1126).

To animate text on a motion path:

1. Create a layer or select a layer in the image window.
2. From the Text On Path section of the Text menu, enable On Path.



   By default, the text appears on a path with three vertices.
3. Adjust and animate the shape of the path using the Text On Path controls.

The Text On Path controls are described as follows.

Select:	To:
Move	Modify the shape of the spline by changing the position of the vertex or tangent.
Add	Add more vertices to the spline to create a more complex shape with text. Click anywhere on the path to add a vertex.
Delete	Remove a vertex from the spline.

Select:	To:
Break	Separate two tangent handles and move them independently. Click a tangent; the tangent handle changes from a dashed to a solid line to indicate it is broken.
Auto	Connect broken tangents automatically. Click a tangent to reconnect it.

Saving and Loading Path Setups

You can use a garbage mask spline as a shape for the motion path of text on a path. You can save and load the Shape channel as a list in raw shape data (*.raw* file), which is compatible with garbage mask raw data. Raw setups between text paths and garbage masks are interchangeable.

To load a raw shape data file:

1. In the Text On Path controls, enable On Path.
2. Click Load.
   The Text load options and file browser appear.
3. In the file browser, select Path and then select Default from the Text Path loading boxes.



   A set of path files is visible when the Path and Default options are selected in the Load Text menu.
4. Browse to find the file you want to load.

   TIP Gmask setup files are typically located in the */usr/discreet/<product_name>/path/default* directory.

5. Click Load.
   The spline appears with its new shape in the image area.



Defining Styles

Styles are sets of text attributes and specifications that can be replicated when you need to use the same format again. You create and modify styles, and save and load them from the file browser using the Styles menu. The following character attributes are saved with the style: font, colour, outline, shadow, blur, scaling, rotation and shear. In addition, the layer's background colour and opacity are saved as well.

To access the Styles menu, click the Styles button in the Text menu.



(a) Style Mode box (b) Style Option box

To define a style:

1. Select a character with the attributes you want to save.

   You can select multiple characters, but it is the first one that defines the style.
2. From the Style Option box, select Define Style and then click the style button to which you want to assign the style.

3 Click Name and enter a name for the style. Then click the style button to which you want the name to apply.

The new name appears on the button.

NOTE You can use `Ctrl+Shift` and `F1` to `F9` to assign text attributes to Style buttons 1 through 9 from any menu.

To modify a style:

1 Select the character with the attributes you want to save.

2 From the Style Option box, select Define Style and then click the style button to which you want to assign the style.

NOTE You can use `Ctrl+Shift` and `F1` to `F9` to assign a style to Style buttons 1 through 9 from any menu.

To clear a style:

1 From the Style Option box, select Clear Style and then click a style button.

NOTE It is sometimes convenient to clear all styles just prior to using AutoStyle to save multiple styles automatically. AutoStyle assigns up to nine style buttons at once. See Saving Multiple Styles Automatically (page 1152).

To apply a style:

1 Select a paragraph or string of characters to which you want to apply a style.

2 Press `Ctrl+F1` to `F9`, or press the style button, to apply the style to the selected text.

Saving and Loading Styles

After you create styles, you may want to save one or all of them. You can load any of them in future sessions.

To save a style:

1 Do one of the following in the Styles menu:
- Select All Styles from the Style Mode box.
- Select One Style from the Style Mode box, and the style you want to save (Style 1 to Style 9).

2 Click Save Style.

The Save Styles menu and file browser appear.

3 Choose a directory for the style.

NOTE You can click the Create Dir button to create a new directory in which to save your styles.

4 Enter the filename.

The style is saved. The file browser automatically closes and the Styles menu reappears.

To load a style:

1 Select an option from the Style Mode box.

Select:	To:
All Styles	Load files saved with multiple styles. The file browser lists setups saved with multiple styles.
One Style	Load files saved with one style only. The file browser lists setups saved with a single style.

2 Click Load Style.
The Load Styles menu and file browser appear.

3 Navigate to the appropriate directory and select the style(s) you want to load.

4 If necessary, click Exit Load Text to exit the file browser.

Saving Multiple Styles Automatically

If you create a section of text with multiple sets of attributes, you can assign each set as a separate style with a shortcut and then save them. The Text Tool automatically detects the different sets of attributes used in the section and assigns each set to a Style button. You can then use shortcuts to apply them.

To assign styles from an existing layer using shortcuts:

1 Select the section of text.

2 From the Style Option box, select AutoStyle.

NOTE If you used more than nine sets of attributes in the selection, only the first nine are assigned to a Style button.

3 Save the styles.

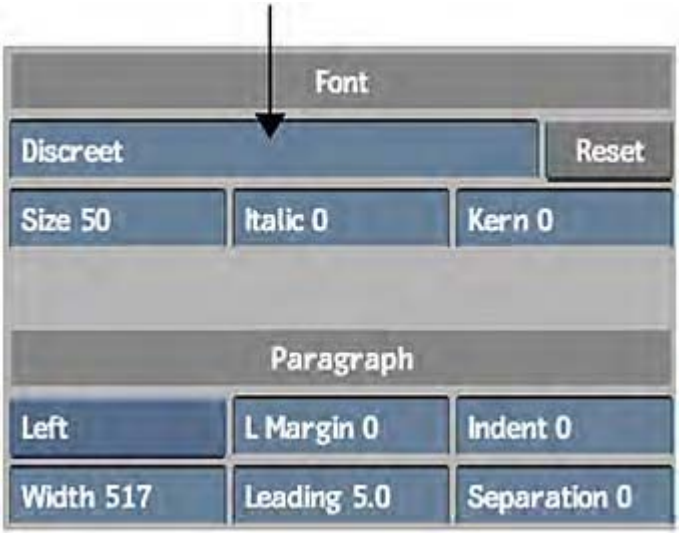
Changing Fonts

When you open the Text tool, the default font is loaded and ready to use. When you choose a different font, it becomes the current font and all text you type appears in the current font. If you edit text that has a different font, the current font is replaced by the font of the text at the cursor position. For example, if you are working with Carta font and you edit text that uses Helvetica, Helvetica becomes the current font and all subsequent text you type appears in Helvetica.

You specify the default font in the TL Effects / BFX tab of the Preferences menu. Also, you can install additional fonts for use in the Text Tool.

To change the current font:

1 In the Font section of the Text menu, click the Font field.



The font library appears.



(a) Font Type box (b) Font Preview window

2 In the Font Type box, select the font type you want to load.

Type	Description
Type 1	A single-byte font type designed for Roman language fonts. Soft-links to the Adobe® Postscript® Type 1 fonts installed with the system or supplied by Autodesk are contained in */usr/discreet/font* directory.
CID	A multi-byte adaptation of Adobe Postscript Type 1 fonts, well-suited to representing the large character sets of Asian languages such as Japanese, Korean and Chinese.
TrueType	A widely used cross-platform font format.
OpenType®	A font developed by Adobe and Microsoft® that supports expanded character sets and layouts.
Auto	Displays all font types, detecting the type automatically.

3 In the file browser, navigate to the directory that contains the font you want to load.

4 Preview a font. Do one of the following:

- Select a font to see it with sample text in the Font Preview window.

The sample text can be the text you selected in the text layer, or text that you enter by clicking the Font Preview window to display the on-screen keyboard.

- In the Proxies/Titles box, select Proxies to preview fonts. If no proxy appears, click Generate Proxies to generate a font proxy.

  This process may take a while, but once you create the proxies, the settings are retained. Each time you return to the font library, you can toggle between viewing font titles and font proxies by selecting Proxies or Titles.

5 Select the font.

6 Click Load.

The Text menu reappears with the name of the new font in the Font field.

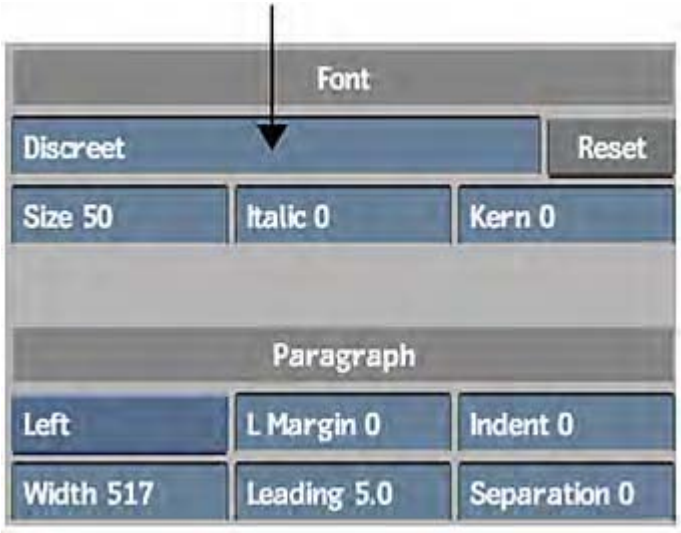
Pasting Text from an External Source

You can copy text from a Linux terminal or another application and paste it directly in a text layer.

To paste text from a terminal to the Text tool:

1 In the Text tool, create a layer for the text you want to paste.

2 Click the Font field and select a font.



3 In the terminal, copy the text.

4 In the Text tool, paste the text in a text layer.

Entering Non-Latin Text

When entering non-Latin text characters, consult your system documentation to make sure the required input method packages are installed. For example, to enter Chinese characters, you can use the *chinput* package.

The following example illustrates entering Japanese text in a text layer with the *kinput2*, *canna*, and *Wnn6* packages.

To enter Japanese text in a text layer:

1. Start the canna daemon by typing **/etc/init.d/canna start** in a Linux terminal.
2. Type the following commands in the terminal:
   - **kinput2 -canna &**
   - **setenv XMODIFIERS @im=kinput2**
   - **setenv LANG ja_JP.eucJP**

   NOTE For detailed instructions on kinput, canna, and Wnn6, consult your system documentation.
3. Start your Autodesk application.
4. Load a clip in the Text tool.

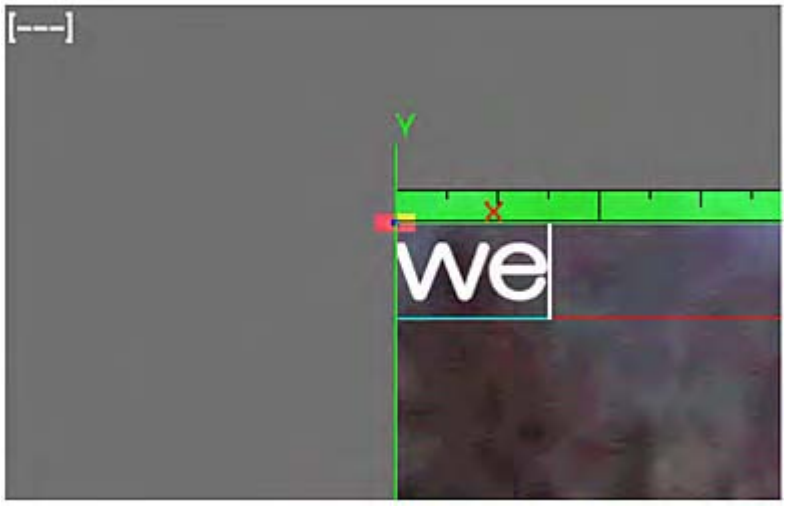
   Notice the message bar that appears at the top of the screen. This indicates you are in Latin input mode.



(a) Message bar

5. Create a layer and click the Font field to select a Japanese font from the file browser. From the Text Mode box, select Edit and then select the layer in the image window.

   When you type text, notice that you are still in Latin input mode.



6. Press your Input Method keyboard shortcut.

   The message bar changes to indicate that you are in Japanese text mode.



(a) Message bar

7 Type text, as required.

8 Press `Enter` to accept the characters and add them to the text layer.



9 To revert back to Latin input mode, press the Input Method keyboard shortcut again.

Accessing OTF Fonts

In order to access symbols of the OTF font type, access the on-screen keyboard, enable the Symbols button (in the lower right of the keyboard), then use the arrow keys to browse through the pages of symbols until you find the character set you need.

Warping and Morphing 24

About Distort

Use the Distort menu to warp a clip and create morphs between two clips. Both of these effects are created using splines, which are drawn and animated in a similar way to garbage masks.

Warping using the Distort Tool

Warps are created by drawing a source spline, and then offsetting a second corresponding destination spline. When you draw a source spline, an identical destination spline is automatically created. When you move the destination spline, the offset of the destination from its source spline is what warps the clip. For example, you can draw a source spline around the eye of an actor, and then edit the points of the corresponding destination spline to make the eye bulge.



Source spline drawn around eye

Image courtesy of Behavior Communications Inc.



Offsetting destination spline creates warp

Image courtesy of Behavior Communications Inc.

Distort Warping Workflow

The general warping workflow in the Distort tool includes the following steps.

Step:	Refer to:
1. Load clips into the Distort tool.	Accessing Distort (page 1160).
2. Draw a source spline.	Drawing Splines (page 1166). **TIP** Use Distort's schematic as you work with splines to help organize your work. See Using Distort's Schematic (page 1172).
3. Edit the source spline.	Editing Splines (page 1170).
4. Animate the source spline.	Animating and Tracking Splines (page 1180).
5. Unlink the destination spline from the source to enable warping.	Linking and Unlinking Splines and Axes (page 1169).
6. Edit the destination spline.	Editing Splines (page 1170).
7. Animate the destination spline.	Animating and Tracking Splines (page 1180).
8. Refine the warping effect.	Refining Distortions (page 1184).
9. Set what appears at edges of clip if the distortion effect pulls the clip away from the border, revealing empty space, or the background clip.	Filling in Gaps at Edges of Distorted Clips (page 1190).

NOTE Repeat this workflow for each warped element of the clip.

Morphing in Distort

Morphs are created by creating corresponding splines on two clips and linking them together. You then set interpolation values controlling how the features contained in the first set of splines morph into the features contained in the second set of splines. This method is called source interpolation. Finally, you blend the two clips over time to complete the morph. For example, you can draw splines around the outline of an actor and each of the actor's facial features. Then duplicate these splines onto a second clip containing the actor to whom the first actor will morph into. Edit the copied splines to more accurately contour the features of the second actor, link the corresponding splines, and then blend the two clips to complete the morph.

Both sets of splines, those on the first clip, and those on the second clip, can be animated. Additionally, you can edit and animate the interpolated splines.



Source splines drawn on actor

Image courtesy of Behavior Communications Inc.



Corresponding splines on second actor are linked

Image courtesy of Behavior Communications Inc.

Distort Morphing Workflow

The general morphing workflow in the Distort tool incudes the following steps.

Step:	Refer to:
1. Load clips into the Distort tool.	Accessing Distort (page 1160).
2. Draw a source spline on the Front1 clip.	Drawing Splines (page 1166). **TIP** Use Distort's schematic as you work with splines to help organize your work. See Using Distort's Schematic (page 1172).
3. Edit the Front1 source spline.	Editing Splines (page 1170).
4. Animate the Front1 source spline.	Animating and Tracking Splines (page 1180).
5. Display the Front2 clip.	Toggling Between Inputs (page 1164).
6. Duplicate the Front1 spline, then toggle the duplicated spline to become an Front2 spline.	Toggling Spline Nodes from One Input to Another (page 1178).
7. Edit the Front2 source spline.	Editing Splines (page 1170).
8. Animate the Front2 source spline.	Animating and Tracking Splines (page 1180).
9. In the Distort schematic, link the spline on the Front1 clip with the spline on the Front2 clip.	Linking Source Nodes (page 1179).

Step:	Refer to:
10. Set Blend and Interpolation values.	Blending Clips (page 1183) and Interpolating Features (page 1183).
11. Refine the morphing effect.	Refining Distortions (page 1184).

NOTE Repeat this workflow for each morphed feature you want to correlate from the Front1 clip to the Front2 clip.

Mesh-Based Warps and Morphs

While the Distort tool uses splines to define features of a clip you want to warp or morph, you can also use the Warper tool. Instead of splines, the Warper uses meshes to define affected features for similar warping and morphing effects.

Mesh-based warps and morphs created with the Warper can be more time-consuming since meshes have many more points to work with and usually cover the entire clip. The Warper can be useful, however, because you have more direct control over the parts of the clip being altered since you manipulate the mesh directly.

Accessing Distort

Use Distort to create warps and morphs of clips. The node uses spline-based shapes to create these effects.



To access the Distort editors, use:

- Batch, then select a node from the Node bin. (page 1161)
- Timeline, then use Batch FX. (page 1161)
- Tools, then select the menu. (page 1161)

Distort accepts two front clips, two matte clips and one back clip as inputs, and outputs a result and an outmatte.

In Batch, you can create a more convincing morph, for example, by animating colour transitions between the first and second clips using Colour Corrector nodes.

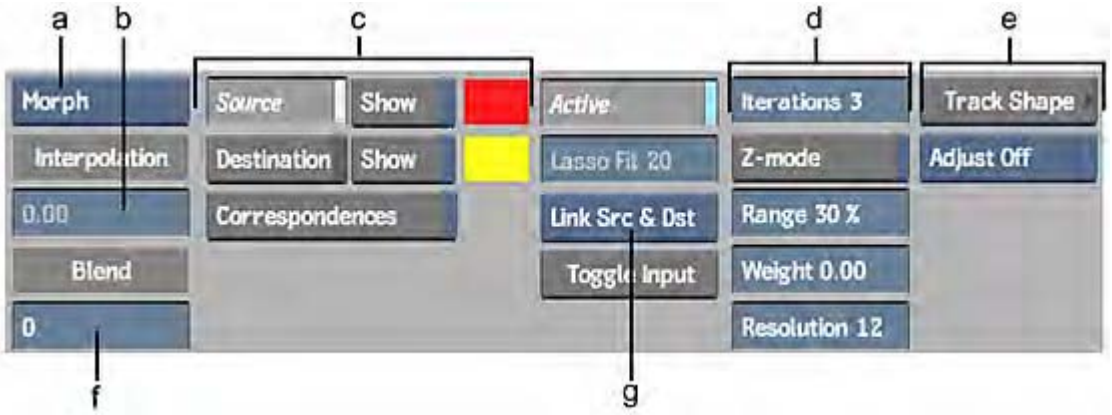
The Distort controls are described as follows.

Clip Display buttons Controls which clips, of those you load into Distort, are displayed. Use the Clip Display buttons to produce various effects, or to toggle on or off clips and mattes to facilitate your work.

Axis controls Edits the axis position, scale, and shear value of a spline.

Axis Stabilizer controls Gives access to the Stabilizer, for automatically tracking motion in the axis.

View box Sets the view in the image window. Two views available from this box, Input and Matte, each correlate with two distinct views, Front1 and Front2, or Matte1 and Matte2. You toggle between the input or matte views using the Input box.



(a) Warper button (b) Interpolation field (c) Spline Display controls (d) Refining controls (e) Shape Stabilizer controls (f) Blend field (g) Link box

TIP The distortion grid can be displayed by enabling Draw Grid in Distort's Setup menu. This can be useful when working with the Iterations and Range fields.

Accessing Distort from Batch

To access the Distort menu from Batch:

1. In Batch, drag and drop the Distort node from the node bin to the schematic.
2. Connect the node to a clip or image. You can connect up to five sockets: Front1, Front2, Matte1, Matte2, and Background. The Front2, Matte1, Matte2, and Background clips are optional.
3. Double-click the Distort node.

   The Distort node menu appears. The Distort node outputs a warped or morphed result and a warped or morphed matte.

Accessing Distort from Batch FX

To access Distort through Batch FX:

1. Select Timeline.
2. Select the segment.
3. Click the FX button.
4. Click the Create Batch FX button.
5. Double-click on, or drag, a Distort node into the schematic.

   The Distort node is now in the schematic.
6. Double-click the Distort node.

   You are in the Distort node editor.

Accessing Distort from Tools

To access the Distort menu from Tools:

1. Select **Tools ➤ Filter ➤ Distort**.

The cursor changes to Pick Front.

2 From the Inputs box that appears, select the clips or mattes you will use.

Select:	To load:	To produce:
Front1	One input clip only	A warp of a single clip.
Front1 Matte1	An input clip with its matte	A warp of a single clip using a matte over a black background. A warped matte is also produced.
Front1 Matte1 Back	An input clip with its matte, as well as a background	A warp of a single clip using a matte over a background clip. A warped matte is also produced.
Front1 Front2	Two inputs	A morph from one clip to another over a black background.
Front1 Front2 Back	Two inputs, as well as a background	A morph from one clip to another over a background clip.
F1 F2 M1 M2	Two inputs with their respective mattes	A morph from one clip to another over a black background. A morphed matte is also produced.
F1 F2 M1 M2 Back	Two inputs with their respective mattes, as well as a background	A morph using two clips and their mattes on a background clip instead of on black. A morphed matte is also produced.

3 Select the clips as indicated by the cursor.

For example, if you selected Front1 Front2 Matte1 Matte2 Bkg to morph between two clips, select clips as follows:

- The clip where your morph originates is Front1.
- The clip containing the subject that will be morphed to is Front2.
- The matte for Front1 is Matte1.
- The matte for Front2 is Matte2.
- The background clip.

When the clip selection is complete, the cursor will change to Render Here.

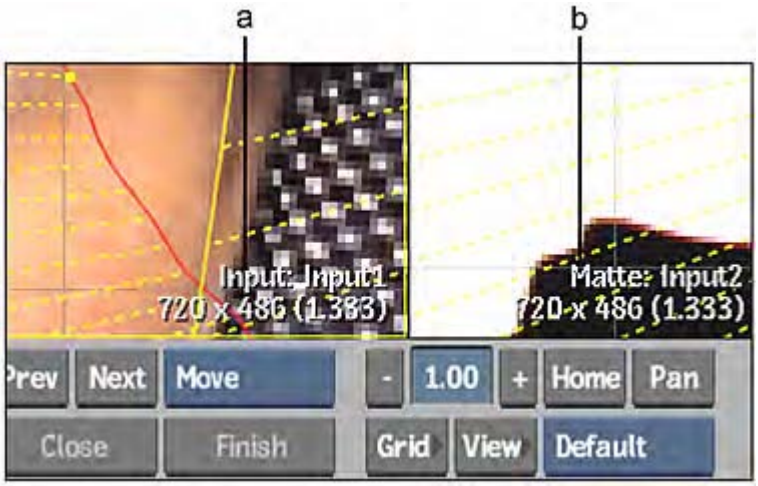
4 Click on any free (or a grey) area on the workspace.

You are now in the Distort Tool editor.

Using Distort's Views and Clip Display

You can change the view using the Distort menu as you create your effect.

Distort makes it easy to toggle between Front1 and Front2 clips or mattes independently in each viewport. For example, you can display Front1 in one viewport, and the matte of Front2 in another viewport.



(a) Front1 (Input1) clip displayed (b) Matte of Front2 (Input2) displayed

Within a particular view, you can also toggle on or off clips to facilitate your work or create different effects. For example, if you loaded two inputs and their mattes, as well as a background clip, you can turn on or off any of the five clips that together make up the composite.

Changing Views

Use the appropriate view, available from the Distort menu's View box, for the task you are performing.

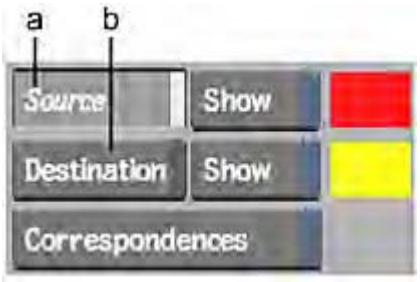


(a) View box (b) Input box

Select:	To view:
Input	The Front1 or Front2 clip. You toggle between them using the Input box.
Matte	Matte1 or Matte2. You toggle between them using the Input box.
Background	The background clip used in the warping or morphing effect.
Result	The result clip of the warping or morphing effect.
ResultMatte	The result matte that is created when you blend the IFront1 and Front2 clips when morphing.
DistortSchm	The Distort schematic.

Each of the Input and Matte views can correlate with two views if you loaded two inputs and their mattes into the Distort tool. See Toggling Between Inputs (page 1164).

While in the Input view, you can also toggle between viewing the unwarped input and the warped input by clicking between the Source and Destination buttons in the Warp and Morph menus.



(a) Source button (b) Destination button

Toggling Between Inputs

When you are morphing between clips, you can toggle between them so either Front1 or Front2 appears in the image window. The splines drawn on the clip are also displayed. For example, if you select Front1, the Front1 clip, along with its splines, is displayed in the image window. If you then select Front2, the Front2 clip appears with the splines drawn for Front2.

You can similarly toggle between Matte1 and Matte2.

To toggle between s or mattes:

1 From the Warp or Morph menu in the Distort tool, select an option from the Input box.

Select:	To:
Front1	Display the Front1 clip if Input is selected in the View box, or Matte1 if Matte is selected in the View box, along with the Front1 source and destination splines.
Front2	Display the Front2 clip if Input is selected in the View box, or Matte2 if Matte is selected in the View box, along with the Front2 source and destination splines.

The corresponding clip with its splines is displayed. As well, the splines that are not displayed, for example, splines added to the Front1 clip if you select Front2, are indicated as hidden in Distort's schematic.

Displaying Clips

Depending on the clips you load into Distort, you can use the Clip Display buttons to produce various effects, or to simply toggle on clips and mattes to facilitate your work. The following table shows how to set up the clips to produce various effects.

Front1	Front2	Matte1	Matte2	Back	Effect
On	Off	Off	Off	Off	Warps theFront1 clip only. This is the same as loading only one input clip.
On	On	Off	Off	Off	Morphs between two clips. This is the same as loading two input clips.

Front1	Front2	Matte1	Matte2	Back	Effect
On	Lock	Off	Off	Off	Wipes (page turns, rolls); regional warps (if the same clip is loaded as both the Front1 and Front2 clip).
On	Off	On	Off	Off	Warps the Front1 clip and Matte1 clip, composited on black.
On	On	On	Off	Off	Warps over a background with a matte clip.
On	Lock	On	Off	Off	Warps the Front1 and Matte1 clips composited on the Front2 clip.
On	On	On	On	On	Morphs between two clips and their mattes onto the selected background clip.

Creating Splines

You draw open or closed splines to match the features of a clip that you want to warp or morph. Splines are drawn in a similar way as garbage masks, by clicking to add points, or by dragging to draw freehand segments. You can also load a saved garbage mask setup to use as distort splines.

You draw a spline around a feature at the start of an effect. When you add a spline, both a source and destination spline is added to the clip. Since these splines are initially linked together and overlap, your manipulations to the source spline affect the destination spline in the same way. After you have drawn and animated the source spline, you can unlink the splines and edit the vertices and tangents of the destination spline independently.

When creating a morph between two clips, you draw splines on the Front1 clip, then optionally duplicate them to the Front2 clip. When you link the splines, those duplicated to the Front2 clip become the destination for splines drawn on the Front1 clip.

NOTE When morphing, you can also draw source spline on the Front2 clip to link to corresponding Front1 splines. Only splines with the same number of points, and drawn in the same orientation (clockwise or counter-clockwise), will create proper morphs.

You can show or hide source and destination splines, as well as change their colours.

Using Garbage Masks as Distort Splines

Garbage mask setups are created through the GMask node in Batch. You can import a saved garbage mask setup into Distort and use it as a spline. Note the following about how Distort interprets settings in the imported garbage mask setup:

- If the imported garbage mask has animation on its axes, the animation is recreated in axes above the main axis in the Distort schematic.
- If the imported garbage mask has shape animation or explicit animation on the gmask splines, the same animation on the source and destination splines is transformed into shape animations.
- If the imported garbage mask includes tracking data, the offset parameters on the geometry are duplicated on the offset parameters of both the source and destination splines.

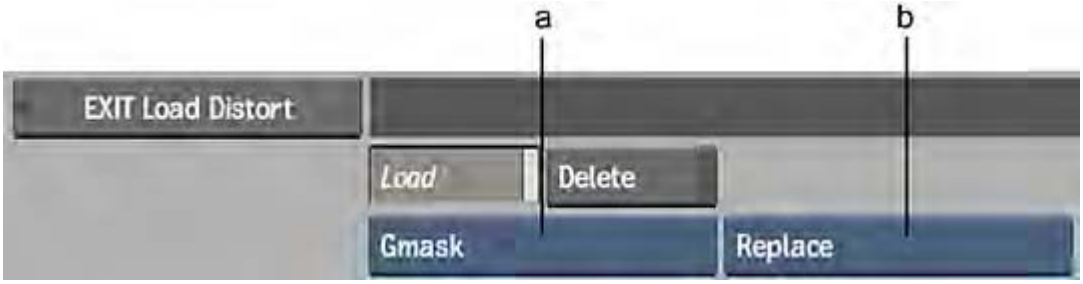
- Garbage mask setups that use multiples axes to manipulate instances of splines are translated to Distort instances. See Instancing Distort Splines (page 1171).

For help creating garbage masks, see Drawing a Mask (page 897).

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.

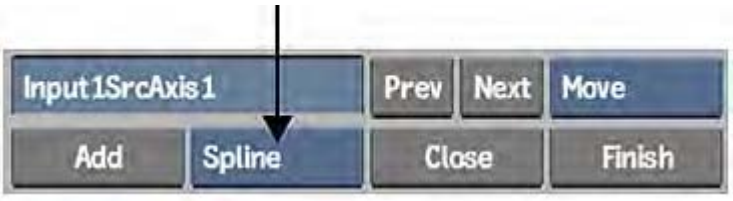
   TIP A library of preset garbage mask setups is available. To load a preset garbage mask, navigate to the */usr/discreet/<product_name>/gmask/default* directory.

Drawing Splines

Draw single-point, open, or closed splines around the features you want to warp or morph at the beginning of the effect.

To draw a closed spline:

1. Go to the frame where you want the effect to begin.
2. Select Spline from the Node box.



3. If you are creating a morph, select the input where you want to draw the spline. To add a spline to Front1, for example, select Front1 in the Morph menu's Input box.



NOTE When creating a morph, a common strategy is to create the spline on just one input, and then duplicate it to the other. This has important advantages over creating splines on both inputs independently. See Toggling Spline Nodes from One Input to Another (page 1178).

4 Click Add.

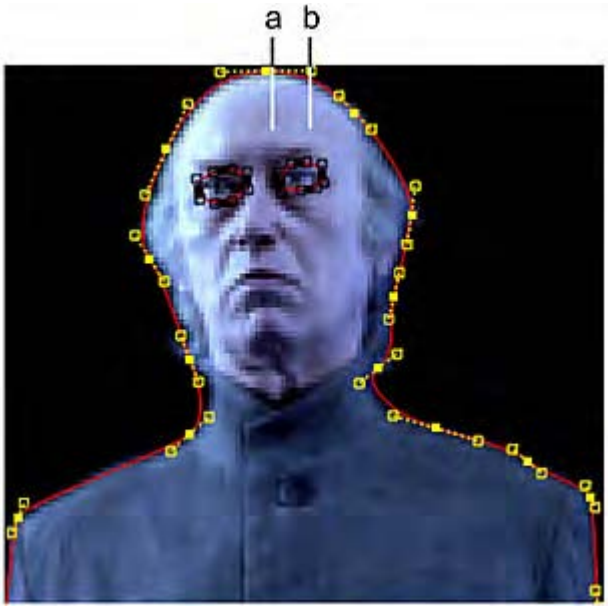
5 Do any of the following:

- Click to add vertices.
- `Shift`-drag to add freehand segments to the spline. Vertices are added where you drag, and appear after you release `Shift`. You can then use the Lasso Fit parameter to increase or decrease the number of vertices that define the freehand segments of the spline. If you add or delete vertices on the spline, or exit, the influence of the Lasso Fit parameter on the number of vertices is lost. See Adjusting the Number of Points in Freehand Segments (page 901).

6 To close the spline, do one of the following:

- Click Close.
- Press `C`.
- Click the first vertex.
- While holding `Shift`, drag to the first vertex.

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



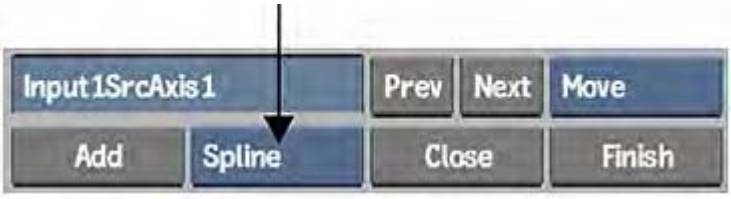
(a) Vertex (b) Tangent

Image courtesy of Behavior Communications Inc.

To draw a single-point or open spline:

1 Go to the frame where you want the effect to begin.

2 Select Spline from the Node box.



3 If you are creating a morph, select the input where you want to draw the spline. To add a spline to Front1, for example, select Front1 in the Morph menu's Input box.



NOTE When creating a morph, a common strategy is to create the spline on just one input, and then duplicate it to the other. This has important advantages over creating splines on both inputs independently. See Toggling Spline Nodes from One Input to Another (page 1178).

4 Click Add.

5 Do any of the following:

- Click to add vertices. If you are drawing a single-point spline, proceed to the next step.
- `Shift`-drag to add freehand segments to the spline. Vertices are added where you drag, and appear after you release `Shift`. You can then use the Lasso Fit parameter to increase or decrease the number of vertices that define the freehand segments of the spline. If you add or delete vertices on the spline, or exit, the influence of the Lasso Fit parameter on the number of vertices is lost. See Adjusting the Number of Points in Freehand Segments (page 901).

6 To finish the spline, do one of the following:

- Click Finish.
- Press `F`.
- Click the last vertex.

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

If you want to view the nodes that are added with each new spline, select DistortSchm from the View box. You can use Distort's schematic to access a menu, create parent-child relationships between splines and axes, delete splines, link Front1 and Front2 splines to create morphs, as well as perform other organizational tasks. See Using Distort's Schematic (page 1172).

To show and hide splines:

1 From the Warp or Morph menu in the Distort tool, do one of the following:

- To work with source splines, click Source.
- To work with destination splines, click Destination.
- To work with source splines while seeing the destination splines as a reference, click Destination Show.
- To work with destination splines while seeing the source splines as a reference, click Source Show.

NOTE To toggle between splines drawn on the Front1 and Front2 clip, select Front1 or Front2 from the Input box.

To colour splines:

1 Click the Source colour pot or the Destination colour pot, and use the colour picker that appears.
The colour you pick is applied to all source or destination splines accordingly.

Changing the Colour of Tangents

You can change the colour of tangents on the splines you draw. This is useful to better contrast the spline's tangents from the clip so that they are easier to work with.

To change the colour of tangents on splines:

1 In the Distort menu, click Setup.
2 In the Setup menu Display section, click the Tangents colour pot.



3 From the colour picker that appears, select a colour.
Tangents are displayed in the colour you set.

NOTE All spline tangents are initially displayed in magenta after they are drawn until a source spline and its corresponding destination spline are offset from one another.

Linking and Unlinking Splines and Axes

Warps are created by offsetting a destination spline from its source. By default, when you draw a spline, it is linked to its corresponding source or destination. After it is drawn, you might animate it, for example, so that it follows a subject in the clip. When the splines are linked, both the source and destination splines are animated in the same way.

When you are ready to create the warp, you unlink the destination spline from its source. Manipulations then affect each spline separately, so you can alter the shape of the destination. The differences between the two corresponding splines are what creates the warp.

You can also independently unlink the axis of a source spline and the axis of a destination spline. By default, the axis nodes are linked, even if the source and destination splines themselves are unlinked. You might unlink only the axes of corresponding splines, for example, if you want their offset to create the warping effect while the actual shapes of the splines remain the same.

You use a different linking method when using source interpolation to create a morph. See Linking Source Nodes (page 1179).

To link and unlink source and destination splines or axes:

1 Do one of the following:
- Select a spline by clicking it in a viewport or by clicking its spline node in the Distort schematic.

- Select an axis node in a spline node tree that is a parent to a spline node.

See Using Distort's Schematic (page 1172).

2 From the Warp or Morph menu in the Distort tool, click the Link box to toggle between Lnk Src & Dst and Distort.



Select:	To:
Lnk Src & Dst	Link source and corresponding destination splines, or their axes. With linked splines, if you move a spline or any of its vertices both the source and corresponding destination splines are affected in the same way. With linked axes, if you move the parent axes of a spline, the parent axes of the destination spline is affected in the same way.
Distort	Allow source and destination splines or axes to be offset from one another, which creates the warping effect.

Vertices on linked splines appear in magenta. Axes that are linked appear in magenta.

NOTE If you select Lnk Src & Dst and manipulate a vertex that has already been offset from its corresponding vertex, both the source and destination vertex snap to the same position. Similarly, if you manipulate the Axis values of a spline that has been offset from its corresponding spline with Lnk Src & Dst selected, all vertices and the splines' axes snap together.

Editing Splines

You edit splines in the same way as you edit garbage masks, working with their vertices and tangents. See Moving Vertices and Tangents (page 904).

If Lnk Src & Dst is selected from the Link box, your edits affect both the source and destination spline regardless of where you make the edits. See Linking and Unlinking Splines and Axes (page 1169).

TIP The Link box has no effect if spline nodes are linked in the Distort schematic while morphing using source interpolation. See Linking Source Nodes (page 1179).

To edit a spline:

1 From the Edit Mode box, select one of the following modes to edit a spline's vertices.

Select:	To:
Select	Select one or more vertices.

Select:	To:
Move	Move vertices to a new position.
Scale	Scale vertices in relation to the axis centre of the spline.
Delete	Remove vertices from the spline.
Break	Break tangents of vertices.

Transforming Splines

You can apply transformations such as translation, rotation, scaling, and shearing to a spline. The transformation data for the spline is stored in its Axis node. See Axis Nodes (page 1176).

All transformation parameters can be animated. You can also use a motion path to animate the position of a spline.

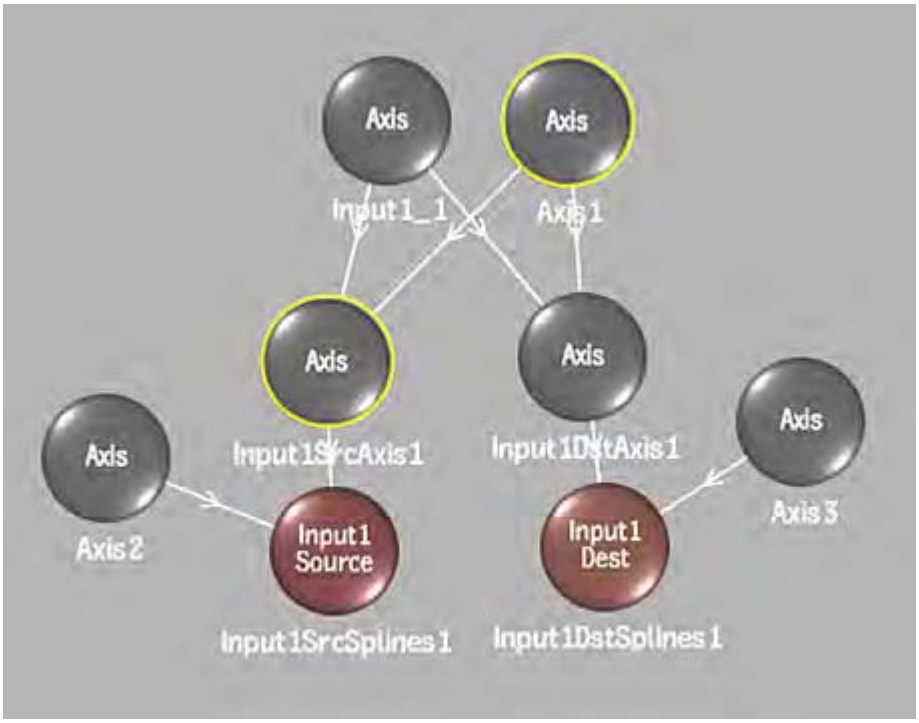
To transform a spline:

1 Use the Transformation fields to modify the position, shape, and size of a spline.



Instancing Distort Splines

When you add multiple axes above a pair of source and destination splines, each axis can be used to manipulate a separate instance of the pair of splines. You can then use the Distort settings for each axis to manipulate the instance of each spline. In the following example, the Input1_1 and Axis1 axes can be used to change instances of the source and destination splines. Axis2 and Axis3 cannot be used in this manner, since they are only attached to one input spline.



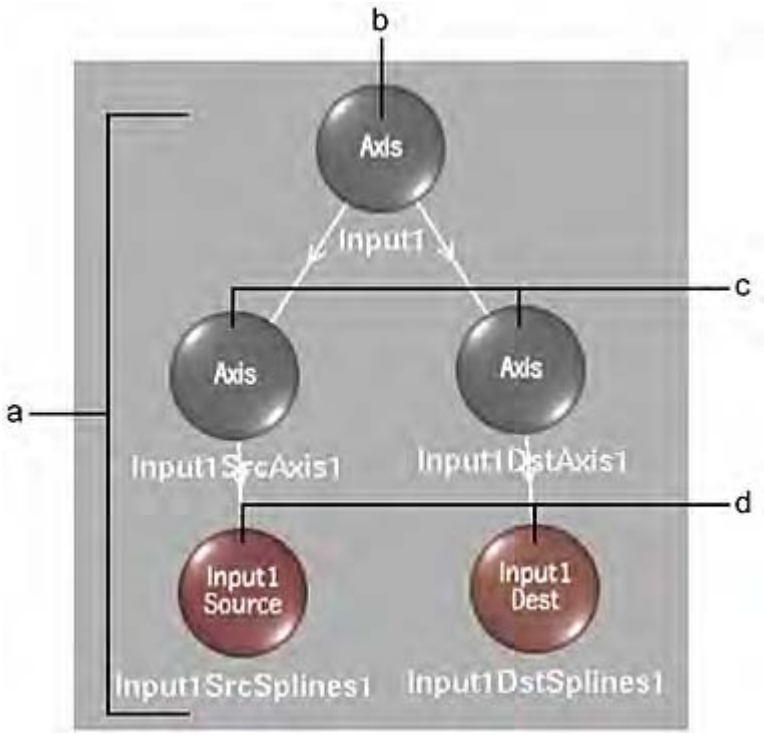
Using Distort's Schematic

The Distort schematic uses nodes to represent all the splines in the scene. The schematic shows relationships between source and destination splines, and between splines applied to the Front1 and Front2 clips.

Use the schematic to control all the splines in the scene, set parent-to-child relationships among nodes, as well as to select splines more easily.

You also use the schematic to link Front1 and Front2 source splines to create morphs using source interpolation. See Linking Source Nodes (page 1179).

When you add a spline to the Front1 clip, for example, the following spline node tree appears.



(a) Spline node tree representing corresponding splines added to Front1 (Input1) (b) Axis node for an Front1 (Input1) spline: the parent of corresponding source and destination splines (c) Axis nodes for each spline (d) Spline nodes for corresponding source and destination splines added to Front1 (Input1)

Each spline added to an input clip results in its own spline node tree in the schematic, with its own corresponding source and destination splines.

You can change an Front1 spline to an Front2 spline or an Front2 spline to an Front1 spline using the Toggle Input button. See Toggling Spline Nodes from One Input to Another (page 1178).

Visibility of the splines is controlled from the Warp or Morph menu using the Spline Display controls. When you toggle between the Front1 and Front2 clips with the Input box, splines are hidden for the inactive clip, which is also reflected in the schematic.

You can also set the transparency of the schematic. This is useful if you want to dim its display so as not to interfere with the main focus of your work.

To view Distort's schematic:

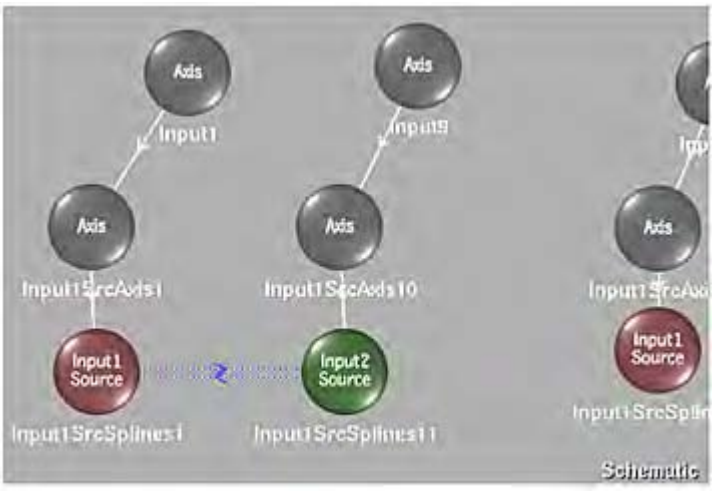
1 Do one of the following:

- Select DistortSchm in the View box.



- Press the ~ key. Pressing the ~ key a second time returns to the previous view.

The Distort schematic appears.



TIP Display Distort's schematic and the input clips in a multiple viewport setup. This way, you can see the splines in the clips as you select and work with them in the schematic.

To set the transparency of the Distort schematic:

1. In the Distort menu, click Setup.
2. In the Setup menu Schematic section, adjust the value in the Transparency field.

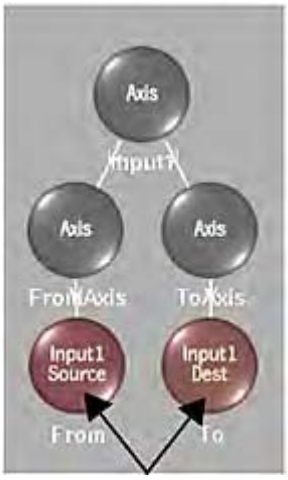


NOTE The schematic transparency has a maximum value of 90%.

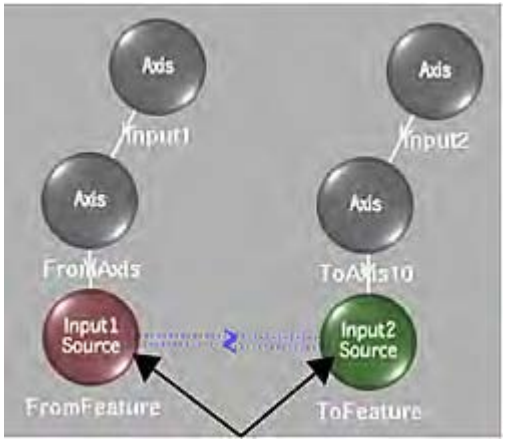
Node Types

In the Distort schematic, a node can be a spline or an axis.

Nodes help you organize warps that occur between corresponding source and destination splines in a spline node tree, and morphs that occur between linked Front1 and Front2 source nodes in different spline node trees.



Warps occur between corresponding source and destination splines that are offset from one another



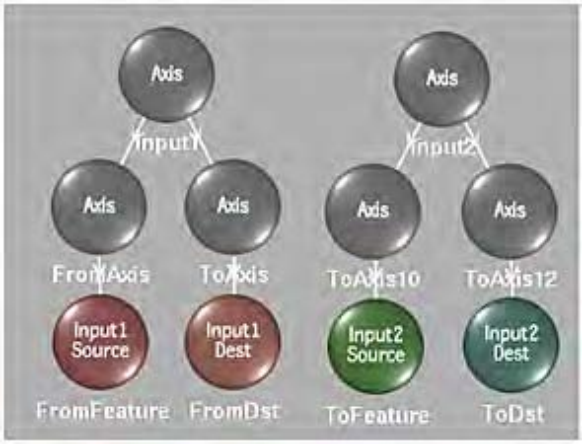
Morphs occur between corresponding Front1 (Input1) and Front2 (Input 2) source splines that are linked together

Spline Node Trees

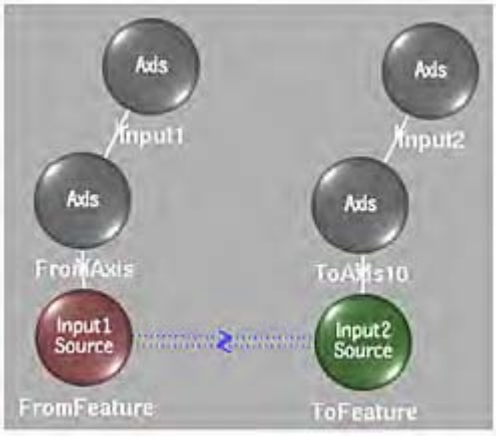
When you add a spline to the Front1 or Front2 clip, axis and spline nodes are added to the Distort schematic. These nodes represent the source spline and its corresponding destination spline, and are parented to axes in a spline node tree.

Warps are created when the source spline and destination spline in a spline node tree are offset from one another.

Morphs are created when you link a source spline from an Front1 spline node tree to a source spline from an Front2 spline node tree.



A spline node tree for a pair of corresponding splines on the Front1 (Input1) clip, and a spline node tree for a pair of corresponding splines on the Front2 (Input2) clip



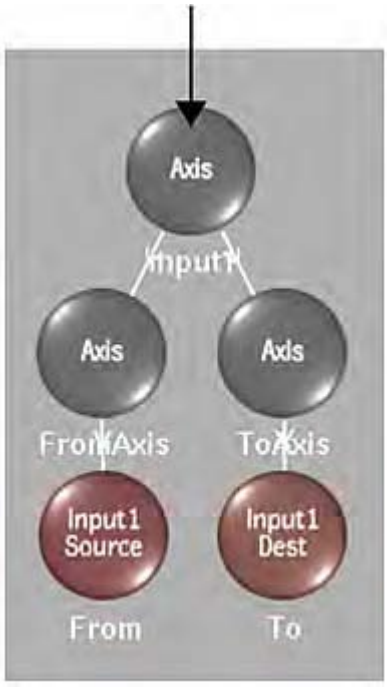
When you link source splines between Front1 (Input1) and Front2 (Input2) for source interpolation, the destination splines and their respective Axis nodes are removed from the spline node trees

Axis Nodes

Use Axis nodes to control the position, rotation, scale, and shear values of splines.

Initially when you add a spline, its source and destination node is parented to its own Axis node, and these Axis nodes are in turn parented to a common Axis node. So, for example, you can control the position of a particular Front1 source spline via its Axis node. Click the Axis node, the direct parent of the Spline node, then use the Axis menu in the Distort tool to apply tracking data to the node.

If you want to affect both a source spline and its corresponding destination spline, you can use the Axis node at the top of the spline node tree.



When you add splines, they are initially parented to their own spline node tree. You can, however, add more Axis nodes to the schematic, and parent splines from different spline node trees to the same Axis node. This can be useful, for example, if you create splines for the different features of an actor's face and want them to move in tandem with the actor.

Axis nodes for corresponding splines can be unlinked from each other by selecting a spline's axis node, then selecting Enable Warping from the Link box.

Spline Nodes

Use the Spline node to select the spline itself, including all its points. Each spline added to the schematic results in a source and destination Spline node. Each Spline node is representative of a particular spline, which consists of a closed shape defined by vertices. If Link Src and Dst is selected from the Link Box, transformations applied to a source or destination spline also affect the corresponding spline. This does not apply when Front1 and Front2 source splines are linked for source interpolation. See Linking Source Nodes (page 1179).

When you add a spline, it is applied to the Front1 or Front2 clip, depending on what is selected in the Input box. You can then select it and change it from an Front1 to an Front2 spline, or from an Front2 to an Front1 spline. This changes the input for both the source and corresponding destination spline. See Toggling Spline Nodes from One Input to Another (page 1178).

You can add more Axis nodes to the schematic and link the spline nodes to them, for example, if you want to affect the children splines in the same way.

You can, for example, track all the points for the selected spline using the Stabilizer. You can also select the Spline node for a spline if you are having trouble distinguishing between splines in a complex setup.

TIP When using source interpolation, you can select either one of the linked spline nodes when you want to set the interpolation value between them.

Adding Spline Node Trees

When you add a spline, a spline node tree is added for Front1 or Front2. The spline node tree includes both Spline and Axis nodes.

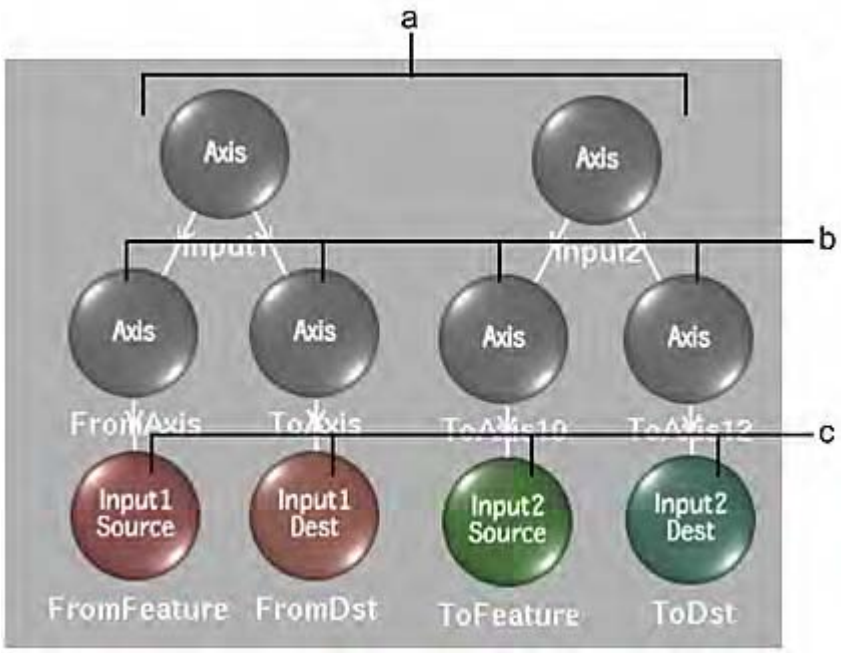
To add a spline node tree:

1. Select Front1 or Front2 from the Input box, depending on where you want to add splines.

   NOTE Splines can be changed from Front1 to Front2, or from Front2 to Front1. See Toggling Spline Nodes from One Input to Another (page 1178).

2. Select Spline from the Node box.
3. Click Add.
4. Draw a spline. See Drawing Splines (page 1166).

   A spline node tree is added for either Front1 or Front2 in the schematic. It includes source and destination spline nodes, each parented to their own Axis node, which are in turn parented to a common axis.



(a) Spline node trees for one Front1 (Input1), and one Front2 (Input2) spline (b) Axis nodes for individual spline wireframes (c) Spline nodes for corresponding source and destination splines

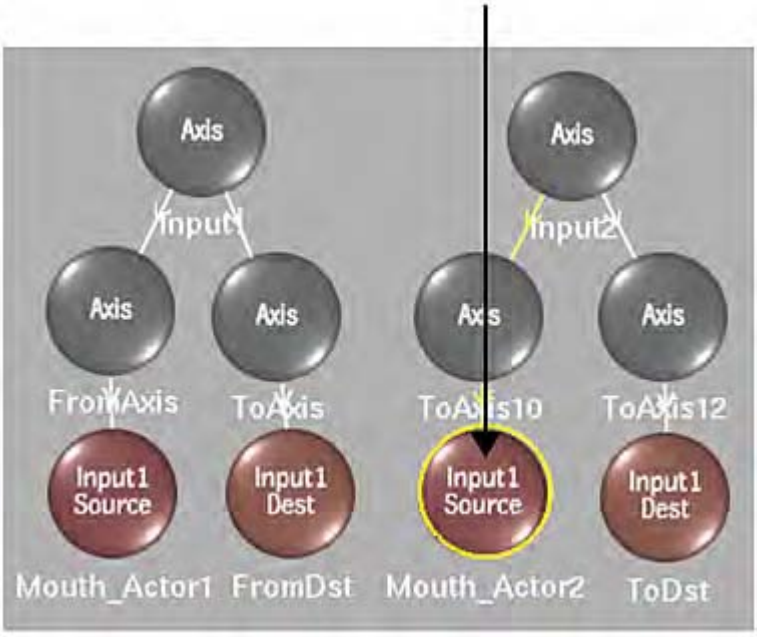
Toggling Spline Nodes from One Input to Another

Morphs work best when the source spline and destination spline have the same orientation—that is, were both drawn clockwise or counterclockwise—and the same number of vertices. You can ensure this is the case by drawing your spline on one input, then duplicating it to the other. To do this, first duplicate the spline, and then "move" the duplicate using the Morph menu's Toggle Input button. It toggles the selected spline from one input to the other.

TIP When morphing, you can duplicate splines and then toggle their input, which results in corresponding splines on the Front1 and Front2 clips that can then be linked for source interpolation.

To toggle a Spline node's input:

1 Select the source or corresponding destination Spline node you want to change from Front1 (Input1) to Front2 (Input2), or from Front2 (Input2) to Front1 (Input1).

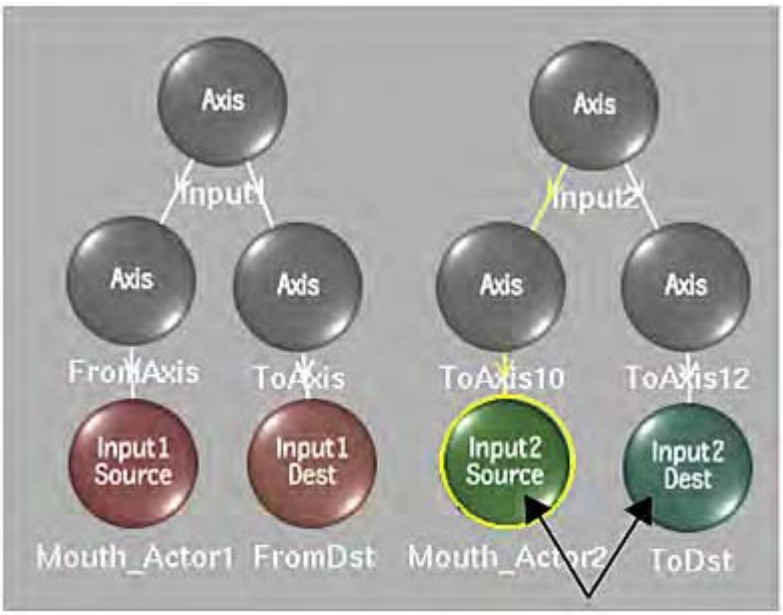


NOTE You can also select the spline directly in the clip.

2 Click Toggle Input.



The source and destination spline nodes change either from Front1 (Input1) to Front2 (Input2), or from Front2 (Input2) to Front1 (Input1).



Linking Source Nodes

You link a spline source node from Front1 to a spline source node from Front2 to create interpolation between them for a morph. Destination splines are no longer used and cleared from the schematic. The source of the Front2 spline node tree becomes the destination of the Front1 spline node tree.

You can create as many relationships between Front1 and Front2 spline nodes by linking them in the schematic, as there are features to correlate between the clips.

To link source nodes:

1. Select Link from the Edit Mode box.
2. Drag from an Front1 source spline node to an Front2 source spline node.

   NOTE You can only link a source spline from an Front1 to a source spline from Front2.

   The splines are linked to each other, as indicated by the blue dotted connector lines. The destination splines, and their respective Axis nodes, are removed from the two spline node trees. Once Spline nodes are linked, you can set an interpolation value to determine the percentage that the feature defined by the Front1 spline mixes with the feature defined by the Front2 spline. See Interpolating Features (page 1183).

To unlink nodes:

1. Select Link from the Edit Mode box.
2. Drag the cursor across the line that joins the two linked source spline nodes.

   The spline nodes are unlinked, and the destination splines and their axes reappear for each of the affected spline node trees.

Animating and Tracking Splines

If the image you are warping or morphing is in motion, it will be necessary to animate each spline so it moves with the feature of interest. To animate a spline, set keyframes for its axes or vertices at various places in the clip. The motion between the keyframes is automatically interpolated. Alternately, use the Stabilizer to automate the process by tracking a reference point and connecting an axis or vertex to the resulting tracking data. Whichever method you choose, you can further control the shape of the spline by adding or removing vertices throughout the animation.

NOTE You can also animate correspondence points to refine the mapping of the source spline to its destination over time. See Working with Correspondence Points (page 1185).

For example, if you are creating a warp, you can load the Front1 clip into the Stabilizer and track all of the source spline's vertices. With Enable Warping selected from the Link box, you can then manually edit the points of the destination splines along the course of the clip to create the desired warp.

Similarly, if you are creating a morph, you can track the vertices of both source splines on the Front1 clip, as well as the source splines on the Front2 clip. When you then link the splines together in the Distort schematic for source interpolation, the morph animation is already set. You can refine the morph by further animating the destination spline.

At any frame, you can control the shape of the spline by adding and removing vertices. If the feature you are tracking gets larger, for example, you can add vertices to adjust the shape of the spline. Similarly, you can remove vertices that are no longer needed. Make them inactive from one keyframe to the next using the Active button, or permanently delete them. See Manipulating Vertices and Tangents (page 903).

Animating Splines

Set keyframes for spline vertices and tangents over the course of the clip.

To animate a spline:

1. Move to the frame where you want to begin animating the spline.
2. If necessary, do one of the following to display the spline you want to animate:
   - From the Warp or Morph menu, click Source to display source splines, or click Destination to display destination splines. You can also use the Show buttons to display both source and destination splines.
   - If you are working with two clips, with either Input, Matte, or Result selected in the View box, toggle the Input box to Front1 or Front2, depending on which spline you want to animate.
3. Select the spline in the clip or its node in the Distort schematic.
4. Enable Auto Key in the Animation menu.
5. Select Move mode in the Edit box, then select and move one or more vertices to a new location. You can also move vertex tangents.
6. Advance the positioner to a different frame then move some more vertices.
7. Continue to move vertices and tangents as you progress through the clip, so the spline follows the movement of the feature you are warping or morphing.

Loading a Clip into the Stabilizer

Use the Stabilizer controls in the Warp menu or in the Morph menu to select the clip you want to track, the spline you want to apply the tracking data to, and the tracking mode (Axis or Shape). See How the Stabilizer Works (page 860).

Select from the Track Clip box:	To track:
Front1	The Front1 clip.
Front2	The Front2 clip.
Matte1	The Matte1 clip.
Matte2	The Matte2 clip.
Background	The background clip.

Tracking by Axis

To make the spline follow the movement in the clip without changing shape, apply the tracking data to the spline's axis.

To track by axis:

1. Set the Stabilizer controls to apply the tracking data to either the source (Src) or destination (Dst) spline for either the Front1 or Front2 clip. Select the Axis node in the Distort schematic that is the parent of the spline you want to track.
2. Select the clip you want to track from the Track Clip box.



3. Click Track Axis.

   The Stabilizer menu appears.
4. Position the tracker on the image. Tracker 1 controls the position. Tracker 2 controls the rotation or scale. See Working with Trackers (page 862).
5. To use two trackers, click Tracker 2, click Active, and position the second tracker on the image.
6. Click Analyse to generate the tracking data.
7. If necessary, fine-tune the analysis.

8 When you are satisfied with the tracking results, click Return to apply the results to the axis and return to Distort.

Tracking by Vertices

To make the spline follow the movement and shape change in the clip, apply the tracking data to the spline's vertices.

To track by vertices:

1 Select the spline's vertices that you want to affect. You can first select a spline by clicking its Spline node in the Distort schematic.

TIP Double-clicking a Spline node displays the Warp or Morph menu.

2 Set the Stabilizer controls to apply the tracking data to either the source (Src) or destination (Dst) spline for either the Front1 or Front2 clip.

3 Click Track Shape.



The Stabilizer menu appears. A tracker appears on each vertex that you selected. The reference frame is the frame from which you opened the Stabilizer.

4 If necessary, adjust the position of the trackers to ensure that each tracker has a good reference point.

5 Click Analyse to generate the tracking data, and if necessary, fine-tune the analysis.

For example, disable a problem tracker and re-analyse, as described in Tracking Difficult Shots and Correcting Errors (page 881).

6 When you are satisfied with the tracking results, click Return to apply the results and return to Distort.

Duplicating and Pasting Tracking Data

You can duplicate and paste tracking data from one mesh to another using the Channel Editor. For example, if tracking was done using Axis, you can duplicate the translation of the Front1 source spline and apply it to the Front2 source spline.

Example: To duplicate translation data:

1 Click Animation to display the Channel Editor.

2 Select the X and Y channels in the Input1 _src translate folder.

3 Click Duplicate.

4 Select the X and Y channels in the Input1_dst translate folder.

5 Click Paste.

The translation channels containing the tracking data from the Front1 source spline are pasted into the Front1 destination spline.

Blending Clips

When you morph features from an Front1 clip to features from an Front2 clip or a background clip, you add a blend to the Front1. By blending the clip over time, the Front1 dissolves into the Front2 or the background. Matching features, for example, the mouth of an actor on the Front1 clip, and the mouth of an actor on the Front2 clip, as defined by the corresponding source and destination splines, can correspondingly blend from one to the other as the second clip becomes visible.

To blend the Front1 clip into the Front2 clip or background:

1 From the Morph menu, adjust the value in the Blend box. A value of zero results in the Front1 clip being totally opaque, while a value of 100 results in the clip being completely transparent.



TIP The blend value you set can be animated over time, and is displayed in the Channel Editor by a transparency channel in the Morph folder.

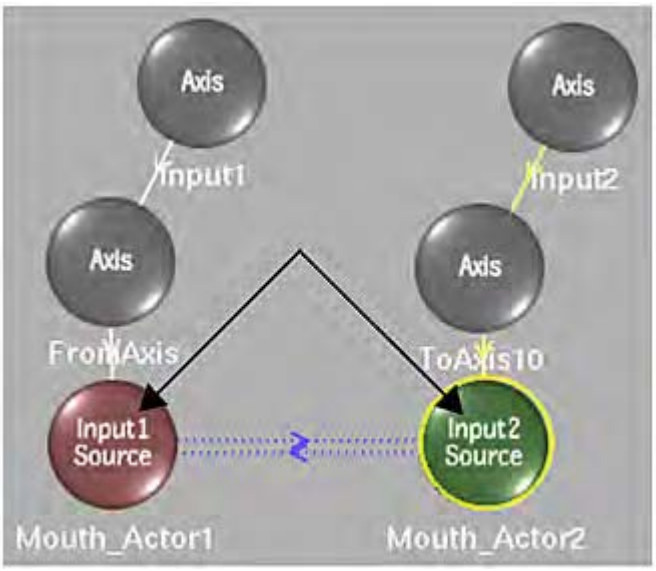
Interpolating Features

Set interpolation values for each set of Spline nodes you linked in the Distort schematic.

The interpolation value determines the percentage that the feature defined in the Front1 source spline mixes with the feature defined in the Front2 source spline.

To set the interpolation value between linked splines:

1 Select the Front1 (Input1) or Front2 (Input2) source Spline node of a linked pair in the Distort schematic.



The current interpolation value for the linked splines appears in the Interpolation field.



2 Set the value for the interpolation. A value of 0 results in the Front1 feature remaining in its original form, while a value of 100 results in the Front1 feature distorting into the shape defined by the Front2 spline.

Refining Distortions

There are numerous methods and techniques to refine your distortions. You can adjust the correspondence between source and destination splines using correspondence points if a feature from Front1 is not mapped correctly to the corresponding Front2 feature. You can work with the number of iterations to increase the warping effect, or limit the affected region of the clip. As well, you can adjust clip resolution to affect how well clips distort.



Working with Correspondence Points

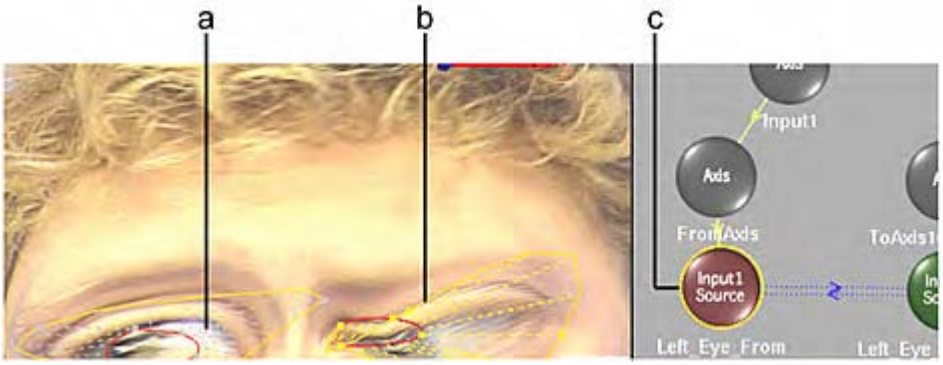
Correspondence points indicate how one frame is mapped to another in relation to source and destination splines, or Front1 and Front2 source splines when using source interpolation. If part of a clip is pulled or shifted too much during a distort, you can realign it by adjusting the correspondence points.

By default, correspondence points are added automatically. You can display the points, move them around the splines, animate them, as well as add and delete points to create a correspondence that better suits your needs. The more correspondence points a spline has, the more control you have over how the warp or morph is mapped.

To show correspondences:

1. Toggle on the visibility of the source splines and destination splines using the Show buttons.
2. In the Distort tool, click Correspondences.

   Correspondence points are shown on the source and destination splines, and are connected by dotted lines. Selected splines show the connections with yellow dotted lines; unselected splines shown them in grey.



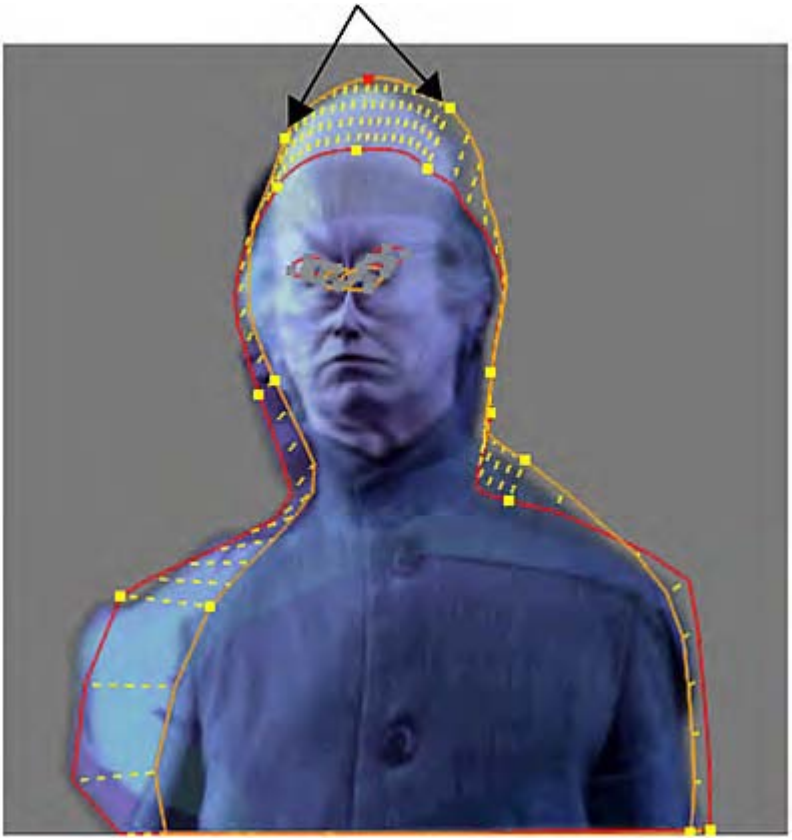
(a) Correspondences of unselected spline (grey) (b) Correspondences of selected spline (yellow) (c) Selected spline in Distort schematic

To move correspondence points:

1. Click Correspondences to display the correspondence points.
2. Select Move mode from the Edit box, then click and move a point on either the source or destination spline.

   You can also select multiple points by `Shift`-clicking them.

   Points can only be moved between the two points that surround it on the spline.



The correspondence point can only be moved between the two surrounding points

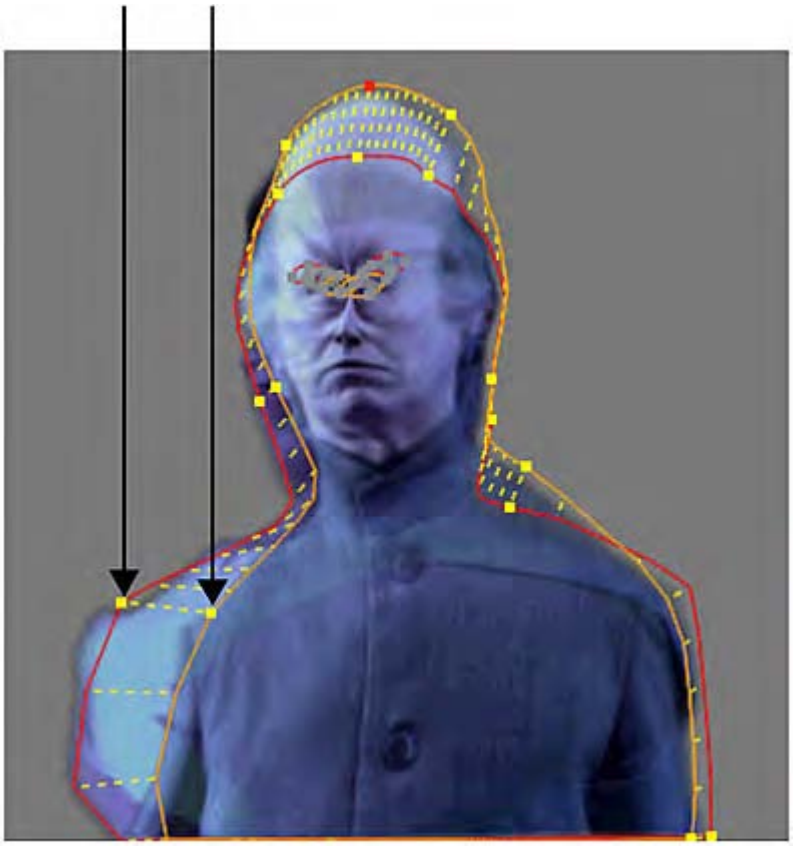
Image courtesy of Behavior Communications Inc.

The warped or morphed region is remapped according to the changes made to the correspondence points.

To add correspondence points:

1. Click Correspondences to display the correspondence points.
2. Select Add from the Edit Mode box.
3. In the region of the clip where you want to improve the mapping, click the source or destination spline to add a new correspondence point.

   A point is added. As well, a corresponding point is added to the corresponding spline. You can then move either of these points to improve the mapping of the clip. The more correspondence points there are, the more refined the distortion.



Adding a correspondence point to a source spline adds one to the destination spline

Image courtesy of Behavior Communications Inc.

To animate correspondence points:

1. Move to the frame where you want to begin animating correspondence points.
2. Display a source and destination spline by clicking the Source Show and Destination Show buttons.
3. Click Correspondences to display correspondence points.

   TIP The source and destination splines should be offset from one another to best be able to animate their correspondence points.

4. Enable Auto Key in the Distort menu.
5. Select Move mode in the Edit box, and then move one, or more, correspondence points to a new location.
6. Advance the positioner to a different frame and then move a correspondence point again.
7. Continue to move correspondence points as you progress through the clip to refine the warp or morph over time.

   Correspondence point animation for each spline is kept in a corresPts channel in the Channel Editor. See Distort Animation Channels (page 1191).

To delete correspondence points:

1. Click Correspondences to display the correspondence points.
2. Select Delete from the Edit Mode box.
3. Click a correspondence point on the source or destination spline.

The point you clicked and the corresponding point on the corresponding spline are deleted.

Adjusting Resolution

Resolution affects the sampling of the clips loaded into the Distort tool. The resolution value represents the number of pixel squares that the image is sampled at. Decrease the resolution value if you find your clips are not deforming smoothly.

NOTE Decreasing the resolution value slows down system performance.

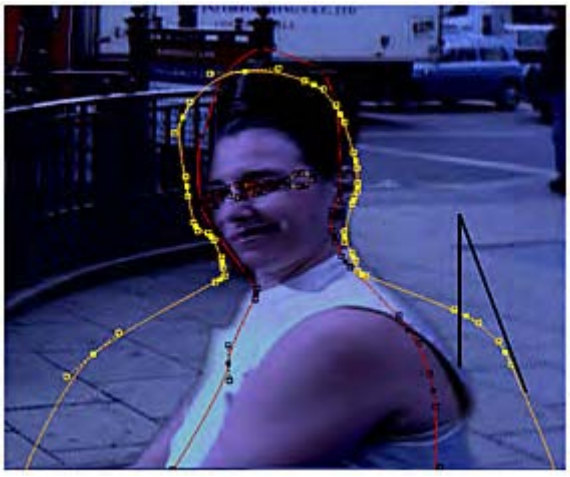
To adjust resolution:

1 Use the Resolution field to increase or decrease resolution. A smaller value increases the number of samples, producing a higher quality distortion.

Adjusting the Degree of Distortion

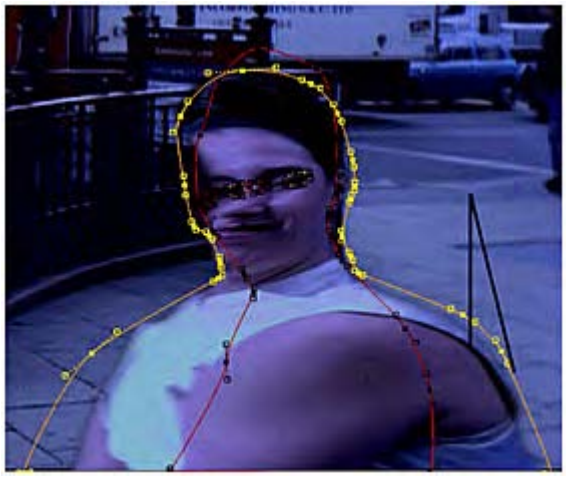
The Distort tool includes parameters that allow you to easily control the degree of distortion in a warp or morph. You can adjust the Iterations parameter so the distorted source pixels more closely approach the destination spline. Similarly, you can adjust the Range parameter to affect all the pixels in the image, or just those in the immediate region of the warp. Displaying the distortion grid can help you evaluate the effects of your changes.

To create a warp or morph, invisible iteration points are added between the source and destination splines. These iteration points determine how many timesthe pixels between the splines is examined when the calculations are performed. Adjusting the number of iteration points is relevant when the distance between the splines is great; that is, when you are trying to achieve a large warp. With a small iteration number, the warp will not “follow” the destination spline closely. Increasing the number of iterations will result in a more obvious distortion. For smaller warps or morphs, there is no advantage in using a greater number of iterations; it simply increases processing time unnecessarily.



With a low number of iterations, a distortion can sometimes be too subtle

Image courtesy of Behavior Communications Inc.



Increasing the number of iterations results in a more obvious distortion

Image courtesy of Behavior Communications Inc.

By adjusting the range, you can control the region affected by the distortion. A lower value decreases the region affected by the warp, while a higher one increases it. For example, a value of 100% affects all the pixels in the image, a value of 33% affects one-third of the image, a value of 25% affects one-quarter of it. To restrict the distortion to the area immediately surrounding the vertex, choose a higher region number. To broaden the effect, choose a lower number. This parameter has a smaller effect on processing time than the number of iteration points.

To display the distortion grid:

1 In Distort's Setup menu, enable Draw Grid.



To adjust the amount the distortion "follows" the destination spline:

1 In the Warp or Morph menu, change the value in the Iterations field.



To change the size of the region affected:

1 In the Warp or Morph menu, change the value in the Range field.



Filling in Gaps at Edges of Distorted Clips

As you warp surfaces, sometimes their edges are pulled away from the absolute edge of the image, revealing either the background clip, or blank space.

You can fill this revealed area with the warped clip by repeating it.



Without Fill

Image courtesy of Das Werk and The House



With Fill, set to Part

Image courtesy of Das Werk and The House

To fill in gaps at edges of a distorted clip:

1 Click Setup.



(a) Repeat button (b) Repeat value (c) Amount box

2 In the Texture Tiling section, Enable Repeat.

3 Toggle the Repeat value to set the number of times the clip is repeated to fill the gaps:
- Select 4 to repeat the clip once on each side: top, bottom, left, and right.
- Select 8 to repeat the clip twice on each side.

4 Toggle the Amount button to control the portion of the clip to use to fill the gaps:
- Select Full to squeeze the whole clip into each of the gaps.
- Select Partial to use only a mirrored corresponding, and proportional portion of the clip to fill each of the gaps.

Warping Example

You can create many warps on a single input clip. Each destination spline offset from its source creates a warp on the clip. The following is an example of a simple warp.

To create a warp:

1. Once a spline shape has been made, click Warp to display the Warp menu.
2. From the View box, display the Result view to see the spline shape over the input clip.
3. Set the timebar positioner to the effect's first frame.
4. From the Warp menu, enable Source.

   All Front1 source splines are displayed.
5. If necessary, adjust the vertices of the shape by moving points or curve handles.

   TIP To prevent the warping of the clip as you edit a spline, make sure Link Src & Dst is selected in the Link box. Otherwise, as you edit either a source or destination spline, it will be offset from its corresponding source or destination—this is what will eventually create your warp. You can toggle between Link Src & Dst and Enable Warp by pressing the `W` keyboard shortcut.
6. Set the timebar positioner to the effect's final frame.
7. From the Link box, select Distort.

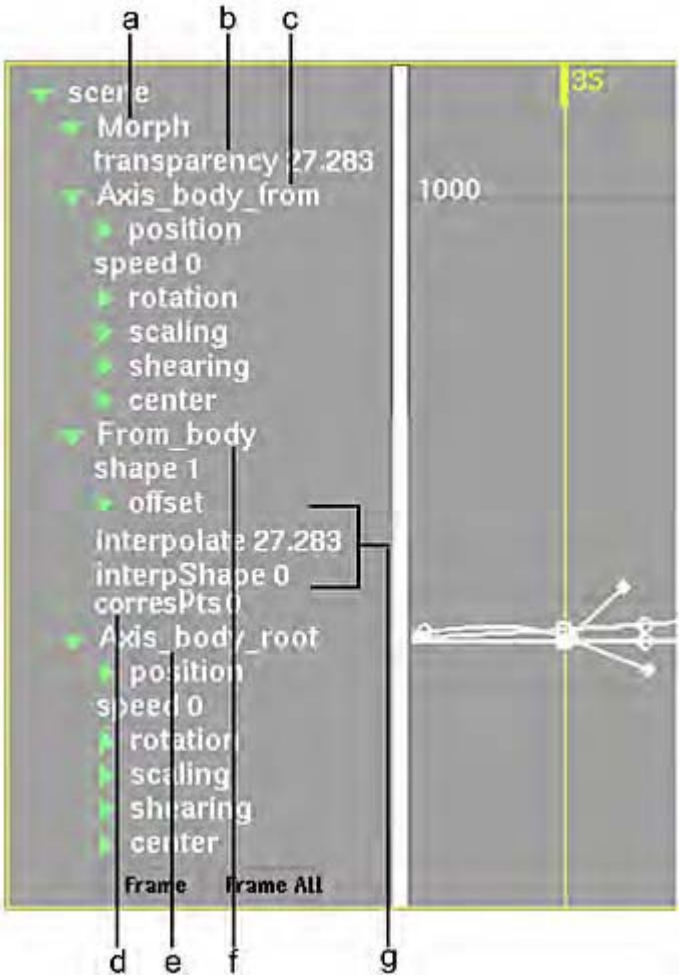


8. Enable Destination, and then adjust the vertices of the shape by moving points or curve handles. Edits to the destination spline-based shape will produce immediate warping.
9. Refine the warp as necessary. See Refining Distortions (page 1184).

Distort Animation Channels

The Distort tool includes channels for the Blend value as well as for all the Axis nodes in the schematic. Each spline in the effect contains a shape channel, with its vertex information, and an offset channel. As well, each spline contains a correspondence points channel containing point location information. All these channels are in the Morph folder in the Channel Editor.

When splines are linked, two additional channels appear under the Front1 spline's folder in the Channel Editor: an Interpolation channel corresponding to the value in the Interpolation field for the pair of linked spline nodes, and an interpShape channel. The interpShape channel contains the vertex information for the interpolated spline, which is the destination of the morph.

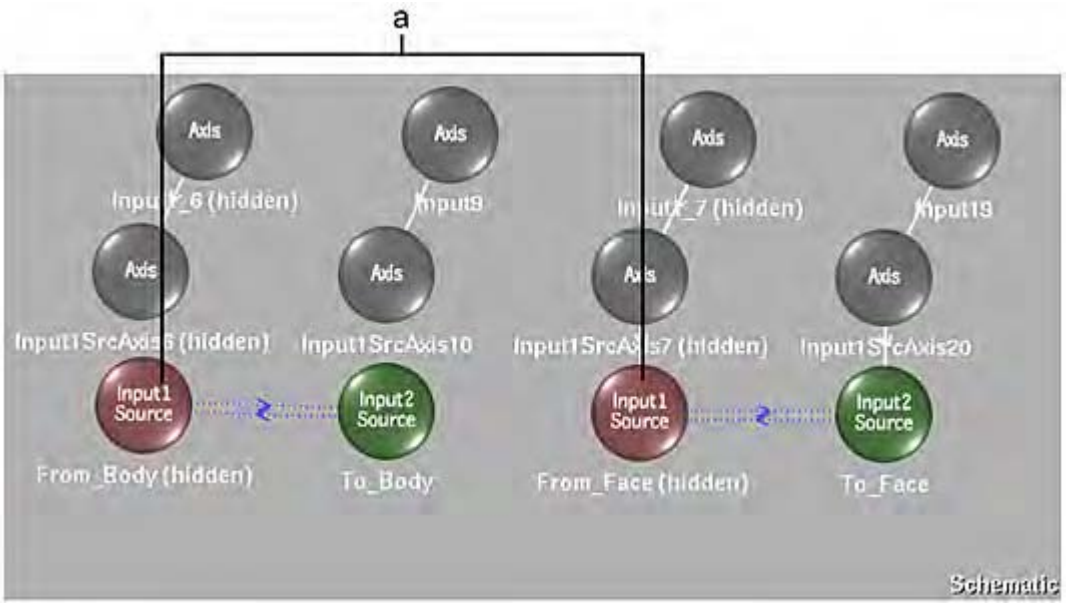


(a) Morph folder contains all folders and channels for both warps and morphs (b) Transparency channel corresponds to Blend parameter in Morph menu (c) Axis node folder for parent of Spline node (d) Correspondence points channel contains correspondence point information for the spline (e) Axis node folder for root of source and corresponding destination spline (f) Spline node folder for spline shape (g) Interpolation channels appear when spline nodes are linked in the schematic, under the Front1 (Input1) spline node's folder

Interpolation and InterpShape Channels

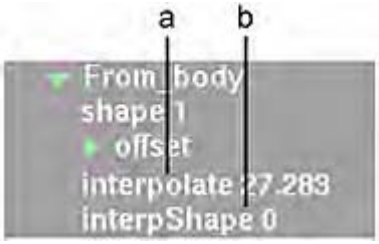
You can adjust the Interpolation and interpShape channels in the Channel Editor for each set of linked Front1 and Front2 splines to refine a morphing effect.

Each pair of linked Spline nodes produces an interpolated spline, which in turn, can be edited and animated independently.



(a) Interpolation and interpShape channels appear for linked spline nodes

The two interpolation channels, corresponding to linked spline nodes, are always in the Front1 source Spline node's folder in the Channel Editor.



(a) Corresponds to value in Interpolation field of Warp or Morph menu (b) Contains vertex information for interpolated spline



With Result selected in the View box, Front1 (Input1) selected in the Input box, and Source enabled, the source spline on Front1 (Input1) is displayed. It has a shape channel containing its vertex positions over time.

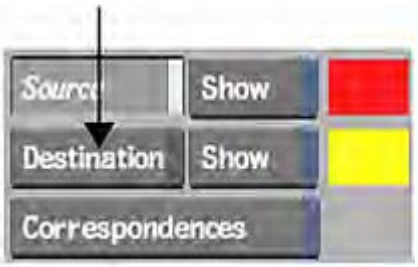


With Result selected in the View box, Front2 (Input2) selected in the Input box, and Source enabled, the source spline on Front2 (Input2) is displayed. It also has its own shape channel containing its vertex positions over time.

Image courtesy of Behavior Communications Inc.

Image courtesy of Behavior Communications Inc.

When an Front1 and Front2 spline are linked in the schematic, you select Result in the View box, Front2 in the Input box, and Destination to view the interpolated spline. The interpolated spline is the result of mixing the Front1 and Front2 source splines. Since it is the target of the morphing effect, you must click Destination to display it.



The interpolated spline's vertex information is contained in the interpShape channel in the corresponding Front1 spline's folder. Although the spline is a result of mixing the Front1 and Front2 source splines according to the Interpolation value, it can be controlled and animated independently.



(a) Morph follows contour of interpolated spline

Image courtesy of Behavior Communications Inc.

TIP Since each Interpolation channel, for every pair of linked Front1 and Front2 source splines, is independent, try using the Channel Editor's Link button on multiple Interpolation channels. This allows a common interpolation value for many sets of linked splines.

Using Distort in Batch

Use other nodes in combination with the Distort node in Batch to create more seamless warps and morphs that can be composited into media in Action.

Creating a Distortion to Feed into Action Media

You can create a morph effect in Distort, then feed the result, and the ResultMatte, into media in an Action node, to composite with additional media.

To create a distortion feeding into Action media:

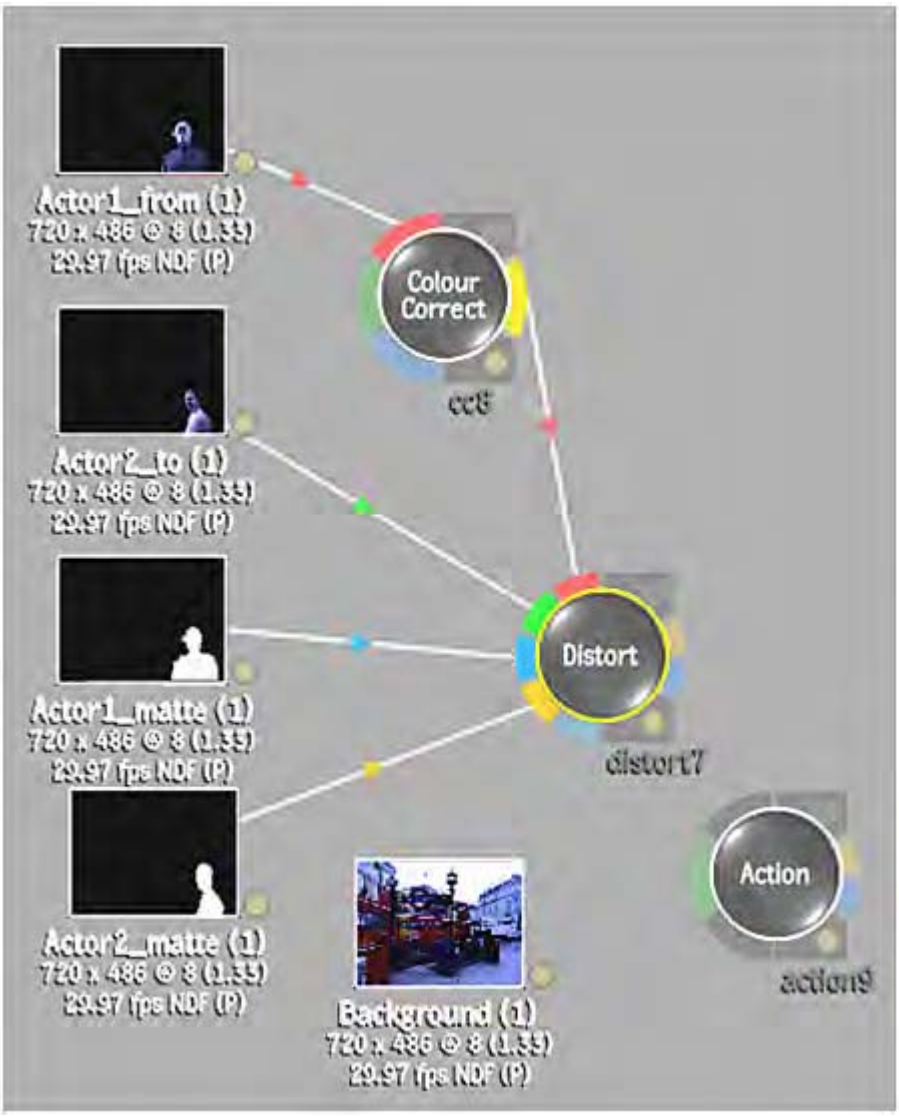
1. In Batch, add an Front1 clip and its corresponding matte, and an Front2 clip and its corresponding matte.
2. Add a background clip.



Images courtesy of Behavior Communications Inc.

NOTE The background clip will feed into the Action node, not the background socket of the Distort node.

3. Add a Distort node.
4. Add a Colour Corrector node.
5. Add an Action node.
6. Connect the Front1 clip to the Colour Corrector node.
7. Connect the Colour Corrector node to the Front1 socket of the Distort node.
8. Connect the other clips to their corresponding sockets on the Distort node: Front2, Matte1, and Matte2. There is no background connected to the background socket since Distort will feed into media of the Action node.

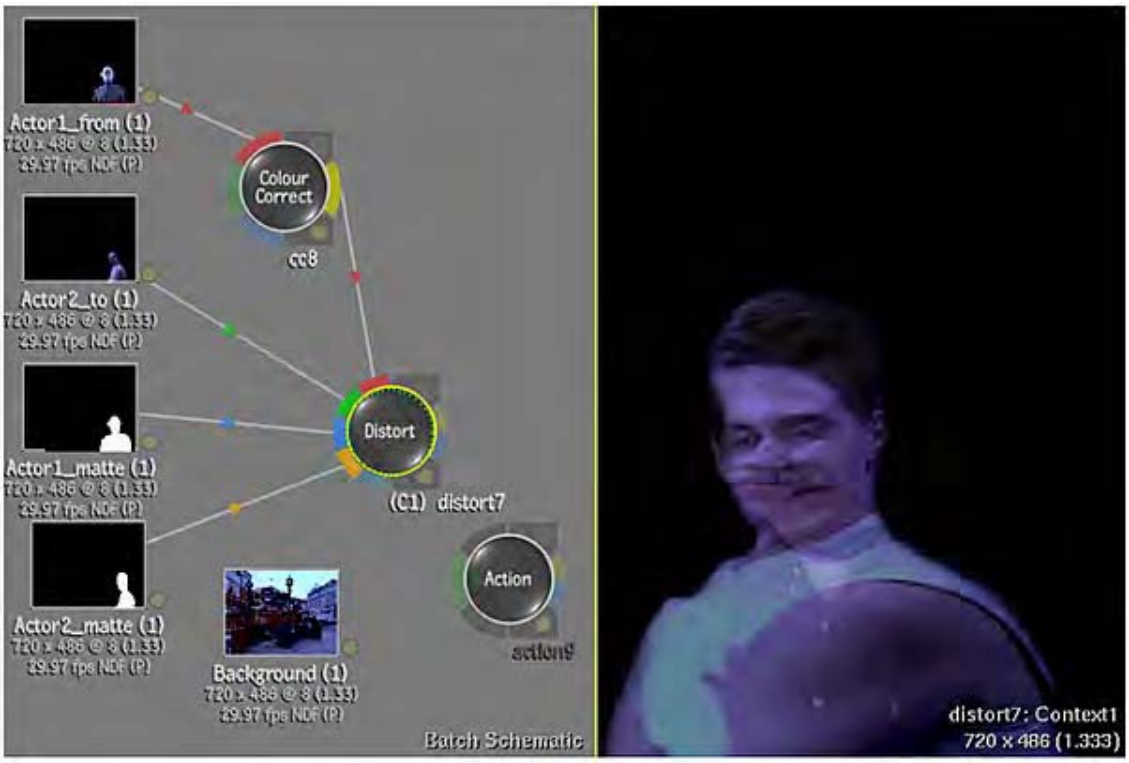


Images courtesy of Behavior Communications Inc.

9 Double-click the Distort node to load its controls into the work area.

10 Create the morph between the Front1 and Front2 clips.

11 Split the screen and create a context view for the Distort result and display it in a viewport.



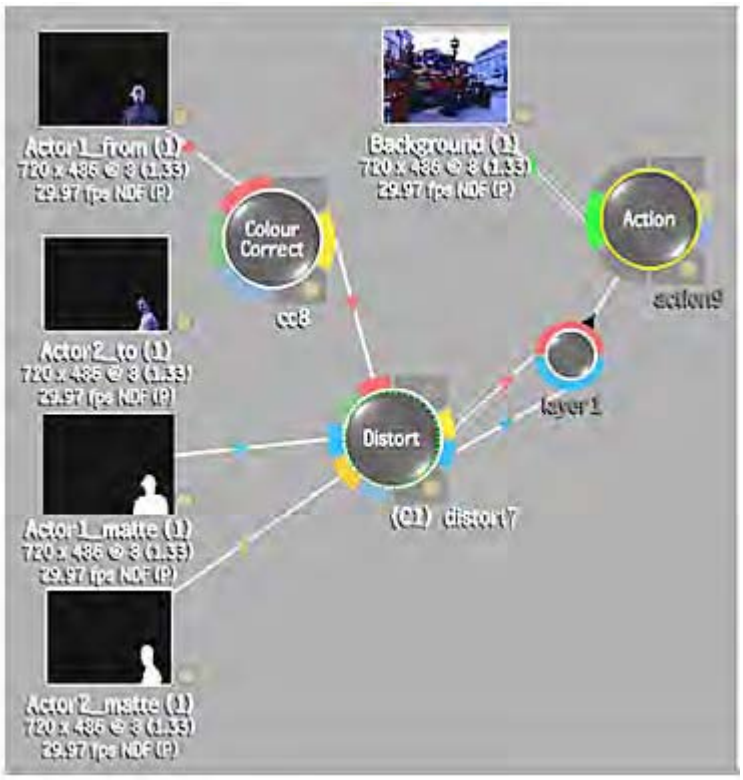
Images courtesy of Behavior Communications Inc.

12 In the other viewport, access the result of the Colour Corrector. Create an animated colour correction for the Front1 clip to gradually transition to the colours in the Front2 clip.

13 In the Action node, connect a background clip to it.

14 Within the Action node, `Ctrl`-click Add to add media that feeds into the Action node in the Batch schematic.

15 Feed the Result and the ResultMatte of Distort into the front and matte of the media, in turn feeding into Action.



Images courtesy of Behavior Communications Inc.

16 In Action, add a surface for the media.

17 Animate the media.

Saving Preferences

You can save preferences for the Distort tool and node.

The state of the following parameters are saved as preferences:

- Autokey
- Icons
- Icon Transparency
- Fill
- Source/Destination
- Show Destination
- Axis Motion Path
- Axis Scale Proportional

You can save preferences as a file you can reload, or as the tool's or node's defaults which are loaded when you reset all, or create new Distort nodes in Batch.

For more information on saving and loading preferences, and managing default preferences, see Setting Preferences (page 1671).

Saving and Loading Distort Setups

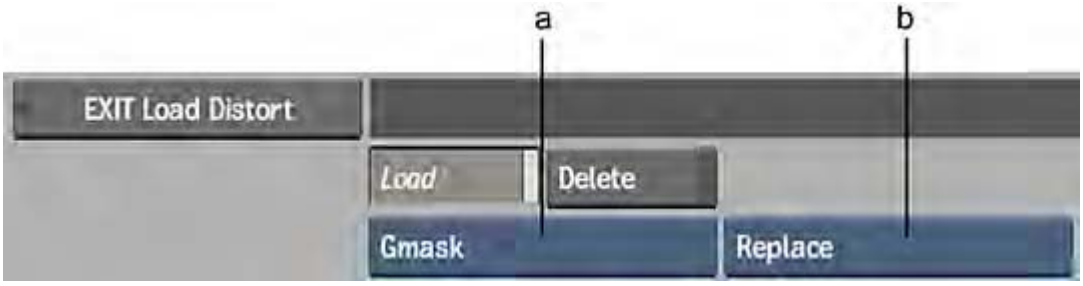
Distort setups can be saved and loaded as setup files, with spline and other information preserved. You can also exchange Distort setups between the Distort tool and the Distort node in Batch.

To save a Distort setup:

1. Do one of the following:
   - In the Distort menu, click Save to open the file browser.
   - From the Distort node in Batch, click NodeSetup to display Distort's Setup menu, then click Save Node to open the file browser.
2. Select Setup as the type of file to save.
3. Type the name of the setup, and click Save.

To load a Distort setup:

1. Do one of the following:
   - In the Distort menu, click Load to open the file browser.
   - From the Distort node in Batch, click NodeSetup to display Distort's Setup menu, then click Load Node to open the file browser.
2. From the Load Type box, select the type of files to load.



(a) Load Type box (b) Load Mode box

3. From the Load Mode box, select whether to want the setup to append to or replace any existing Distort splines in your scene.
4. From the file browser, select the name of the setup to load.

   NOTE If the Distort setup you are loading was created in a different resolution than the current project, click Scale Setup to scale the setup's splines.

About the Warper

Use the Warper to warp a clip or morph from a source clip to a result clip. Warping is free-form distortion of an image; it is a transition effect that matches the morphology of one clip gradually to that of another clip, such as when a human is morphing into an alien.

Accessing the Warper

You must select the required clips to access the Warper. You can enter the Warper with one to five clips. The number of clips that you load depends on the effect that you want to produce.

To access the Warper:

1. Select **Tools ➤ Filter ➤ Warper**.
   The cursor changes to Pick Front.
2. Select the types of clips you want to load into the Warper from the Input Mode box.

Select:	To load:	To produce:
Front1	Front clip only	A warp of a single image.
Front1/Front2	Front and back clips	A morph, a wipe, or a regional warp.
Front1/Matte1	Front and matte clips	A warp of a single image using a matte.
Front1/Front2/Matte1	Front, back, and matte clips	A warp over a background using a matte.
F1/F2/M1/M2/Back	Front, matte, back, back matte, and background clips	A morph using two clips and their mattes on a background clip instead of on black.

3. Select the front clip.
   The cursor changes to Render Here.
4. Click on any free (or a grey) area on the workspace.
   You are now in the Warper Tool.

   NOTE Regardless of which clips you load into the Warper, there are always two compositing layers available: a front layer and a back layer. If you do not load a back clip, black is used for the back layer.

To reset the Warper:

1. Click Reset All, and click Confirm when prompted.
   Warper options are restored to their default settings.

Setting Up the Clips

Use the Clip Setup boxes to specify how each clip loaded into the Warper will be used.



For Front and Back clips, the Clip Setup box contains On, Off, and Lock options. For Matte and Back Matte clips, the Clip Setup box contains On, Off, and Invert options. Background clips contain On and Off options.

Select:	To:
On	Warp the clip.
Off	Disable the clip. When a clip is disabled, it is not displayed and is not used in the processed result.
Lock	For Front and Back only, process the front or back clip in the final composite without warping it.
Invert	For Matte and Back Matte only, invert the front or back matte.

Activating Clips

The following table shows how to set up the clips to produce various effects.

Front1	Front2	Matte1	Matte2	Back	Effect
On	Off	Off	Off	Off	Warps the front clip only. This is the same as loading a front clip only.
On	On	Off	Off	Off	Morphs between two clips. This is the same as loading a front clip and a back clip.
On	Lock	Off	Off	Off	Wipes (page turns, rolls); regional warps (if the same clip is loaded as the front clip and the back clip).
On	Off	On	Off	Off	Warps the front clip and matte clip, composited on black.
On	On	On	Off	Off	Warps over a background with a matte clip.
On	Lock	On	Off	Off	Warps the front and matte clips composited on the back clip.
On	On	On	On	On	Morphs between two clips and their mattes onto the selected background clip.

Creating Meshes

After you load and set up your clips in the Warper, you create meshes to define the shape and position of the input image—the original image before the warp or morph—and the output image—the result of the warp or morph. The meshes use control points to determine the change, in shape and position, and create a convincing transition from the input image to the output image.

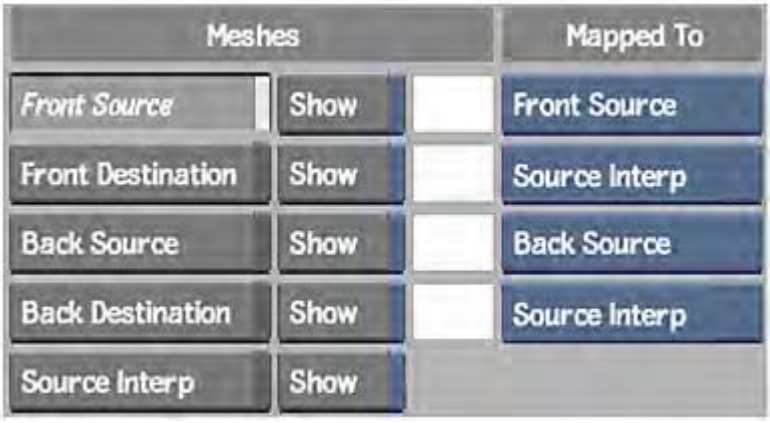
You use two meshes, a source mesh and a destination mesh, to generate the warp effect. For instance, if you want to morph a square into a circle, the source mesh defines the input image, which is a square, and the destination mesh defines the output image, which is a circle.

Defining a Mesh

When creating a mesh, you anchor the mesh to the surrounding area. To ensure this, create the mesh over a slightly larger area of the image than the part you want to affect. For example, to warp a person's mouth, extend the mesh to cover the lower half of the person's face. This “anchors” the area surrounding the warp so that it is not affected by the warp. You begin by defining the number of patches in the mesh and then placing it over the required area in the image.

To define a mesh:

1. Go to the frame where you want the effect to begin.
2. Specify which mesh will be created.



Click:	To create:
Front Source	The front source mesh.
Front Destination	The front destination mesh.
Back Source	The back source mesh in a morph.
Back Destination	The back destination mesh in a morph.

For information on when to create each type of mesh, see Warping (page 1214) and Morphing (page 1216).

3. Click Add Mesh.



4 In the Xcells and Ycells fields, set the number of patches for the mesh. For example, to create a mesh that is 10 patches wide by 5 patches high, enter 10 in the Xcells field and 5 in the Ycells field.

   Enable the Proportional button to automatically set the X and Y cells fields to the same value.

5 Click the image and drag the cursor to draw the mesh.

   TIP If you cannot see the mesh properly against the colours in the image, you can customize the colours of the various meshes using the colour pots beside each mesh name.

Adding Patches

As you build the mesh, you may need to add patches to cover more of the image.

To add patches:

1 In the Warper menu, click Add Mesh.
2 To add only one patch, click the edge of the patch that is closest to where you want to add a patch.

   The new patch is approximately the size of the adjacent patch.
3 To add several patches, drag from the edge of an existing patch over the area in which to add patches.

   Each new patch added is approximately the size of its adjacent patch.

Deleting Patches

You can delete patches if you make a mistake while creating the mesh.

To delete patches:

1 In the Warper menu, click Delete.



2 To delete a patch that lies along the boundary of the mesh, click the middle of the patch.
3 To delete a line that connects two patches that are not along the boundary, click both sides of the line.

Modifying a Mesh

Use the modelling controls in the Warper menu to modify the meshes to define the original and the final shape of the image. You can modify the shape and size of the patches or the location and orientation of the mesh, divide the patches into smaller subpatches, and split the edges into smaller segments.



NOTE The more detail you add to the mesh, the more control you have when warping or morphing a clip. However, if you have too much detail, the meshes may be difficult to modify.

If you cannot see the mesh properly against the colours in the image, you can customize the colours of the various meshes, as well as those of the splines, tangents, and vertices. See Display Settings (page 1222).

Moving Control Points, Edges, and Tangent Handles

You can manipulate control points and tangent handles on the mesh to add precision to your warp effect. Click the control points to access their tangent handles, then click the tangent handles to change the shape of a patch.

Click the Move button to move a vertex, an edge, or a tangent handle.

To:	Click:
Move a control point	The control point and drag. Note that the X, Y, and Z coordinates of the selected vertex appear in the corresponding fields in the menu.
Move an edge	Near the centre of the edge and drag.
Move a tangent handle	The corresponding vertex to display its tangents, then click the tangent handle and drag.

Selecting Multiple Control Points

Select multiple control points to translate, scale, or rotate all or part of the mesh while maintaining the spatial relationship between the selected control points.

To select multiple control points:

1 In Warper menu, click Select.



2 Hold down the `Ctrl` key and drag a selection box over the control points that you want to select.

NOTE To deselect individual points, click the points. To deselect all, click outside the area of selected control points.

Translating, Scaling, and Rotating the Mesh

Use the Translate, Scale, and Rotate options to change the position, size, and orientation of part or all of the mesh.

To translate, scale, or rotate the mesh:

1 Select the appropriate area of the mesh. See Selecting Multiple Control Points (page 1204).

2 In the Warper menu, select Translate, Scale, or Rotate.



3 Click Move.

4 Click one of the selected control points and drag.

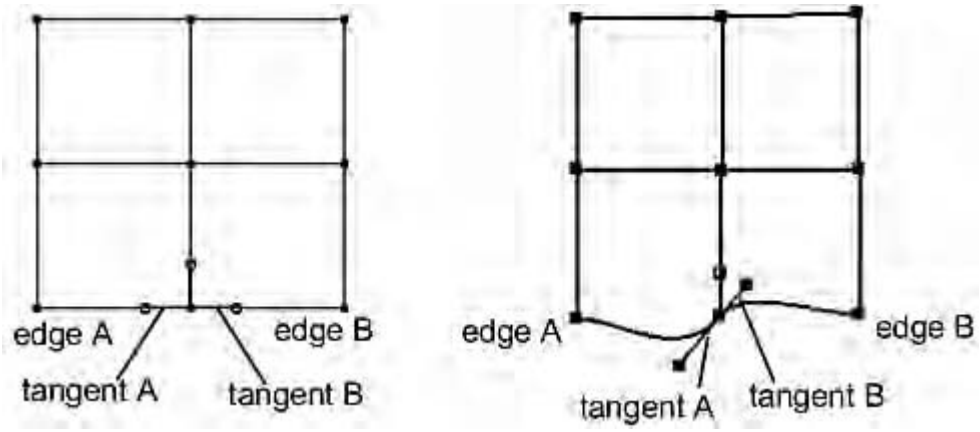
The selected area of the mesh is translated, scaled, or rotated.

NOTE You can also translate, scale, and rotate the entire mesh using the Translate, Scale, and Rotate channels in the Channel Editor. However, you cannot use the Channel Editor to affect only part of the mesh.

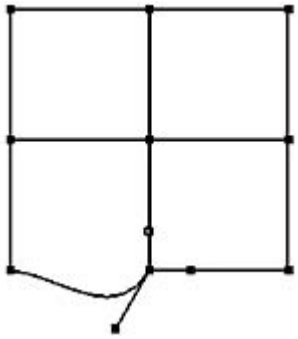
Breaking Tangents

By default, a vertex's tangents work together. Moving one tangent has the inverse effect on the opposite one to maintain a smooth curve at the vertex. In certain cases, you may want the tangents to move independently of each other.

Consider the mesh in the following example.



Suppose that you want to move tangent A to change the shape of edge A. Moving tangent handle A also causes tangent B to move in the opposite direction (as shown above); tangent A and tangent B are locked. You can use the Break command to break the tangents and move them independently (as shown here).



After using Break to break two tangents, you can use the Auto command to rejoin them. Note that resetting the tangents also resets the corresponding edges to their default positions.

To break tangent handles:

1. In the Warper menu, click Move.
2. Click a vertex to display its tangents.
3. Click Break, then click a tangent handle.

   The tangent and the opposite one can now be moved independently. The tangent handles are solid indicating they are broken.

   NOTE If the vertex has four tangents, click one handle of each pair to make all four tangents move independently.

To reset broken tangent handles:

1. In the Warper menu, click Move.
2. Click a vertex to display its tangents.
3. Click Auto, then click a tangent handle.
   The tangents and edges are reset.

Subdividing a Patch

You can divide patches in a mesh into smaller subpatches and split the edges into smaller segments using the Subdivide button.

To subdivide a patch:

1. In the Warper menu, click Subdivide.



2. Click the edge of a patch and drag the cursor to the opposite edge.

   NOTE To delete an edge created with Subdivide, enable Delete and click the edge.

Adding a Vertex

For more control when manipulating an edge, add a vertex to the edge.

To add a vertex:

1. In the Warper menu, click Split.



2 Click the edge of a patch where you want to add a vertex.

NOTE To delete a vertex created using the Split button, click Delete, then click the vertex.

Using the Freehand Mesh Tool

Use the Freehand Mesh tool to perform intricate mesh modelling.

To create a freehand mesh:

1 In the Warper menu, click Freehand.



2 Click the edge of a patch, and continue clicking until you complete the mesh.

3 `Ctrl`-click the final vertex.

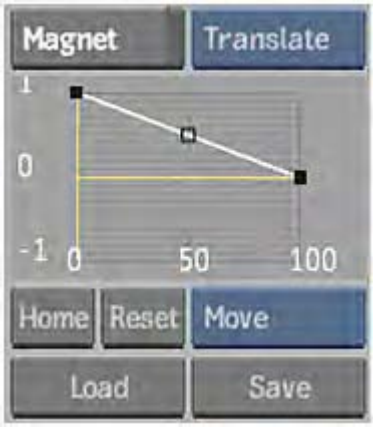
To delete a freehand mesh:

1 Click Delete.

2 Click the spline or vertex you want to delete, or click Undo.

Warping with the Magnet Tool

Use the Magnet tool to define the shape of the warped image. The Magnet tool warps the area of the mesh under the Magnet. You can set the size of the Magnet and define its effect on the selected area.



Note that the Magnet's effect is most noticeable when the selected area includes many control points (that is, when the grid has many patches, when you are using a large Magnet, or both).

To use the Magnet tool:

1. In the Warper menu, click Magnet.
2. Place the cursor over the mesh and adjust the Magnet size by pressing `Ctrl+S` and dragging.
3. Place the Magnet over the control points you want to warp and click.
   The affected control points turn red.
4. Drag the Magnet to produce the effect you want.

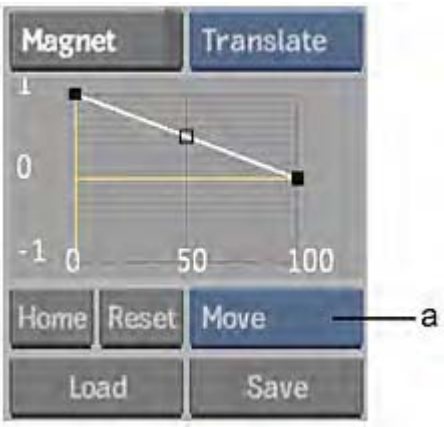
   NOTE By default, the Magnet translates the mesh. To scale or rotate the mesh, select the Scale or the Rotate option in the selection box beside the Magnet button.

Customizing the Magnet's Effect

You can customize the Magnet's effect by changing the shape of the curve in the Magnet Curve Editor. Note that, by default, the Magnet has the greatest effect on the pixels at the centre of the Magnet and the least effect on the pixels at its perimeter.

To customize the Magnet's effect:

1. In the Warper menu, click MEditor.
   The Magnet Curve Editor appears.



(a) Edit Mode box

2 Adjust the shape of the Magnet curve.
You can also use the Edit Mode options to modify the curve.

NOTE You can save and load custom Magnet settings using the Save and Load buttons in the Magnet Curve Editor.

Tips and Tricks

You can experiment with the Magnet curve to achieve different effects.



(a) Point representing effect at the Magnet's centre (b) Point representing effect at the Magnet's perimeter

- Invert the curve to affect the pixels at the perimeter of the Magnet more than those at the centre.
- Flatten the curve along the horizontal axis to have the same effect on all pixels under the Magnet.
- Add extra control points and create a sine or wave curve to make the control points move in opposite directions.
- Move the curve into the negative horizontal axis to create a magnet with an opposing magnetic effect.

Modifying the Meshes for Live-Action Clips

If you are warping or morphing a live-action or moving clip, you must define the source mesh at several frames in the clip. The Warper interpolates the location and shape of the mesh between frames.

Creating a mesh over the length of a live-action or moving clip can be time-consuming. To simplify the mesh-building process, create the mesh for the first frame of the required clip, then use the Stabilizer to generate the successive meshes.

NOTE You can track the source or destination mesh for either the front or the back clip.

With the Stabilizer, you can track the movement in the clip, and then apply the tracking data to the mesh (tracking by Axis). This translates the mesh so that it follows the movement in the clip without changing shape. Alternatively, you can track the movement of selected points in the image, and apply the tracking data to the mesh (tracking by Vertices). This translates and changes the shape of the mesh so that it matches the position and shape of the image over the length of the clip.

Loading a Clip into the Stabilizer

Use the Stabilizer controls in the Warp menu (if you are warping) or in the Morph menu (if you are morphing) to select the clip you want to track, the mesh you want to apply the tracking data to, and the tracking mode.



Select:	To:
Front	Track the front clip.
Back	Track the back clip.
Axis	Apply the tracking data to the axis of the mesh. You can use one or two trackers in the Stabilizer.
Vertices	Apply the tracking data to the mesh control points (vertices). You must select the control points you want to track before entering the Stabilizer. You can track up to 1000 control points. Note that tracking by vertices sets a shape keyframe at every frame of the clip.
Source	Apply the tracking data to the source mesh of the selected clip.
Destination	Apply the tracking data to the destination mesh of the selected clip. This option can be useful if front and back destination meshes are different.

Tracking by Axis

To make the mesh follow the movement in the clip without changing shape, apply the tracking data to the mesh axis.

To track by axis:

1. Set the Stabilizer controls to apply the tracking data to either the Source or Destination mesh for either the Front or Back clip.
2. Select Axis.
3. Click the Stabilizer button.

   The Stabilizer menu appears.
4. Position the tracker on the image. See Working with Trackers (page 862).
5. To use two trackers, click Tracker 2, click Active, and position the second tracker on the image.
6. Click Analyse to generate the tracking data.
7. If necessary, fine-tune the analysis.
8. When you are satisfied with the tracking results, click Return to apply the results and return to the Warper.

Tracking by Vertices

To make the mesh follow the movement and shape change in the clip, apply the tracking data to the mesh vertices.

To track by vertices:

1. Select the mesh control points that you want to affect. See Selecting Multiple Control Points (page 1204).
2. Set the Stabilizer controls to apply the tracking data to either the Source or Destination mesh for either the Front or Back clip.
3. Select Vertices.
4. Click the Stabilizer button.

   The Stabilizer menu appears. A tracker appears on each vertex that you selected. The reference frame is the frame from which you opened the Stabilizer.
5. If necessary, adjust the position of the trackers to ensure that each tracker has a good reference point.
6. Click Analyse to generate the tracking data, and if necessary, fine-tune the analysis.

   For example, disable a problem tracker and re-analyse, as described in Tracking Difficult Shots and Correcting Errors (page 881).
7. When you are satisfied with the tracking results, click Return to apply the results and return to the Warper.

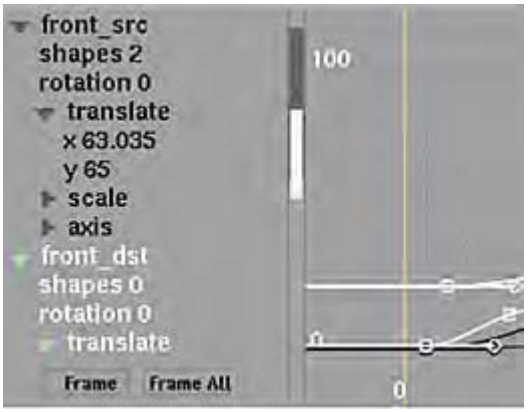
Copying and Pasting Tracking Data

You can copy and paste tracking data from one mesh to another using the Channel Editor. For example, if tracking was done using Axis, you can copy the translation of the front source mesh and apply it to the back source mesh.

To copy translation data:

1. Click Animation to view the Channel Editor.
2. Select the X and Y channels in the front_src translate folder.
3. Click Copy.
4. Select the X and Y channels in the front_dst translate folder.
5. Click Paste.

   The translation channels containing the tracking data from the front source mesh are pasted into the front destination mesh.



If tracking was done using Vertices, you can copy the shape animation of the front source mesh and apply it to the front destination mesh.

Example: To copy shape animation data:

1. Select the front_src shapes channel.
2. Click Copy.
3. Select the front_dst shapes channel.
4. Click Paste.

   The shapes channel containing the tracking data from the front source mesh is pasted into the front destination mesh.

Copying and Pasting Meshes for a Live-Action Clip

For a live-action clip, use the Channel Editor to copy the shapes curve of one mesh and paste it into another mesh. The shape and translation of the mesh at each frame are copied.

To copy and paste meshes for a live-action clip:

1. Create the mesh at the first frame of a clip and modify it over several frames to define the movement in the clip.
2. Click Animation to view the Channel Editor.
3. Select the shapes channel for the mesh you are copying (front_src, front_dst, back_src, or back_dst).
4. Click Copy.
5. Select the shapes channel for the mesh that you want to be the same as the copied mesh (front_src, front_dst, back_src, or back_dst).
6. Click Paste.

   NOTE If Auto Key in the Setup menu is enabled, a keyframe is created in the associated animation channel at the current frame every time the mesh is copied to a new frame and modified. When you copy the shapes curve to a new mesh, you copy the mesh and all its keyframes.

Copying and Pasting Meshes for a Still Clip

For a still clip, use the Copy and Paste buttons in the Warper menu to copy a mesh and paste it into another mesh.

To copy and paste meshes for a still clip:

1. Create the mesh at the first frame of the clip.
2. Click Copy in the Warper menu to copy the mesh—including its translation, scale, and rotation values.
3. Go to the required frame and display the mesh (source or destination) where you want to paste the copy.
4. Click Paste.

Warping

Warping changes the shape of the image in a clip. When you warp an image, you start by drawing a mesh over the region of the image you want to warp. You then modify the mesh over the image at specific keyframes to distort the image.

To create the warp, define a source mesh and a destination mesh for the front clip. The source mesh defines the area of the image that you want to warp; it outlines the shape of the non-distorted image in each frame of the clip. The destination mesh defines what the warped image looks like; it outlines the shape of the distorted image in each frame of the clip. Typically, you set the destination mesh at several frames of the clip. The Warper generates a smooth transition between frames.

The general steps for warping:

1. Load the appropriate clips into the Warper. If you are warping just a region of the clip, load the same clip as the Front and the Back clip. See Warping a Region of a Clip (page 1216).
2. In the Warper menu, click Warp.
3. Set up the clips. See Setting Up the Clips (page 1200). In general:
   - Set Front to On.
   - If you are warping just a region of the clip, set Back to Lock.
4. Click Front to view the front clip.
5. Define the source mesh for the original image.
6. Define the destination mesh for the warped image.
7. Process the clip.

Defining the Source Mesh

Use the modelling tools to define the source mesh so that it outlines the original shape of the image in the front clip.

To define the source mesh for the original image:

1. Go to the first frame of the clip.
2. Click Front Source to define the source mesh for the front clip.
3. Add a mesh over the area to warp and modify its shape to match the original image.

4 If you are using a live-action or moving clip, the source mesh must follow the movement of the image. You can either modify the mesh manually throughout the clip, or use the Stabilizer to track the movement in the clip. See Modifying the Meshes for Live-Action Clips (page 1210).

Defining the Destination Mesh

Set the destination mesh to be the same as the source mesh at the first frame, then change the shape of the destination mesh at different frames of the front clip to outline the required shape of the distorted image.

To define the destination mesh for the warped image:

1 Click Front Source.
2 Go to the first frame in the clip.
3 Click Copy to copy the source mesh.
4 Click Front Destination to select the destination mesh for the front clip.
5 Click Paste to paste the copied mesh into the destination mesh.
6 At several frames in the clip, modify the destination mesh to outline the required shape of the warped image.

TIP In the view box, select Result to see the pixels move as you move the mesh points.

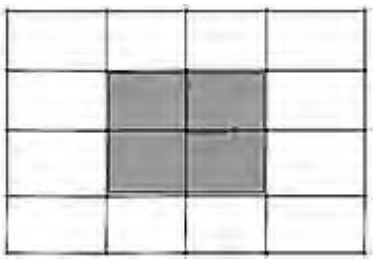
7 When you finish modifying the meshes, go to frame 1 and click Process to process the clip.

Warping a Still Clip

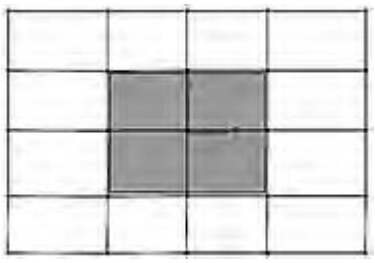
To warp a still clip, you define the source mesh only for the first frame of the clip since the object to warp has the same shape and location in each frame of the clip. Typically, you have to define the destination mesh only for the first and the last frames of the clip. The Warper generates a smooth transition between the two frames.

The following illustration shows the source and destination meshes used to warp a square into an oval.

Source Mesh:

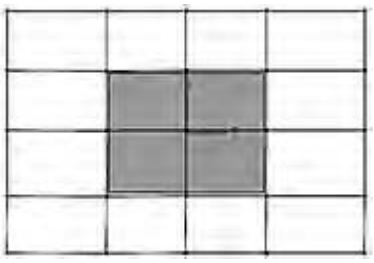


First Frame

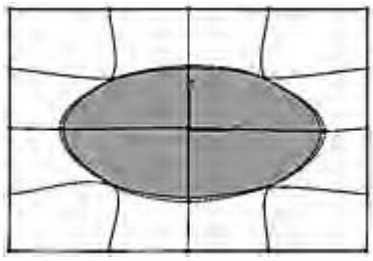


Last Frame

Destination Mesh:



First Frame



Last Frame

Warping a Region of a Clip

When a clip is processed, only the area defined by the mesh is processed. If you want to warp only a region in an image (known as regional warping), but you want to see the entire image in the result, you must load the same clip as the front and the back clip, and then lock the back clip.

Morphing

Morphing gradually transforms an image in a front clip into an image in a back clip. The effect is achieved by warping the two images and dissolving between the front and back clips. Unlike warping, which requires only a front source mesh and a front destination mesh, morphing requires a source mesh and a destination mesh for both the front clip and the back clip.

The front source mesh defines the original shape of the image in each frame of the front clip. Similarly, the back source mesh defines the original shape of the image in each frame of the back clip.

NOTE You should create the mesh over a slightly larger area of the image than just the part you want to affect. See Defining a Mesh (page 1202).

Both the front and the back destination meshes correspond to the warped image. Since the front clip transforms into the back clip:

- At the first frame, the front and back destination meshes correspond to the shape of the front clip.
- At the last frame, the front and back destination meshes correspond to the shape of the back clip.

The general steps for morphing:

1. Load the appropriate clips into the Warper. You need to load at least a front clip and a back clip.
2. In the Warper menu, click Morph.
3. Set up the clips (see Setting Up the Clips (page 1200)). In general:
   - Set Front to On.
   - Set Back to On.
4. Click Front to view the front clip.
5. Define the front source mesh to match the shape of the front clip at each frame.
6. Define the back source mesh to match the shape of the back clip at each frame.
7. Define the front and back destination meshes. You can either use Source interpolation to automatically modify the destination meshes or copy and paste the meshes and set the keyframes manually.

8 When you finish modifying the meshes, go to frame 1 and click Process to process the clip.

Defining the Front Source Mesh

The front source mesh defines the original shape of the image in each frame of the front clip.

To define the source mesh for the front clip:

1 Go to the first frame of the clip.
2 In the Morph menu, click Front Source.
3 Add a mesh over the area to be morphed and modify its shape to match the image.
4 Make the front source mesh match the image at each frame of the front clip. You can either modify the mesh manually or use the Stabilizer to track the movement in the front clip.

Defining the Back Source Mesh

The back source mesh always contains the same number of control points and patches as the front source mesh. Copy the front source mesh to use as a starting point, then adapt the mesh shape to the image at each frame of the back clip.

If you are using the Stabilizer and you need more detail for the back source mesh, add subdivisions to the mesh. Be sure to add the same subdivisions to the front source mesh.

NOTE Enable the Fixed button in the Stabilizer menu.

To define the source mesh for the back clip:

1 Go to the first frame of the clip.
2 Click Front Source, then click Copy to copy the front source mesh.
3 In the Morph menu, click Back Source to select the back source mesh.
4 Click Paste to paste the copied mesh into the back source mesh.
5 Click Back to view the back clip.
6 Modify the back source mesh to match the image in the back clip at each frame. You can either modify the mesh manually or use the Stabilizer to track the movement in the back clip.

Using Source Interpolation

Use source interpolation to automatically define the destination meshes. Source interpolation automatically creates the transition from the front source mesh at the first frame to the back source mesh at the last frame of the clip.

NOTE If you want to add another effect, such as scaling, during the morph, do not use source interpolation. You must define the front and back destination meshes manually at several frames. See Setting Keyframes Manually (page 1218).

Dissolve and Interpolation Curves

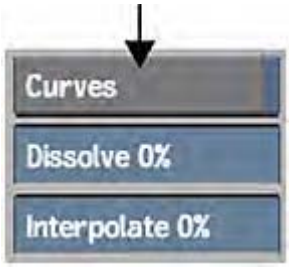
Source interpolation uses two predefined animation curves—dissolve and interpolation—to calculate the percentage change from the front source mesh to the back source mesh over the length of the clip. These values are set in the Dissolve and Interpolate fields:

- The Dissolve field defines the percentage change in the colour of the pixels at the current frame.
- The Interpolate field defines the percentage change in the position of the pixels at the current frame.

The default values in these fields are 0% at the first frame and 100% at the last frame. Using the default values results in a smooth dissolve from the pixels in the front image to those in the back image, as well as a smooth transition from the shape of the front image to that of the back image. You can change the rate of the morph by changing the interpolation and dissolve values.

To use source interpolation:

1. Click Morph to display the Morph menu.
2. Under Mapped To, select Source Interp to use source interpolation for the Front Destination mesh and Back Destination mesh.
3. Enable Curves to activate the interpolation and dissolve curves.



4. In the View box, select Result to view the result, and play the clip.

To change the rate of the morph:

1. In the Warper menu, click Animation to display the Channel Editor.
2. Select the dissolve or the interpolate channel.
3. Modify the curve.

Setting Keyframes Manually

If you are not satisfied with the source interpolation result, you can set the keyframes manually. However, you will have to set the keyframes throughout the clip—you cannot use source interpolation in conjunction with manual keyframes.

To set a keyframe manually:

1. Go to the frame corresponding to the keyframe you want to set.
2. Click Front Destination to display the front destination mesh.
3. Modify the mesh.
4. Copy the front destination mesh and paste it into the back destination mesh for that frame.

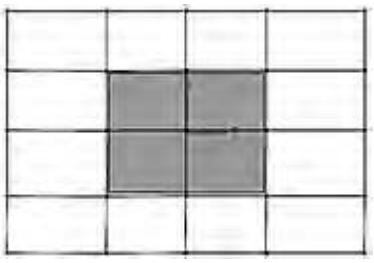
   Alternatively, if you want the back destination mesh to always match the front destination mesh, select the Front Destination under Mapped To adjacent to the Back Destination Meshes button.

Morphing Between Two Still Clips

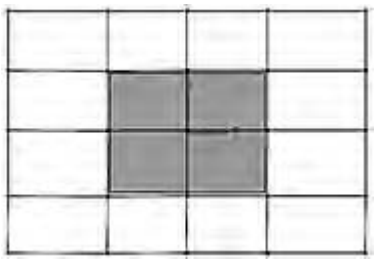
To morph between two still clips, you define the source meshes only for the first frame of the front and back clips (since the objects have the same shape and location in each frame of the clips). You can use source interpolation to create a smooth morph.

The following illustration shows the source and destination meshes needed to morph a square into a circle.

Front Source Mesh:

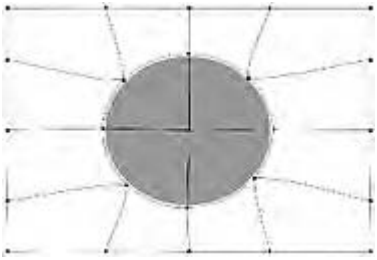


First Frame

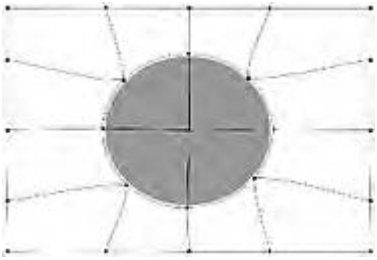


Last Frame

Back Source Mesh:

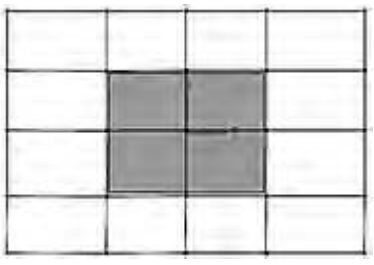


First Frame

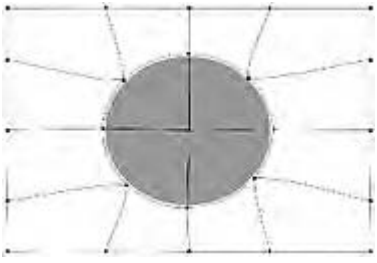


Last Frame

Front and Back Destination Meshes:



First Frame



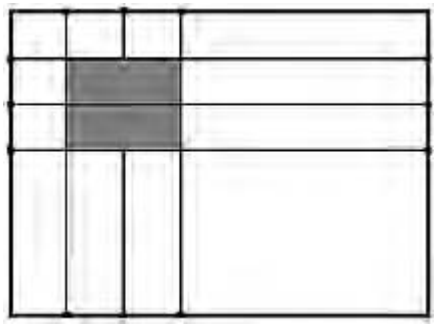
Last Frame

Morphing Between Two Live-Action Clips

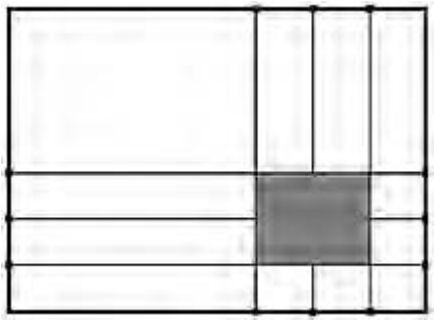
To morph between two live-action clips, you have to define the source and destination meshes for the front and back clips for the duration of the clips—since the objects change shape and location in each frame of the clips. You can use the Stabilizer to track the movement of the objects and source interpolation to create a smooth morph.

The following illustration shows the source and destination meshes needed to morph a square into a circle, both moving from the upper-left to the lower-right corner of the image window.

Front Source Mesh:

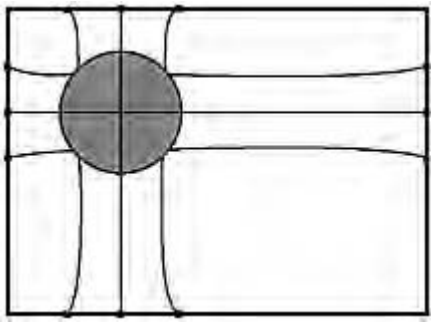


First Frame

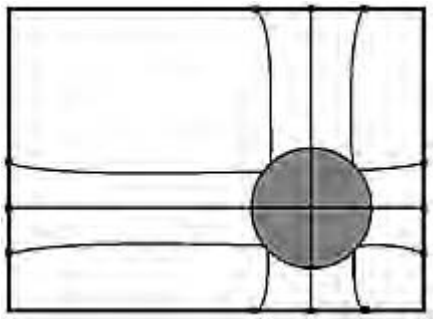


Last Frame

Back Source Mesh:

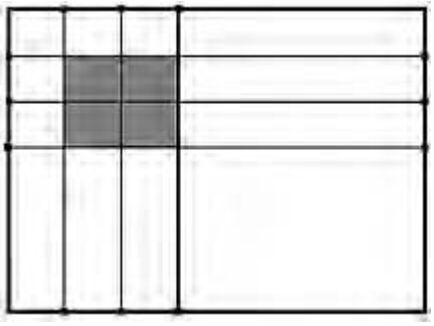


First Frame

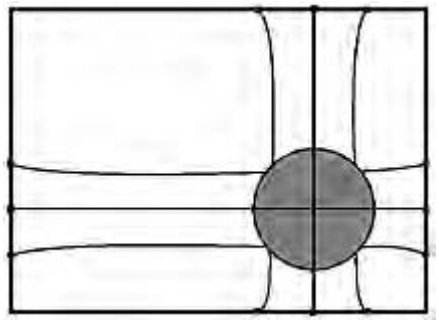


Last Frame

Front Destination Mesh:

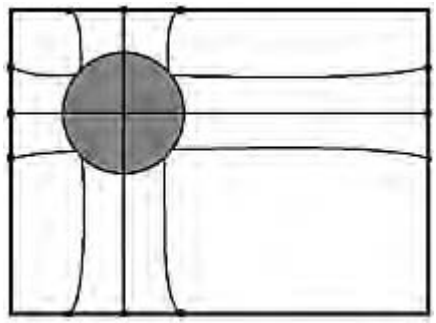


First Frame

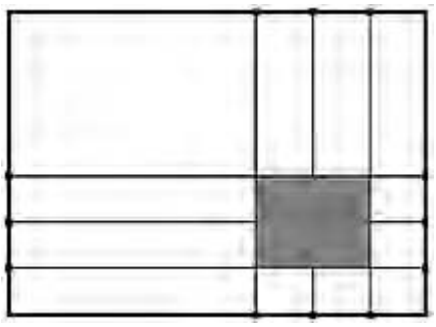


Last Frame

Back Destination Mesh:



First Frame



Last Frame

Setup Options

From the Setup menu, you can customize display colours and set processing options.



Rendering Settings

Use the Rendering settings to improve the final output quality of your image.

Texture Settings

Use the following controls to choose between hardware texture-mapping or polygons for processing.

Repeat Specifies how pixels are handled when a source mesh goes outside of the borders of an image. When this button is disabled, anything outside of the image is considered black. When the button is enabled, the image is repeated. TexRepeat is disabled by default.

Warper Settings

Use these settings to define general warper properties.

Slip Settings

Slips a clip by a specified number of frames. Click in the field for the clip you want to slip and enter the number of frames. The first frame of the clip will be repeated by the number specified to result in a delayed start.

Select:	**To slip:**
Front	The front clip.
Back	The back clip.
Matte	The front matte.
BMatte	The back matte.

Display Settings

Enable the Icons button to display splines, tangents, and vertices in the image. You can also customize the colours of splines, tangents, and vertices using the colour pot next to each button.

Effects and Tools Reference

25

This section provides information about the effects and tools that you can use on your media.

2D Histogram

Use the 2D Histogram to display the luminance distribution of the matte.



To access the 2D Histogram menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

2D Histogram Menu Settings



Lift field Displays the value added to the resulting pixels to create the final matte. Editable.

Gain field Displays the value that the resulting pixel values are multiplied by to create the final matte. Editable.

Histogram Displays a bar graph that is used to adjust the luminance values of the image. In the Ranges menu, the curves for the three luma ranges are also displayed.

Minimum Input field Displays the lower limit of the luminance values. Pixels with lower values are mapped to black.

Minimum Output field Displays the lower limit of the luminance values for black pixels.

Maximum Input field Displays the upper limit of the luminance values. Pixels with higher values are mapped to white.

Maximum Output field Displays the upper limit of the luminance values for white pixels.

Out Range box Select a curve that is constant (Clamp) or linear (No Clamp) before the first point of the curve and after the last point of the curve.

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

Frame Selection box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Destination button Enable to show a histogram of the colour values in the result or destination clip.

The destination colour values are obtained from the current frame. If you input both front and matte clips, the colour values in the front clip that are defined by the white part of the matte are displayed.

Reset button Resets to default histogram settings.

Zoom field Displays the vertical zoom value of the histogram.

You can also zoom horizontally by pressing `Ctrl+spacebar` and dragging left or right in the histogram. To pan horizontally, click `spacebar` and drag left or right in the histogram.

2D Transform

Use 2D Transform to apply basic axis transformations and camera shake effects to clips.



To access the 2D Transform menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front and a matte clip, and outputs a result and an outmatte.

2D Transform Menu Settings

General Settings



Transform Type box Select Perspective to modify all available settings, or Pan & Scan to modify only Position and Scale. In Pan & Scan mode, some Axis and Camera Shake settings are unavailable.

Camera Shake button Enable to simulate camera shake in the image.

Show Icons button Enable to display the vertex editing tool in the image window.

Scan Mode box Select the scan mode of rendered clips.

Axis Settings



Position X field Displays the X-axis offset applied to the clip. Editable.

Position Y field Displays the Y-axis offset applied to the clip. Editable.

Position Z field Displays the depth offset of the clip. Available when Transform Type is set to Perspective. Editable.

Rotation X field Displays the degree of rotation on the X axis. Available when Transform Type is set to Perspective. Editable.

Rotation Y field Displays the degree of rotation on the Y axis. Available when Transform Type is set to Perspective. Editable.

Rotation Z field Displays the degree of rotation along the Z axis. Available when Transform Type is set to Perspective. Editable.

Scale X field Displays the X-axis scaling offsets applied to the clip. Editable.

Scale Y field Displays the Y-axis scaling offsets applied to the clip. Editable.

Scale Z field Displays the Z-axis scaling offset. Available when Transform Type is set to Perspective. Editable.

Proportional button Enable to affect the scale fields proportionally.

Shear X field Displays the offset value for the horizontal edges of the clip. Available when Transform Type is set to Perspective. Editable.

Shear Y field Displays the offset value for the vertical edges of the clip. Available when Transform Type is set to Perspective. Editable.

Shear Z field Displays the depth value of the horizontal edges of the clip. Available when Transform Type is set to Perspective. Editable.

Centre X field Displays the centre point value of the clip along the horizontal axis. Available when Transform Type is set to Perspective. Editable.

Centre Y field Displays the centre point value of the clip along the vertical axis. Available when Transform Type is set to Perspective. Editable.

Centre Z field Displays the centre point value of the clip along the Z axis. Available when Transform Type is set to Perspective. Editable.

Camera Shake Settings



Translate X Amount field Displays the amount of horizontal movement applied to the clip. Editable.

Translate Y Amount field Displays the amount of vertical movement applied to the clip. Editable.

Rotation Z Amount field Displays the degree of rotation on the Z axis. Available when Transform Type is set to Perspective.Editable.

Scale Amount field Displays the amount of scaling applied to the clip. Editable.

Translate X Frequency field Displays the frequency of horizontal movement applied to the clip. Editable.

Translate Y Frequency field Displays the frequency of vertical movement applied to the clip. Editable.

Scale Frequency field Displays the frequency of scaling applied to the clip. Editable.

Rotation Z Frequency field Displays the frequency of rotation on the Z axis. Available when Transform Type is set to Perspective.Editable.

Translate X Phase field Displays the phase of horizontal movement applied to the clip. Editable.

Translate Y Phase field Displays the phase of vertical movement applied to the clip. Editable.

Rotation Z Phase field Displays the phase of rotation on the Z axis. Editable.

Scale Phase field Displays the phase of scaling applied to the clip. Editable.

X Centre field Displays the centre point value of the clip along the horizontal axis. Available when Transform Type is set to Perspective. Editable.

Y Centre field Displays the centre point value of the clip along the vertical axis. Available when Transform Type is set to Perspective.Editable.

Stabilization Settings



Enter Stabilizer button Enable to enter the Stabilizer editor.

Invert Stabilization button Enable to apply an inverse of the stabilization data to recreate the original camera shake.

Framing Mode box Select a method to fill or remove the area where the image has been shifted after stabilization.

Position X button Enable to include the X-axis position in the stabilization.

Position X field Displays the amount of stabilization data found in the animation channel for the X position.

Position Y button Enable to include the Y-axis position in the stabilization.

Position Y field Displays the amount of stabilization data found in the animation channel for the Y position.

Scaling button Enable to include scaling in the stabilization.

Scaling field Displays the amount of stabilization data found in the animation channel for scaling.

Rotation button Enable to include the rotation in the stabilization.

Rotation field Displays the amount of stabilization data found in the animation channel for rotation.

Set Frame button Enable to determine which frame has no transformation data.

Frame field Displays the frame that has no transformation data. Non-editable.

Perspective and Random Seed Settings



Perspective field Displays the field of view. Available when Transform Type is set to Perspective. Editable.

Random Seed field Displays the random seed value of the camera shake. Editable.

Texture Settings



Filter box Displays available filters to control the quality of resize operations. The list of filters changes when you switch between Perspective and Pan & Scan Transform Types.

Precision field Displays the frequency cut-off point used during resize. Editable only when doing a pan and scan with Shannon or Lanczos filters.

Crisp/Soft field Displays the amount of blurring used during resize. Editable only when doing a pan and scan with Mitchell, Shannon or Lanczos filters.

Repeat Mode box Select an option to fill the empty portions of the frame.

Select:	To:
Repeat Last	Repeats last pixel color.
Tile Repeat	Wrap the image around to fill the area.
Mirror Repeat	Repeats the source pixels both vertically and horizontally.
Repeat Off	Use a colour to fill the area. Use the adjacent colour pot to pick the colour. You will lose texture and will fill the rest of the image with a black border.

Anti-Aliasing Settings



Anti-Aliasing button Enable to activate sampling and softness.

Sampling box Select the number of samples to use in the anti-aliasing process.

Softness field Displays the level of softness of the samples. Editable.

Motion Blur Settings



Motion Blur button Enable to apply a motion blur to a selected clip.

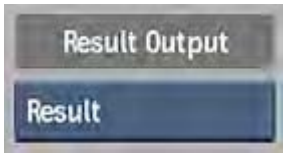
Samples field Displays the number of frames to sample when creating the blur. The samples include the current frame, and an equal distribution of past and future frames. Editable.

Phase field Displays the level of softness of the samples. Editable.

Shutter field Displays the number of frames for which the shutter stays open. For example, when the shutter value is set to 3, every third frame is a sample. Editable.

Lock Transform button Enable to lock the transformation of the motion blur effect.

Result Output Settings



Result Output box Select an output option. Select Premultiplied to output alpha values in the colour channels of the result.

Canvas Resolution Settings



Resize Fit Mode box Select whether to resize the clip with width and height values.

Same as Input	The output resolution remains the same as the input resolution.
Custom	You define the output resolution.

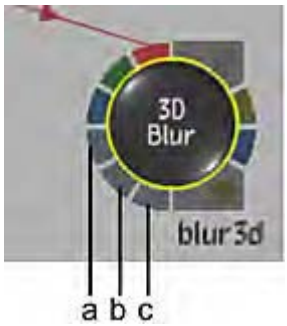
Auto Scale	The output resolution changes dynamically, which insures that any transformation applied will not crop the image across the whole length of the clip.

Width field Displays the width of the image. Editable.

Height field Displays the height of the image. Editable.

3D Blur

Use the 3D Blur to create a plausible blur effect using Z-depth map information.



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

to access the 3D Blur menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).

This node accepts a front, back, and matte as input. Additional inputs are available for the Z-depth map, forward vector, and kernal media. It outputs a result and an outmatte.

The 3D Blur can be used in the following ways:

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

3D Blur Menu Settings

Creating a Lens Blur

Lens blur simulates the blur created by a camera lens, such as a rack defocus.

The primary input of the 3D blur node is the front clip and the matte clip. A blur effect can be created with a front clip only. However a matte clip allows you to determine how the matte is used. The Blending settings also determine how the input is composited in the Result view.



Divide Front button Enable to unpremultiply the front clip using the matte clip. This button is active when a matte is in use.

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

Use the 3D Blur type settings to refine and constrain the amount of blur to apply to the image and to constrain the amount of blur. Higher blur values increase rendering time. You can increase rendering performance by decreasing the blur quality. When you are ready to create a result clip and an output matte, you can select the highest quality blur.



Blur Type box Select the type of 3D Blur to apply. Some of the settings differ based on the blur type you choose.

Blur Width field Displays the horizontal blur amount in pixels. Editable.

Blur Height field Displays the vertical blur amount in pixels. Editable.

Blur Proportional button Enable to constrain blur amount proportions.

Sides field Displays the number of sides in a Gaussian blur shape.Available when Gaussian is chosen as the blur type. Editable.

Rotation field Displays the angle of rotation of a Gaussian blur shape. Available when Gaussian is chosen as the blur type.Editable.

Kernel Precision box Select the quality and speed of the defocus blur. For example, a 1x1 Kernel results in a fast, but coarse defocus blur.Available when Defocus is chosen as the blur type.

Z Slicing field Displays the amount of slices to blend to determine the Z-Depth interpolation of a defocus blur. Available when Defocus is chosen as the blur type.Editable.

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



Adjust button Enable to activate highlight controls.

Gain field Displays the light values of the image. Editable.

Offset field Displays each colour value adjusted by this increment. Editable.

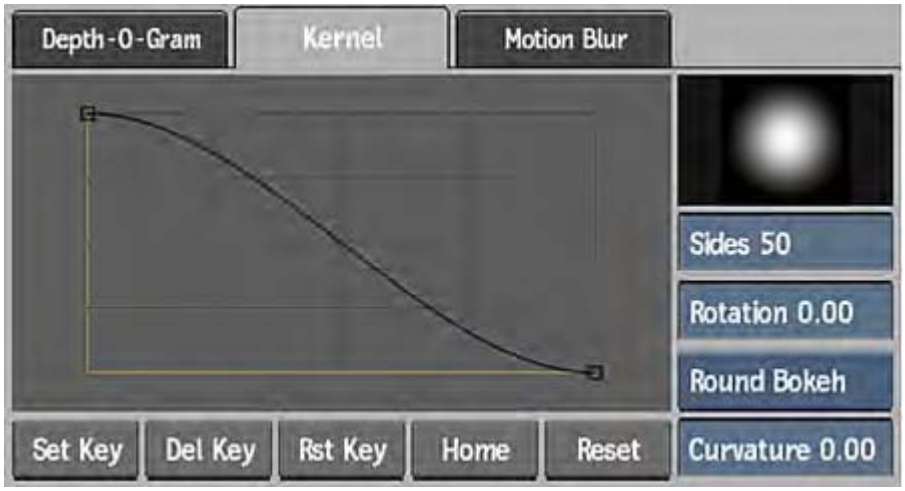
Threshold field Displays the minimum colour value included in highlights. Editable.

Ramp Range field Displays the difference between the Threshold value and the value at which the highlights take full effect. Editable.

Max Effect At field Displays the value at which the highlights take full effect. Non-editable.

Changing the Blur Pattern

Use the Kernal tab to edit the blur kernel pattern.



The kernel is the basic blur shape. The shape of the kernel is determined by its number of sides, its rotation, and the shape of its S-curve. This curve represents the shape of the pattern, from its centre to the outside. The default S-curve defines the softness of the blur. You can change the curve by manipulating the two points that define the curve, or you can add points to the curve. Use the Edit Mode box to add and delete points on the curve.

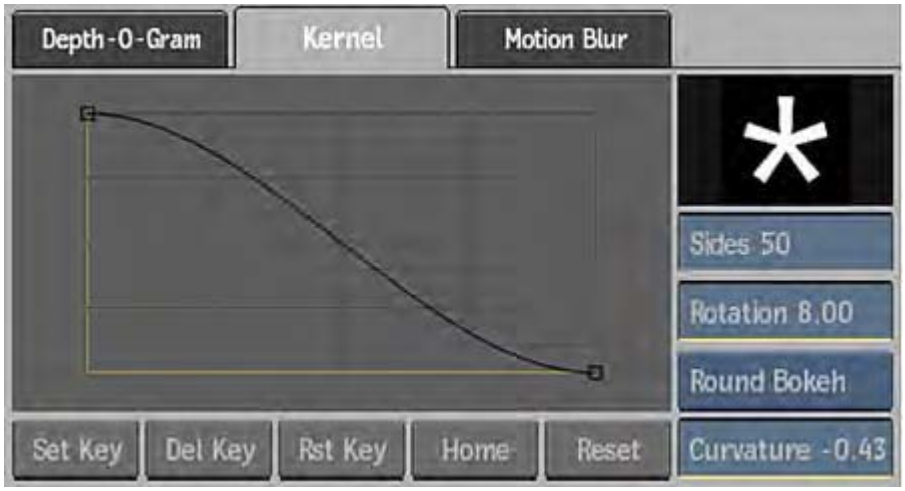
Sides field Displays the number of sides in the kernel shape. Editable, if kernel information is not attached to the node.

Rotation field Displays the angle of rotation of the kernel shape. Editable, if kernel information is not attached to the node.

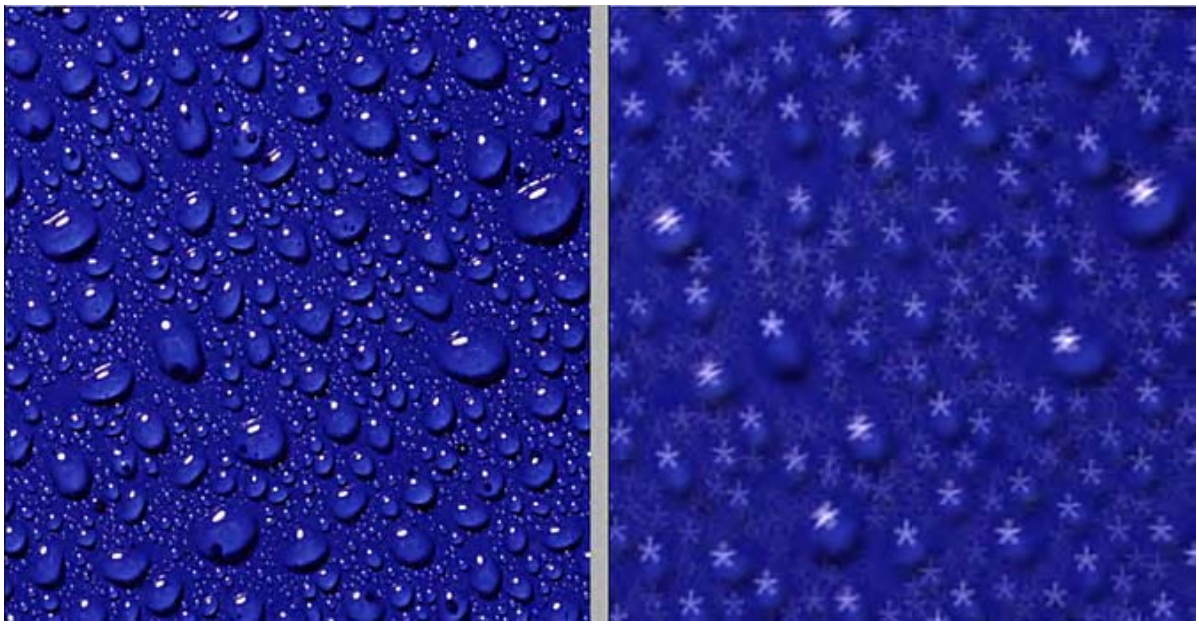
Bokeh Type box Select whether to use a round or angle bokeh curve to define the kernel shape.

Curvature field Displays the amount of curvature applied to a round bokeh kernel shape. Editable.

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

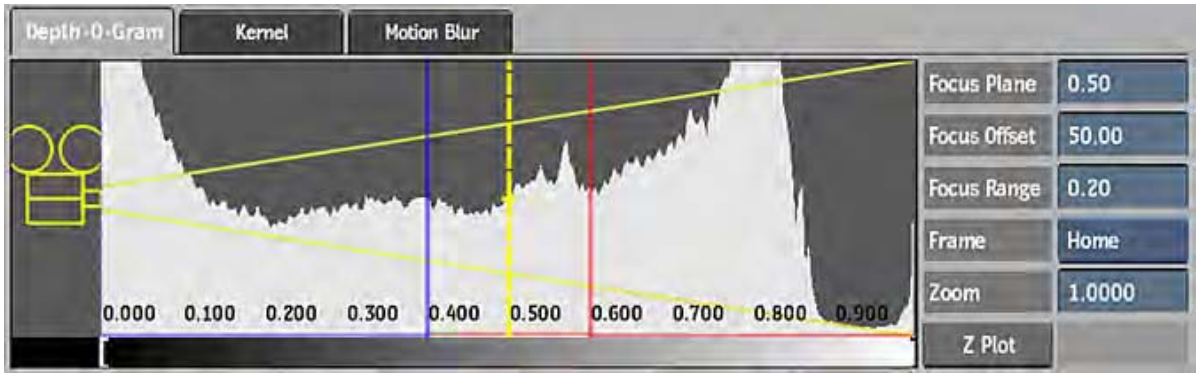


External blur information is not editable. Like the default blur shape selections, the shape determines the influence of the kernel's pixels on each pixel in the front input.

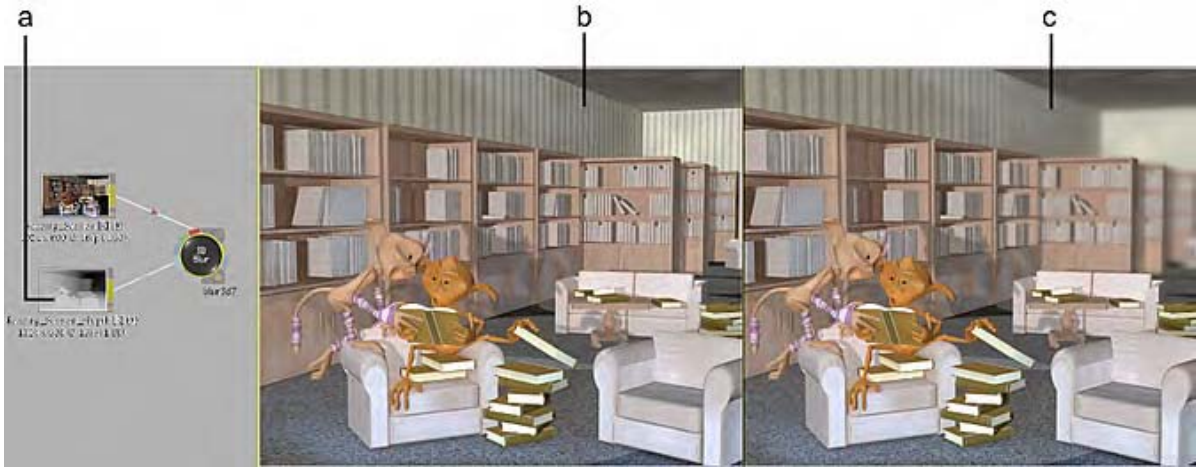


Simulating a Depth of Field Effect

Depth blur effects control the amount of blur based on a depth of field matte. Depth blur settings are displayed in the Depth-O-Gram tab.



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



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

Image courtesy of Jean-Marc Belloncik

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



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

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

Slopes field Displays the gamma value. Applies a gamma curve to the Z-depth map before it is used. Editable.

Gain field Displays the white balance value. Applies a gain curve to the Z-depth map before it is used. Editable.

Max Blur Field Displays the maximum total horizontal and vertical blur amount to clamp the value and avoid unnecessary rendering. The cumulative effect of the high blur amount (Basic tab), and gamma and gain correction applied to the Depth map (Depth tab) may result in very large values, which greatly increases the rendering time. Editable.

Blur Z-Depth button Enable to allow pre-blurring of the Z-Buffer. Disable to increase performance time.

Edge Blur field Displays the minimum blur value of "in focus" pixels. In some instances, an out-of-focus object that appears closer to the camera (that is, in front of the focus plane) can have hard edges. In this case, increase the value in the field to add a blur to these edges. Editable.

Use the Depth-O-Gram settings to refine focus values.



Focus Plane field Displays the distance of your focus point, which is the point at which there is no blur on the image. Editable.

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

Focus Range field Displays the distance the near and far offset. Editble.

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

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

Z Plot button Click to activate the pick cursor. Use to select a pixel in the image to display its depth.

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

Use On-Screen Widgets section in the NodeSetup menu to define display settings.



Near/Far button Enable to display the focus offset plane in the image window.

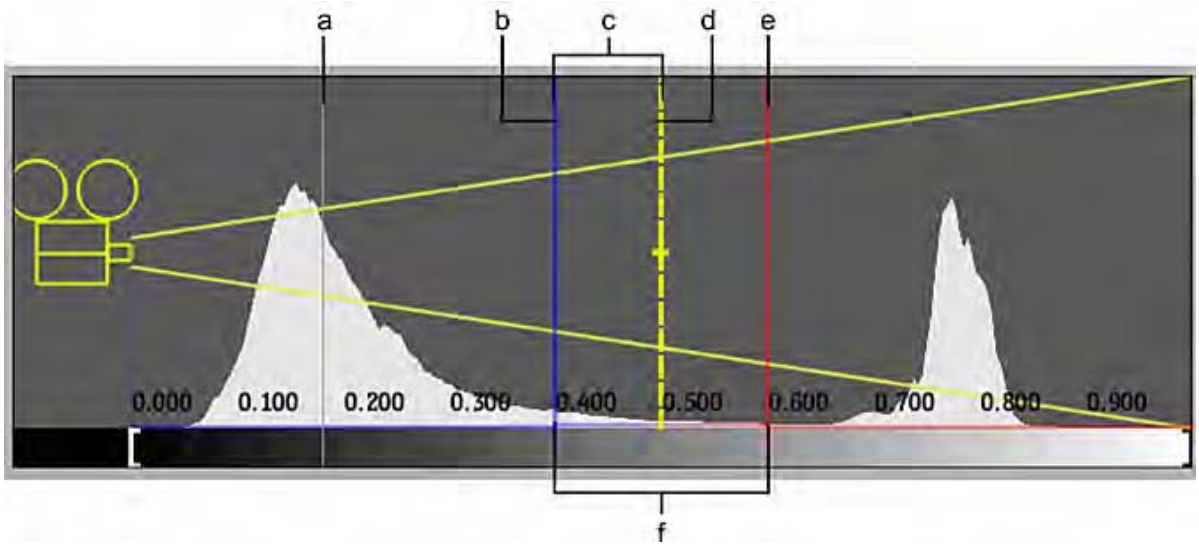
Near and Far colour pots Select the colours that indicate the nearest and furthest points of focus.

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

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

Modifying Depth of Field Gesturally

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

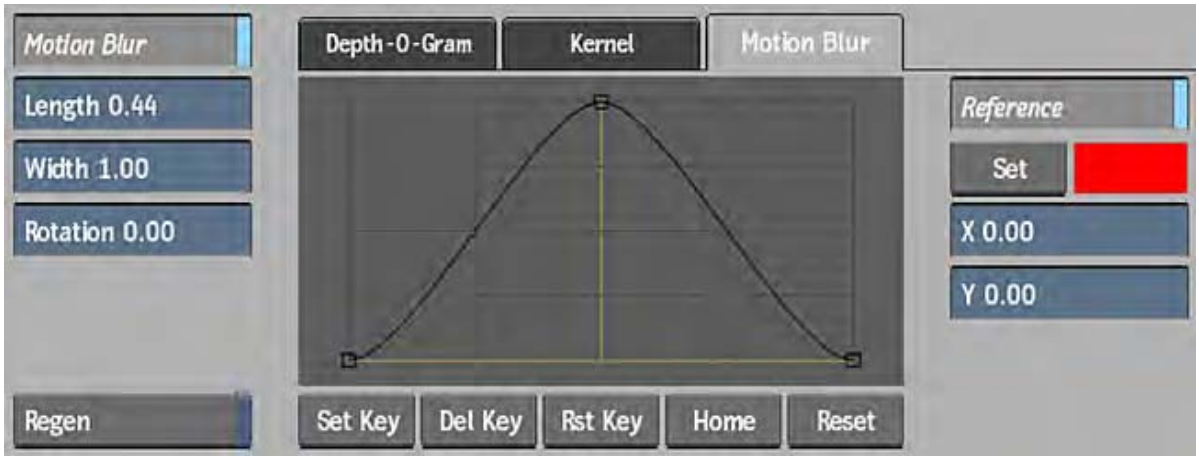


(a) Plot Value Plane (b) Near Focus Offset Plane (c) Focus Offset (d) Focus Plane (e) Far Focus Offset Plane (f) Focus Range

Simulating a Motion Blur Effect

The motion blur simulates the blur created by a fast-moving object or camera. A blur effect can be created with front clip input and a clip with forward motion data input into the Forward Flow tab.

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



Motion Blur button Enable to activate the motion blur controls and apply the effect to the image.

Length field Displays the amplitude of the motion blur. Editable.

Width field Displays the width of the blur. Editable.

Rotation field Displays the angle of rotation to apply to motion vectors. Motion is rotated counterclockwise. Editable.

Reference button Enable to activate the reference controls and apply the effect to motion blur.

Set button Enable to apply the motion blur to objects that do not have the same relative motion as the selected pixel. Enabled if the X or Y field is edited.

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

X field Displays the horizontal position of the pixel to use as a reference point at the selected frame. A reference point can be selected directly in the frame using the cursor. Editable.

X field Displays the vertical position of the pixel to use as a reference point at the selected frame. A reference point can be selected directly in the frame using the cursor. Editable.

Auto Matte

Use the Auto Matte to generate a high-contrast matte from a clip.



Image courtesy of Optimus

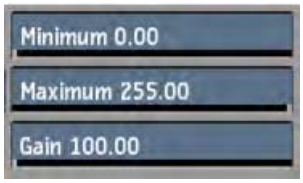
To access the Auto Matte menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip as input, and outputs a result.

Auto Matte Menu Settings

General Settings



Minimum field Displays the minimum luma value considered to be part of the opaque area of the matte. Editable.

Maximum field Displays the maximum luma value considered to be part of the transparent area of the matte. Editable.

Gain field Displays a value, in percentage, that is multiplied to the pixel values in the resulting matte to reduce or increase the grey levels.

Auto Stabilize

Use the Auto Stabilize to analyse the movement in a clip and remove camera instability. For example, you can remove camera jitter or lock an object's position over a sequence of frames to make it appear motionless.



To access the Auto Stabilize menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts front and matte clips, and outputs a result and an outmatte.

Auto Stabilize Menu Settings

Analysis Settings



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

Progress field Displays the percentage of the analysis that is complete. Non-editable.

Scene Flexibility field Displays the level of focus on rigid objects that move in the clip. Enter 0% to focus on a single rigid object as it moves or 100% to analyse all visible motion paths. If the analysis has no scene flexibility, the stabilization will focus on a single rigid object as it moves in the clip. Use the following values as a general guideline:

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

Use Matte button Enable to exclude black areas on the matte from analysis on the front clip, and constrain the Analysis region.

Range From field Displays the value of the first frame to include in the analysis. Editable.

Range To field Displays the value of the last frame to include in the analysis. Editable.

Stabilization Method box Select whether to perform a two-dimensional or perspective analysis.

Negate Stabilization button Enable to apply the inverted parameter values.

Region of Interest button Enable to display a rectangle that can be resized to indicate the region you want to analyse on the front clip. Select Front from the View box.

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

Stabilization Settings

Position X button Enable to include the X-axis position in the stabilization.

Position X Option box Select whether to lock the X-axis position at the reference frame to all frames in the analysis (Fixed), or to apply smoothness to the stabilization curve on the X-axis.

Position Y button Enable to include the Y-axis position in the stabilization.

Position Y Option box Select whether to lock the Y-axis position at the reference frame to all frames in the analysis (Fixed), or to apply smoothness to the stabilization curve on the Y-axis.

Rotation button Enable to include the rotation in the stabilization.

Rotation Option box Select whether to lock the rotation at the reference frame to all frames in the analysis (Fixed), or to apply smoothness to the stabilization curve on the rotation.

Scaling button Enable to include scaling in the stabilization.

Scaling Lock Option box Select whether to lock the scaling at the reference frame to all frames in the analysis (Fixed), or to apply smoothness to the stabilization curve on the scaling.

NOTE You can change the centre of rotation and scaling for the image. By default, the centre of rotation and scaling is the centre of the image, represented by a yellow crosshair. Switch to Front view, then hold down C and click the new centre point.

Perspective button Enable to include perspective in the stabilization.

Perspective Option box Select whether to lock the perspective at the reference frame to all frames in the analysis (Fixed), or to apply smoothness to the stabilization curve on the perspective.

Smoothness field Displays the level of smoothness of the stabilization curves generated by the analysis and determines the mix between the original motion and the completely stabilized transformation. Editable.

Mix field Displays the percentage of smoothness used in the final stabilization output and determines how smooth the stabilization curves are. Editable.

Reference and Tracker Settings



Set Frame button Click to set the current frame as the reference frame for the stabilization.

Reference Frame field Displays the current reference frame. Non-editable.

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

Show Cloud button Enable to display the track point cloud.

Delete and Update button Click to delete selected track points and update stabilization curves. Selected points are displayed in red in the image.

Offset Settings



Position X field Displays the X-axis offset applied to the clip. Editable.

Position Y field Displays the Y-axis offset applied to the clip. Editable.

Rotation field Displays the rotation offset applied to the clip. Editable.

Scaling field Displays the scaling offsets applied to the clip. Editable.

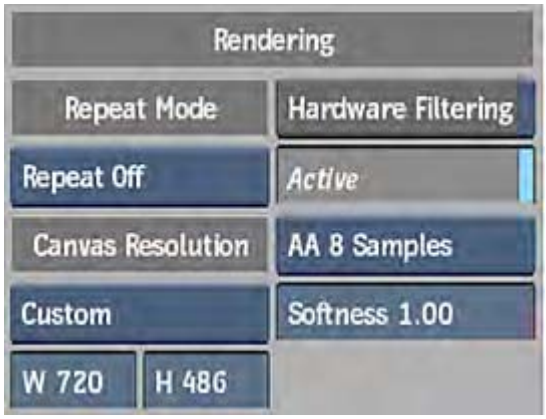
Project X field Displays the X-axis projection offsets applied to the clip. Editable.

Project Y field Displays the Y-axis projection offsets applied to the clip. Editable.

Anamorphic field Displays the anamorphic offsets applied to the clip. Editable.

Shear field Displays the shearing offsets applied to the clip. Editable.

Rendering Settings



Repeat Mode box Select an option to fill the empty portions of the frame.

Select:	To:
Repeat Last	Repeats last pixel color.

Select:	To:
Tile Repeat	Wrap the image around to fill the area.
Mirror Repeat	Repeats the source pixels both vertically and horizontally.
Repeat Off	Use a colour to fill the area. Use the adjacent colour pot to pick the colour. You will lose texture and will fill the rest of the image with a black border.

Resize Fit Mode box Select whether to resize the clip with width and height values.

Select:	To:
Same as Input	The output resolution remains the same as the input resolution.
Custom	You define the output resolution.

Width field Displays the width of the image. Editable.

Height field Displays the height of the image. Editable.

Hardware Filtering button Enable to filter subpixel information.

Anti-Aliasing button Enable to activate sampling and softness.

Sampling box Select the number of samples to use in the anti-aliasing process.

Softness field Displays the level of softness of the samples. Editable.

Common

Clear Buffer button Clears the Undo/Redo list of performed actions.

Use Proxies button Enable to use proxy-sized clips instead of the resolution clip. Increases interactivity performance.

Backup button Enable to automatically save backups of Tools data.

Tools Save field Displays the time delay for an automatic save when you are using a tool that you have accessed from the Tool tab. Non-editable.

Append Type button Appends the output type (Result or OutMatte) to the filenames of rendered clips.

Average

Use Average to simulate the motion blur of a moving object in a clip. An image averaging function is applied to the frames to create the motion blur.



To access the Average menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip, and outputs a result.

Average Menu Settings

The Average Node processes frames based on data from preceding and subsequent sample frames. A frame set to No Media with missing media may include a sample.



General Settings

Average Over field Displays the number of frames used to calculate the average and create the transparency for the motion blur effect. Editable.

Weighted/Uniform box Select whether to uniformly average surrounding frames or give frames closest to the current frame a greater weight when calculating the transparency for the motion blur effect.

Frames box Select whether the blur is calculated from previous frames only or future frames as well.

Blend & Comp

Use Blend & Comp to apply basic blending and compositing between two inputs, and their mattes, over a background.



To access the Blend & Comp menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts up to two front and matte clips, as well as a back clip, and outputs a result and outmatte.

Blend & Comp Menu Settings

General Settings



Blend Type box Select the set of blend mode operations available in the Blend Mode box.

Blend Mode box Select a blend operation. Options determined by Blend Type box.

Swap Inputs button Enable to switch the rendering order of the inputs.

Input Settings

NOTE Input 1 and 2 settings are identical. Activate and use based on inputs selected for Blend & Comp.



Active button Enable to activate the input.

Clamping box Select a clamping option.

Matte box Select how the matte is used when blending.

Pre-multiplication box Select whether the colour values in the input are pre-multiplied or divided from the alpha, or left as-is when blending.

Transparency field Displays the transparency value. Editable.

Gain Trackball Adjusts the gain of the input.

Saturation field Displays level of colour purity in the image. Editable.

Gamma field Displays the gamma level. Editable.

Offset field Displays a value that modifies all of the colour parameters. Editable.

Red Gain field Set the percentage of colour values in the red channel. Editable.

Green Gain field Set the percentage of colour values in the green channel. Editable.

Blue Gain field Set the percentage of colour values in the blue channel. Editable.

Luma Gain field Set the percentage of luma gain value to display. Editable.

Proportional button Enable to adjust the gain of the colour values proportionally.

Matte Blending Settings



Composition box Select how the input mattes are combined.

Correlation box Select whether the mattes are related to each other.For example, select Correlated if the mattes are from different, but continuous parts on the same object.

Result Output Settings



Blend With box Select whether you want to blend the inputs with the second input clip, the background clip or a colour.

Blend Type box Select the set of blend mode operations available in the Blend Mode box.

Clamping box Select a clamping option.

Blend Mode box Select a blend operation. Options determined by Blend Type box.

Transparency field Displays the transparency value. Editable.

Gain Trackball Adjusts the gain of the input.

Saturation field Displays level of colour purity in the image. Editable.

Gamma field Displays the gamma level. Editable.

Offset field Displays a value that modifies all of the colour parameters. Editable.

Red Gain field Set the percentage of colour values in the red channel. Editable.

Green Gain field Set the percentage of colour values in the green channel. Editable.

Blue Gain field Set the percentage of colour values in the blue channel. Editable.

Luma Gain field Set the percentage of luma gain value to display. Editable.

Proportional button Enable to adjust the gain of the colour values proportionally.

Blur

Use Blur to apply and customize a blur filter to a clip.



To access the Blur menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front and a matte clip, and outputs a result and outmatte. The outmatte clip can have a different level of blur than the result clip.

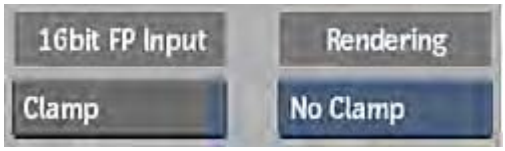
Blur Menu Settings

General Settings

Rendering box Select whether to render in Progressive or Interlaced mode.

Regen button Enable to get dynamic updating of the image as you make changes.

Setup Settings



Clamp Input button Enable to clamp colour and luminance values on input in the 16-bit floating point rendering pipeline.

Clamp Render box Select a clamping option for colour and luminance values on output in the 16-bit floating point rendering pipeline.

Blur Type Settings



Blur Type for Gaussian



Blur Type for Radial

Blur Type box Select the type or shape of blur filter to apply to the clip. Depending on the blur type, some of the other blur settings may vary.

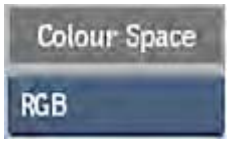
Bokeh Blur field Displays the amount of smoothness applied to sharp bokeh edges. This creates the blur that is applied to out-of-focus points of light to simulate a shallow focus. Editable.

Bias field Displays the direction of a blur. Enter a positive value for forward, a negative value for backward, or 0 for a blur that moves in both directions. Available when Directional or Radial Stamp is chosen as the blur type.

Radial Mode box Select whether a radial blur or glow moves in one circular direction (Spin), or two rotating directions (Twist). Available when Radial is chosen as the blur type.

Samples field Displays the quality of a Radial Stamp blur or glow. Editable. Available when Radial Stamp is chosen as the blur type.

Color Space Settings



Colour Space box Select whether to blur in RGB or YUV space. The Channels settings change to reflect this setting.

Centre Settings (Radial Only)



Stabilizer button Opens the Stabilizer menu to track the centre of the blur from the source clip. See Tracking (page 878) for more information.

Centre X field Displays the X position of the centre of the radial blur (or gesturally move the red circle in the image). Editable.

Centre Y field Displays the Y position of the centre of the radial blur (or gesturally move the red circle in the image). Editable.

NOTE You can also move the red circle on the image to set the position of the centre of the blur. The Centre X and Y fields update accordingly.

Absolute/Relative box Select whether to position and offset the centre of the radial blur in a relative mode (expressed as a percentage) or absolute mode (expressed in pixels).

Offset X field Offsets the centre along the X axis (or press Ctrl and gesturally move the red circle in the image).This field is useful to apply changes to the centre after tracking has been performed. Editable.

Offset Y field Offsets the centre along the Y axis (or press Ctrl and gesturally move the red circle in the image). This field is useful to apply changes to the centre after tracking has been performed. Editable.

NOTE You can also press `Ctrl`and move the red circle on the image to set the offset of the centre blur.

Front Settings



Width field Displays the width of the blur.Increasing the blur increases the render time. Editable.

Height field Displays the height of the blur.Increasing the blur increases the render time. Editable.

Proportional button Enable to affect the width and height proportionally.

Pixel Ratio button Enable to blur the image using the same proportion as its aspect ratio.

Gain field Displays a value by which pixel colour values are multiplied. The offset value is added to this value to determine the final colour. Editable.

Offset field Displays the value to add to current pixel colour values. The resulting colour value is clipped at 0. Editable.

Length field (Directional) Displays the radius amount of a directional blur. Editable.

Angle field (Directional) Displays the angle of a directional blur. Editable.

Amount field (Radial, Radial Stamp) Displays the amount of radial blur. Editable.

Rotation field (Radial, Radial Stamp) Displays the angle of rotation for a radial blur. Editable.

Blooming Settings (Defocus Only)



Basic/Additive Blooming button Switch between basic blooming and additive blooming bokeh effects. The basic blooming mode displays the gain applied to highlights in non-HDR images, to allow the creation of bokeh patterns without affecting colour integrity. The additive blooming mode allows you to create higher intensity bokeh patterns from any source image, using minimum and maximum thresholds for highlight segregation.

Basic Blooming field Displays the gain applied to highlights in non-HDR images, to allow the creation of bokeh patterns without affecting colour integrity. This creates the glow effect that is applied to the bright spots of the image to simulate light bleeding, or blooming, over the edges. Editable.

Additive Blooming field Displays the amount of high intensity bokeh patterns that can be created from any source image generating extreme highlight content . Editable.

Additive Blooming minimum field Displays the minimum threshold for highlight segregation. Editable.

Additive Blooming maximum field Displays the maximum threshold for highlight segregation. Editable.

Matte Settings



Matte Settings for Gaussian, Defocus, or Box



Matte Settings for Radial or Radial Stamp



Matte Settings for Directional

Matte Width field Displays the width of the blur for the matte. Editable.

Matte Height field Displays the height of the blur for the matte. Editable.

Matte Length field Displays the radius amount of a directional blur for the matte. Editable.

Matte Angle field Displays the angle of a direction blur of the matte. Editable.

Matte Amount field Displays the amount of radial blur for the matte. Editable.

Matte Rotation field Displays the angle of rotation for a radial blur for the matte. Editable.

Lock To Front button Enable to keep the matte values the same as their corresponding values for the front clip.

Invert Matte button Enable to blur the region outside the area defined by the matte.

Channel Settings



RGB Channels



YUV Channels

Weight field Displays the weighted value of the channel.

Position X field Displays the horizontal offset of the channel.

Position Y field Displays the vertical offset of the channel.

YUV Colour box Select whether to affect Luma or Chroma only, or include both when blurring in YUV colour space.

Blending Settings

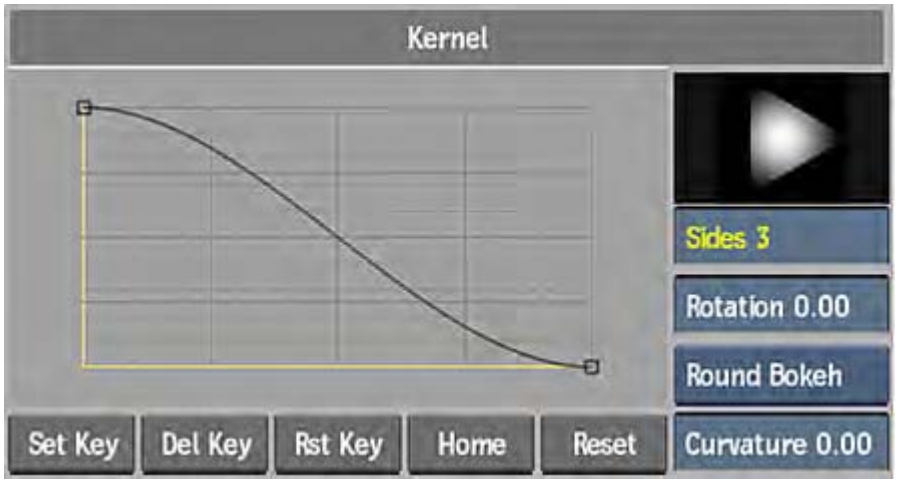


Screen option box Select a logical operation that can be used to blend the front clip and the result clip.

Transparency field Displays the percentage of blending when the result is composed on the front clip. Editable.

Use Matte button Enable to apply the blur with the areas defined by the matte.

Kernal Settings (Defocus Only)



The kernel is the defocus blur shape. The shape of the kernel is determined by its number of sides, its rotation, and the shape of its S-curve. This curve represents the shape of the pattern, from its centre to the outside. The default S-curve defines the softness of the blur. You can change the curve by manipulating the two points that define the curve, or you can add points to the curve. Use the Edit Mode box to add and delete points on the curve.

Sides field Displays the number of sides in the kernel shape. Editable, if kernel information is not attached to the node.

Rotation field Displays the angle of rotation of the kernel shape. Editable, if kernel information is not attached to the node.

Bokeh Type box Select whether to use a round or angle bokeh curve to define the kernel shape.

Curvature field Displays the amount of curvature applied to a round bokeh kernel shape. Editable.

Bump Displace

Use the Bump Displace effect to create displacement-like emboss effects using light and surface bumps. You can select Front or Front/ZDepth or Front/ZDepth/Matte from the Input Mode box.



To access the Bump Displace menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip and a displacement clip (usually a matte), and outputs a result.

Bump Displace Menu Settings

Bump Settings



Minimum Bumps field Displays the lower limit of the bump depth. Editable.

Maximum Bumps field Displays the upper limit of the bump depth. Editable.

Bump Height field Displays the height value for the effect. Positive numbers create an embossed look. Negative numbers create an engraved look. Editable.

Bump Softness field Displays a blurring value for the effect. Editable.

Light Settings



Light X Position field Displays the horizontal position of the light. Editable. You can also use the light icon in the image window to change the position of the light.

Light Y Position field Displays vertical position of the light. Editable. You can also use the light icon in the image window to change the position of the light.

Light Y Position field Displays the depth of the light. Editable. You can also use the light icon in the image window to change the position of the light.

TIP You can also use the light icon in the image window to gesturally position the light.

Light Scale field Displays a value that you can modify to see the results of moving your light when using extreme X, Y or Z values. Editable.

Shaded field Displays the level of intensity for the image. Editable.

Ambient field Displays the percentage of overall lighting for the image. Editable.

Threshold field Displays the percentage of bumps filtered out by using light exposure. Editable.

Red Light field Displays the red value of the light. Editable.

Green Light field Displays the green value of the light. Editable.

Blue Light field Displays the blue value of the light. Editable.

Light Colour Pot Displays the colour of the light. Click to open the Colour Picker and specify a value.

Texture Settings



Diffuse Gain field Displays a value for the contrast in the image. Use to increase or decrease the intensity of highlights and shadows in surface features. Editable.

Shininess field Displays a value for the amount of shine in the image. Editable.

Specular field Displays a value for the reflectiveness of the image. Editable.

RGB Distortion Settings

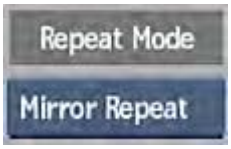


Red Distortion field Displays the amount of distortion in the red channel. Editable.

Green Distortion field Displays the amount of distortion in the green channel. Editable.

Blue Distortion field Displays the amount of distortion in the blue channel. Editable.

Repeat Mode Settings



Repeat Mode Settings Select an option to fill the empty portions of the frame.

Anti-Aliasing Settings



Anti-aliasing button Enable to activate sampling and softness.

Sampling box Select the number of samples to use in the anit-aliasing process.

Softness field Displays the level of softness of the samples. Editable.

Burn-In Letterbox

Use Burn-In Letterbox to burn a letterbox into a clip.



To access the Burn-In Letterbox menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip, and outputs a result.

Burn-In Letterbox Menu Settings

General Settings



Aspect Ratio box Select a preset or custom aspect ratio.

Offset field Displays the vertical offset. Drag left or right to apply the letterbox to a lower or higher portion of the image. Editable.

Border colour pot View the current border colour. Editable.

Ratio field Displays the custom aspect ratio. Editable.

NOTE The Ratio field is only available when Custom is selected from the Aspect Ratio box.

Burn-In Timecode

Use Burn-In Timecode to burn-in clip information.



To access the Burn-In Timecode menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip, and outputs a result.

Burn-In Timecode Menu Settings

General Settings



Text colour pot Select the current colour of burned-in text. Select to open the colour picker.

Shadow button Enable to add a drop shadow to burned-in information.

Shadow colour pot Displays the current shadow colour. Select to open the colour picker.

Background button Enable to display a rectangular background to burned-in text.

Background colour pot Displays the current background colour. Select to open the colour picker.

Font box Displays the font used for burned-in text.

Name field Displays the name of the clip. Click to enter new name or comment. Editable.

Scan Format box Select to scan a frame-based clip with or without interlacing, or use the scan format of the front clip.

Offset box Select to display the burned in information within the full frame, or a safe area. This box is only active when position is set using the Horizontal Position and Vertical Position boxes.

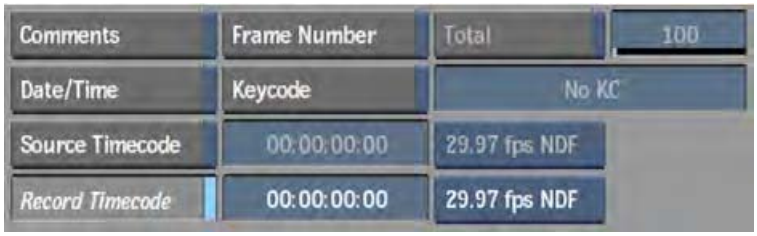
Horizontal Position box Select the horizontal placement of the burned-in text.

Horizontal Position field Set the position on the Y-axis. Drag left or right to move the text in the same direction. Editable.

Vertical Position box Select the vertical placement of burned-in text.

Vertical Position field Set the position on the X-axis. Drag left or right to move the text up or down. Editable.

Size field Displays the font size of burned-in text. Drag left or right to decrease or increase the point size. Editable.



Comments button Enable to include the current clip name or comment in burned-in text.

Frame Number button Enable to include the current frame number in burned-in text.

Frame Total button Enable to include the total number of frames in burned-in text.

Total Frames field Click to enter a value for the total number of frames in the clip. Editable.

Date/Time button Enable to include the current date and time in burned-in text.

Keycode button Enable to include the keycode in burned-in text.

Keycode field Click to enter a keycode. Editable.

Source TC button Enable to include the source timecode in burned-in text.

Source TC field Click to enter a new source timecode. Editable.

Source TC Frame Mode box Select the style of source frame code notation.

Record TC button Enable to include the record timecode in burned-in text.

Record TC field Click to enter a new record timecode. Editable.

Record TC Frame Mode box Select the style of record frame code notation.

Clamp

Use the Clamp node to clamp 16-bit floating point OpenExr clips.

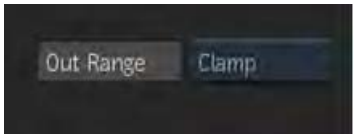


To access the Clamp menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

Clamp Menu Settings

General Settings



Out of Range box Select a curve that is constant (clamp) or clamps negative values below 0.

Color Corrector

About Colour Corrector

The Colour Corrector includes tools that provide precise control over colour values. You can modify luma ranges in a clip (shadows, midtones, and highlights), sample colours, and adjust the colour balance. You can rewire colour channels and suppress colours, as well as animate a colour correction by manipulating the animation curve in the Channel Editor.

Accessing the Colour Corrector

To access the Colour Corrector, you must load clips of the same resolution. If the clips you want to load have different resolutions, resize them so that they have the same resolution.

You can load a front clip, a front and back clip, or a front, back, and matte clip for colour correction. Changes in colour are applied to the front clip.

The Colour Corrector can be accessed from the following locations:

- Timeline, then use a Colour Corrector timeline effect. (page 933)
- Batch, then select a node from the Node bin. (page 934)
- Modular Keyer, Action, Paint tool. (page 934)
- Timeline, then use Batch FX. (page 934)

Colour Correct Menu Settings

Main Menu

Crop button Displays a crop box to view colour changes on a limited region of the clip.

Setup button Opens the Setup menu where you setup either Colour Correct or Colour Warper options.

Animation button Opens the Animation menu where you display the animation channel.

Master button Modifies the luma range for the entire image after individual range modifications.

Shadows button Modifies the luma range for the dark areas in the image.

Midtones button Modifies the luma range for the midlevel areas in the image.

Highlights button Modifies the luma range for the light areas in the image.

Regen button Enable to get dynamic updating of your colour corrections.

Rewire box Select an option to rewire the RGB channels of an image, or create a monochrome or negative image.

Red Rewiring box Select a color or luma channel to replace the red values with the selected channel values. Select 1-R to replace red values with its inverse values. Active when Master range is selected.

Green Rewiring box Select a color or luma channel to replace the green values of an image with its inverse (1-R), red, blue, or luma values. Active when Master range is selected.

Blue Rewiring box Select a color or luma channel to replace the blue values of an image with its inverse (1-R), red, green, or luma values. Active when Master range is selected.

Reset Basics Resets colour correction controls.

Hue field Displays the colour range. Editable.

Saturation field Displays level of colour purity in the image. Editable.

Pivot field Displays the percentage value of the colour range around which the contrast pivots. The default value is 50% for integer input and 18% for floating point input. Editable.

Hue field Displays the colour range. Editable.

Gain field Drag left or right to increase or decrease light greys from the white area of the image.

RGB Gamma field Adjust the grey values. Editable.

Red Gamma field Adjust the grey values in the red channel. Editable.

Green Gamma field Adjust the grey values in the green channel. Editable.

Blue Gamma field Adjust the grey values in the blue channel. Editable.

RGB Gain field Set the percentage of original colour values to use in the image. Editable.

Red Gain field Set the percentage of colour values in the red channel to display in the image. Editable.

Green Gain field Set the percentage of colour values in the green channel to display in the image. Editable.

Blue Gain field Set the percentage of colour values in the blue channel to display in the image. Editable.

RGB Offset field Adjust the colour values by adding an offset value. Editable.

Red Offset field Adjust the colour values in the red channel by adding an offset value. Editable.

Green Offset field Adjust the colour values in the green channel by adding an offset value. Editable.

Blue Offset field Adjust the colour values in the blue channel by adding an offset value. Editable.

Contrast field Adjust the gradations between the light and dark areas in the image. Editable.

Red Contrast field Adjust the gradations between the light and dark areas in the red channel of image. Editable.

Green Contrast field Adjust the gradations between the light and dark areas in the green channel of image. Editable.

Blue Contrast field Adjust the gradations between the light and dark areas in the blue channel of image. Editable.

Red Suppression button Enable to suppress red colour values in the image.

Green Suppression button Enable to suppress green colour values in the image.

Blue Suppression button Enable to suppress blue colour values in the image.

Cyan Suppression button Enable to suppress cyan colour values in the image.

Magenta Suppression button Enable to suppress magenta colour values in the image.

Yellow Suppression button Enable to suppress yellow colour values in the image.

Front colour patch Displays colour sampled from the front clip.

Result colour patch Displays the result of front clip colour correction.

Back colour patch Displays colour sampled from the back clip.

Sampling box Select the type of colour values to display for sampled colours.

Sample Data Type box Select the type of measurement to use for RGB values of sampled colours.

Histogram Menu

Histogram tab Displays the Histogram menu where you adjust the red, green, blue, and luminance channels of the image.

Histogram Displays a bar graph that is used to adjust the luminance values of the image. In the Ranges menu, the curves for the three luma ranges are also displayed.

Minimum Input field Displays the lower limit of the luminance values. Pixels with lower values are mapped to black.

Minimum Output field Displays the lower limit of the luminance values for black pixels.

Maximum Input field Displays the upper limit of the luminance values. Pixels with higher values are mapped to white.

Maximum Output field Displays the upper limit of the luminance values for white pixels.

Channel Selection box Select the luminance or an RGB channel to extract its values for the key.

Out Range box Select a curve that is constant (Clamp) or linear (No Clamp) before the first point of the curve and after the last point of the curve.

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

Frame Selection box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Source button Enable to show a histogram of the source colour values.

The source colour values are obtained from the current frame. If you input both front and matte clips, the colour values in the front clip that are defined by the white part of the matte are displayed.

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

Destination button Enable to show a histogram of the colour values in the result or destination clip.

The destination colour values are obtained from the current frame. If you input both front and matte clips, the colour values in the front clip that are defined by the white part of the matte are displayed.

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

When viewing the Source and Destination histograms at the same time in Mono mode, the Destination histograms are lighter; in RGB mode, the Destination histograms are displayed with a white outline.

Reset button Resets to default histogram settings.

Zoom field Displays the vertical zoom value of the histogram.

You can also zoom horizontally by pressing `Ctrl+spacebar` and dragging left or right in the histogram. To pan horizontally, click `spacebar` and drag left or right in the histogram.

Curves Menu

Curves tab Displays the Curves menu to display luminance and RGB curves.

Curves Display the curves for the luminance and each RGB channel.

Home button Restores the position of panned or zoomed curves to the default setting.

Move button This setting is available when you access the Colour Corrector via the Timeline. Select an option to work directly in the image window or schematic.

Match button Matches colours in the front and back colour patches, and assigns a number to the current match.

Match field Displays which match is currently displayed.

ReMatch button Overrides the selected match operation with current settings.

Out Range box Select a curve that is constant (Clamp) or linear (No Clamp) before the first point of the curve and after the last point of the curve.

Reset button Resets to default curve settings.

Ranges Menu

Ranges tab Displays the HLS Ranges menu where you can adjust highlight, midtone, and shadow ranges.

Reset button Resets to default histogram and curve settings.

Setup

Border button Enable to display a crop box border.

Border colour pot Displays the border colour. Editable.

Top field Displays the position for the top of the crop box. Editable.

Bottom field Displays the position for the bottom of the crop box. Editable.

Left field Displays the position for the left side of the crop box. Editable.

Right field Displays the position for the right side of the crop box. Editable.

Invert Matte button Enable to colour correct the region outside the area defined by the matte.

Background field Displays the background brightness value. Editable

Colour Curves

Use Colour Curves to access a clip's colour menu, remove colour spill from a front clip, and perform a hue shift.



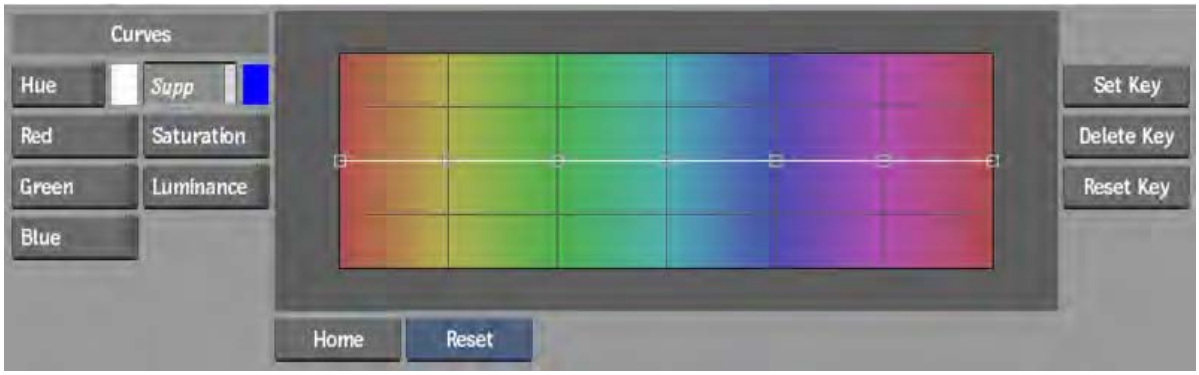
To access the Colour Curves menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Colour Curves Menu Settings

General Settings



Curve Channel button Enable this button to select the colour curve for a specific channel. Only one Curves button can be enabled at a time. Each Curves button is active when you modify its curve in the graph.

Hue colour pot Displays the colour to shift toward when the curve is modified. Editable.

Suppression colour pot Displays the colour to suppress in the clip. Editable.

Home button Reverts to the original view.

Reset box Resets the curve view.

Set Key button Sets a keyframe at the selected frame.

Delete Key button Deletes the selected keyframe.

Reset Key button Resets the curve at the selected keyframe.

Color Warper

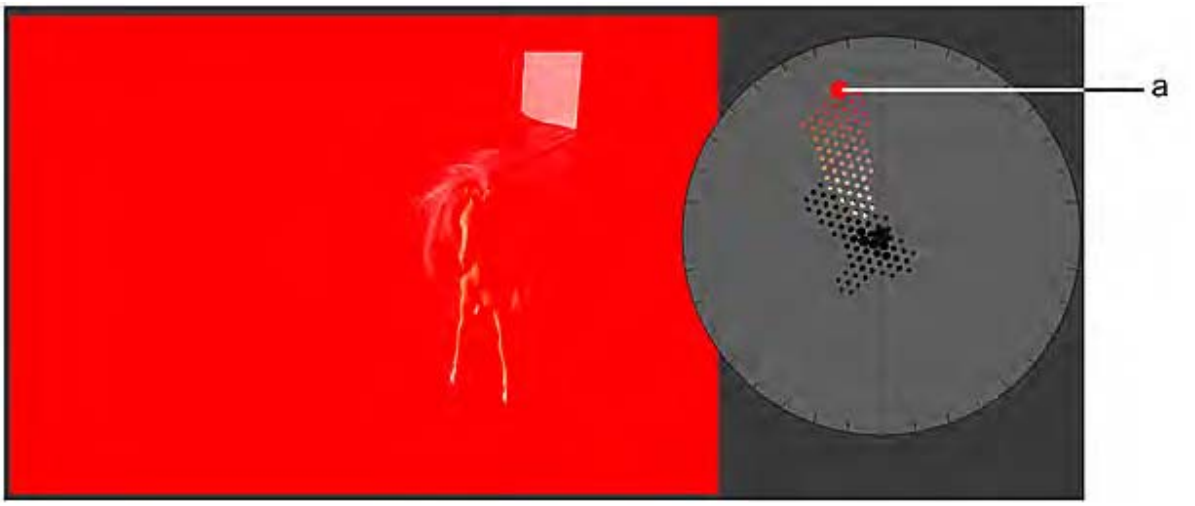
About the Colour Warper

When you colour correct an image or clip, use the Colour Warper to perform advanced colour corrections and create artistic colour effects. The way in which you approach these tasks depends on your goal, the number of clips you are using, and the type of clips being used.

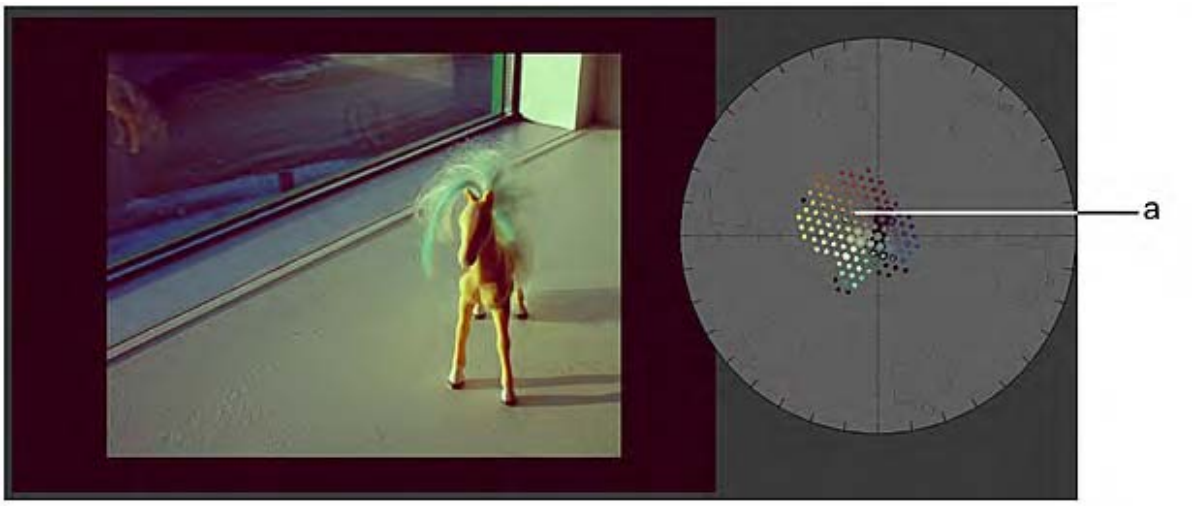
Clips created from source material shot with the same camera equipment under the same lighting conditions may be colour corrected quickly and easily to correct lighting and colour imbalances. Clips created from source material shot at different times of the day, in different seasons, at different locations, or using different equipment require more work. With the Colour Warper, you can manipulate colours with precision and ease, working on the entire clip as you would with traditional tools or working with a matte to adjust a range of colour in the clip.

Use the Colour Warper to gesturally set black and white levels, adjust specific colours and colour ranges, and accurately match colours in one clip to another. You can also perform hue shifts and suppress colour to remove colour spill or create visual effects such as a colour cast. While you manipulate the colour content of a clip, you can monitor reference clips as well as changes in the colour distribution to ensure that you achieve the result you want. Original data is always preserved, so you can adjust colours without the risk of permanently losing colour information.

When working with 16-bit floating point images in the Colour Warper, you can plot colours outside of the 0 to 1 range. Even when working with integer images, clamped colour information (colours that go beyond the RGB range) can be retrieved using the Colour Warper controls.



(a) Clamped colour



(a) Colour information restored

Accessing the Colour Warper

Use the Colour Warper to modify the colour content in your clips. You can manipulate colour content with intuitive controls that provide precise colour correction, and view histograms that help you visualize your image's colours. You can also output a matte corresponding to selected colours.

You can access the Colour Warper as a Batch or Batch FX node that you drag and drop into the process tree or the Modular Keyer's processing pipeline. Access the Colour Warper from the Modular Keyer to remove colour spill, or access it from Batch or Batch FX to modify the colour content of your clips.

The Colour Warper can be accessed from the following locations:

- The Timeline, using a Colour Correct timeline effect. (page 964)
- Batch, Then select a node from the Node bin. (page 964)
- Modular Keyer, Action or Paint. (page 965)
- Batch FX. (page 965)

When working with large images, you can free up additional screen space with the Overlay user interface. This feature is exclusive to the Colour Corrector and the Colour Warper. The Colour Warper's Overlay user interface is not available when accessing the Colour Warper as a Batch or Batch FX node.

For details on using the Overlay user interface, see Overlay User Interface (page 935).

You can access the following menus from the Colour Warper menu.

NOTE If you are accessing the Colour Warper from Batch or Batch FX or the Modular Keyer, you use the Setup menu for the node to adjust vectorscope and hue cube settings, as well as update colour information. You cannot use the Setup menu for the node to crop the colour correction area or invert a matte.

Color Warper Menu Settings

Colour box Select whether to open the Colour Corrector or the Colour Warper.

Basics button Opens the Basics menu where you perform basic colour correction, warping, suppression and saturation of your clips.

Subsetups button Opens the menu where you store, compare, and share Colour Warper subsetups.

Basics Menu

Colour correct your clips. You can adjust shadows, midtones, and highlights, and control black and white levels, hue, and saturation. You can also perform colour warping, colour suppression, and colour saturation on a limited range of colour as well as adjust gamma and luminance curves.

Home button Positions a vectorscope at its default location and size.

Scope button Enable to display a 2D or 3D vectorscope in the Result view.

Work On box Select an option to apply colour corrections to all or part of an image. Select Master to apply colour corrections to the entire image or select a selective to generate a matte and perform selective colour correction.

Clear /Reset box Select whether to clear the value at the current frame or reset the value for the entire animation curve.

Selective View box Select whether to view the matte, selective, or result image.

Clear /Reset box Select whether to clear the value at the current frame or reset the value for the entire animation curve.

Black field Displays the luminance value in the image shadows without affecting the chrominance values. Editable.

White field Displays the luminance value in the image highlights without affecting the chrominance values. Editable.

Hue field Displays the luminance value in the image shadows without affecting the chrominance values. Editable.

Saturation field Displays the global saturation value. Editable.

Red Gamma field Displays the gamma level in the red channel.

Clear /Reset box Select whether to clear the value at the current frame or reset the value for the entire animation curve.

Green Gamma field Displays the gamma level in the green channel.

Clear /Reset box Select whether to clear the value at the current frame or reset the value for the entire animation curve.

Blue Gamma field Displays the gamma level in the blue channel.

Clear /Reset box Select whether to clear the value at the current frame or reset the value for the entire animation curve.

Select button Activates the pick cursor. Use to sample an area to adjust. Activate the cursor again and select the area to match.

Plot button Activates the pick cursor. Use to sample an area to display its result clip colour value.

Ref button Activates the pick cursor. Use to sample an area to display its reference clip colour value.

Trackball option box Select a method of adjustment for a range of colour.

Luma field Displays the luminance of the destination colour. Editable.

Edit Mode box Select an edit mode to work with.

Home button Restore the histogram to its default zoom and pan settings.

Clear /Reset box Select whether to clear the value at the current frame or reset the value for the entire animation curve.

Source button Enable to show a histogram of the colour values in the front, or source clip.

Destination button Enable to show a histogram of the colour values in the result clip.

Zoom field Displays the zoom ratio of the histogram. Editable.

Selective menu

Define colour ranges using softness and tolerance to create a matte and apply colour correction to a selected region.

1 Enable to apply the matte for Selective 1 to the result.

2 Enable to apply the matte for Selective 2 to the result.

3 Enable to apply the matte for Selective 3 to the result.

Invert button Enable to invert the matte or selective.

Gaussian button Enable to apply a Gaussian blur to the matte. Disable to apply a box blur.

X field Displays the width of the blur. Editable.

Y field Displays the height of the blur. Editable.

Pick Custom button Define the tolerance range based on a sample from the image. Drag the cursor over the image to define initial tolerance.

Red button Define the tolerance range based on the red channel.

Green button Define the tolerance range based on the green channel.

Blue button Define the tolerance range based on the blue channel.

Cyan button Define the tolerance range based on the cyan channel.

Magenta button Define the tolerance range based on the magenta channel.

Yellow button Define the tolerance range based on the yellow channel.

Shadows button Define the tolerance range based on the dark areas in the image.

Midtones Define the tolerance range based on the midlevel areas in the image.

Highlights button Define the tolerance range based on the light areas in the image.

Adjusting box Select Tolerance to add tolerance to the matte, +Softness to add softness to the matte, or -Softness to remove softness from the matte.

Sharpness field Displays the percentage of sharpness of the matte. Editable.

Tolerance button Enable to apply the grey tolerance indicator on the hue cube.

Softness button Enable to apply the black softness indicator on the hue cube.

Move/Zoom box Select Move to adjust the softness and tolerance by moving the handles on the hue cube or Zoom to zoom in by dragging on the cube.

Low Softness field Displays the minimum value for the softness range. Editable.

Low Tolerance field Displays the minimum value for the tolerance range. Editable.

High Tolerance field Displays the maximum value for the tolerance range. Editable.

High Softness field Displays the maximum value for the softness range. Editable.

Frame Options box Select Home to return the hue cube to its default position, Plot Colour to enlarge the gradient to include the plot and reference colours, or Autoframe to view the complete gradient range.

Reset box Select CW to reset the Colour Warper, CC to reset the Colour Corrector, or Reset All to restore all default settings.

Auto Key button Set keyframes automatically when changes are made to a frame.

Gamma Options

RGB button Enable to display the RGB gamma curve.

Red button Enable to display the red gamma curve.

Green button Enable to display the green gamma curve.

Blue button Enable to display the blue gamma curve.

Luminance button Enable to display the luminance curve.

Warp Options

Source colour pot Displays the colour you want to modify. Editable.

Destination colour pot Displays the corrected colour.

Subsetups Menu

Store, compare, and share Colour Warper subsetups.

Current button Access the most recently committed setup.

Load button Open the file browser to load a group of subsetups within a directory.

Save button Open the file browser to save active subsetups in a specific directory.

Subsetup A LED button Indicates the subsetup is stored.

Subsetup A button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup B LED button Indicates the subsetup is stored.

Subsetup B button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup C LED button Indicates the subsetup is stored.

Subsetup C button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup D LED button Indicates the subsetup is stored.

Subsetup D button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup E LED button Indicates the subsetup is stored.

Subsetup E button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup F LED button Indicates the subsetup is stored.

Subsetup F button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup G LED button Indicates the subsetup is stored.

Subsetup Gbutton Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup H LED button Indicates the subsetup is stored.

Subsetup H button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup I LED button Indicates the subsetup is stored.

Subsetup I button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Subsetup J LED button Indicates the subsetup is stored.

Subsetup J button Apply an active subsetup to the current frame. Hold down to replace an existing subsetup.

Name field Displays the name of the subsetup. Editable.

Setup Menu

Customize your work environment by adjusting vectorscope and hue cube settings, updating colour information, cropping the area you want to colour correct, and inverting a matte.

Scope box Displays the vectorscope you want to use to view the clip.

Canvas button Enable to show the vectorscope canvas. When the canvas is off, the vectorscope is transparent, but can be outlined.

Lines button Enable to show the vectorscope outline.

Bars box Select the SMPTE bars setting for the 2D vectorscope.

Size box Select the size of the coloured squares or cubes that make up the histogram.

Clear Buffer button Clears the Undo buffer of all previous undo operations.

Levels field An upper limit for the number of undo or redo operations. Editable.

Coloured Frame

Use Coloured Frame to generate a colour bar, noise, gradient, or colour clip that can be used as the clip for other nodes.



To access the Coloured Frame menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node outputs a result.

The Coloured Frame node includes eight colour pots for storing and selecting customized colours for creating frames. You can specify a frame's resolution and bit depth.

Coloured Frame Menu Settings

General Settings



Source Type box Select whether to generate frames of solid colour, noise, or colour bars.

Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

Width field Displays the custom width resolution of the clip. Editable.

Height field Displays the custom width resolution of the clip. Editable.

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

Aspect Ratio field Displays the custom render/output aspect ratio. Editable

Frame Depth box Select the render/output frame depth of clips.

Scan Mode box Select the scan mode of clips.

Current Colour bar Displays the pattern applied to the colour source.

Luminance box Select whether to apply 75% or 100% luminance to the colour frames.

Combine

Use Combine to combine the individual color space channels of three different source clips (RGB, YUV, or HSL).



To access the Combine menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front, back and matte clip, and outputs a result.

Combine Menu Settings

General Settings



Colour Space Type box Select which type of colour space to combine.

Compound

Use Compound to combine the contents of several frames into a single frame.



To access the Compound menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip, and outputs a result.

Compound Menu Settings

General Settings



First Rendered Frame field Displays the first frame at which output is rendered from the node. Unrendered output does not display any media.Editable.

Compound Over field Displays the number of frames that are combined into one frame. Editable.

Damage

Use Damage to apply a large variety of film or video degradation effects to a clip.



To access the Damage menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip, and outputs a result.

Damage Menu Settings

General Settings



Damage Type box Select an option for the media type of damage effects to use. Selecting one of the effects groups from the Damage Type box will display a menu of options for this effect.

Random Seed field Displays the number used to generate random variations in the damage effects. Editable.

Timing Offset field Displays the number used for timing offset in the damage effects. Editable.

Damage Effect buttons Enable to display options for an effect. Click LED to switch between On (blue) and Off (grey).

Film Settings

Use the Film Damage effects to apply a large variety of film degradation effects to a clip.



When you select Film from the Damage type box, the Film Damage Effects are displayed, along with the most useful adjustment box to the right of each.



Blotches Settings

Blotches button Use this effect to simulate blotches on old film.



Blotch Amount field Displays the amount of blotches applied to the image. Editable.

Blotch Transparency field Displays the percentage of transparency applied to the blotches. Editable.

Blotch Transparency Variation field Displays the percentage of variation for the transparency applied to the blotches. Editable.

Blotch Size field Displays the average size of the blotches. Editable.

Blotch Size Variation field Displays the percentage of variation in the size of the blotches. Editable.

Blotch Softness field Displays the percentage by which the blotches are out of focus. Editable.



Blotch Negative Mix field Displays the amount of negative blotches in the mix. Editable.

Blotch colour pot Displays the tint applied to the blotches in the image. Editable.

Colour Settings

Colour button Use this effect to simulate colour fading and variation in old film.



Colour Gain trackball Drag to adjust the colour gain. This is an alternate control combining the Red Gain, Green Gain, Blue Gain, and Luma fields.

Colour Saturation field Displays the level of colour purity in the image. Editable.

Colour Gamma field Displays the amount of gamma correction applied to the image. Editable.

Colour Offset field Displays the amount by which all the colour parameters are modified. Editable.

Red Gain field Displays the amount of gain applied to the red channel. Editable.

Green Gain field Displays the amount of gain applied to the green channel. Editable.

Blue Gain field Displays the amount of gain applied to the blue channel. Editable.

Luma field Displays the amount of gain applied to the luminance channel. Editable.

Proportional button Enable to synchronize changes in red, green, blue, and luma channels.

Hue field Displays the colour range. Editable.

Shadows colour pot Displays the tint applied to shadows in the image. Editable.

Highlights colour pot Displays the tint applied to highlights in the image. Editable.

Minimum Colour Value field Displays the luminance value in the image shadows. Editable.

Maximum Colour Value field Displays the luminance value in the image highlights. Editable.

Dust Settings

Dust button Use this effect to simulate dust particles on old film.



Dust Amount field Displays the amount of dust particles applied to the image. Editable.

Dust Transparency field Displays the percentage of transparency applied to the dust particles. Editable.

Dust Transparency Variation field Displays the percentage of variation in transparency applied to the dust particles. Editable.

Dust Size field Displays the average size of the dust particles. Editable.

Dust Size Variation field Displays the percentage of variation applied to the size of the dust particles. Editable.

Dust Softness field Displays the percentage by which the dust is out of focus. Editable.



Dust Negative Mix field Displays the amount of negative dust particles in the mix. Editable.

Dust colour pot Displays the tint applied to the dust particles in the image. Editable.

Grain Settings

Grain button Use this effect to simulate grain on old film.



Grain Coloured button Enable for the grain to be coloured. Disable for the grain to be monochrome.

Grain Transparency field Displays the percentage of transparency applied to the grain. Editable.

Grain Blur field Displays the amount of blur applied th the grain. Editable.

Grain Width field Displays the width of the grain in pixels. Editable.

Grain Height field Displays the height of the grain in pixels. Editable.

Grain Proportional button Enable to effect the width and height proportionally. Editable.

Scratches Settings

Scratches button Use this effect to simulate scratches in old film.



Scratch Amount field Displays the amount of scratches applied to the image. Editable.

Scratch Width field Displays the average width of the scratches. Editable.

Scratch Width Variation field Displays the percentage of variation in the width of the scratches. Editable.

Scratch Length field Displays the average length of the scratches. Editable.

Scratch Length Variation field Displays the percentage of variation in the length of the scratches. Editable.

Scratch Transparency field Displays the transparency of the scratches. Editable.

Scratch Discontinuity field Displays the percentage of gaps that occur in the scratches. Editable.

Scratch Roughness field Displays the amount of irregularities and changes in alignment of the scratches. Editable.



Scratch Negative Mix field Displays the amount of negative scratches in the mix. Editable.

Scratch colour pot Displays the tint applied to the scratches. Editable.



Scratch Travel field Displays the amount in pixels that the scratches move along. Editable.

Scratch Travel Variation field Displays a percentage of variation that the scratches travel. Editable.

Scratch Angle Variation field Displays the angle at which the scratches occur. Editable.

Scratch Spread field Displays the area of the frame over which the scratches are spread. Editable.

Scratch Horizontal Offset field Displays the horizontal offset of the scratches. Editable.



Scratch Duration field Displays the duration in frames of the scratch effect. Editable.

Scratch Duration Variation field Displays the variation in duration for the scratch effects. Editable.

Splices Settings

Splices button Use this effect to simulate splices on old film.



Splice Auto box Select whether the splice occurs automatically or is manually inserted. Select Manual to display the Create Splice and Delete Splice buttons.

Splice Type box Select what kind of splice to be applied to the image.



Splice Border Width field Displays the width of the splice border. Editable.

Splice colour pot Displays the colour used for the splice effects. Editable.



Splice Projector Roll field Displays the amount of space that the frame will travel as a percentage. Editable.

Splice Motion Blur field Displays the amount of motion blur applied to the splice effects. Editable.

Splice Jitter field Displays the amount of jitter applied to the splice effects. Editable.

Splice Defocus field Displays the amount of defocus applied to the splice effects. Editable.



Splice Spacing field Displays the spacing in frames between splice effects. Editable.

Splice Spacing Variation field Displays the percentage of variation in spacing between splice effects. Editable.

Splice Duration field Displays the duration in frames of the splice effects. Editable.

Splice Duration Variation field Displays the percentage of variation in duration for the splice effects. Editable.

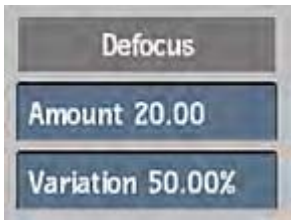


Create Splice button Creates a splice at the current frame. Available when Splice Auto is set to Manual.

Delete Splice button Deletes the splice at the current frame. Available when Splice Auto is set to Manual.

Defocus Settings

Defocus button Use this effect to simulate projection defocus on old film.



Defocus Amount field Displays the amount of defocus applied to the image. Editable.

Defocus Amount Variation field Displays the percentage of variation in the amount of defocus applied to the image. Editable.



Defocus Spacing field Displays the spacing in frames between defocus effects. Editable.

Defocus Spacing Variation field Displays the percentage of variation in the spacing between defocus effects. Editable.

Defocus Duration field Displays the duration in frames of the defocus effects. Editable.

Defocus Duration Variation field Displays the percentage of variation in duration for the defocus effects. Editable.

Flicker Settings

Flicker button Use this effect to simulate frame flicker on old film.



Flicker Amount field Displays the amount of flicker added to the image. Editable.

Flicker Amount Variation field Displays the percentage of variation in the amount of flicker added to the image. Editable.

Flicker Balance field Displays the ratio of dark and bright flickers used for the effect. Editable.



Flicker Spacing field Displays the spacing in frames between flicker effects. Editable.

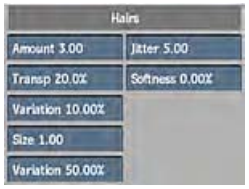
Flicker Spacing Variation field Displays the percentage of variation in the spacing between flicker effects. Editable.

Flicker Duration field Displays the duration in frames of the flicker effects. Editable.

Flicker Duration Variation field Displays the percentage of variation in duration for the flicker effects. Editable.

Hairs Settings

Hairs button Use this effect to simulate hairs on old film.



Hair Amount field Displays the amount of hairs applied to the image. Editable.

Hair Transparency field Displays the transparency applied to the hairs. Editable.

Hair Transparency Variation field Displays the percentage of variation in transparency applied to the hairs. Editable.

Hair Size field Displays the average size of the hairs. Editable.

Hair Size Variation field Displays the percentage of variation in size of the hairs. Editable.

Hair Jitter field Displays the amount of jitter applied to the hairs. Editable.

Hair Softness field Displays the percentage by which the hair is out of focus. Editable.



Hair Duration field Displays the duration in frames of the hair effects. Editable.

Hair Duration Variation field Displays the percentage of variation in duration for the hair effects. Editable.

Jitter Settings

Jitter button Use this effect to simulate projection jitter on old film.



Jitter X Amount field Displays the amount of horizontal movement applied to the jitter effect. Editable.

Jitter Y Amount field Displays the amount of vertical movement applied to the jitter effect. Editable.

Jitter Amount Variation field Displays the percentage of variation applied to the movement of the jitter effect. Editable.



Jitter Spacing field Displays the spacing in frames between jitter effects. Editable.

Jitter Spacing Variation field Displays the percentage of variation in the spacing between jitter effects. Editable.

Jitter Duration field Displays the duration in frames of the jitter effects. Editable.

Jitter Duration Variation field Displays the percentage of variation in duration for the jitter effects. Editable.

Vignette Settings

Vignette button Use this effect to simulate vignetting on old film.



Vignette Scale field Displays the size of the vignette effect compared to the size of the frame. Editable.

Vignette Ratio field Displays the ratio of width to height of the vignette effect. Editable.

Vignette Transparency field Displays the amount of transparency applied to the vignette effect. Editable.

Vignette Softness field Displays the amount of softness added to the edge of the vignette effect. Editable.

Analog Video Settings

Use the Analog Video Damage effects to apply a large variety of analog video degradation effects to a clip.



When you select Analog Video from the Damage type box, the Analog Video Damage Effects are displayed, along with the most useful adjustment box to the right of each.



Analog Drop Settings

Analog Drops button Use this effect to simulate analog drops causing noise or shash when playing back old tapes on VTRs. .



Drop Type box Select an option for the type of noise.



Drop Amount field Displays the amount of shash applied to the image. Editable.

Drop Amount Variation field Displays the percentage of variation in the amount of shash applied to the image. Editable.

Drop Transparency field Displays the percentage of transparency applied to the shash. Editable.

Drop Transparency Variation field Displays the percentage of variation in the transparency applied to the shash. Editable.



Drop Type Variation field Displays the percentage of variation applied to the drop type. Editable.

Distortion Settings

Distortion button Use this effect to simulate edge distortion when playing back old tapes on VTRs.



Edge Distortion Amount field Displays the amount of edge distortion applied to the image. Editable.

Edge Distortion Amount Variation field Displays the percentage of variation in the amount of edge distortion. Editable.

Edge Distortion Transparency field Displays the percentage of transparency applied to the edge distortion effect. Editable.

Edge Distortion Transparency Variation field Displays the percentage of variation in transparency applied to the edge distortion effect. Editable.

Edge Distortion Coverage field Displays the percentage of vertical spread for the edge distortion effect. Editable.

Edge Distortion Scale field Displays the size scaling applied to the edge distortion effect. Editable.

Edge Distortion Vertical Offset field Displays the vertical offset applied to the edge distortion effect. Editable.



Edge Distortion Speed field Displays the speed at which the edge distortion effect moves across the image. Editable.

Edge Distortion Speed Variation field Displays the percentage of variation in the speed at which the edge distortion effect moves. Editable.

Edge Distortion Speed Variation Period field Displays the time period in frames within which the speed variation occurs. Editable.

Interference Settings

Interference button Use this effect to simulate electromagnetic Interference when playing back old tapes on VTRs.



Interference Amount field Displays the amount of electromagnetic interference applied to the image. Editable.

Interference Transparency field Displays the percentage of variation in the amount of electromagnetic interference applied to the image. Editable.

Interference Coverage field Displays the percentage of vertical spread applied to the electromagnetic interference effect. Editable.

Interference Scale field Displays the scaling applied to the electromagnetic interference effect. Editable.

Interference Vertical Offset field Displays the vertical offset applied to the electromagnetic interference effect. Editable.



Interference Spacing field Displays the spacing in frames between electromagnetic interference effects. Editable.

Interference Spacing Variation field Displays the percentage of variation in spacing between electromagnetic interference effects. Editable.

Interference Duration field Displays the duration in frames of the electromagnetic interference effects. Editable.

Interference Duration Variation field Displays the percentage of variation in duration for the electromagnetic interference effects. Editable.

Lines Settings

Lines button Use this effect to simulate ones caused by noise artifacts when playing back old tapes on VTRs.



Lines Distance Amount field Displays the amount of distance between the noise artifact lines applied to the image. Editable.

Lines Jittering Amount Variation field Displays the percentage of variation in the jitter applied to the lines. Editable.

Lines Transparency field Displays the percentage of transparency applied to the noise artifact lines. Editable.

Lines Transparency Variation field Displays the percentage of variation in the transparency of the lines. Editable.

Lines Angle field Displays the angle of the noise artifact lines. Editable.



Lines Speed field Displays the speed at which the noise artifact lines move around. Editable.

Lines Speed Variation field Displays the percentage of variation in the speed of the noise artifact lines. Editable.

Lines Speed Variation Period field Displays the time period in frames within which the noise artifact lines occur. Editable.

Play Effects Settings

Play Effects button Use this effect to simulate play effects caused by misaligned or dirty tape heads when playing back old tapes on VTRs.



Play Effects Amount field Displays the amount of play effects added to the image. Editable.

Play Effects Speed field Displays the speed at which the play effects occur. Editable.

Play Effects Freeze field Displays the number of repeat frames added to create a jerky play effect. Editable.



Play Effects Spacing field Displays the spacing in frames between play effect errors. Editable.

Play Effects Spacing Variation field Displays the percentage of variation in spacing between play effect errors. Editable.

Play Effects Duration field Displays the duration in frames of the play effect errors. Editable.

Play Effects Duration Variation field Displays the percentage of variation in duration of the play effects. Editable.

Vertical Offset Settings

Vertical Offset button Use this offset to simulate vertical rolls when playing back old tapes on VTRs.



Vertical Offset Roll Amount field Displays the amount of vertical roll applied to the image. Editable.

Vertical Offset Roll Amount Variation field Displays the percentage of variation in the amount of the vertical roll. Editable.

Vertical Offset Jitter field Displays the amount of jitter applied to the vertical roll. Editable.

Vertical Offset Border Width field Displays the width of the vertical roll border. Editable.

Vertical Offset Border Artifacts field Displays the amount of artifacts within the vertical roll border. Editable.



Vertical Offset Spacing field Displays the spacing in frames between vertical roll errors. Editable.

Vertical Offset Spacing Variation field Displays the percentage of variation in spacing between vertical roll errors. Editable.

Vertical Offset Duration field Displays the duration in frames of the vertical roll errors. Editable.

Vertical Offset Duration Variation field Displays the percentage of variation in duration for the vertical roll errors. Editable.

Blur Settings

Blur button Use this effect to simulate blur from poor signal reception.



Blur Amount field Displays the amount of blurring added to the image. Editable.

Blur Amount Variation field Displays the percentage of variation in the amount of the blur effect. Editable.

Colour Settings

Colour button Use this effect to simulate analog colour degradation, shift and variations from poor signal reception.



Analog Colour Degrade Amount field Displays the amount of colour degradation applied to the image. Editable.

Analog Colour Degrade Amount Variation field Displays the percentage of variation in the amount of the colour degradation. Editable.



Analog Colour Contrast Transparency field Displays the amount of contrast applied to the image. Editable.

Analog Colour Brightness Transparency Variation field Displays the amount of brightness applied to the image. Editable.

Analog Colour Saturation field Displays the amount of colour saturation applied to the image. Editable.

Analog Colour Shadows colour pot Displays the tint applied to shadows in the image. Editable.

Analog Colour Highlights colour pot Displays the tint applied to highlights in the image. Editable.



Analog Colour Red Channel Gain field Displays the amount of gain applied to the red channel. Editable.

Analog Colour Green Channel Gain field Displays the amount of gain applied to the green channel. Editable.

Analog Colour Blue Channel Gain field Displays the amount of gain applied to the blue channel. Editable.

Analog Colour Red Channel X Shift field Displays the amount of horizontal offset applied to the red channel. Editable.

Analog Colour Green Channel X Shift field Displays the amount of horizontal offset applied to the green channel. Editable.

Analog Colour Blue Channel X Shift field Displays the amount of horizontal offset applied to the blue channel. Editable.

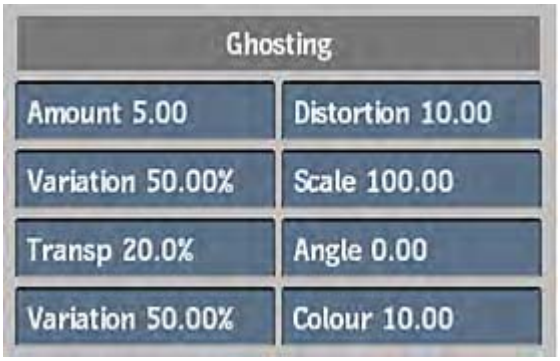
Analog Colour Red Channel Y Shift field Displays the amount of vertical offset applied to the red channel. Editable.

Analog Colour Green Channel Y Shift field Displays the amount of vertical offset applied to the green channel. Editable.

Analog Colour Blue Channel Y Shift field Displays the amount of vertical offset applied to the blue channel. Editable.

Ghosting Settings

Ghosting button Enable to simulate ghosting from poor signal reception.



Ghosting Amount field Displays the number of repeated ghost images applied to the original image. Editable.

Ghosting Amount Variation field Displays the percentage of variation in the number of ghost images. Editable.

Ghosting Transparency field Displays the percentage of the transparency applied to the ghosted images. Editable.

Ghosting Transparency Variation field Displays the percentage of variation in the transparency of the ghosted images. Editable.

Ghosting Distortion field Displays the amount of distortion applied to the ghosted images. Editable.

Ghosting Scale field Displays the amount of scaling applied to the ghosted images. Editable.

Ghosting Angle field Displays the angle applied to the ghosted images. Editable.

Ghosting Colour Degradation field Displays the amount of colour degradation applied to the ghosted images. Editable.



Ghosting Spacing field Displays the spacing in frames between ghosting errors. Editable.

Ghosting Spacing Variation field Displays the percentage of variation in spacing between ghosting errors. Editable.

Ghosting Duration field Displays the duration in frames of the ghosting errors. Editable.

Ghosting Duration Variaton field Displays the percentage of variation in the duration of ghosting errors. Editable.

Scanlines Settings

Scanlines button Use this effect to simulate scanlines from poor signal reception.



Scanlines Scale field Displays the size of the scanlines applied to the image. Editable.

Scanlines Variation field Displays the percentage of variation in the size of the scanlines. Editable.

Scanlines Variation Speed field Displays the speed at which the scanline variation occurs. Editable.

Scanlines Transparency field Displays the percentage of transparency applied to the scanlines.

Scanlines Vertical Offset field Displays the vertical offset for the scanline effect. Editable.

Scanlines Softness field Displays the amount of softness applied to the scanlines. Editable.

Scanlines Angle field Displays the angle at which the scanlines occur. Editable.

Snow Settings

Snow button Enable to simulate snow from poor signal reception.



Snow Amount field Displays the amount of snow applied to the image. Editable.

Snow Amount Variation field Displays the percentage of variation in the amount of snow applied to the image.

Snow Transparency field Displays the percentage of transparency or softness, applied to the snow. Editable.

Snow Transparency Variation field Displays the percentage of variation in the transparency or softness, applied to the snow. Editable.

Snow Sparseness field Displays the density of the snow. Editable.

Snow Width field Displays the width in pixels of the snow particles. Editable.

Snow Height field Displays the height in pixels of the snow particles. Editable.

Snow Proportional button Enable to effect the width and height proportionally. Editable.

TV Distort Settings

TV Distort button Use this effect to simulate TV lens distortion from a poor video camera.



TV Distort Amount field Displays the amount of TV lens distortion applied to the image. Editable.

TV Distort Amount Variation field Displays the percentage of variation in the amount of TV lens distortion applied to the image.

TV Distort Radius field Displays the size of the image to which the TV lens distortion is applied. A value of 1 includes the entire image. Editable.

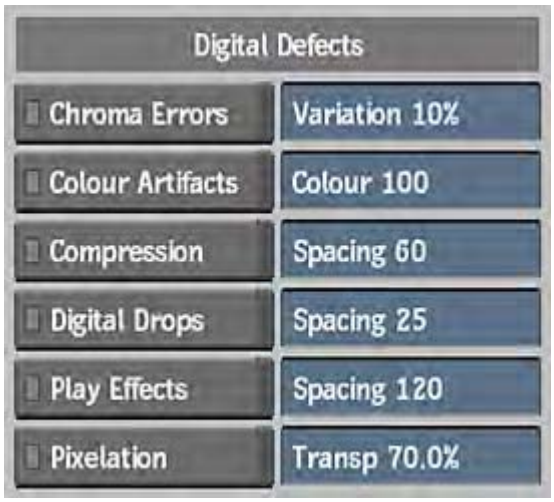
TV Distort Ratio field Displays the ratio of height to width that is affected by the TV lens distortion effect. Editable.

Digital Video Settings

Use the Digital Video Damage effects to apply a large variety of digital video degradation effects to a clip.



When you select Digital Video from the Damage type box, the Digital Video Damage Effects are displayed, along with the most useful adjustment box to the right of each.



Chroma Errors Settings

Chroma Errors button Use this effect to simulate chroma errors in badly encoded digital video transmissions.



Chroma Amount box Displays the value representing the change to the chroma channel. Editable.

Luma Amount box Displays the value representing the change to the luminance channel. Editable.

Variation Amount box Displays the percentage of variation applied to the Chroma and Luma channels.

Colour Artifacts Settings

Colour Artifacts button Use this effect to simulate digital colour artifacts and degradation appearing in badly encoded digital video transmissions.



Colour Degradation Amount box Displays the amount of digital colour degradation applied to the image. Editable.

Colour Degradation Variation box Displays the percentage of variation in the amount of colour degradation applied to the image. Editable.



Contrast box Displays the amount of contrast applied to the image. Editable.

Brightness box Displays the amount of brightness applied to the image. Editable.

Tint Colour box Displays the tint value applied to the colour in the image. Editable.

Shadows colour pot Displays the tint applied to shadows in the image. Editable.

Highlights colour pot Displays the tint applied to highlights in the image. Editable.

Compression Settings

Compression button Use this effect to simulate digital compression errors in badly encoded digital video transmissions.



Compression Amount field Displays the amount of the digital compression effect applied to the image. Editable.

Compression Amount Variation field Displays the percentage of variation in the amount of digital compression applied to the image. Editable.



Compression Edges Active button Enable to activate edge detection errors in compression.

Compression Edges Minimum field Displays the minimum threshold value used to detect edges. Available when Active button is enabled. Editable.

Compression Edges Maximum field Displays the maximum threshold value used to detect edges. Available when Active button is enabled. Editable.

Compression Edges Width field Displays the width of the edge in pixels. Available when Active button is enabled. Editable.

Compression Edges Softness field Displays the softness of the edge. Available when Active button is enabled. Editable.



Compression Spacing field Displays the spacing in frames between digital compression errors. Editable.

Compression Spacing Variation field Displays the percentage of variation in spacing between compression errors. Editable.

Compression Duration field Displays the duration in frames of the compression errors. Editable.

Compression Duration Variation field Displays the percentage of variation in duration of the compression errors. Editable.

Digital Drops Settings

Digital Drops button Use this effect to simulate digital drop errors in badly encoded digital video transmissions.



Digital Drops Amount field Displays the amount of digital drop errors applied to the image. Editable.

Digital Drops Coherence field Displays the amount of coherence to digital drop errors in the image. Editable.

Digital Drops Amount Variation field Displays the percentage of variation in the amount of digital drop errors applied to the image. Editable.



Digital Drops Spacing field Displays the spacing in frames between digital drop errors. Editable.

Digital Drops Spacing Variation field Displays the percentage of variation in spacing between digital drop errors. Editable.

Digital Drops Freeze Frame button Displays the number of repeat frames added to create a jerky digital drop effect. Editable.

Digital Drops Duration field Displays the duration in frames of the digital drop errors. Editable.

Digital Drops Duration Variation field Displays the percentage of variation in duration for the digital drop errors. Editable.

Play Effect Settings

Play Effects button Use this effect to simulate digital play effects in badly encoded digital video transmissions.



Play Effects Speed field Displays the amount of change to the speed of playback applied to the image. Editable.

Play Effects Shuttering field Displays the amount of shuttering during playback that is applied to the image. Editable.



Play Effects Spacing field Displays the spacing in frames between play effects errors. Editable.

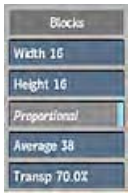
Play Effects Spacing Variation field Displays the percentage of variation in spacing between play effects errors. Editable.

Play Effects Duration field Displays the duration in frames of the play effects errors. Editable.

Play Effects Duration Variation field Displays the percentage of variation in the duration for play effects errors. Editable.

Pixelation Settings

Pixelation button Use this effect to simulate reduced resolution and digital pixelation errors in badly encoded digital video transmissions.



Pixelation Width Amount field Displays the width in pixels of the new pixelation block. Editable.

Pixelation Height Amount field Displays the height in pixels of the new pixelation block. Editable.

Pixelation Proportional button Enable to effect the width and height proportionally. Editable.

Pixelation Average button Displays the amount of averaging applied between the pixel block and its surroundings. Editable.

Pixelation Transparency field Displays the percentage of transparency, or softness, applied to the pixel blocks. Editable.



Pixelation X Offset field Displays the horizontal offset for the pixelation effect. Editable.

Pixelation Y Offset field Displays the vertical offset for the pixelation effect. Editable.



Pixelation Size Variation field Displays the percentage of variation in the size of pixels during playback. Editable.

Deal

Use Deal to deal out the frames of a single clip evenly to any number of destination clips.



To access the Deal menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip, and outputs a result.

The Clip field defines the number of times the clip is split; the Frame determines which of the dealt clips is output. The First Processed Frame field set the value at which output is processed from the node. Unprocessed output does not display any media.

Deal Menu Settings

General Settings



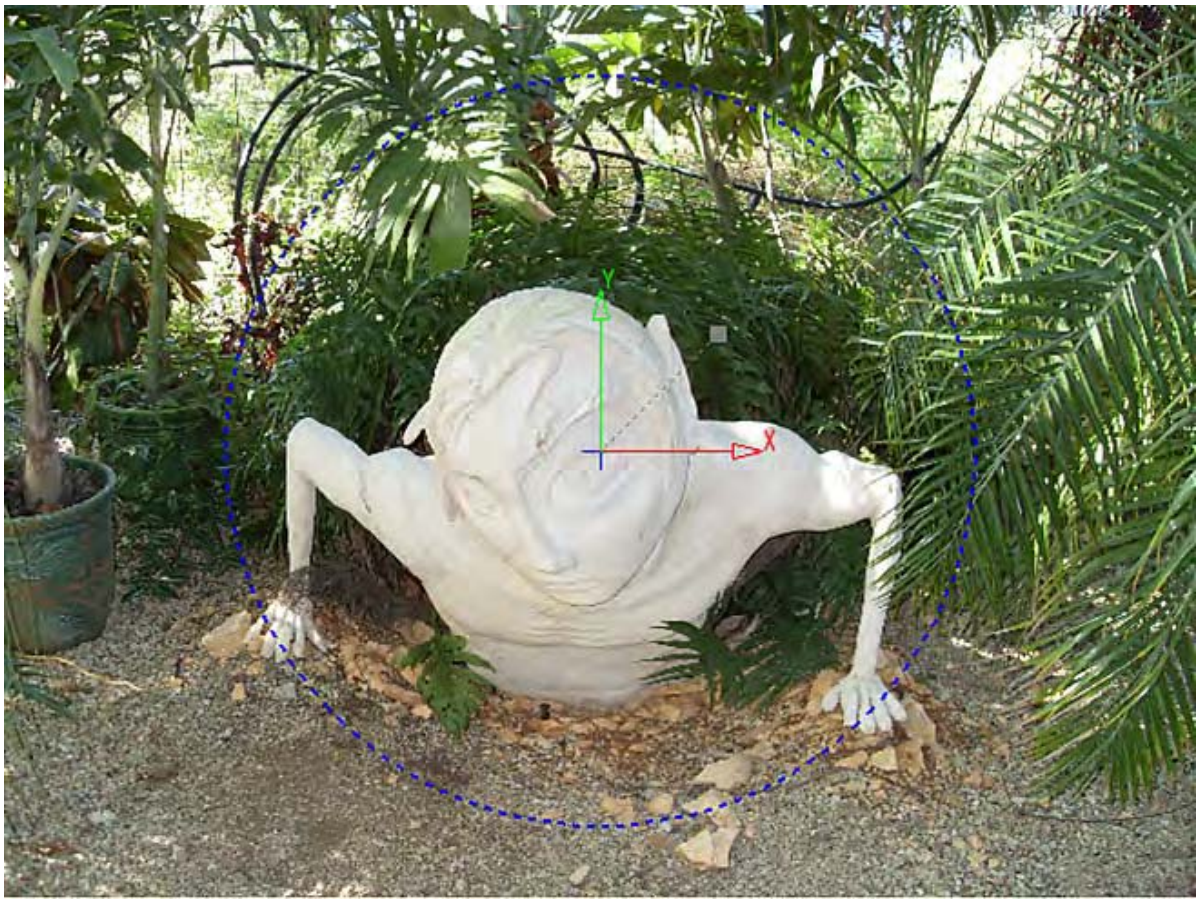
First Rendered Frame field Displays the first frame at which output is rendered from the node. Unrendered output does not display any media.Editable.

Clips field Displays the number of times the clip is split. Editable.

Start Frame field Displays which frame of the input clip used to determine which dealt clip is output. Editable.

Deform

Use Deform to apply various types of deformation effects to clips.



To access the Deform menu, use:

- Batch, then select a node from the Node bin.

- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front and a matte clip, and outputs a result and an outmatte.

Deform Menu Settings

General Settings



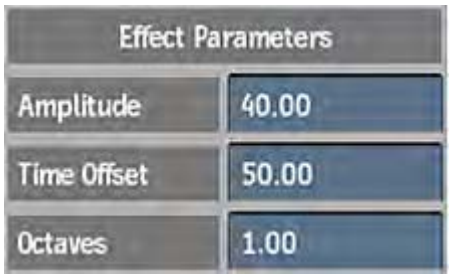
Deform Type button Select the type of deformation you want to create. The available effect parameters change based on the selection.

Deform Ripple box (not shown) Select how you want the ripples created. Pond Ripples create ripples with a 45 degree offset. Out from Center pushes the ripples away from the center, extending the first half, and compressing the second half of each ripple. Around Center rotates the crest of each ripple in a clockwise direction around the center.

Repeat Mode box Select an option to fill the empty portions of the frame.

Effect Parameters

NOTE The following settings are available when Crumple is selected from the Deform Type button.

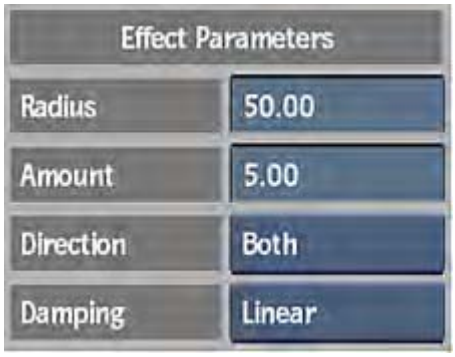


Amplitude field Displays the amount of deformation. Increase the value to increase the effect. Editable.

Time Offset field Displays the time offset interval of the crumpling. Editable.

Octaves field Displays a value for the number of layers summed in the operation, from zero to 10. Increase the value to increase the fractal effect. Editable.

NOTE The following settings are available when Magnify is selected from the Deform Type button.



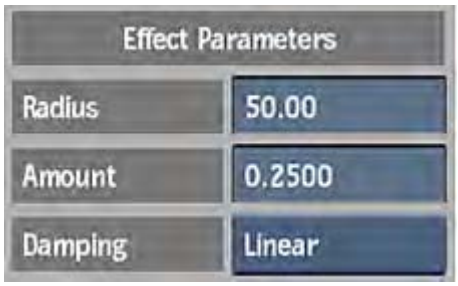
Radius field Displays the size of the affected area. You can also drag the circle in the image window to change the size of the radius. Editable.

Amount field Displays the amount of the deformation. Editable.

Direction box Select Horizontal, Vertical, or Both to indicate the direction of the magnification.

Damping box Select how the effect is applied within the radius. Select None to apply no damping, Linear to decrease the effect linearly to 0, or Quadratic to decrease the effect quadratically to 0.

NOTE The following settings are available when Pinch is selected from the Deform Type button.

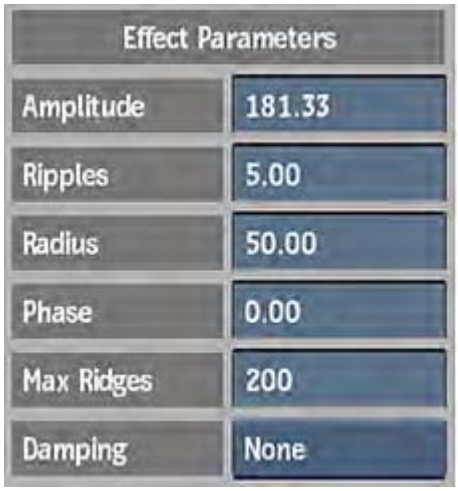


Radius field Displays the size of the affected area. You can also drag the circle in the image window to change the size of the radius. Editable.

Amount field Displays the amount of the deformation. Editable.

Damping box Select how the effect is applied within the radius. Select None to apply no damping, Linear to decrease the effect linearly to 0, or Quadratic to decrease the effect quadratically to 0.

NOTE The following settings are available when Ripple is selected from the Deform Type button.



Amplitude field Displays the amount of deformation. Increase the value to increase the effect. Editable.

Ripples field Displays the frequency of the ripples (from zero to 60). Editable.

Radius field Displays the size of the affected area. You can also drag the circle in the image window to change the size of the radius. Editable.

Phase field Displays the spread of the ripples to the centre. Use this value to animate the ripple effect. Editable.

Max Ridges field Displays the total number of ripples that can be generated. For example, set to 5 to create 5 rings of distortion. Editable.

Damping box Select how the effect is applied within the radius. Select None to apply no damping, Linear to decrease the effect linearly to 0, or Quadratic to decrease the effect quadratically to 0.

NOTE The following settings are available when Twirl is selected from the Deform Type button.

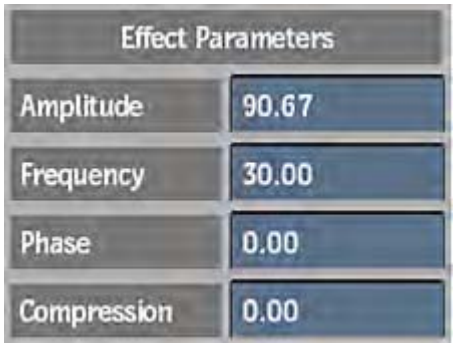


Radius field Displays the size of the affected area. You can also drag the circle in the image window to change the size of the radius. Editable.

Twirl Angle field Displays the direction of the twirl. Editable.

Damping box Select how the effect is applied within the radius. Select None to apply no damping, Linear to decrease the effect linearly to 0, or Quadratic to decrease the effect quadratically to 0.

NOTE The following settings are available when Wave is selected from the Deform Type button.



Amplitude field Displays the amount of deformation. Increase the value to increase the effect. Editable.

Wave Frequency field Displays the number of waves. Editable.

Phase field Displays the spread of the ripples to the centre. Use this value to animate the ripple effect. Editable.

Compression field Displays a pixel flattening value (negative values flatten to the left, and positive values flatten to the right). Editable.

Transform Settings



NOTE Transform settings for Input, Deform and Output are identical.

Transform X Position field Displays the horizontal position of the transformation. Enable the Icons button to change the position by dragging the vertex tool in the image window. Editable.

Transform Y Position field Displays the vertical position of the transformation. Editable. Enable the Icons button to change the position by dragging the vertex tool in the image window. Editable.

Transform Rotation field Displays the rotation of the transformation. Editable. Enable the Icons button to change the rotation by dragging the vertex tool in the image window. Editable.

Transform Centre X field Displays the centre point value of the transform along the horizontal axis. Editable.

Transform Centre Y field Displays the centre point value of the transform along the vertical axis. Editable.

Transform X Scale field Displays the horizontal scale factor. Editable.

Transform Y Scale field Displays the vertical scale factor. Editable.

Proportional button Enable to scale X and Y values proportionally.

Icons button Enable to display the vertex editing tool in the image window.

Anti-Aliasing Settings



Active button Enable to activate software anti-aliasing.

Anti-Aliasing Sampling box Select the number of samples to use in the anti-aliasing process.

Anti-Aliasing Softness field Displays the softness value for software anti-aliasing. Editable.

Degrain

Use Degrain to remove grain from the RGB channels of a selected colour in an image.



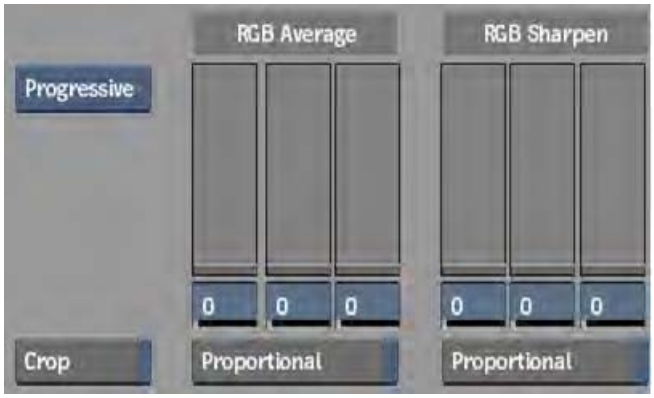
To access the Degrain menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Degrain Menu Settings

General Settings



Render Method box Select whether you are removing grain from progressive or interlaced media.

Crop box Applies the Degrain to a specific region in the clip.

Red Average slider Displays the amount of blur applied to the red channel.

Red Average field Displays the amount of blur applied to the red channel. Editable.

Green Average slider Displays the amount of blur applied to the green channel.

Green Average field Displays the amount of blur applied to the green channel. Editable.

Blue Average slider Displays the amount of blur applied to the blue channel.

Blue Average field Displays the amount of blur applied to the blue channel. Editable.

Proportional RGB Average button Enable to blur RGB channels proportionally.

Red Sharpen slider Displays the amount of sharpness applied to the red channel.

Red Sharpen field Displays the amount of sharpness applied to the red channel. Editable.

Green Sharpen slider Displays the amount of sharpness applied to the green channel.

Green Sharpen field Displays the amount of sharpness applied to the green channel. Editable.

Blue Sharpen slider Displays the amount of sharpness applied to the blue channel.

Blue Sharpen field Displays the amount of sharpness applied to the blue channel. Editable.

Proportional RGB Sharpen button Enable to sharpen RGB channels proportionally.

Deinterlace

Use Deinterlace to separate the odd and even scanlines of a clip.



To access the Deinterlace menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip as input, and outputs a result.

For each frame of the clip, the result clip contains one frame with odd scanlines (Field 1) and one frame with even scanlines (Field 2).

Deinterlace Menu Settings

General Settings



First Rendered Frame field Displays the first frame at which output is rendered from the node. Unrendered output does not display any media.Editable.

Field Dominance box Select Field 1 or Field 2 dominance, or Auto to have the application detect automatically the field dominance.

Interpolation box Enable to interpolate a blend between adjacent lines and fill in the isolated scan lines. This also reduces interlacing artifacts.

Denoise

Use the Denoise effect to reduce or remove noise and grain from your source media.



To access the Denoise menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result clip.

Denoise Menu Settings

Setup Settings



Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Export button Click to export a Denoise setup to be used as a Lustre Degrain preset.

Import button Click to open the browser to select a Lustre Degrain preset to import.

Grain Analysis Settings



Add Bounding Box button Adds a new sub-region. Click and drag the bounding box in the image window to set the location.

Remove Bounding Box box Select an option to remove the currently selected bounding box or all the bounding boxes.

Analyze button Click to determine the grain structure.

Reference frame field Displays the frame number that is analysed. You can use this frame analysis as a reference that can be edited and applied to the clip.

Show Icons button Enable to display all the defined bounding boxes.

Bounding Box colour pot Displays the colour used for the border of the bounding boxes. Click to change the colour.Editable.

Denoise Settings



Denoise button Enable to use noise removal on the clip.

Red Gain field Displays the gain for red channel colour values. Editable.

Green Gain field Displays the gain for green channel colour values. Editable.

Blue Gain field Displays the gain for blue channel colour values. For film scans, the grain is often greater in this channel. Editable.

Proportional button Enable to change gain values proportionally for all three colour channels.

Grain Size field Displays a value in pixels that is proportional to the size of the grain. The default value is 3, but may be higher for 4K images. Editable.

Smoothing Radius field Displays the blur radius. For smoother results, a higher value will add more pixels to the blur, but increase rendering time. Editable.

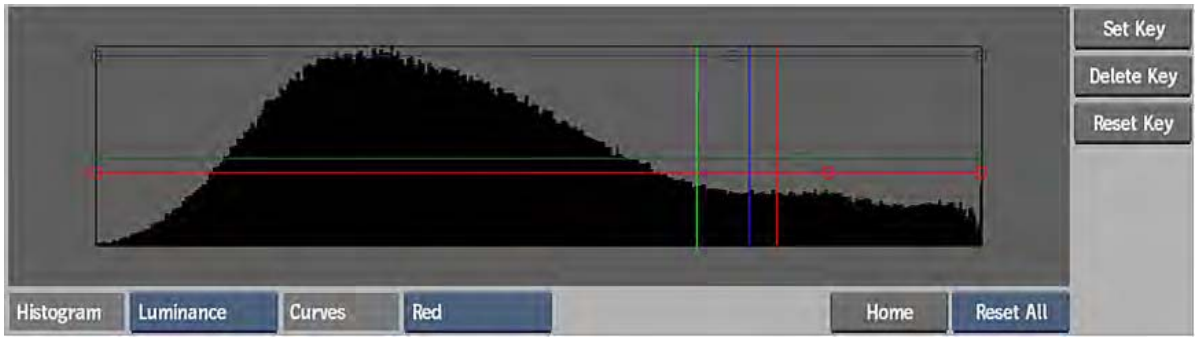
Detail field Displays the amount of detail to preserve when the Smoothing Radius is set to a high value. It is recommended you enter 0.05 to 0.15 as an initial value. Editable.

Opacity field Displays a percentage of the level of opacity between the source image and the output with the applied grain filter. Editable.

Use Past button Enable to compare with pixel data from previous frames.

Use Future button Enable to compare with pixel data from subsequent frames.

Histogram Settings



Histogram box Select to display the red, green, blue, or luminance histogram in the graph. Select Current Curve to display the histogram for the currently selected Curves Channel.

Curves Channel box Select to highlight the red, green, or blue channel curve in the graph.

Home button Reverts to the original view.

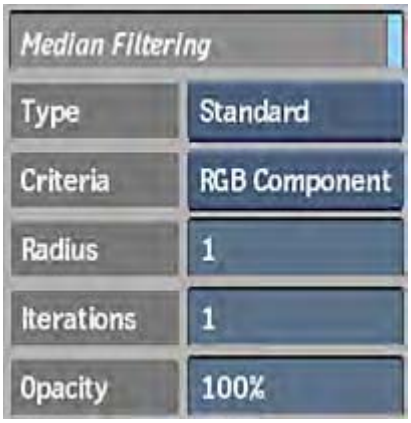
Reset box Resets the curve view.

Set Key button Sets the current values for the selected channels in the current frame.

Delete Key button Deletes the selected keyframe or curve.

Reset Key button Select to reset the current curve or all the curves to default.

Median Filtering Settings



The Median Filter removes noise by calculating the median value for each pixel (the most probable pixel value) and applies an edge-preserving smoothing filter.

Median Filtering button Enable to use the median filter.

Type box Select the filtering mode. Options are:

- Standard: Applies the standard fitering algorithm.
- Advanced: Applies a more advanced filtering algorithm, but is more resource intensive. Use the Advanced Filter Mode on particularily noisy shots.

Criteria box Select the ranking criteria by which the median value is chosen. Options are:

- Luminance :Uses the pixel luminance value to calculate the median value.
- RGB Vector: Uses the RGB coordinate values to calculate the median value.
- RGB Component: Uses the R, G and B channels independently to calculate the median value.

Radius field Displays the size of the filtering region in pixels from the centre. A higher value results in more pixels being taken into account when calculating the median value. Editable.

Iterations field Displays the number of times that the filter is applied recursively. Editable.

Opacity field Displays how much the result of one median iteration is combined with the original input. Editable.

Depth of Field

Use Depth of Field to create a blur that is applied to out-of-focus points of light to simulate a shallow focus.



To access the Depth of Field menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front, Z-depth and matte clip, and outputs a result, out Z-depth and outmatte clip. The output matte clip can have a different level of depth of field than the result clip.

Depth of Field Menu Settings

Blur and Blooming Settings



Blur Width field Displays the horizontal blur amount in pixels. Editable.

Blur Height field Displays the vertical blur amount in pixels. Editable.

Blur Proportional button Enable to constrain blur amount proportions.

Bokeh Blur field Displays the amount of smoothness applied to sharp bokeh edges. This creates the blur that is applied to out-of-focus points of light to simulate a shallow focus. Editable.

Basic/Additive Blooming button Switch between basic blooming and additive blooming bokeh effects. The basic blooming mode displays the gain applied to highlights in non-HDR images, to allow the creation of bokeh patterns without affecting colour integrity. The additive blooming mode allows you to create higher intensity bokeh patterns from any source image, using minimum and maximum thresholds for highlight segregation.

Basic Blooming field Displays the gain applied to highlights in non-HDR images, to allow the creation of bokeh patterns without affecting colour integrity. This creates the glow effect that is applied to the bright spots of the image to simulate light bleeding, or blooming, over the edges. Editable.

Additive Blooming field Displays the amount of high intensity bokeh patterns that can be created from any source image generating extreme highlight content . Editable.

Additive Blooming minimum field Displays the minimum threshold for highlight segregation. Editable.

Additive Blooming maximum field Displays the maximum threshold for highlight segregation. Editable.

Bokeh Settings



Sides field Displays the number of sides in the kernel shape. Editable, if kernel information is not attached to the node.

Rotation field Displays the angle of rotation of the kernel shape. Editable, if kernel information is not attached to the node.

Bokeh Type box Select whether to use a round or angle bokeh curve to define the kernel shape.

Curvature field Displays the amount of curvature applied to a round bokeh kernel shape. Editable.

Slices and Edge Artifacts Settings



Slices Number field Displays the number of slices to blend to determine the Z-Depth interpolation of a depth-of-field focus blur. You can see the slices in the depth-o-gram to allow a better understanding of the focus plane behaviour, and to help define the number of slices required, depending on your Z-depth colour information. Editable.

Slices Overlap field Displays the amount of blending overlap between adjacent slices. Editable.

Foreground Expand field Displays the amount of mixing between the edges of selected objects and their surroundings in the foreground. Editable.

Background Blend field Displays the amount of mixing between the edges of selected objects and the background image. Editable.

Z-Depth Settings



Z-Depth Minimum field Displays the lower limit of the Z-Depth values. Pixels with lower values are mapped to black. Editable.

Z-Depth Z Maximum field Displays the upper limit of the Z-Depth values. Pixels with higher values are mapped to white. Editable.

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

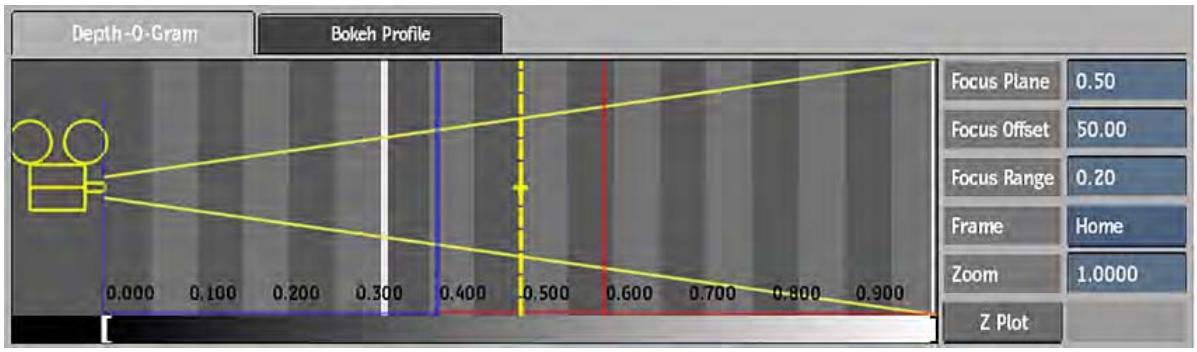
Premultiplied Depth button Enable to have the alpha channel information premultiplied with the colour depth data. Use to prevent a black halo from forming around blurred areas.

Max Blur Field Displays the maximum total horizontal and vertical blur amount to clamp the value and avoid unnecessary rendering. The cumulative effect of the high blur amount (Basic tab), and gamma and gain correction applied to the Depth map (Depth tab) may result in very large values, which greatly increases the rendering time. Editable.

Slopes field Displays the gamma value. Applies a gamma curve to the Z-depth map before it is used. Editable.

Depth-O-Gram Settings

Use the Depth-O-Gram settings to refine focus values.



Depth blur effects control the amount of blur based on a depth of field matte. Depth blur settings are displayed in the Depth-O-Gram tab.

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

Focus Plane field Displays the distance of your focus point, which is the point at which there is no blur on the image. Editable.

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

Focus Range field Displays the distance the near and far offset. Editble.

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

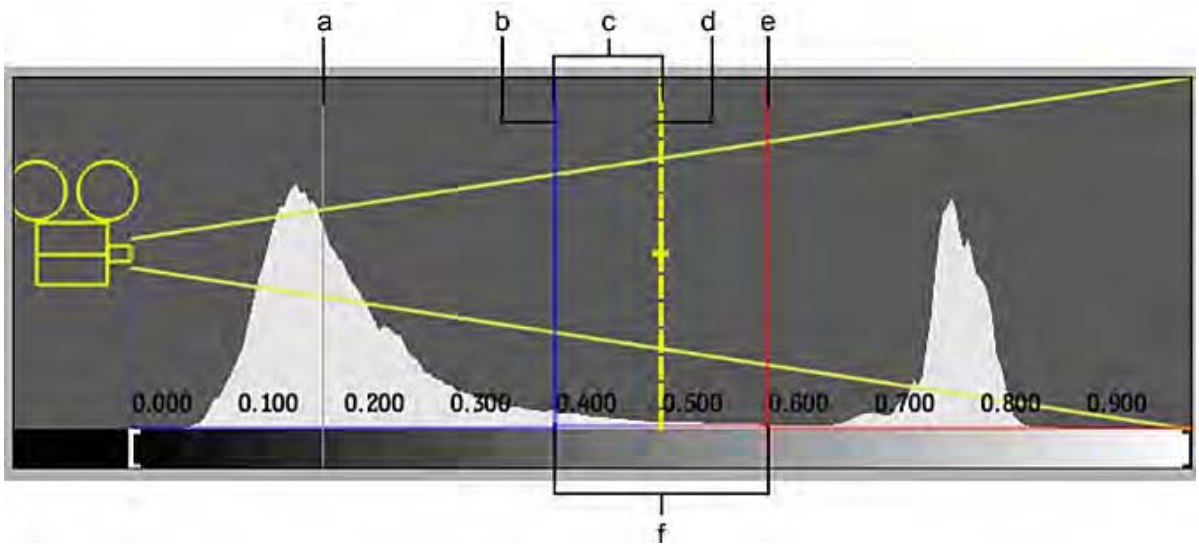
Zoom field Displays a vertical zoom value for the histogram to display. You can zoom horizontally by pressing Ctrl+spacebar and dragging left or right in the histogram. To pan horizontally, click spacebar and drag left or right in the histogram.Editable.

Z Plot button Click to activate the pick cursor. Use to select a pixel in the image to display its depth.

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

Modifying Depth of Field Gesturally

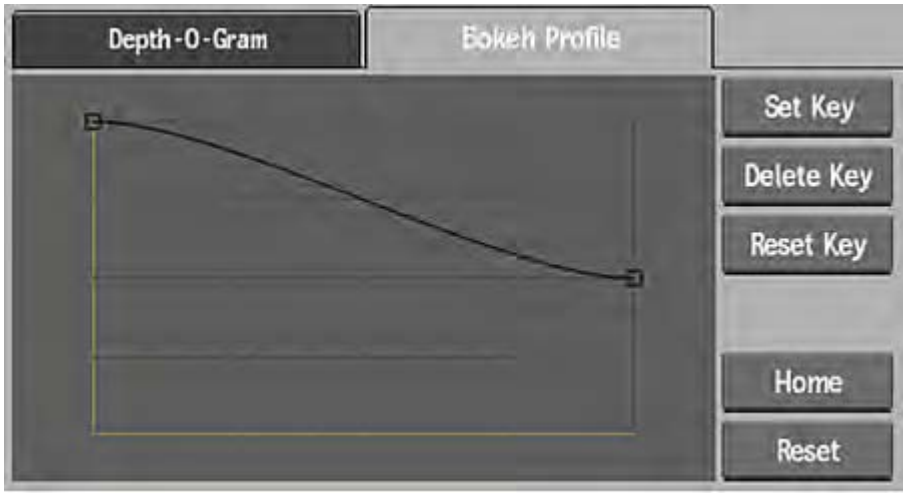
Depth blur effects can be modified gesturally by dragging the focus, near offset, and far offset planes in the graphic representation of the depth of field. As you drag these elements, the planes are also displayed in the Result view as a preview of the areas that will be in focus. Use the depth control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



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

Bokeh Profile Settings

Use the Bokeh Profile tab to edit the blur kernal pattern.



The kernel is the basic blur shape. The shape of the kernel is determined by its number of sides, its rotation, and the shape of its S-curve. This curve represents the shape of the pattern, from its centre to the outside. The default S-curve defines the softness of the blur. You can change the curve by manipulating the two points that define the curve, or you can add points to the curve. Use the Edit Mode box to add and delete points on the curve.

Set Key button Sets a keyframe at the selected frame.

Delete Key button Deletes the selected keyframe.

Reset Key button Resets the curve at the selected keyframe.

Home button Resets the profile view.

Reset button Resets the blur pattern profile curve to its default values.

Difference Matte

Use the Difference Matte to generate a matte clip from two source clips with the same background but different foreground elements.



To access the Difference Matte menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

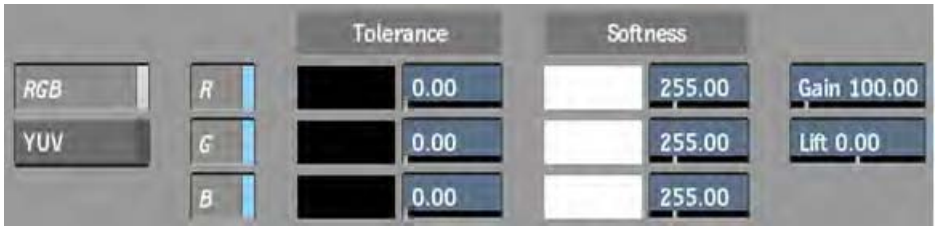
This node accepts a front and back clip as input, and outputs a result.

The Difference Matte allows you to remove an image from one context and add it to another. The matte is created using the Tolerance and Softness values that you specify.

Difference Matte Menu Settings

General Settings

You can enable the RGB or the YUV settings to generate a matte clip.



RGB Settings

RGB button Enable to use values in the RGB colour space to generate a matte.

R button Enable to use the red colour channel to create the matte.

G button Enable to use the green colour channel to create the matte.

B button Enable to use the blue colour channel to create the matte.

Red Tolerance colour pot Displays the value of the red channel used to create the matte. Editable.

Red Tolerance field Displays the value of the red channel used to create the matte. Editable.

Green Tolerance colour pot Displays the value of the green channel used to create the matte. Editable.

Green Tolerance field Displays the value of the green channel used to create the matte. Editable.

Blue Tolerance colour pot Displays the value of the blue channel used to create the matte. Editable.

Blue Tolerance field Displays the value of the blue channel used to create the matte. Editable.

Red Softness colour pot Displays the softness value applied to the red channel to create the matte. Editable.

Red Softness field Displays the softness value applied to the red channel to create the matte. Editable.

Green Softness colour pot Displays the softness value applied to the green channel to create the matte. Editable.

Green Softness field Displays the softness value applied to the green channel to create the matte. Editable.

Blue Softness colour pot Displays the softness value applied to the blue channel to create the matte. Editable.

Blue Softness field Displays the softness value applied to the blue channel to create the matte. Editable.

Gain field Displays the value that the resulting pixel values are multiplied by to create the final matte. Editable.

Lift field Displays the value added to the resulting pixels to create the final matte. Editable.

YUV Settings

YUV button Enable to use values in the YUV colour space to generate a matte.

Y button Enable to use the Y channel (luminance) to create the matte.

U button Enable to use the U channel (chrominance) to create the matte.

V button Use the V channel (chrominance) to create the matte.

Y Tolerance colour pot Displays the value of the Y channel used to create the matte. Editable.

Y Tolerance field Displays the value of the Y channel used to create the matte. Editable.

U Tolerance colour pot Displays the value of the green channel used to create the matte. Editable.

U Tolerance field Displays the value of the U channel used to create the matte. Editable.

V Tolerance colour pot Displays the value of the V channel used to create the matte. Editable.

V Tolerance field Displays the value of the V channel used to create the matte. Editable.

Y Softness colour pot Displays the softness value applied to the Y channel to create the matte. Editable.

Y Softness field Displays the softness value applied to the Y channel to create the matte. Editable.

U Softness colour pot Displays the softness value applied to the U channel to create the matte. Editable.

U Softness field Displays the softness value applied to the U channel to create the matte. Editable.

V Softness colour pot Displays the softness value applied to the V channel to create the matte. Editable.

V Softness field Displays the softness value applied to the V channel to create the matte. Editable.

Gain field Displays the value that the resulting pixel values are multiplied by to create the final matte. Editable.

Lift field Displays the value added to the resulting pixels to create the final matte. Editable.

Distort

Use Distort to create warps and morphs of clips. The node uses spline-based shapes to create these effects.



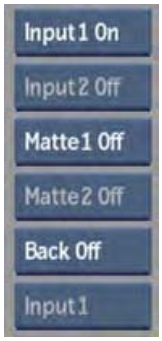
To access the Distort editors, use:

- Batch, then select a node from the Node bin. (page 1161)
- Timeline, then use Batch FX. (page 1161)
- Tools, then select the menu. (page 1161)

Distort accepts two front clips, two matte clips and one back clip as inputs, and outputs a result and an outmatte.

Distort Menu Settings

Accessing the Distort Menu



Input1 Display box Select a display mode for Input1.

Input2 Display box Select a display mode for Input2.

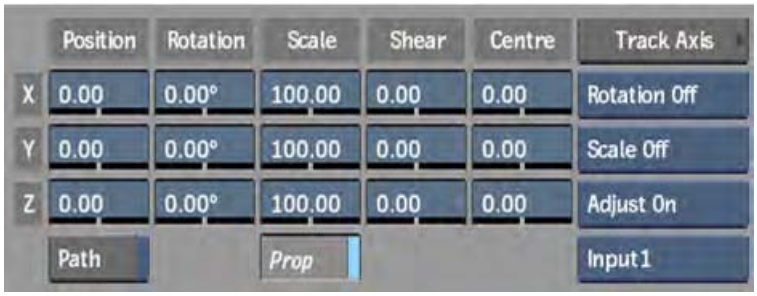
Matte1 Display box Select a display mode for Matte1.

Matte2 Display box Select a display mode for Matte2.

Back Display box Select a display mode for the background clip.

Input box Toggles between Input1 and Input2 clips based on the selection in the View box.

If Input is selected in the View box, this box toggles between Front1 and Front2. If Matte is selected in the View box, this box toggles between Matte1 and Matte2. If Result is selected in the View box, this box toggles between showing the splines for Front1 and Front2 drawn on top of the result.



X Position field Displays the position of the X axis. Editable.

Y Position field Displays the position of the Y axis. Editable.

Z Position field Displays the position of the Z axis. Editable.

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

X Rotation field Displays the rotation value of the X axis. Editable.

Y Rotation field Displays the rotation value of the Y axis. Editable.

Z Rotation field Displays the rotation value of the Z axis.

X Scale field Displays the scale value of the X axis. Editable.

Y Scale field Displays the scale value of the Y axis. Editable.

Z Scale field Displays the scale value of the Z axis. Editable.

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

X Shear field Displays the shear value of the X axis. Editable.

Y Shear field Displays the shear value of the Y axis. Editable.

Z Shear field Displays the shear value of the Z axis. Editable.

X Centre field Displays the centre value of the X axis. Editable.

Y Centre field Displays the centre value of the Y axis. Editable.

Z Centre field Displays the centre value of the Z axis. Editable.

Track Axis button Opens the Stabilizer menu to apply tracking data to an axis.

Tracking Rotation box Select whether tracking rotation is On, Off, or Inverted.

Tracking Scale box Select whether tracking scaling is On, Off, or Inverted.

Tracking Adjust box Select whether an offset is applied relative to the spline's axis.

Track Clip box Select the clip to track.



Warper button Select whether to display the Warp or Morph menu.

Interpolation field Displays the percentage of mixing between the Input1 spline and the Input2 spline when source splines are linked for source interpolation. Editable.

You independently set an interpolation value for each set of linked Front1 and Front2 source splines.

Blend field Displays the level of blend between the Input1 and Input2 clips when morphing. Editable.

NOTE The Blend field is only available from the Morph menu.



Source button Enable to display source splines for manipulation. Also use to display the unwarped input clip.

Source Show button Enable to display source splines for reference when working with destination splines.

Source colour pot Displays the colour of all source splines. Editable.

Destination button Enable to display destination splines for manipulation. Also use to display the warped input clip.

Destination Show button Enable to display destination splines for reference when working with source splines.

Destination colour pot Displays the colour of all destination splines. Editable.

Correspondences button Enable to display correspondence points and connector lines, which indicate how the source spline maps to the destination spline.

You can add, edit, animate, and delete correspondence points. The more correspondence points a spline has, the greater effect it has on the overall warp.



Active button Enable to allow greater control over an animated spline by toggling selected vertices on and off.

Lasso Fit field Displays the number of vertices in freehand segments of a spline. Editable.

Increasing the value decreases the number of vertices, while decreasing the value increases the number of vertices. Only segments of the spline drawn using freehand mode are affected, while segments created by simple clicks remain unaffected. The Lasso Fit parameter loses its influence over freehand segments of a spline if you edit vertices.

Link box Controls whether source and destination splines are linked together or manipulated independently.

Set this to Lnk Src & Dst to keep the source and destination splines linked as you draw and animate the source spline. Set this to Enable Warping to control the destination spline independently to create a warp.

You can also independently link and unlink the axis nodes that are the parents of a source spline and its corresponding destination spline.

Toggle Input button Switch between an Input1 spline and an Input2 spline.

This is useful when you have animated a spline on one input and wish to apply it to the other.



Iterations field Displays a value for the distance that the distortion follows the destination spline. Editable.

If Z-mode is disabled, the maximum distance of the distortion is limited to avoid overlapping. When there is a large distance between a source and destination spline, you will achieve better results by increasing the number of iterations. For smaller warps or morphs, there is no advantage in using a greater number of iterations—it will increase processing time unnecessarily.

Z-mode button Enable to allow overlapping in the image to create 3D-like effects where parts of the image are pulled over other parts.

NOTE When Z-mode is enabled, the Iterations field is disabled.

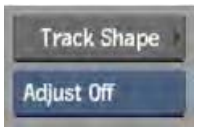
Range field Displays the size of the region affected by the warping effect. Editable.

The greater the value, the larger the portion of the image affected. A value of 100% affects the entire image, a value of 33% affects one-third of the image, for example.

Weight field Displays the relative weight on the selected spline, to control its influence. Editable.

For example, you can set a higher weight on a spline in an area of the image that you do not want to move.

Resolution field Displays the pixel resolution of the clip being warped or morphed. A lower pixel resolution creates warps and morphs that better follow the contours of the distortion splines, although it can slow down system performance.



Track Shape button Opens the Stabilizer menu to apply tracking data to a spline's vertices.

Adjust box Select whether an offset is applied relative to the spline's tangents.

Distort Setup Settings

Repeat button Enable to fill in gaps by repeating the clip.

Repeat Value box Select 4 to fill gaps by repeating the clip once on each side, or 8 to repeat the clip twice on each side.

Amount box Select the amount of clip to repeat.

Draw Grid button Enable to display the distortion grid.

Scale Setup button Enable to scale a loaded setup's splines to the resolution of the current project.

Icons button Enable to display splines, tangents, and vertices in the image.

Icons Transparency field Displays the transparency level of the icons in the image. Editable.

Tangent colour pot Displays the colour of tangents on the splines. Editable.

Schematic Transparency field Displays the transparency level of the schematic. Editable.

Undo Levels field Displays the levels of Undo available in the Undo/Redo lists. Editable.

Miscellaneous Settings

Exit button Exits the Distort module.

Load button Loads a setup.

Save button Saves a setup.

Setup Name field Displays the name of the last saved setup.

Revert button Reverts to the last saved setup.

Delete button Deletes the most recently processed clip.

Player button Opens the Player to view the last rendered clip.

Step Process field Displays the frame number interval to be processed (to see intermediate results, for example). Editable.

Preview button Displays the distort effect without processing.

Setup button Opens the Setup menu, where you can set various preference and display settings.

Animation button Opens the channel editor, where you can animate various distort settings.

Set Key button Sets a keyframe at the selected frame.

Delete Key button Deletes the selected keyframe.

Copy button Copies an object, a branch (object and its children), or all, depending on what is selected in the Selection Mode box.

Delete button Deletes an object, a branch (object and its children), or all, depending on what is selected in the Selection Mode box.

Selection Mode box Select whether to copy, delete, or reset.

Reset button Resets an object, a branch (object and its children), or all, depending on what is selected in the Selection Mode box.

Node Name field Displays the name of the selected node in the schematic. Editable.

Previous button Selects the previous node in the schematic.

Next button Selects the next node in the schematic.

Add button Adds an axis or spline, depending of the selection of the Node box.

Node box Select whether to add an axis or spline, then click Add.

Close button Closes a spline at the first vertex.

Finish button Finishes a single-point or open spline at the last vertex.

Duplicate

Use Duplicate to create a copy based on the copy parameter selected.



To access the duplicate menu, use:

- Tools, then select from the menu.

Edge Detect

Use Edge Detect to trace the edges in a clip based on colour. This data can then be used to create a range of artistic effects in various colour spaces.



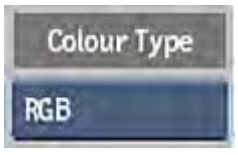
To access the Edge Detect menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front and matte clip, and outputs a result and an outmatte.

Edge Detect Settings

General Settings



Colour Type box Select the colour space that you want to use to isolate the edge.



Result Output box Select Result to output the composite of the input image, input matte, and the matte generated by the edge detection. Or, select Edge Only to output only the result of compositing the input image with the matte generated by the edge detection.

Matte Output button Select Input to output the input matte or Edge Matte to output the matte generated by the edge detection.

Edge Detection Settings



Red Weight field Displays a value for the relative intensity of red used to detect edges. This value is scaling factor, not the color value of the channel itself. Editable.

Green Weight field Displays a value for the relative intensity of green used to detect edges. This value is scaling factor, not the color value of the channel itself. Editable.

Blue Weight field Displays a value for the relative intensity of blue used to detect edges. This value is scaling factor, not the color value of the channel itself. Editable.

Proportional Weight button Enable to adjust the red, green, and blue weights proportionally.

Minimum Edge Detection Threshold field Displays the lower limit for edge detection. Editable.

Maximum Edge Detection Threshold field Displays the upper limit for edge detection. Editable.

Softness Width field Displays a value for the width of the edge. Editable.

Softness Gain field Displays a value for the softness of the edge. Editable.

Edge Effects Settings



Double Edge button Enable to do a second edge detection pass and create twice as many edges.

Minimum Edge Detection Threshold field Displays the lower limit for edge detection. Editable.

Maximum Edge Detection Threshold field Displays the upper limit for edge detection. Editable.

Softness Width field Displays a value for the width of the edge. Editable.

Softness Gain field Displays a value for the softness of the edge. Editable.

Directional button Enable to create the edge in the direction indicated in the Angle field.

Edge Angle field Displays a value for the direction of the edge. Editable.

Colour Offset pot Displays a colour offset value for the image outside of the detected edges.

Colour Correction Settings



Saturation field Displays level of colour purity in the image. Editable.

Gamma field Displays the gamma level. Editable.

Offset field Displays a value that modifies all of the colour parameters. Editable.

Red Gain field Set the percentage of colour values in the red channel. Editable.

Green Gain field Set the percentage of colour values in the green channel. Editable.

Blue Gain field Set the percentage of colour values in the blue channel. Editable.

Luma Gain field Set the percentage of luma gain value to display. Editable.

Proportional button Enable to adjust the gain of the colour values proportionally.

Blending Settings



Blend Mode option box Select an operation to blend the input image with the matte created by the edge detection, and the input matte (if the Use Matte button is enabled).

Edge Transparency field Displays a value for the transparency applied to the edges. Editable.

Use Matte button Enable to use the input matte to constrain the blending of the input image and the matte created by the edge detection.

Exposure

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



Front view of a 16-bit floating point image



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

To access the Exposure menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

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

NOTE To adjust the exposure, contrast, and image data type of the image for display purposes only, set the image display viewer options in the View menu or edit them gesturally in the current viewport. See Displaying Multiple Views (page 262).

Exposure Menu Settings

Exposure Node

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



(a) Input Data Type box

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

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

Red Exposure field Displays the exposure offset of the red channel.

Green Exposure field Displays the exposure offset of the green channel.

Blue Exposure field Displays the exposure offset of the blue channel.

Proportional button Enable to change the exposure offset value for a channel and update the values for the other channels proportionally.

Red Contrast field Displays the contrast level of the red channel.

Green Contrast field Displays the contrast level of the green channel.

Blue Contrast field Displays the contrast level of the blue channel.

Proportional button Enable to change the contrast value for a channel and update the values for the other channels proportionally.

Red Pivot field Displays the red value used as a pivot when generating contrast. The pivot value affects the way the contrast is calculated. The default value is mid-gray on a logarithmic scale.

Green Pivot field Displays the green value used as a pivot when generating contrast. The pivot value affects the way the contrast is calculated. The default value is mid-gray on a logarithmic scale.

Blue Pivot field Displays the blue value used as a pivot when generating contrast. The pivot value affects the way the contrast is calculated. The default value is mid-gray on a logarithmic scale.

Proportional button Enable to change the pivot value for a channel and update the values for the other channels proportionally.

Field Merge

Use Field Merge to remove field jitter by merging the fields of a clip.



To access the Field Merge menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip, and outputs a result.

Field Merge Menu Settings

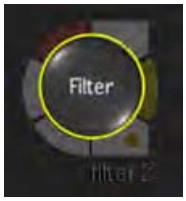
General Settings



Level field Enter the percentage of blending between fields, or drag on the field to modify its value.

Filter

Use Filter to apply different effects to a clip, including textures, blurring, edge detection, embossing, and sharpening.



To access the Filter menu:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip, and outputs a result.

Filter Menu Settings

General Settings



Filter 1 button Modifies the set of image operations in the first filter element that is applied in the final composite filter.

Filter 2 button Modifies the set of image operations in the second filter element that is applied in the final composite filter.

Filter 3 button Modifies the set of image operations in the third filter element that is applied in the final composite filter.

Filter 4 button Modifies the set of image operations in the fourth filter element that is applied in the final composite filter.

Filter 5 button Modifies the set of image operations in the fifth filter element that is applied in the final composite filter.

Use Filter button Enable to apply the filter element to the final composite filter.

Effect field Displays the level of filtering required. Editable.

Bypass Chroma button Enable to ignore hue and saturation channels of an image.

Bypass Luminance button Enable to ignore luminance channels of an image.

Filter Type box Select whether to use matrix calculations or a predefined procedure to alter the image.

Matrix Settings

Gain field Displays the light values of the image. Editable.

Automatic button Enable to preserve the average luminance of the clip.

Balanced button Enable to activate a compensating algorithm when a value is entered in the matrix, where the value is divided by the remaining fields, and then the result is subtracted from each value in the matrix.

Symmetry Type box Select an arrangement used to change symmetrical elements.

Same button Enable to change a symmetrical field to the same value as a field that is being edited.

Invert button Enable to change a symmetrical field to the equal and opposite value as the field value that is being edited.

Copy button Click to copy the current matrix setup.

Paste button Click to paste a copied matrix setup.

Reset Matrix button Reset all values in the matrix.

3x3 button Enable to apply a matrix of three rows and three columns. The field in the centre represents the pixel that is currently being evaluated; the others represent surrounding pixels. The matrix displays the relative influence of each pixel surrounding the current one.

5x5 button Enable to apply a matrix of five rows and five columns. The field in the centre represents the pixel that is currently being evaluated; the others represent surrounding pixels. The matrix displays the relative influence of each pixel surrounding the current one.

7x7 button Enable to apply a matrix of seven rows and seven columns. The field in the centre represents the pixel that is currently being evaluated; the others represent surrounding pixels. The matrix displays the relative influence of each pixel surrounding the current one.

9x9 button Enable to apply a matrix of seven rows and seven columns. The field in the centre represents the pixel that is currently being evaluated; the others represent surrounding pixels. The matrix displays the relative influence of each pixel surrounding the current one.

11x11 button Enable to apply a matrix of eleven rows and eleven columns. The field in the centre represents the pixel that is currently being evaluated; the others represent surrounding pixels. The matrix displays the relative influence of each pixel surrounding the current one.

Procedural Settings

Invert button Enable to invert the image.

Rect button Enable to blur the image.

Width field Displays the height of the blur. Editable.

Height field Displays the width of the blur. Editable.

Sobel button Enable to apply an edge-detection filter that uses the Sobel operator.

Sobel Direction box Select the direction in which the filter is applied.

Prewitt button Enable to apply an edge-detection filter that uses the Prewitt operator.

Prewitt Direction box Select the direction in which the field is applied.

Flip

Use Flip to generate a mirror image of a clip. You can flip frames in a clip horizontally, vertically or both.



To access the Field Merge menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip, and outputs a result.

Flip Menu Settings

General Settings



Flip Direction box Select to flip the clip horizontally or vertically.

Reverse Dominance box Select an option for correcting the field dominance when flipping clips vertically.

Garbage Mask

Use the Garbage Mask to isolate particular areas of an image to include with, or exclude from, the opaque area of the matte.



To access the Garbage Mask menu, use:

- Batch, then select a node from the Node bin. (page 894)
- Timeline, then use Batch FX (page 894).
- Modular Keyer, then select a node from the Node bin. (page 894)

This node accepts a front and a matte clip as input, and outputs a result.

The Garbage Mask menu includes the Tracer and the region of Interest (ROI) functions, which can also be found in the Modular Keyer. The node processes gaps in clips set to No Media as black frames. An unconnected

front clip will return an error, while and unconnected back clip will process black frames. You can save and load GMask setups directly in Batch or Batch FX.

Garbage Mask Menu Settings

Garbage Mask General Settings



Regenerate button Enable to get dynamic updating of your mask as you make changes.

X Position field Position the X axis.

Y Position field Position the Y axis.

Z Position field Position the Z axis.

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

X Rotation field Rotate the X axis.

Y Rotation field Rotate the Y axis.

Z Rotation field Rotate the Z axis.

X Scale field Scale the X axis.

Y Scale field Scale the Y axis.

Z Scale field Scale the Z axis.

Proportional button Enable to constrain proportions while scaling.

X Shear field Shear the X axis.

Y Shear field Shear the Y axis.

Z Shear field Shear the Z axis.

X Centre field Centre the X axis.

Y Centre field Centre the Y axis.

Z Centre field Centre the Z axis.

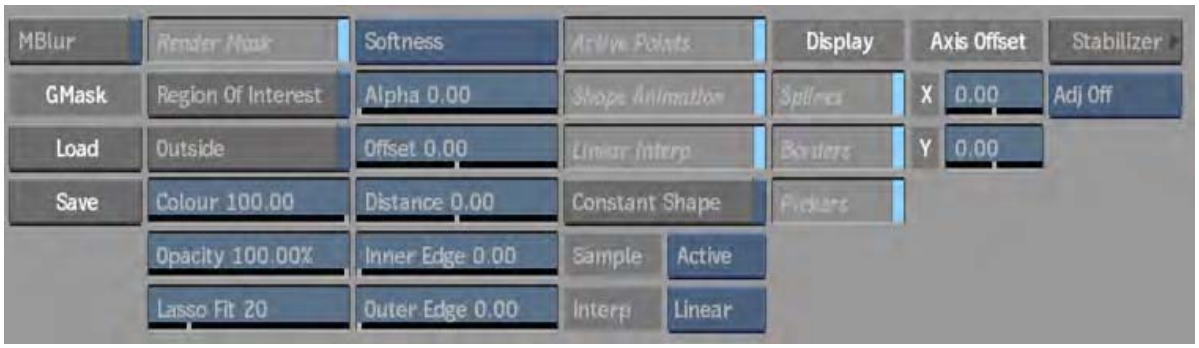
Stabilizer button Opens the Stabilizer menu where you track reference points in a clip.

Tracking Rotation option box Select Rotation Off for one-point tracking, Rotation On for two-point tracking, Rotation Inv to invert the rotation data.

Tracking Scale option box Select Scaling Off for one-point tracking, Scaling On for two-point tracking, Scale Inv to invert the scaling data.

Adjust box Select Adj Offset if the mask is parented to one axis. Select Adj Axis if the mask is parented to a hierarchy of objects. Select Adj Tangents to have the tangents for the selected points adjusted while the points are tracked.

Motion Blur button Enable the blur effect for the selected axis (Motion Blur must be enabled in the Setup menu).



Motion Blur button Toggles the blur effect for the selected geometry (Motion Blur must be enabled in the Setup menu).

Load button Loads a garbage mask setup.

Save button Saves a garbage mask setup.

Render Mask button Enable to see the mask and its effect on the image.

Region of Interest button Enable to reveal the matte of multiple garbage masks.

Outside button Enable to apply the effect to the part of the image outside the mask.

Colour field Set the blend between the outgoing and incoming image inside the mask.

Opacity field Set the transparency of the mask.

Lasso Fit field Set the number of points used in the freehand part of the mask.

Edge Softness box Select Softness to create a uniform gradient around the mask edge. Select Tracer to key out details around the mask edge.

Alpha field Set the transparency of the gradient from the mask edge.

Offset field Set the position of the gradient's border from the mask edge.

Distance field Specify the area over which the Inner and Outer Edge adjustments have an effect.

Inner Edge field Smoothen the softness gradient towards the inside of the mask edge.

Outer Edge field Smoothen the softness gradient towards the outside of the mask edge.

Active Points button Enable to use all points as a reference for the mask. Disable at a specific frame to mute picker values previously sampled at another frame.

Shape Animation button Enable to animate a mask using the Shape channel.

Linear Interpolation button Enable to use linear interpolation of the mask border between keyframes. Disable to use rounded interpolation. This button is active when the Shape Animation button is enabled.

Constant Shape button Enable to modify the mask's shape without setting keyframes.

Sample Status box Select how picker values are sampled. Active resamples picker values at every frame (the default). Passive disables resampling for one or more frames. This box is active when the Shape Animation button is disabled.

Interpolation Mode box Select the type of interpolation between keyframes when the Sample Status box is set to Active.

Splines button Enable to display inner and outer splines of the mask.

Borders button Enable to display inner and outer borders for advanced gradients.

Pickers button Enable to display elements that allow for detailed chroma and luma analysis when refining a detailed key.

X Axis Offset field Offset the mask from its X axis.

Y Axis Offset field Offset the mask from its Y axis.

Stabilizer button Opens the Stabilizer menu where you apply tracking data to selected vertices.

Adjust box Select Adj Tangents to have the tangents for the selected vertices adjusted while the points are tracked.

Add button Click to add a gmask axis or geometry node.

Node Type box Select whether to add an axis or geometry node.

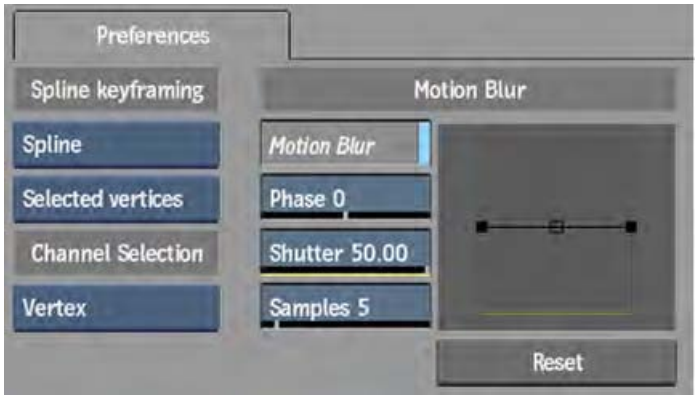
Close button Click to close the mask shape automatically.

Object Node Name field Displays the name of the selected object node.

Previous Node button Click to scroll to the previous similar node.

Next Node button Click to scroll to the next similar node.

Garbage Mask Node Setup Settings



Spline Keyframing box Select the parameters to use when animating a Garbage Mask spline. When Shape Animation is disabled in the Garbage Mask general settings, then the Spline & Tracer, Spline and Item options become available.

Vertex Keyframing box Select the conditions under which parameters are applied to specified vertices in the mask.

Channel Selection box Select which channels are selected in the Channel Editor when you select one or more vertices.

Motion Blur button Enable to apply motion blur to garbage masks that can be used to match the movement of objects in a clip.

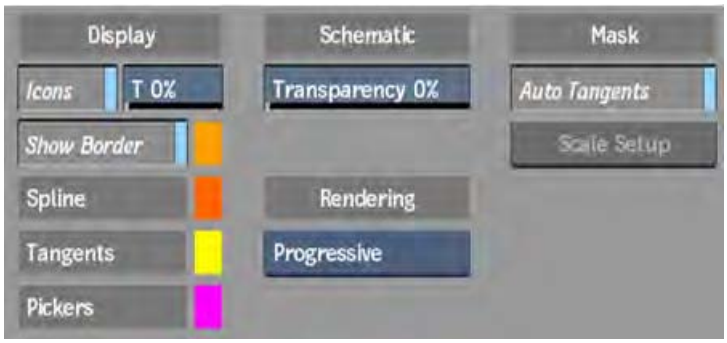
Phase field Displays whether the motion blur is based on the movement before or after the current frame. -100 places the motion blur before the frame; a value of 100 places it after the frame.

Shutter field Displays the duration of the motion blur at each frame. Editable.

Samples field Displays the number of samples taken at each frame to create the motion blur. Editable.

Motion Blur curve Displays the transparency of the samples that create the blur effect. Drag the curve or its handles to edit it gesturally.

Reset button Resets all motion blur controls and disables the Motion Blur button.



Icons button Enable to display garbage mask splines and axes.

Transparency field Displays the transparency of garbage mask splines and axes. Editable.

Show Border button Enable to display the Softness Offset wireframe border, defined in the Shape menu.

Show Border colour pot Select the colour for the Softness Offset wireframe.

Wireframe colour pot Select the colour of garbage mask splines.

Control Points colour pot Select the colour of garbage mask control points.

Pickers Display colour pot Select the colour of the pickers.

Transparency field Displays the transparency of nodes in the schematic. Editable.

Auto Insert button Enable to automatically insert a node when dragged between two connected nodes. When disabled, press Shift to auto insert.

Rendering box Select the rendering method.

Auto Tangents Enable to create tangents for new points.

Scale Setup button Enable to scale the GMask.

Glow

Use Glow to create and customize a glow effect on a clip.



To access the Glow menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

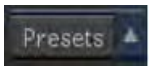
This node accepts a front and matte clip, and outputs a result an outmatte. You can add a Glow node to any part of the pipeline except to the Front and Matte pipes of MBLend nodes and the Matte pipes of CBlend nodes.

Glow Menu Settings

Using Glow Presets

A number of Glow presets are included, such as a basic glow effect or edge boost effect. These presets can help you learn how the glow effect works, or provide a good starting point to change settings to create better glow effects.

To use the Glow preset, select a preset from the Presets drop-down list in the Glow Setup menu. The Glow menu settings are changed to reflect the chosen preset.



Presets button Opens the Presets browser where you can select a preset.

Presets dropdown list Select a preset from the dropdown list. Menu settings are changed to reflect the chosen preset.

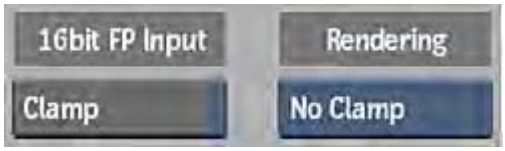
General Settings



Rendering box Select whether to render in Progressive or Interlaced mode.

Regen button Enable to get dynamic updating of the image as you make changes.

Setup Settings



Clamp Input button Enable to clamp colour and luminance values on input in the 16-bit floating point rendering pipeline.

Clamp Render box Select a clamping option for colour and luminance values on output in the 16-bit floating point rendering pipeline.

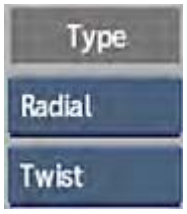
Glow Type Settings



Glow Type for Gaussian



Glow Type for Directional



Glow Type for Radial



Glow Type for Radial Stamp

Glow Type box Select the type or shape of glow filter to apply to the clip. For example, a Box blur has rectangular edges.Depending on the glow type, some of the other glow settings vary.

Radial Mode box Select whether a radial blur or glow moves in one circular direction (Spin), or two rotating directions (Twist). Available when Radial is chosen as the blur type.

Bias field Displays the direction of a blur. Enter a positive value for forward, a negative value for backward, or 0 for a blur that moves in both directions. Available when Directional or Radial Stamp is chosen as the blur type.

Samples field Displays the quality of a Radial Stamp blur or glow. Editable. Available when Radial Stamp is chosen as the blur type.

Centre Settings (Radial only)



Stabilizer button Opens the Stabilizer menu to track the centre of the blur from the source clip. See Tracking (page 878) for more information.

Centre X field Displays the X position of the centre of the radial blur (or gesturally move the red circle in the image). Editable.

Centre Y field Displays the Y position of the centre of the radial blur (or gesturally move the red circle in the image). Editable.

NOTE You can also move the red circle on the image to set the position of the centre of the blur. The Centre X and Y fields update accordingly.

Absolute/Relative box Select whether to position and offset the centre of the radial blur in a relative mode (expressed as a percentage) or absolute mode (expressed in pixels).

Offset X field Offsets the centre along the X axis (or press Ctrl and gesturally move the red circle in the image).This field is useful to apply changes to the centre after tracking has been performed. Editable.

Offset Y field Offsets the centre along the Y axis (or press Ctrl and gesturally move the red circle in the image). This field is useful to apply changes to the centre after tracking has been performed. Editable.

NOTE You can also press Ctrl and move the red circle on the image to set the offset of the centre of the glow. The Offset X and Y fields update accordingly.

Input Settings



Hue field Displays the colour range in the image before the glow effect is applied. Editable.

Saturation field Displays the colour purity level in the image before the glow effect is applied. Editable.

Contrast field Displays the level of gradations between light and dark areas before the glow effect is applied. Editable.

Gamma field Displays the level of grey in the image before the glow effect is applied. Editable.

Gain field Displays a value by which pixel colour values are multiplied. The offset value is added to this value to determine the final colour. Editable.

Offset field Displays the value to add to current pixel colour values. The resulting colour value is clipped at 0. Editable.

Front Settings



Front Settings for Gaussian or Box



Front Settings for Directional



Front Settings for Radial or Radial Stamp

Width field Displays the width of the blur.Increasing the blur increases the render time. Editable.

Height field Displays the height of the blur.Increasing the blur increases the render time. Editable.

Proportional button Enable to affect the width and height proportionally.

Pixel Ratio button Enable to blur the image using the same proportion as its aspect ratio.

Amount field (Radial, Radial Stamp) Displays the amount of radial blur. Editable.

Rotation field (Radial, Radial Stamp) Displays the angle of rotation for a radial blur. Editable.

Length field (Directional) Displays the radius amount of a directional blur. Editable.

Angle field (Directional) Displays the angle of a directional blur. Editable.

Matte Settings



Matte Settings for Gaussian or Box



Matte Settings for Directional



Matte Settings for Radial or Radial Stamp

Matte Width field Displays the width of the blur for the matte. Editable.

Matte Height field Displays the height of the blur for the matte. Editable.

Matte Amount field Displays the amount of radial blur for the matte. Editable.

Matte Rotation field Displays the angle of rotation for a radial blur for the matte. Editable.

Matte Length field Displays the radius amount of a directional blur for the matte. Editable.

Matte Angle field Displays the angle of a direction blur of the matte. Editable.

Lock To Front button Enable to keep the matte values the same as their corresponding values for the front clip.

Expand field Displays the percentage of additional blur to the matte. Editable when the Lock Expand button is disabled.

Lock Expand button Enable to make the Expand value directly proportional to the value in the Width field.

Auto Matte button Enable to generate matte values from the front clip.

Max Level field Displays the upper limit of the luminance values included in the glow effect. Editable.

Min Level field Displays the lower limit of the luminance values included in the glow effect. Editable.

Invert Matte button Enable to apply the glow to the region outside the area defined by the matte.

Premultiply button Enable to multiply the matte clip to the front clip.

Colour Settings



Colour 1 box Enable to define the first colour used for the glow. If two colours are used, they are combined in additive mode to create the glow effect.

Colour 1 Colour pot Displays the hue and saturation colour of the first glow colour. Click to open the colour picker to select a different colour.

Colour 1 Trackball Adjusts the hue and saturation of colour 1.

Colour 1 Hue field Displays the colour range of colour 1. Editable.

Colour 1 Saturation field Displays the level of colour purity of colour 1. Editable.

Colour 1 Intensity field Displays the level of brightness of colour 1. Editable.

Colour 2 box Enable to define the second colour used for the glow. If two colours are used, they are combined in additive mode to create the glow effect.

NOTE If you are using both Colour 1 and Colour 2, they are combined in additive mode to create the glow effect.

Colour 2 Colour pot Displays the hue and saturation colour of the second glow colour. Click to open the colour picker to select a different colour.

Colour 2 Trackball Adjusts the hue and saturation of colour 2.

Colour 2 Hue field Displays the colour range of colour 2. Editable.

Colour 2 Saturation field Displays the level of colour purity of colour 2. Editable.

Colour 2 Intensity field Displays the level of brightness of colour 2. Editable.

Channels Settings



Weight field Displays the weighted value of the channel.

Position X field Displays the horizontal offset of the channel.

Position Y field Displays the vertical offset of the channel.

Blending Settings



Screen option box Select a logical operation that can be used to blend the front clip and the result clip.

Transparency field Displays the percentage of blending when the result is composed on the front clip. Editable.

Use Matte button Enable to include the matte in the blending.

Result Settings



Result Output button Select whether to output the combined result (with blending), or only the glow effect itself.

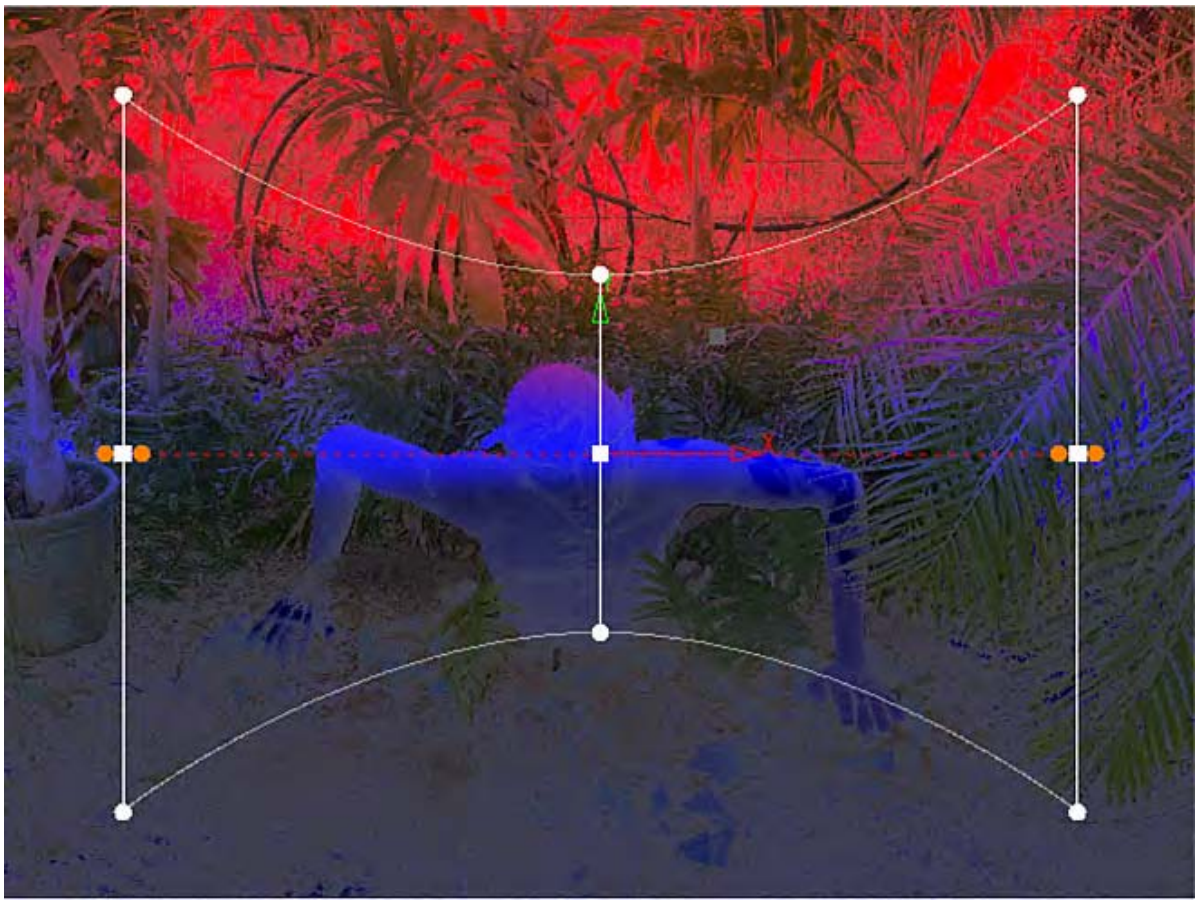
Matte Output Settings



Matte Output button Switch between blend matte and glow luma. Blend matte allows you to output the matte used internally to perform the blending of the glow effect over the input image. Glow luma allows you to generate a straight-forward luminance conversion of the actual glow effect.

Gradient

Use the Gradient to create an effect based on colours and patterns of your choosing.



To access the Gradient menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front and matte clip, and outputs a result and outmatte.

Gradient Menu Settings

Setup Settings



Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

Width field Displays the custom width resolution of the clip. Editable.

Height field Displays the custom width resolution of the clip. Editable.

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

Aspect Ratio field Displays the custom render/output aspect ratio. Editable

Frame Depth box Select the render/output frame depth of clips.

Scan Mode box Select the scan mode of clips.



Icons button Enable to display vertices, splines, and other gradient selection tools in the image window.

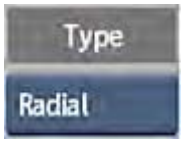
Transparency field Displays the transparency level of the gradient icons in the image window. Editable.

Vertex Colour Pot Displays the colour of vertices in the image window. Click to open the colour picker to select a different colour.

Spline Colour Pot Displays the colour of splines in the image window. Click to open the colour picker to select a different colour.

Amplitude Colour Pot Displays the colour of amplitude in the image window. Click to open the colour picker to select a different colour.

Type Settings



Gradient Type box Select the type of gradient to apply. Some of the gradient settings differ based on the gradient type you choose.

Select:	To create a gradient based on:
Directional	A single straight line spline.
Radial	Two circular geometries.
Spline	A spline that can be manipulated with vertices and tangents.
Point	Multiple coloured points.

For each gradient type, you can gesturally create your gradients in the image window with various widgets, such as splines and vertices. Make sure that Show Widgets is enabled in the Gradient Setup menu.

Orientation Settings



Swap UV button Enable to change the direction of the gradient.

Gradient Controls Settings



Position X field Displays the position of the gradient along the X axis. Editable.

Position Y field Displays the position of the gradient along the Y axis. Editable.

Rotation field Displays the degree of rotation of the gradient. Editable.

NOTE You can also manipulate the position and rotation of the gradient directly in the image window if Show Widgets is available.

Scale field Displays the scale of the gradient. Editable.

Centre field Directional gradient only: Displays the offset value of the centre of the spline as a percentage of its position on the spline. Editable.

Amplitude field Directional gradient only: Displays the pixel length of the spline. You can also drag either end of the spline directly in the image window. Editable.

Roundness field Radial gradient only: Displays the relative shape of the inner and outer circular patterns. A value of 1 displays the full rounded shape, whereas a value of 0 indicates a square shape. You can also enter a negative value to produce a concave shape. Editable.

Sharpness field Point gradient only: Displays the level of edge sharpness between the gradient point colours. Editable.

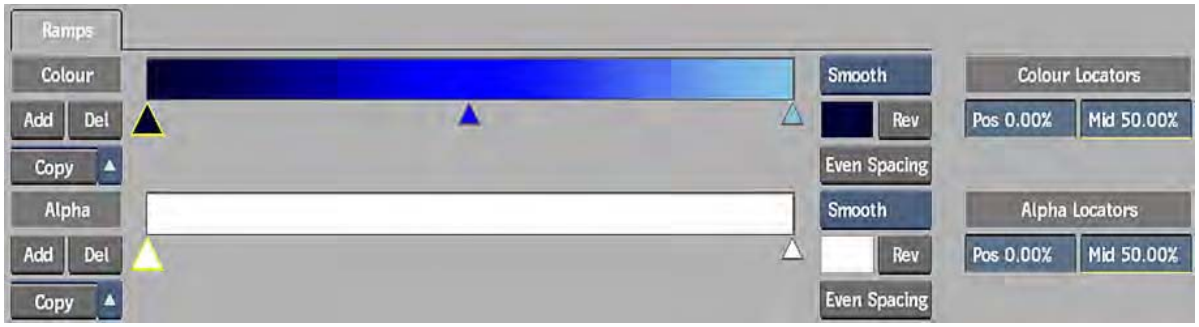
Softness field Displays the level of blur added to the gradient. A higher value produces a smoother transition between gradient colours. Editable.

Extrapolation box Select an extrapolation type for the gradient pattern.

Select:	To:
Constant	Keep the last value as a solid colour.
Repeat	Repeat the same gradient pattern.
Mirror	Repeat the gradient pattern in reverse order until the frame border is reached.

Offset Cycle field Displays the cycle position of the color creating the gradient. Editable.

Ramps Settings



NOTE The Colour and Alpha ramps are available for Directional, Radial and Spline gradients.

Colour Ramp Displays the gradations of the gradient colours. The triangles below the ramp represent each color in the gradient. You can add more colours to your gradient pattern.

A selected triangle's colour is displayed in the colour pot to the right of the ramp, and its position is displayed in the Position field. Also, when selecting a triangle, a smaller triangle appears representing the mid point between the colour and the next triangles' colour. You can move the triangles gesturally to achieve different gradations, and move the mid point triangle to shift the weight between the colours.

Colour Add button Adds a new colour triangle to the right of the currently selected triangle. Press Ctrl while clicking add to clone the selected triangle colour.

Colour Delete button Deletes the selected triangle.

Colour Ramp Copy list Select whether to copy the Colour ramp information to the Alpha ramp, or to preserve the Colour ramp information when switching gradient types.

NOTE The Alpha ramp under the Colour ramp is identical in functionality, except that it displays gradations in gradient transparency.

Alpha Ramp Displays the gradations of the gradient transparency. The triangles below the ramp represent each alpha colour in the gradient. You can add more alpha colours to your gradient pattern.

Alpha Add button Adds a new colour triangle to the right of the currently selected triangle. Press Ctrl while clicking add to clone the selected triangle colour.

Alpha Delete button Deletes the selected triangle.

Alpha Ramp Copy list Select whether to copy the Alpha ramp information to the Colour ramp, or to preserve the Alpha ramp information when switching gradient types.

Colour Interpolation Type box Choose a Linear or Smooth gradient interpolation curve.

Colour Colour pot Displays the colour of the selected triangle. Click to open the colour picker to select a different colour.

Reverse Colour button Click to reverse the colours of the gradient proportionally.

Colour Even Spacing button Click to space the ramp triangles evenly.

Alpha Interpolation Type box Choose a Linear or Smooth gradient interpolation curve.

Alpha Colour pot Displays the alpha colour of the selected triangle. Click to open the colour picker to select a different colour.

Reverse Alpha Colour button Click to reverse the alpha colours of the gradient proportionally.

Alpha Even Spacing button Click to space the ramp triangles evenly.

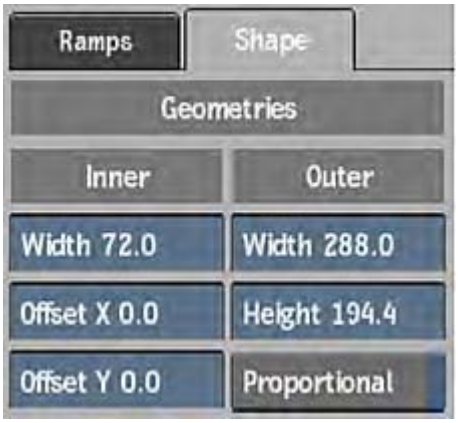
Colour Locator Position field Displays the location of the selected colour triangle along the colour ramp. Editable.

Colour Locator Mid field Displays the weight level between colours on the ramp (represented by the small triangle). Editable.

Alpha Locator Position field Displays the location of the selected alpha colour triangle along the alpha ramp. Editable.

Alpha Locator Mid field Displays the weight level between alpha colours on the ramp (represented by the small triangle). Editable.

Radial Shape Settings



Use these settings if you wish to further affect the shape of a radial gradient.

NOTE Radial Shape settings are available when Radial is selected from the Gradient Type box.

Inner Width field Displays the width of the inner radial circle. You can also gesturally drag the red square on the inner circle. Editable.

Inner Offset X field Displays the offset along the X axis of the inner radial circle. You can also gesturally drag the + inside the inner circle. Editable.

Inner Offset Y field Displays the offset along the Y axis of the inner radial circle. You can also gesturally drag the + inside the inner circle. Editable.

Outer Width field Displays the width of the outer radial circle. You can also gesturally drag the right-most red square on the outer circle. Editable.

Outer Height field Displays the height of the outer radial circle. You can also gesturally drag the left-most red square on the outer circle. Editable.

NOTE Drag the middle red square of the outer radial circle to affect both the width and the height at the same time.

Proportional button Enable to affect the outer radial width and height proportionally.

Spline Shape Settings



Use these settings if you wish to further affect the shape of a spline gradient. All of these settings have gestural equivalents in the image window. Ensure that Icons is enabled in the Gradient Setup menu.)

NOTE Available when Spline is selected from the Gradient Type box.

Vertex X Position field Displays the position along the X axis of the selected vertex. Editable.

Vertex Y Position field Displays the position along the Y axis of the selected vertex. Editable.

Tangent X Position field Displays the position along the X axis of the selected tangent. Editable.

Tangent Y Position field Displays the position along the Y axis of the selected tangent. Editable.

Amplitude Position field Displays the position of the spline as a percentage of the distance between the first and last vertex. Editable.

Amplitude Start field Displays the start position of the spline. Use to set the pixel length of the spline. Editable.

Amplitude End field Displays the end position of the spline. Use to set the pixel length of the spline. Editable.

Reset Shape button Resets the spline shape.

Points Settings



NOTE Available when Point is selected from the Gradient Type box.

Points List Use the rows to set the colour, position, and other settings for each color of a Point gradient.

Points Column Displays the number of the point. By default, four points are added to a gradient; however, you can add more.

Colour Column Displays the colour of the point. Click the colour pot to open the colour picker to select a different colour.

Alpha Column Displays the alpha colour of the point. Click the colour pot to open the colour picker to select a different colour.

X Column Displays the position of the point along the X axis. You can also gesturally drag the point in the image window to position it.

Y Column Displays the position of the point along the Y axis. You can also gesturally drag the point in the image window to position it.

Weight Column Displays the weight of the colour in relation to the nearest point colour in the image.

Radius Column Displays the radius of the point colour.

Add button Adds a new colour point to the list and image.

Delete button Deletes the selected colour point from the list and image.

Clone button Clones the selected colour point settings in a new point in the list and image.

Blending Settings



Screen option box Select a logical operation that can be used to blend the front clip and the result clip.

Transparency field Displays the percentage of blending when the result is composited on the front clip. Editable.

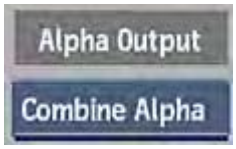
Gain field Displays the level of gain applied to the chosen blending option. Editable.

Result Output Settings



Output Mode box Select whether to output the combined result (with blending), or only the gradient effect itself.

Alpha Output Settings



Alpha Output box Select whether to output the combined alpha result, or only the gradient effect itself.

Interlace

Use Interlace to connect the odd and even scanlines of a clip.



To access the Interlace menu:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

The Interlace node accepts a front clip as input, and outputs a result.

For each pair of frames in the input clip, the Field 1 scanlines of one frame are interlaced with the even scanlines of the second frame to produce a single frame in the generated clip.

Interlace Menu Settings

General Settings



First Rendered Frame field Displays the first frame at which output is rendered from the node. Unrendered output does not display any media.Editable.

Field Dominance box Select Field 1 or Field 2 dominance, or Auto to have the application detect automatically the field dominance.

Keyer 3D

Use the Keyer 3D to key out a given range of colours.



To access the Keyer 3D menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Keyer 3D Menu Settings

General Settings



Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Key button Select to open the Key menu, where you set the elements to key in the RGB viewer, or the Pref menu, where you customize the RGB Viewer display properties.

Regen button Enable to get dynamic updating of the image as you make changes.

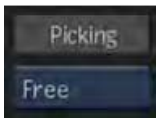
Sampling Settings



Sampling box Select the type of key element value to sample in the image window.

Sampling Reset button Resets the values for the key element selected in the Sampling box.

Picking Settings



Picking box Select the key element to restrict selection in the RGB viewer. Select Free to select any element.

Rotation Axis Settings



NOTE The Rotation Axis Settings are available when you select the Pref menu from the Key button.

Rotation Axis box Select the centre of the RGB viewer or a key element as the axis of rotation of the RGB viewer.

Plot Regions Settings



NOTE The Plot Regions Settings are available when you select the Pref menu from the Key button.

Plot Region 1 button Enable to display the first plotted region.

Plot Region 2 button Enable to display the second plotted region.

Plot Region 3 button Enable to display the third plotted region.

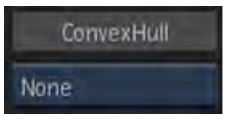
Plot Point button Enable to display plotted colour value in the RGB viewer.

Tolerance Display Settings



Tolerance Display box Select to display the tolerance as an opaque, transparent, or wireframe ellipsoid.

Tolerance ConnexHull Settings



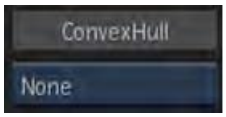
Tolerance Convex Hull option box Select to display the vertices, surfaces, or both the vertices and surfaces of the tolerance convex hull.

Softness Display Settings



Softness Display box Select to display the softness as an opaque, transparent, or wireframe ellipsoid.

Softness ConnexHull Settings



Soft Convex Hull option box Select to display the vertices, surfaces, or the vertices and surfaces of the softness convex hull.

Softness/Noise Settings



Colour box Select a YUV or RGB colour model.

Softness Scaling box Select to scale softness proportionally or to remove noise from softened areas.

Scaling field When Softness Scaling is set to Prop, adjust the X, Y, and Z-axis softness scaling values proportionally. When Softness Scaling is set to Min Noise, adjust the value to increase grain removal. (The scaling value resets to 1.00 after use.) Editable.

Scale X field Displays X-axis softness scaling. Editable.

Scale Y field Displays Y-axis softness scaling. Editable.

Scale Z field Displays Z-axis softness scaling. Editable.

Patches Settings



Patches box Select the Patch you want to sample.

Active button Enable to include the sample in the key.

Display box Select to display the current patch as a wireframe, a convex hull, both a wireframe and convex hull, or a transparent cloud.

Colour field Enter the percentage of the colour value to use to render the patch. Editable.

Opacity field Enter the percentage of opacity of the patch. Editable.

Softness field Enter the softness value of the patch. Editable.

Key Histogram Settings



Plot button Activates a pick cursor which you use to select a plot value in the image window. If Plot Point is enabled in the Pref menu, a cube with a dashed outline surrounds the point in the RGB viewer.

Home button Restores the RGB viewer to its original settings and position.

Invert Point button Inverts the direction of scaling of the softness ellipsoid.

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

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

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

Buffer Value field Displays the Range value stored in the buffer. Non-editable.

Recall button Retrieves the last Range value that was stored in the buffer.

Pref Histogram Settings



NOTE The Pref Histogram Settings are available when you select the Pref menu from the Key button.

Histogram button Enable to display the 3D histogram in the RGB viewer.

Histogram Grid option box Select an option for the histogram grid resolution.

Opacity field Displays the percentage of opacity of the cubes. Editable.

Scaling field Resizes the cubes. Cubes representing less prevalent colours are scaled down more than those representing more prevalent colours. Editable.

Threshold field Enter a value to edit the display of colours that are less prevalent in the image. As you increase the threshold, less prevalent colours are increasingly removed from the histogram. Editable.

Canvas button Enable to display the canvas.

Canvas Light field Sets the lighting in the RGB viewer. The light source emanates from behind the viewer.

Lines button Enable to display a white outline of the borders of the RGB viewer, and a green wireframe around selected ellipsoids.

Cloud Opacity field Displays the overall opacity for all ellipses and patches.

Key Element box Select an element type to change its opacity.

Patch Opacity field Displays the opacity for the third patch in the RGB viewer.

Plot button Activates a pick cursor which you use to select a plot value in the image window. If Plot Point is enabled in the Pref menu, a cube with a dashed outline surrounds the point in the RGB viewer.

Home button Restores the RGB viewer to its original settings and position.

Invert Point button Inverts the direction of scaling of the softness ellipsoid.

Keyer Channel

Use the Keyer Channel to extract a key from a red, green, or blue channel, or from a custom value.



To access the Keyer Channel menu, use:

- Batch, then select a node from the Node bin.

- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Keyer Channel Menu Settings

General Settings



Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Channel Mode box Select a predefined or custom colour channel.

Key colour pot Activates the pick cursor. Use to sample an area of the image to display an average colour value for a custom channel.

Softness field Displays the percentage of transparency of the key-in clip. Editable.

More button Enable to enhance the keying effect.

Spread field The Spread field is available when Custom is selected from the Channel Mode box.Displays a value for the range of colours extracted from the key-in clip. Editable.

Regen button Enable to get dynamic updating of the image as you make changes.

Keyer HLS

Use the Keyer HLS to extract a key by adjusting tolerance and softness using hue, luminance, and saturation ranges.



To access the Keyer HLS menu, use:
- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).

■ The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Keyer HLS Menu Settings

General Menu Settings



Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Regen button Enable to get dynamic updating of the image as you make changes.

Picking Controls Settings



Key Colour colour pot Activates the pick cursor. Use to sample an area of the image to display the average colour value.

Tolerance button Activates the pick cursor. Use to select the maximum and minimum values for the tolerance range in the image window.

Master Tolerance field Displays the value added to the existing tolerance of each channel. Resets to 0 after each use.

+ Increases tolerance values.

- Decreases tolerance values.

Softness button Activates the pick cursor. Use to select the maximum and minimum values for the softness range in the image window.

Master Softness field Displays the value added to the existing softness of each channel. Resets to 0 after each use.

+ Increases softness values.

- Decreases softness values.

Colour Control Settings



Hue button Enable to extract hue values for the key.

Minimum Softness field Displays the minimum hue value of the softness range. Editable.

Minimum Tolerance field Displays the minimum hue value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum hue value of the tolerance range. Editable.

Maximum Softness field Displays the maximum hue value of the softness range. Editable.

Luminance button Enable to extract luminance values for the key.

Minimum Softness field Displays the minimum luminance value of the softness range. Editable.

Minimum Tolerance field Displays the minimum luminance value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum luminance value of the tolerance range. Editable.

Maximum Softness field Displays the maximum luminance value of the softness range. Editable.

Saturation button Enable to extract saturation values for the key.

Minimum Softness field Displays the minimum saturation value of the softness range. Editable.

Minimum Tolerance field Displays the minimum saturation value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum saturation value of the tolerance range. Editable.

Maximum Softness field Displays the maximum saturation value of the softness range. Editable.

Plot button Activates the pick cursor. Use to sample an area of the image to display its colour value.

Frame option box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Keyer Luma

Use the Keyer Luma to extract a key from the luminance of a clip.



To access the Keyer Luma menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Keyer Luma Menu Settings

General Settings



Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Master Tolerance field Displays the value added to the existing tolerance of each channel. Enter 100 for an entirely opaque matte.

Master Softness field Displays the softness. Drag left or right to adjust the percentage value or enter a new percentage value.

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

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

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

Plot button Activates the pick cursor. Use to sample an area of the image to display its colour value.

Frame option box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Keyer RGB

Use the Keyer RGB to extract a key by adjusting tolerance and softness using red, green, and blue ranges.



To access the Keyer RGB menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Keyer RGB Menu Settings

General Settings



Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Regen button Enable to get dynamic updating of the image as you make changes.

Picking Controls Settings



Key Colour colour pot Activates the pick cursor. Use to sample an area of the image to display the average colour value.

Tolerance button Activates the pick cursor. Use to select the maximum and minimum values for the tolerance range in the image window.

Master Tolerance field Displays the value added to the existing tolerance of each channel. Resets to 0 after each use.

+ Increases tolerance values.

- Decreases tolerance values.

Softness button Activates the pick cursor. Use to select the maximum and minimum values for the softness range in the image window.

Master Softness field Displays the value added to the existing softness of each channel. Resets to 0 after each use.

+ Increases softness values.

- Decreases softness values.

Colour Control Settings



Red Channel button Enable to extract red channel values for the key.

Minimum Softness field Displays the minimum red channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum red channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum red channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum red channel value of the softness range. Editable.

Green Channel button Enable to extract green channel values for the key.

Minimum Softness field Displays the minimum green channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum green channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum green channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum green channel value of the softness range. Editable.

Blue Channel button Enable to extract blue channel values for the key.

Minimum Softness field Displays the minimum blue channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum blue channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum blue channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum blue channel value of the softness range. Editable.

Plot button Activates the pick cursor. Use to sample an area of the image to display its colour value.

Frame option box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Keyer RGBCMYL

Use the Keyer RGBCMYL to extract a key by adjusting tolerance and softness using red, green, blue, cyan, magenta, yellow, and luminance ranges.



To access the Keyer RGBCMYL menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Keyer RGBCMYL Menu Settings

General Settings



Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Regen button Enable to get dynamic updating of the image as you make changes.

Picking Controls Settings



Key Colour colour pot Activates the pick cursor. Use to sample an area of the image to display the average colour value.

Tolerance button Activates the pick cursor. Use to select the maximum and minimum values for the tolerance range in the image window.

Master Tolerance field Displays the value added to the existing tolerance of each channel. Resets to 0 after each use.

+ Increases tolerance values.

- Decreases tolerance values.

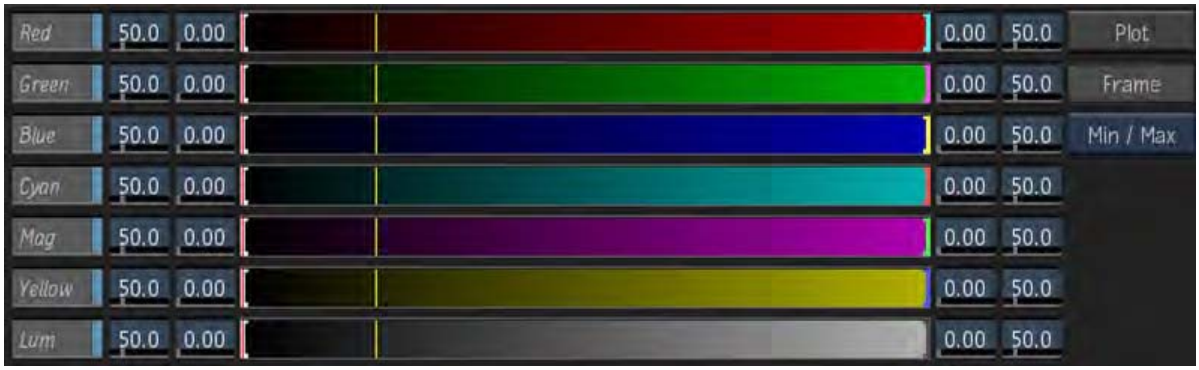
Softness button Activates the pick cursor. Use to select the maximum and minimum values for the softness range in the image window.

Master Softness field Displays the value added to the existing softness of each channel. Resets to 0 after each use.

+ Increases softness values.

- Decreases softness values.

Colour Control Settings



Red Channel button Enable to extract red channel values for the key.

Minimum Softness field Displays the minimum red channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum red channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum red channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum red channel value of the softness range. Editable.

Green Channel button Enable to extract green channel values for the key.

Minimum Softness field Displays the minimum green channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum green channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum green channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum green channel value of the softness range. Editable.

Blue Channel button Enable to extract blue channel values for the key.

Minimum Softness field Displays the minimum blue channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum blue channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum blue channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum blue channel value of the softness range. Editable.

Cyan Channel button Enable to extract cyan channel values for the key.

Minimum Softness field Displays the minimum cyan channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum cyan channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum cyan channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum cyan channel value of the softness range. Editable.

Magenta Channel button Enable to extract magenta channel values for the key.

Minimum Softness field Displays the minimum magenta channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum magenta channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum magenta channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum magenta channel value of the softness range. Editable.

Yellow Channel button Enable to extract yellow channel values for the key.

Minimum Softness field Displays the minimum yellow channel value of the softness range. Editable.

Minimum Tolerance field Displays the minimum yellow channel value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum yellow channel value of the tolerance range. Editable.

Maximum Softness field Displays the maximum yellow channel value of the softness range. Editable.

Luminance button Enable to extract luminance values for the key.

Minimum Softness field Displays the minimum luminance value of the softness range. Editable.

Minimum Tolerance field Displays the minimum luminance value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum luminance value of the tolerance range. Editable.

Maximum Softness field Displays the maximum luminance value of the softness range. Editable.

Plot button Activates the pick cursor. Use to sample an area of the image to display its colour value.

Frame option box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Keyer YUV

Use the Keyer-YUV to extract a key by adjusting tolerance and softness using luma (Y) and video component (U,V) ranges.



To access the Keyer YUV menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip and outputs a result.

Keyer YUV Menu Settings

General Settings



Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

Regen button Enable to get dynamic updating of the image as you make changes.

Picking Controls Settings



Key Colour colour pot Activates the pick cursor. Use to sample an area of the image to display the average colour value.

Tolerance button Activates the pick cursor. Use to select the maximum and minimum values for the tolerance range in the image window.

Master Tolerance field Displays the value added to the existing tolerance of each channel. Resets to 0 after each use.

+ Increases tolerance values.

- Decreases tolerance values.

Softness button Activates the pick cursor. Use to select the maximum and minimum values for the softness range in the image window.

Master Softness field Displays the value added to the existing softness of each channel. Resets to 0 after each use.

+ Increases softness values.

- Decreases softness values.

Colour Control Settings



Y Channel button Enable to extract luminance channel values for the key.

Minimum Softness field Displays the minimum luminance value of the softness range. Editable.

Minimum Tolerance field Displays the minimum luminance value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum luminance value of the tolerance range. Editable.

Maximum Softness field Displays the maximum luminance value of the softness range. Editable.

U button Enable to extract blue-luminance difference values for the key.

Minimum Softness field Displays the minimum blue-luminance difference value of the softness range. Editable.

Minimum Tolerance field Displays the minimum blue-luminance difference value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum blue-luminance difference value of the tolerance range. Editable.

Maximum Softness field Displays the maximum blue-luminance difference value of the softness range. Editable.

V button Enable to extract red-luminance difference values for the key.

Minimum Softness field Displays the minimum red-luminance difference value of the softness range. Editable.

Minimum Tolerance field Displays the minimum red-luminance difference value of the tolerance range. Editable.

Maximum Tolerance field Displays the maximum red-luminance difference value of the tolerance range. Editable.

Maximum Softness field Displays the maximum red-luminance difference value of the softness range. Editable.

Plot button Activates the pick cursor. Use to sample an area of the image to display its colour value.

Frame option box Select whether to frame the histogram based on minimum and maximum slider values, the full range of histogram values, the plot and reference colours, or the [0:1] vertical and horizontal range (Home).

Lens Distort

Use the Lens Distort to rectify or simulate lens distortions caused by camera lens or perspective irregularities that can result in skewed angles.



To access the Lens Distort menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).

This node accepts a front and matte clip, and outputs a result and an outmatte.

Lens Distort Menu Settings

General Settings



Distortion box Select whether to perform a lens distortion or rectification. All other radial distortion controls are implemented based on this selection.

Regen button Enable to get dynamic updating of the image as you make changes.

Centre box Select the size of the crosshair icon in your image.

Centre X field Displays the horizontal position of the centre of the lens. Editable.

Centre Y field Displays the vertical position of the centre of the lens. Editable.

Icon colour pot Displays the colour of the crosshair in your image. Editable.

Radial Settings

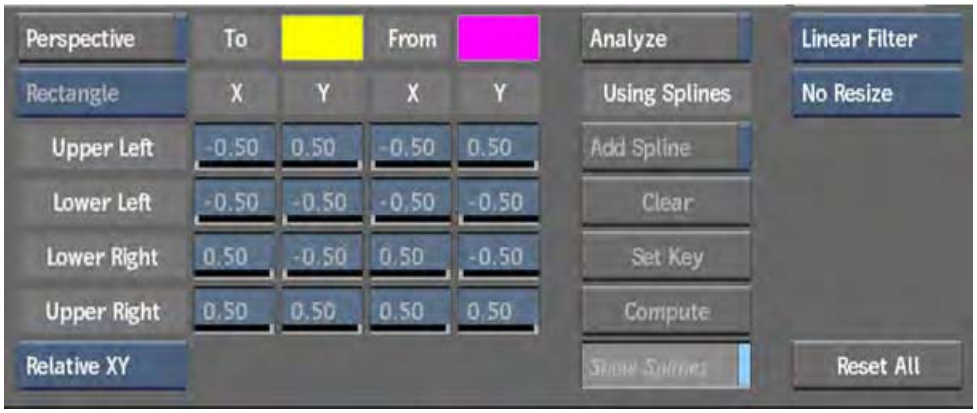


Lens Mag field Displays the magnitude of radial distortion or rectification. Editable.

Lens Adj field Displays the radial distortion or rectification. Editable.

Anamorph field Displays the ratio of distortion or rectification along the X or Y axis. A value greater than 1.00 stretches more on the X axis, while a value less than 1.00 stretches more on the Y axis. Editable.

Lens Distort Controls



Perspective button Enable to perform a perspective rectification.

Perspective Type box Select an option to map the source anchor points to a rectangle, preserving either the horizontal or vertical edges of the image, or mapping one four-corner free-form shape to another. Active when Perspective button is enabled.

Range box Select whether to set the X and Y range from -0.5 to 0.5 (relative) or set the X and Y range to half of the image size (absolute).

Source colour pot Displays the colour of the source anchor points. Editable.

Destination colour pot Displays the anchor points of the destination shape. Editable.

Upper Left X field Displays the horizontal value for the upper left source anchor. Editable.

Lower Left X field Displays the horizontal value for the lower left source anchor. Editable.

Lower Right X field Displays the horizontal value for the lower right source anchor. Editable.

Upper Right X field Displays the horizontal value for the upper right source anchor. Editable.

Upper Left Y field Displays the vertical value for the upper left source anchor. Editable.

Lower Left Y field Displays the vertical value for the lower left source anchor. Editable.

Lower Right Y field Displays the vertical value for the lower right source anchor. Editable.

Upper Right Y field Displays the vertical value for the upper right source anchor. Editable.

NOTE The following fields are available when Free Form is selected from the Perspective Type box.

Upper Left X field Displays the horizontal value for the upper left destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Lower Left X field Displays the horizontal value for the lower left destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Lower Right X field Displays the horizontal value for the lower right destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Upper Right X field Displays the horizontal value for the upper right destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Upper Left Y field Displays the vertical value for the upper left destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Lower Left Y field Displays the vertical value for the lower left destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Lower Right Y field Displays the vertical value for the lower right destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Upper Right Y field Displays the vertical value for the upper right destination anchor. Editable (active when the Perspective Type box is set to Free Form).

Analyse button Enable to use the spline controls.

Add Spline button Enable to draw a spline along a feature that you want to straighten.

Clear button Click to clear splines at the selected keyframe.

Set Key button Click to set a keyframe based on the result of the analysis.

Compute button Click to analyse the image based on the splines you added.

Show Splines button Enable to display splines in the image window.

Filter box Defines the type of filter for your distortion or rectification.

Resize box Assigns the resize option you choose to apply to your image (if any distortion or rectification is present).

Reset All button Click to restore default settings.

Logical Operation

Use the Logical Operation node to process missing media in clips set to No Media, based on the input tab it is connected to.



To access the Logic Op menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front, back, and matte clip, and outpus a result and an output matte.

Logic Op Menu Settings

General Settings



Logical Operations box Select the logical operation you want to use to combine the source images.

Swap Inputs button Enable to switch the ordering of the front and back inputs.

Transparency field Displays the percentage of the back clip that is used in the operation. Editable.

Clamp Render box Select a clamping option for colour and luminance values on output in the 16-bit floating point rendering pipeline.

LUT Editor

Use LUT Editor to convert logarithmic images to linear images or linear images to logarithmic images, while maintaining color accuracy.



To access the LUT Editor menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

The LUT Editor node can also be used to apply a conversion to 16-bit floating-point image in the OpenEXR file format.

When you import or export clips in Batch, default LUT parameters are used and the direction of the conversion is detected automatically. Use the LUT Editor to modify these default settings interactively or to import an existing LUT. You can also export a LUT from the LUT Editor.

When you load an OpenEXR-formatted clip into Batch, you cannot connect it directly to nodes that do not support 16-bit floating-point images. To work with a node that does not support this input, first connect the clip to a LUT Editor node to convert the clip into an 8- or 12-bit linear image. Whether or not you change the clip's bit depth, use the LUT Editor's EXR Display option to control the image's luminance and black point properties.

The LUT Editor node can be set to output a bit depth of 8, 10, 12, or 16 bits. You can also manually set the variables for output.

LUT Editor Menu Settings

General Settings



Conversion LUT Type box Select a basic LUT type, EXR Display, PhotoMap, or Gamma correction.

Destination box Select an output bit-depth.

Export button Opens the Export LUT menu where you select the location to export the LUT.



Basic button Enable to display the basic conversion LUT curve.

Advanced Editing button Enable to display advanced editing curves for each colour channel. RGB curves use blending to distinguish overlapping curves.

Final button Enable to display the final conversion LUT curve.

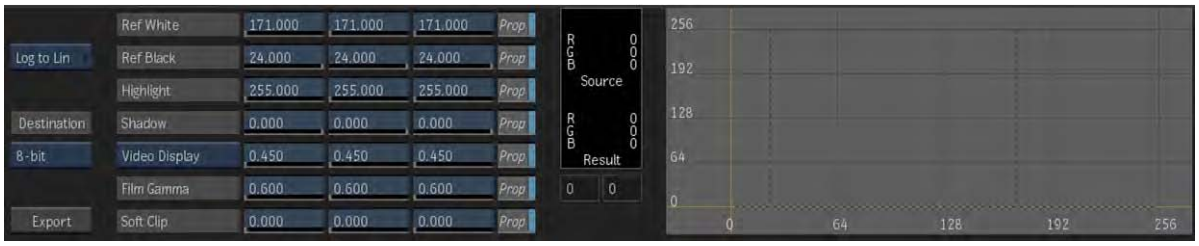
Edit Curve box Select to display the editing curve for the selected colour channel.

Zoom field Displays the zoom percentage of the histogram. Editable.

Home box Select whether to recenter the graph or center the frame.

Reset button Applies default settings.

Log to Lin, Lin to Log, and Gamma Settings



Red Reference White field Displays the lower limit at which red channel values are considered white. Editable.

Green Reference White field Displays the lower limit at which green channel values are considered white. Editable.

Blue Reference White field Displays the lower limit at which blue channel values are considered white. Editable.

Red Reference Black field Displays the upper limit at which red channel values are considered black. Editable.

Green Reference Black field Displays the upper limit at which green channel values are considered black. Editable.

Blue Reference Black field Displays the upper limit at which blue channel values are considered black. Editable.

Red Highlight field Displays the value at which greater red channel values are clamped. Editable.

Green Highlight field Displays the value at which green channel values are clamped. Editable.

Blue Highlight field Displays the value at which blue channel values are clamped. Editable.

Red Shadow field Displays the value at which lower red channel values are clamped. Editable.

Green Shadow field Displays the value at which lower green channel values are clamped. Editable.

Blue Shadow field Displays the value at which blue channel values are clamped. Editable.

Gamma Correction box Select preset gamma correction values (Video Display) or custom values. This setting is available for Log to Lin and Lin to Log.

Red Gamma Correction field Displays the gamma correction value for the red channel. Editable.

Green Gamma Correction field Displays the gamma correction value for the green channel. Editable.

Blue Gamma Correction field Displays the gamma correction value for the blue channel. Editable.

NOTE The following settings are available for Log to Lin and Lin to Log.

Red Film Gamma field Displays the gamma correction value for the red channel. Editable.

Green Film Gamma field Displays the gamma correction value for the green channel. Editable.

Blue Film Gamma field Displays the gamma correction value for the blue channel. Editable.

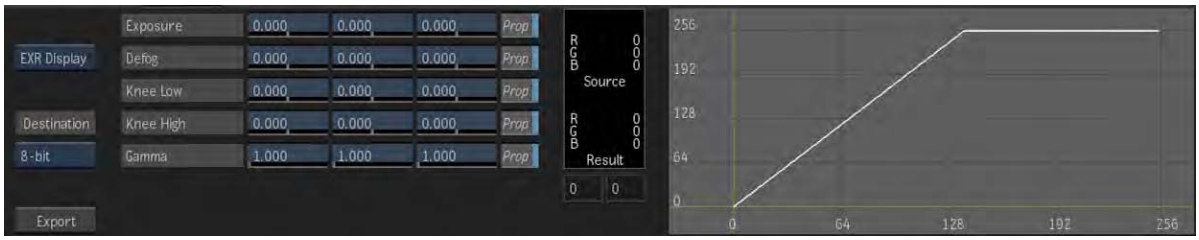
Red Soft Clip field Displays the level of adjustment to the shoulder of the conversion curve for the red channel. The upper limit is dependent on the channel's current reference white value. To create softer highlight, drag right when using a linear curve. Drag left when using a logarithmic curve.

Green Soft Clip field Displays the level of adjustment to the shoulder of the conversion curve for the green channel. The upper limit is dependent on the channel's current reference white value. To create softer highlight, drag right when using a linear curve. Drag left when using a logarithmic curve.

Blue Soft Clip field Displays the level of adjustment to the shoulder of the conversion curve for the blue channel. The upper limit is dependent on the channel's current reference white value. To create softer highlight, drag right when using a linear curve. Drag left when using a logarithmic curve.

Proportional button Enable to constrain channel value proportions.

EXR Display Settings



Red Exposure field Displays the luminance level of the red channel for the display image. Editable.

Green Exposure field Displays the luminance level of the green channel for the display image. Editable.

Blue Exposure field Displays the luminance level of the blue channel for the display image. Editable.

Red Defog field Displays the value to be subtracted from red color values to reduce fogging of the image. Editable.

Green Defog field Displays the value to be subtracted from green color values to reduce fogging of the image. Editable.

Blue Defog field Displays the value to be subtracted from blue color values to reduce fogging of the image. Editable.

Red Knee Low field Displays the lower limit of the compressed pixel range for the red channel. Editable.

Green Knee Low field Displays the lower limit of the compressed pixel range for the green channel. Editable.

Blue Knee Low field Displays the lower limit of the compressed pixel range for the blue channel. Editable.

Red Knee High field Displays the upper limit of the compressed pixel range for the red channel. Editable.

Green Knee High field Displays the upper limit of the compressed pixel range for the green channel. Editable.

Blue Knee High field Displays the upper limit of the compressed pixel range for the blue channel. Editable.

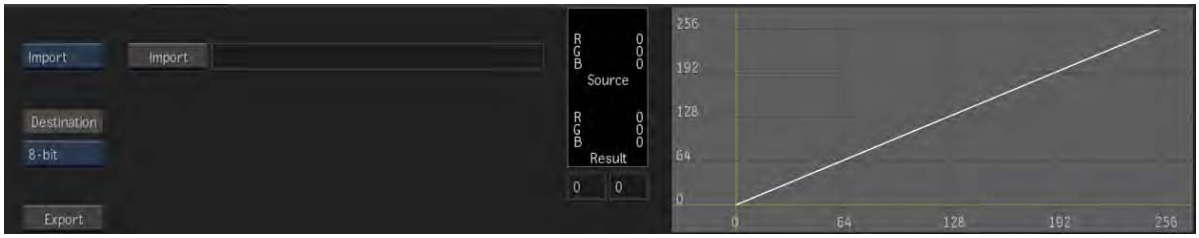
Red Gamma Correction field Displays the gamma correction value for the red channel. Editable.

Green Gamma Correction field Displays the gamma correction value for the green channel. Editable.

Blue Gamma Correction field Displays the gamma correction value for the blue channel. Editable.

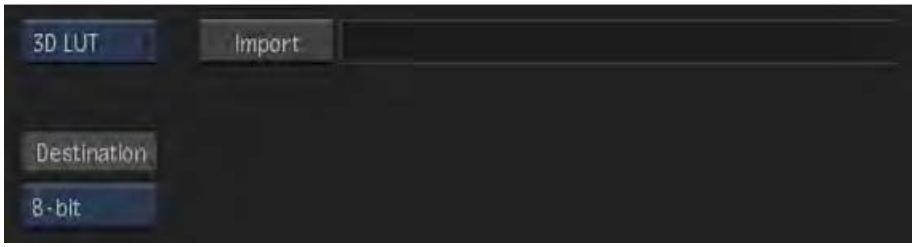
Proportional button Enable to constrain channel value proportions.

Import Settings



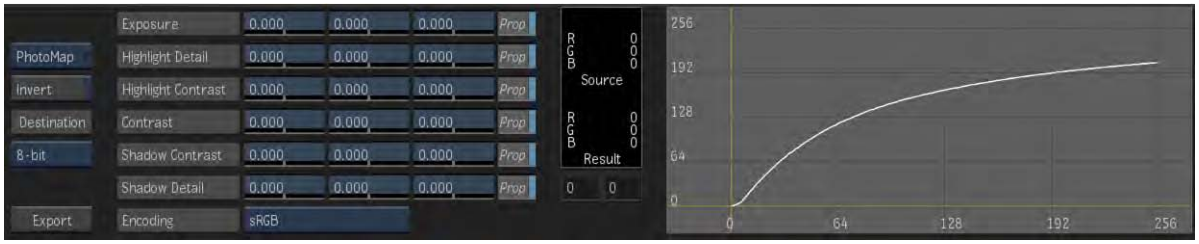
Import button Open the file browser to select a 1D LUT for import.

3D LUT Settings



Import button Open the file browser to select a 3D LUT for import.

PhotoMap



Red Exposure field Displays the luminance level of the red channel for the display image. Editable.

Green Exposure field Displays the luminance level of the green channel for the display image. Editable.

Blue Exposure field Displays the luminance level of the blue channel for the display image. Editable.

Red Highlight Detail field Displays the value for the detail in the light areas of the red channel. Editable.

Green Highlight Detail field Displays the value for the detail in the light areas of the green channel. Editable.

Blue Highlight Detail field Displays the value for the detail in the light areas of the blue channel. Editable.

Red Highlight Contrast field Displays the value for the contrast in the light areas of the red channel. Editable.

Green Highlight Contrast field Displays the value for the contrast in the light areas of the green channel. Editable.

Blue Highlight Contrast field Displays the value for the contrast in the light areas of the blue channel. Editable.

Red Contrast field Displays the value for the midtone contrast in the red channel. Editable.

Green Contrast field Displays the value for the midtone contrast in the green channel. Editable.

Blue Contrast field Displays the value for the midtone contrast in the blue channel. Editable.

Red Shadow Contrast field Displays the value for the contrast in the dark areas of the red channel. Editable.

Green Shadow Contrast field Displays the value for the contrast in the dark areas of the green channel. Editable.

Blue Shadow Contrast field Displays the value for the contrast in the dark areas of the blue channel. Editable.

Red Shadow Detail field Displays the value for the detail in the dark areas of the red channel. Editable.

Green Shadow Detail field Displays the value for the detail in the dark areas of the green channel. Editable.

Blue Shadow Detail field Displays the value for the detail in the dark areas of the blue channel. Editable.

Proportional button Enable to constrain channel value proportions.

Encoding option box Select a standard format for the colour space.

Colour Transform Settings

Import button Use to browse and select a LUT or a colour transform.

Custom button Apply a custom chain of color transforms.



Export button Export a custom chain of colour transforms as a single .CTF file.

Add button Add a new row to the end of the chain. Click in the Type column to select or change the transform type, and then click in the Transform column to select a transform.

Delete button Remove the selected transform from the chain.

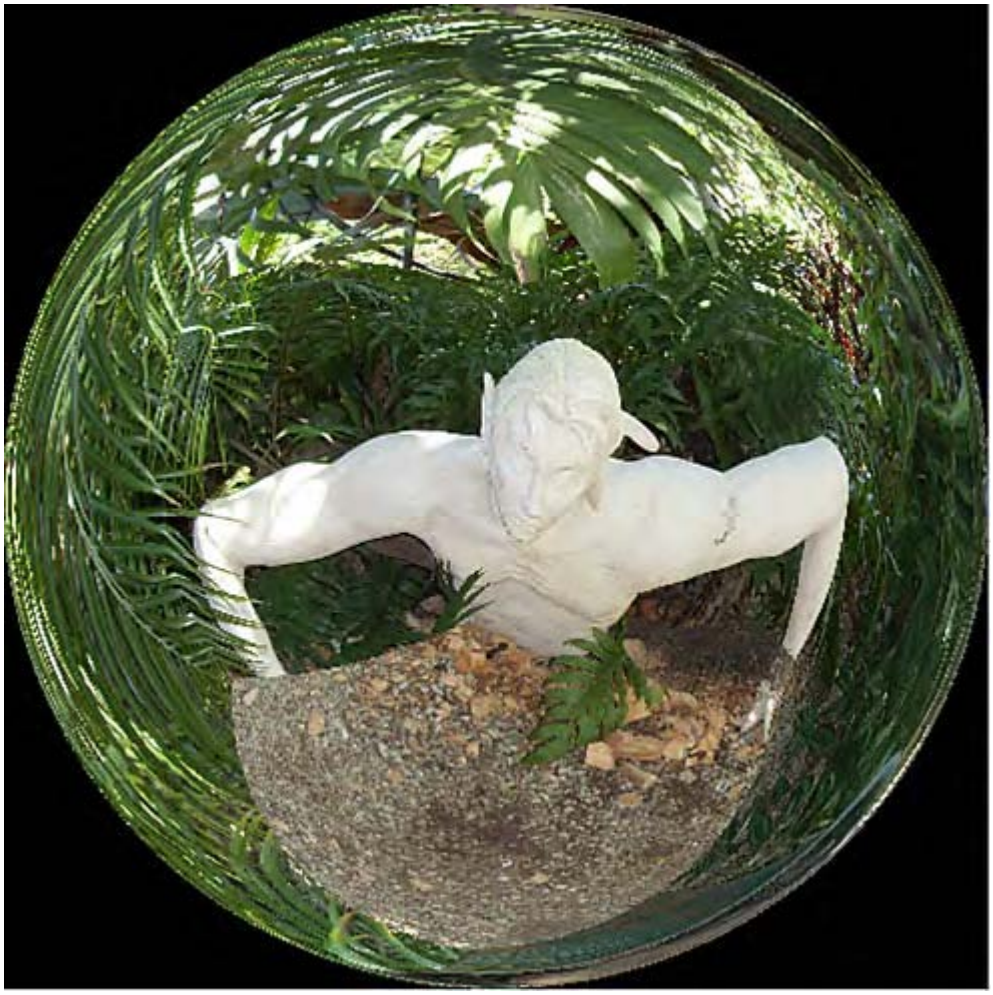
Up button Move the selected transform earlier in the chain.

Down button Move the selected transform later in the chain.

Map Convert

Use Map Conver to provide conversion transformation for image-based lighting (IBL) and environment maps.

You can use the results of Map Convert in Action as an IBL map (page 527).



To access the 2D Transform menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front and matte clip, and outputs a result an outmatte.

Map Convert Menu Settings

Input and Output Format Settings



Input Format box Select the type of image you want to convert.

Cubic Type box Select horizontal or vertical to identify the type of cubic image you are inputting. Select Spliced Faces to stitch together multiple images to create a horizontal cubic image. Available when Cubic is selected in the Input Format box.

Output Format box Select the output format for the image you are converting.

The Map Convert effect accepts the following format types:

Type:	Description:
Spheric	Sometimes referred to as a chrome ball or mirrored ball. An image of a mirrored ball in an assumed orthogonal projection.
Angular	Similar to the mirrored ball format, except that the radial dimension is mapped linearly with the angle, giving better sampling around the edges.
Cylindrical	An image mapped onto an unfolder cylinder (also known as longitude and latitude maps).
Cubic (Horizontal or Vertical)	An unfolded six-sided cube.
Cubic (Spliced Faces)	A type of cubic map available as an input format using a different frames displayed as an unfolded cube showing six sides of the image. To use this input type, frames from the input clip are used as the 6 faces.
Polar	A coordinate system for locating points in which each point on a plane is determined by a distance from a fixed point and an angle from a fixed direction. When using this input format, only Cartesian is available as the output format.
Cartesian	A coordinate system for locating points on a plane by measuring the horizontal and vertical distances from an arbitrary origin to a point. When using this input format, only Polar is available as the output format.

Transform Settings



These Transform settings apply to Cylindrical, Angular, Spheric, and Cubic, (Horizontal or Vertical) input formats.

Transform X Position field Displays the horizontal value of the image. Editable.

Transform Y Position field Displays the vertical value of the image. Editable.

Transform Z Position field Displays the depth value of the image. Editable.



In addition to the common transform settings, a Cubic (Spliced Faces) input format also contains the following settings:

Sequence Length field When creating a cubic map from spliced images, indicate how frequently images change in the sequence. Each change in the sequence gets mapped to a different face of the cube.

Face Assignments fields These locked fields display the order that the face assignment follows when creating the cubic map.



X Position field Displays the horizontal position of the output image. Editable.

Y Position field Displays the vertical position of the output image. Editable.

Scale field Displays the scale of the output image. Editable.

Rotation field Displays the rotation of the output image. Editable.

Repeat Mode box Select an option to fill the empty portions of the frame.



Radius Scale field Displays the scale of the radius for a cartesian to polar conversion. Editable.

Radius Offset field Displays the offset of the radius for a cartesian to polar conversion. Editable

Angle Scale field Displays the scale of the angle for a cartesian to polar conversion. Editable.

Angle Offset field Displays the offset of the Angle Scale for a cartesian to polar conversion. Editable.

Angular Repeat Mode box Select an option to fill the empty portions of the frame.

Output Size Settings



Output Size box Select whether to output the map at the size of the input image (Automatic), or select Custom to change the size of the output image.

Output Width field Displays the width of the output image. Editable when you select a custom output size.

Output Height field Displays the height of the output image. Editable when you select a custom output size.

Ratio field Displays the width to height ratio of automatic outputs. Non-editable.

Master Keyer

Use the Master Keyer to gesturally pull a key.



To access the Master Keyer menu, use:

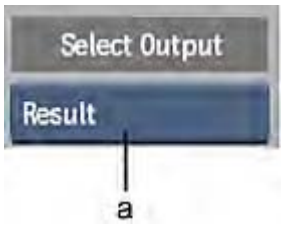
- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front, back and key in clip, and outputs a result and an outmatte.

Add a Master Keyer node to the pipeline when you need to create an accurate key of a clip. Use the Master Keyer node to automatically suppress colour spill, to colour correct, and to remove noise or grain. See Creating and Refining a Key in the Master Keyer (page 828).

While there is no setup menu for the Master Keyer node, it can be saved as part of the Batch setup, or alternatively, as a Custom node. The Master Keyer is also available from the Keyer node.

Select an output type from the Select Output box.



(a) Select Output box

TIP In order for the keyboard shortcuts M and D to correspond to Matte and Degrain, the Master Keyer in Batch must have focus and the viewport cannot be set to a schematic view. If the schematic has focus, then M corresponds to Move and D corresponds to Delete.

The Master Keyer node processes gaps in clips set to No Media based on the input tab receiving the information.

Input	Result
Front	No media
Back	Black frames
Matte	White frames

Master Keyer Menu Settings

MasterK Settings



Primary Sample colour pot Activates a pick cursor. Use to sample an area of the image.

Mix field Displays the mix between the primary and secondary sample. Drag right to include more of the secondary sample or left to include less.

Secondary Sample colour pot Activates a pick cursor. Use to sample an area of the image where you do not want any softness in the matte.

Patch1 button Enable to isolate a range of colours to be included in, or excluded from, the key. This button is active when an area of the image is sampled and Patch1 is selected in the Sampling box.

Patch box Select the areas of the matte to which the patch is applied.

Patch Range field Displays the colour range value. Editable.

Patch Softness field Displays the softness value. Editable.

Patch2 button Enable to isolate a range of colours to be included in, or excluded from, the key. This button is active when an area of the image is sampled and Patch2 is selected in the Sampling box.

Patch box Select the areas of the matte to which the patch is applied.

Patch Range field Displays the colour range value. Editable.

Patch Softness field Displays the softness value. Editable.

Patch3 button Enable to isolate a range of colours to be included in, or excluded from, the key. This button is active when an area of the image is sampled and Patch3 is selected in the Sampling box.

Patch box Select the areas of the matte to which the patch is applied.

Patch Range field Displays the colour range value. Editable.

Patch Softness field Displays the softness value. Editable.

Degrain button Enable to modify the grain and de-sharpen the edges of the key.

Size field Displays the size of the grain in the image. Editable.

Edges field Displays the level of sharpness of edges in the image. Editable.

More button Enable to increase the overall Degrain effect.

Sampling box Select the sampling method required to refine the key.

Reset button Resets sampling parameters (excluding the mix value and key colour).

Primary Reference colour pot Activates a pick cursor. Use to sample an area of the image.

Secondary Reference colour pot Activates a pick cursor. Use to sample an area of the image where you do not want any softness in the matte.

Colour Settings



Luma field Displays the luminance value. Editable.

Edge Size field Displays the range for the blend. Both the Luma field and the Edge Balance trackball are affected. Editable.

Spill colour pot Displays the colour to suppress in the clip. Editable.

Range field Displays the range for the removal of colour spill along the edges of the key. Drag right to soften the edge and remove colour spill further into the key. Drag left to harden, or create a thinner, edge.

Hue field Displays the hue value. Editable.

Auto Colour Correct button Enable to apply colour settings to the front clip.

Reset button Resets the colour settings.

Result Output



Result Output button Select the output mode for the result.

Matchbox

Matchbox is an interactive development tool that allows you to run generic OpenGL Shading Language (GLSL) shader code directly in Batch or Batch FX or through the Tools tab, to add specific functionality, or create custom effects. GLSL is a high-level shading language that is part of the OpenGL specification.

Because of the nature of GLSL fragment shaders, Matchbox works well on image processing effects. You can however, create simulated 3D effects using a number of image processing techniques, like using a Z-depth pass, for example.

The Matchbox node populates the user interface dynamically, based on the parameters required by the shader. You can also design and implement more sophisticated interface elements and naming, through

the use of an XML sidecar file. A utility is included to test your shader code and help you create the sidecar XML UI file, if needed.

Re-purposing of existing effects is easy, since Matchbox shaders are simple generic GLSL fragment shader code, with no encryption, and no required customization. Included are a number of useful example shaders, that can be used as is, or serve as starting points for you to develop your own tools. Some of the included example shaders are:

- Vignetting
- Fabric
- Cross Hatching
- NaN Replace
- Switcher
- Ripples
- Twirl
- Warp
- Posterise
- Z-Glow (multipass shader)
- Z-Rays (multipass shader)
- Median Filter (multipass shader)

To access the Matchbox menu:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node has six physical inputs, but you are not limited to the amount of actual inputs you can use in the effect, since you can use the same image for more than one input. It also outputs a result and an outmatte.

Matchbox Menu Settings

Most of the Matchbox menu is generated dynamically based on the GLSL (and optional XML) code, but there are a few UI elements that are constant.



Name field This locked field displays the name of the current shader.

Change Shader button Click to open the file browser to select a different shader.

Regen button Enable to automatically update the image as settings are changed.



Output Resolution box Select an output resolution for the effect. If you select Custom, settings appear with presets and custom options.

Output Width field Displays the width of the selected output resolution. Non-editable.

Output Height field Displays the height of the selected output resolution. Non-editable.

Creating Your Own Matchbox Effects

A great benefit of working with the Matchbox tool is being able to create your own effects, depending on your particular needs. Creating a Matchbox shader can be as simple as copying and pasting GLSL code snippets, or can be complex multipass effects with multiple inputs and dozens of UI elements. For example, here is the contents of a simple Add effect:

```
uniform sampler2D input1, input2;
uniform float adsk_result_w, adsk_result_h;

void main()
{
   vec2 coords = gl_FragCoord.xy / vec2( adsk_result_w, adsk_result_h );
   vec3 sourceColor1 = texture2D(input1, coords).rgb;
   vec3 sourceColor2 = texture2D(input2, coords).rgb;

   gl_FragColor = vec4( sourceColor1+sourceColor2, 1.0 );
}
```

Here's a quick high-level workflow to follow when creating Matchbox shaders:

1. Write or copy/paste GLSL fragment shader code.
2. Use the provided command line tool to test the shader.
3. Edit the Shader Description output from the test tool.
4. Package the XML and GLSL code together for use in Flame Premium.

Writing and Testing GLSL Fragment Shader Code

You can repurpose existing fragment shader code, or create an effect specific to your needs. In either case, you can use the *test_shader* utility to validate and debug your code, and optionally help you design user interface elements in a sidecar XML file. The *test_shader* utility also has an extensive Help file that lists the available uniforms (including a number of adsk_ custom uniforms) and XML structure.

The *test_shader* utility can be found in */usr/discreet/<product home>/bin*. To access the Help file, from the bin directory, type `test_shader --help`.

To create and test a fragment shader:

1. Write or copy your fragment shader code in a text editor.
2. Save the file with the extension *.glsl*. For example, here is the contents of a scaling effect:

```
1 uniform float size;
2 uniform sampler2D myInputTex;
3
4 void main (void) {
5         vec4 tex0 = texture2D(myInputTex, gl_TexCoord[0] * size);
6         gl_FragColor = vec4 (tex0.rgb, 1.0);
7 }
```

3 Run your code through the test utility. For example, `test_shader scale.glsl` produces this result:

```
0(5) : warning C7011: implicit cast from "vec4" to "vec2"

 XML :
 <ShaderNodeDescription Description="" Name="Preset Name">
   <Shader Index="1">
      <Uniform RepeatMode="Off" InterpolationMode="Linear" Type="sampler2D"
Tooltip="" Name="myInputTex">
      </Uniform>
      <Uniform DisplayName="size" Type="float" Name="size">
        <SubUniform Inc="0.01" Max="1000000.0" Min="-1000000.0" Default="0.0"
 Row="0" Col="0" Page="0" Tooltip="" Name="size">
         </SubUniform>
      </Uniform>
   </Shader>
   <Page Name="Page 1" Page="0">
      <Col Name="Column 1" Col="0" Page="0">
      </Col>
   </Page>
</ShaderNodeDescription>
```

In this case, the first line displays a compilation warning that you might want to fix. In some cases, you'll receive errors that need to be fixed for the shader to work properly in Flame Premium.

4 Fix any errors, and rerun the code through the *test_shader* utility.

5 Optional: Use the XML information in the test_shader output to help you set up the UI of the effect. This can be especially useful if different users are going to be working with these effects.

Simply copy the XML shader node description section of the test output into a new file and save it using the same name, but with an *.xml* extension. In our example, you can edit *scale.xml* to add default values, better names for inputs and other UI elements, and even tooltips to help the user (see the bold sections below):

```
<ShaderNodePreset SupportsAdaptiveDegradation="0" Description="" Name="Next
Generation Scaling">
   <Shader OutputBitDepth="Output" Index="1">
      <Uniform Index="0" NoInput="Error" Tooltip="" DisplayName="Front"
Mipmaps="False" GL_TEXTURE_WRAP_T="GL_REPEAT" GL_TEXTURE_WRAP_S="GL_REPEAT"
GL_TEXTURE_MAG_FILTER="GL_LINEAR" GL_TEXTURE_MIN_FILTER="GL_LINEAR"
Type="sampler2D" Name="myInputTex">
      </Uniform>
      <Uniform ResDependent="None" Max="100.00" Min="-100.00" Default="0.0"
Inc="0.01" Tooltip="Displays the percentage of scaling applied to the image." Row="0"
 Col="0" Page="0" DisplayName="size" Type="float" Name="Scale">
      </Uniform>
   </Shader>
   <Page Name="Page 1" Page="0">
      <Col Name="Effect Settings" Col="0" Page="0">
      </Col>
```

```
    </Page>
</ShaderNodePreset>
```

6 Add your .glsl and optional sidecar .xml file to the same directory. The existing shader example files are located in */usr/discreet/<product home>/matchbox*.

7 Try your effect in Smoke or Flame.

Creating Multipass Shaders

In order to build more efficient, complex, or sophisticated effects, you can split your effects into multiple passes. In order to do this, you can separate your effect into multiple .glsl files using advanced adsk_ uniforms. For example, the existing Median Filter preset consists of *MedianFilter.1.glsl* and *MedianFilter.2.glsl*. In this case, when selecting this effect from the Load Shaders browser in Flame Premium, you need to select the root group *MedianFilter.glsl* file to incorporate all of the passes as one effect.

Optional: Creating Browser Proxy Files

Along with the *.glsl* and optional *.xml* files that comprise a fragment shader, you can also create a file that can display a proxy of your effect in the Load Shaders browser. You can use Flame Premium to create a proxy of your effect, but if you don't have access to Flame Premium, or want to create proxies programmatically, you can use the following header (byteswap). The standard width and height of the proxy is 126x92, and the file is RGB 8-bit. Save your proxy files as *.p*, and place them in the same folder as your *.glsl* and *.xml* files of the same name.

```
typedef struct {
        unsigned short Magic;
        float  Version;
        short  Width;
        short  Height;
        short  Depth;
        float  Unused [ 6 ];
} LibraryProxyHeaderStruct;

#define PROXY_MAGIC 0xfaf0
#define PROXY_VERSION 1.1f
#define PROXY_DEPTH 130
```

Matte Curves

Use Matte Curves to adjust the luminance of input mattes.



To access the Matte Curves menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front, back and matte clip, and outputs a result.

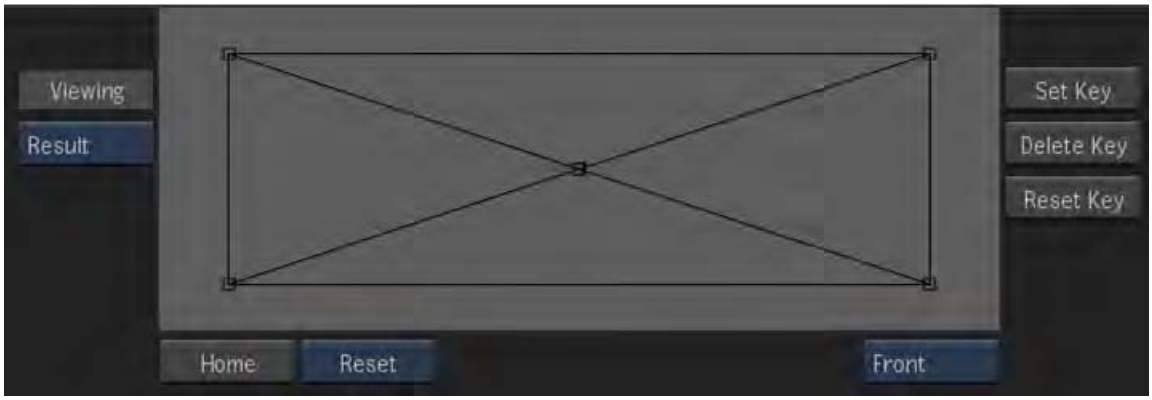
NOTE The Matte Curves node is the same as the Modular Keyer Matte Curves node.

The Matte Curves node processes gaps in clips set to No Media based on the input tab receiving the information.

Input	Result
Front	Black frames
Back	Black frames
Matte	No media

Matte Curves Menu Settings

General Settings



Result box Select the type of clip you want to process.

Home button Reverts to the original view.

Reset box Resets the curve view.

Matte box Select the matte curve you want to adjust.

Set Key button Sets a keyframe at the selected frame.

Delete Key button Deletes the selected keyframe.

Reset Key button Resets the curves at the selected keyframe.

Matte Edge

Use the Matte Edge effect to create an image based on the detected edges, or use it to modify the edges of a matte to help in creating a key.



To access the Matte Edge menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front and matte clip as inputs, and outputs a result. You can add the Matte Edge node to any part of the pipeline except the Front pipe of the CBLend node. The Matte Edge node is not appropriate for this pipe because its result is a matte.

Matte Edge Menu Settings

General Settings

Rendering box Select whether to render in Progressive or Interlaced mode.

Regen button Enable to get dynamic updating of the image as you make changes.

Edges Settings

Edges button Enable to create an image composed of the edges in an image.

Matte Edge Mode box Select whether to use the Basic, Advanced or Smooth edge-detection algorithm. Certain settings change based on this selection.

Basic Settings



Width field Displays the value that affects how the edge-detection algorithm determines whether each pixel forms part of the edge. Editable.

Minimum Input field Displays the lower limit of the luminance values. Pixels with lower values are mapped to black. Editable.

Maximum Input field Displays the upper limit of the luminance values. Pixels with higher values are mapped to white. Editable.

Advanced Settings



Minimum Input field Displays the lower limit of the luminance values. Pixels with lower values are mapped to black. Editable.

Maximum Input field Displays the upper limit of the luminance values. Pixels with higher values are mapped to white. Editable.

Inner Width field Displays the width of the inner edges of the matte. Editable.

Outer Width field Displays the width of the outer edges of the matte. Editable.

Width Proportional button Enable to affect the inner and outer width proportionally.

Inner Softness field Displays the level of softness on the inner edges of the matte. Editable.

Outer Softness field Displays the level of softness on the outer edges of the matte. Editable.

Softness Proportional button Enable to affect the inner and outer softness proportionally.

Smooth Settings



Smooth Minimum field Enter a lower limit for the edge detection. Editable.

Smooth Maximum field Enter the upper limit for the edge detection. Editable.

Smooth Softness field Enter a value for softness of the edge. Editable.

Smooth Gain field Enter a value for the softness gain of the edge. Editable.

Shrink Settings



Shrink button Enable to remove pixels from the edge of the matte.

Shrink Mode box Select whether to use shrink in a single pass or iterative mode. Single Pass mode allows you to control edge softness.

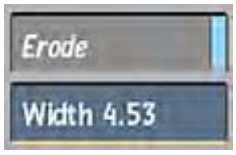
Shrink Width field Displays the width of the border that is removed from the edge of the matte. Editable.

Softness field Displays the amount of softness applied to the edges of the matte.

Minimum Input field Displays the lower limit of the luminance values. Pixels with lower values are mapped to black. Editable.

Maximum Input field Displays the upper limit of the luminance values. Pixels with higher values are mapped to white. Editable.

Erode Settings



Erode button Enable to blend the light and dark edges of the matte.

Erode Width field Displays the width of the matte border to soften. Editable.

Blur Settings



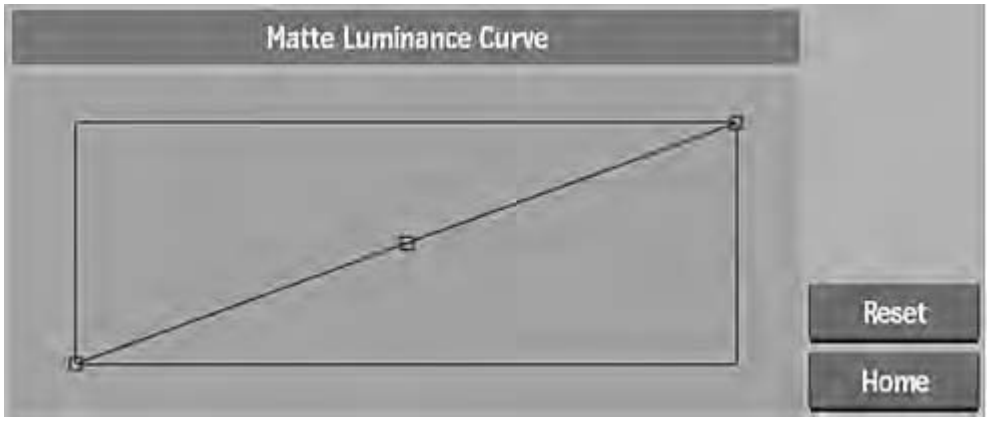
Blur button Enable to apply a softening Gaussian blur filter to the edge of the matte.

Blur Width field Displays the width of the blur applied to the edge of the matte. Editable.

Blur Height field Displays the height of the blur applied to the edge of the matte. Editable.

Proportional button Enable to constrain blur amount proportions.

Matte Luminance Curve Settings



Matte Luminance Curve Controls the blending of the edges.You can add keyframes to the curve using Add mode, move keyframes with Move mode, and modify the curve's shape using tangent handles.

Reset box Resets the curve view.

Home button Reverts to the original view.

Noise Settings



Noise button Enable to add noise to the transparent areas of the matte.

Noise Mode box Select the mode of noise to apply throughout the length of the image. Static adds noise as a still frame; Impulse adds noise that changes at each frame.

Select:	To add noise:
Static	As a still frame. You can move the static noise using the Position X and Y fields.
Impulse	Changing at each frame, but always starting from the same seed.

Weight field Displays the level of noise in the clip. Values below 1 add more white noise, and values above 1 add black noise. Editable.

Softness field Displays the level of Gaussian softness added to the noise. Editable.

Position X field Displays the X position of the noise when using Static mode. Editable.

Position Y field Displays the Y position of the noise when using Static mode. Editable.

Size X field Displays the size of the noise along the X axis. Editable.

Size Y field Displays the size of the noise along the Y axis. Editable.

Proportional button Enable to affect the X and Y sizes proportionally.

Mono

Use Mono to generate a monochrome copy of the front clip.



To access the Mono menu, use:

- Batch, then select a node from the Node bin.

- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

Mono Menu Settings

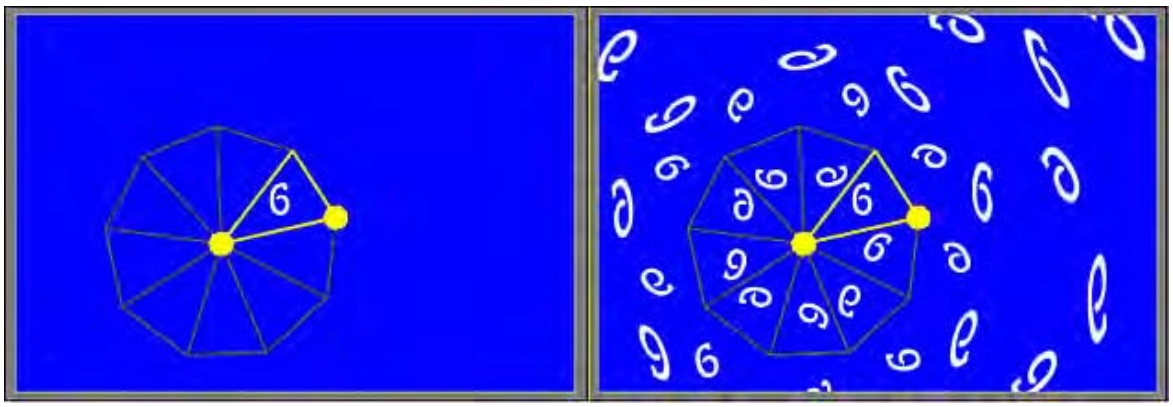
General Settings



Luminance box Select the channel you want to use to create the monochrome clip.

Motif

Use the Motif node to create a tiled symmetrical texture.



To access the Motif menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front and a matte input, and outputs a result clip and a matte output.

Source clips can be transformed before symmetrical effects are applied. Transformations are applied to both the front and matte clips simultaneously. You can select the type of symmetry mode to use on the transformation, and define its parameters.

The radial symmetry mode displays a user-defined widget, which is a polygon with an equal number of sectors to the order of symmetry. You select the sector to use as the originating tile, the basis for a kaleidoscopic texture. Additional parameters allow you to define the center of symmetry of the image, and the angle and radius of the originating tile.



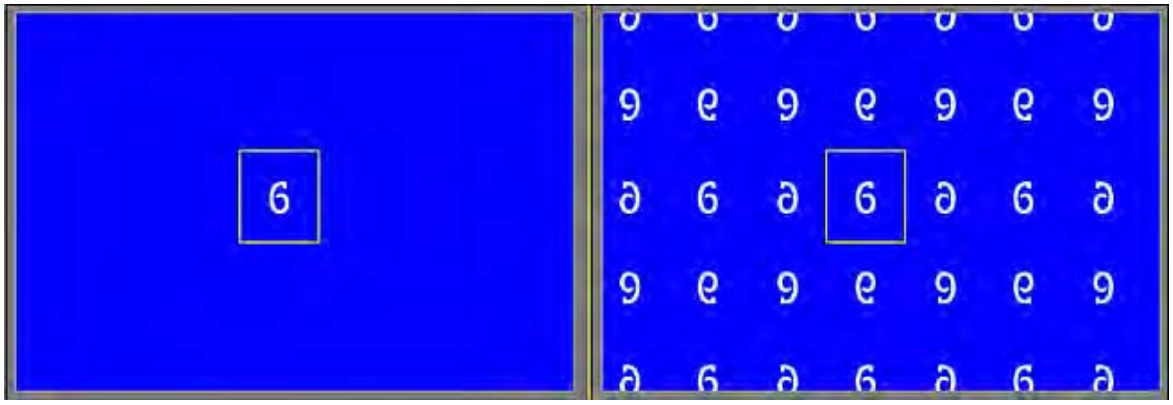
Front and Result views in radial symmetry mode

In Radial mode, you can also reuse the pixel on the edge of a sector to pad the space between outer tiles.



Result views with the repeating disabled and enabled

The region of interest (ROI) effect uses a user-defined rectangular or triangular selection as the originating tile to create a basic mirrored texture.





Front and Result views in region of interest symmetry mode

Image courtesy of The House

Motif Menu Settings

Motif Node

The Motif node displays 2D Texture Transform controls that allow you to simultaneously change the position, scaling, rotation, and type of image padding. The Symmetry Mode controls display a colour pot for the original tile selection.

2D Texture Transform settings



X Position field Displays the horizontal offset in pixels of the input and matte clips. Editable.

Y Position field Displays the vertical offset in pixels of the input and matte clips. Editable.

X Scaling field Displays the amount of horizontal scaling to apply to the front and matte clips.

Y Scaling field Displays the amount of vertical scaling to apply to the front and matte clips. Editable.

Scaling proportional button Enable to effect Scale X and Scale Y proportionally.

Rotation field Displays the angle of rotation of the front and matte clips. Editable.

Repeat button Select fill options to pad the empty portions of the frame with the last line of pixels, a repeated (rolled) image, or black pixels.

Symmetry Mode settings

Mode button Select the type of symmetry effect to apply to the transformation. Additional parameters are displayed in Radial mode.

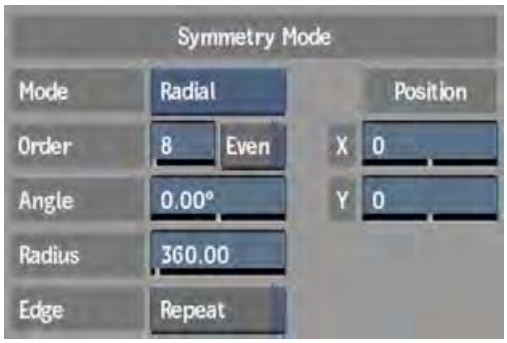
NOTE The following settings are available when the Symmetry Mode button is set to ROI.



Symmetry Mode box Select between a square or triangle region of interest mode.

Proportional button Enable to effect Scale X and Scale Y proportionally.

NOTE The following settings are available when the Symmetry Mode button is set to Radial.



Order field Displays the order of symmetry and indicates the number of sectors or sides of the polygonal widget.An order of 2 creates a basic mirrored image. An order of symmetry can have an odd number as a value. Tiles are mirrored in a counter-clockwise direction, therefor the sector to the right of the highlighted sector may not be its mirror image. Editable.

Even button Enable to maintain an even order of symmetry.If the order of symmetry is an odd value, an extra tile will be added, ensuring that adjacent tiles are mirror images of each other.

Angle field Displays angle of the axis of symmetry. Editable.

Radius field Displays the pixel length of the sector radius. Editable.

Edge Repeat button Enable to use the pixel colour on the sector's edge to pad the space between outer tiles. Displayed in Radial mode.

Position X field Displays the horizontal position of the centre of symmetry. Editable.

Position Y field Displays the vertical position of the centre of symmetry. Editable.

Softness settings



Softness field Displays the amount by which the motif effect is out of focus. Editable.

Display settings



Show Widgets button Enable to display the widgets in the clip.

Widget colour pot Select the colour used to highlight the originating tile. Editable.

Gestural Modifications

Modifications to the originating tile and symmetry mode parameters can be made gesturally in any view.

Drag:	To:
Outer edge of the region of interest	Change the width or height of the selection.
Corner of the region of interest	Change the width and height of the selection.
Inside the region of interest or widget	Move the selection.
The widget centre	Change the centre of symmetry.
Circle on the outer edge of the widget	Change the radius and the angle of symmetry.

Motion Analysis

Use Motion Analysis to analyse image displacement in a frame with respect to the frame before it.



To access the Motion Analysis menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip, and outputs forward and backward vectors.

Motion Analysis Menu Settings

General Settings



Quality box Select Use Full Resolution to render the image at the current resolution, or an option with decreased motion analysis accuracy and increased rendering speed.

Motion Blur

Use Motion Blur to simulate the blur created by fast moving objects.



To access the Motion Blur menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts front, matte and forward vector clips and outputs a result, outmatte, or forward Vector clip.

Motion Blur Menu Settings

Vector Input Type Settings



Vector Input Type button Select whether the vector inputs are Absolute or Normalized.

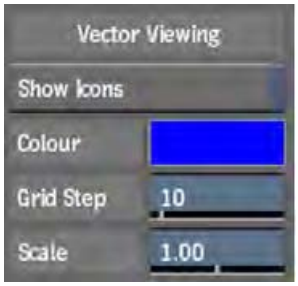
Max Dispersion field Displays the level of motion displacement in the image. Set to the same value that was set in the 3D application. Editable.

Blue Channel Magnitude field Enable to use the blue channel as a magnitude multiplication of the maximum displacement value.

Black As No Movement button Enable to set black pixels as no movement in the motion vector. When enabled, you can set a threshold value for near-black pixels.

Threshold field Displays the level of near-black pixels taken into account in the conversion.

Vector Viewing Settings



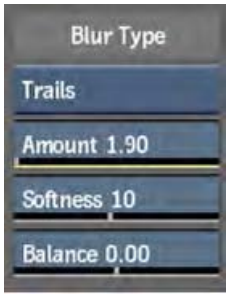
Show Icons button Displays the forward motion vectors in the Result view.

Colour pot Changes the display colour of the forward and backward motion vectors.

Grid Step field Displays the length of the pixel area used to calculate each vector.

Scale field Displays the size of the motion vectors.

Blur Type Settings



Blur Type button Enable this button to select between trails and samples. Choose trails for a pixel-based motion blur. Choose sample for an accumulation-based motion blur.

Blur Type amount field Displays the amount of motion blur applied to the image. Editable.

Blur Type softness field Displays the amount of softness applied to the trails. Only available when trails is selected as the blur type. Editable.

Blur Type balance field Displays the amount of blur that either precedes or follows the image. The more positive the number, the more blur precedes the image. The more negative the number, the more blur follows the image. Editable.

Artefacts Settings



Vector Softness field Displays the amount of softness that is applied to reduce the quality of vectors, and therefor solve precision artefacts. Editable.

Post Blur field Displays the amount of global blur applied after other effects have been applied. Apply conservatively. Editable.

Matte Fill field Display the amount used to fill in gaps in the matte. Best applied after softness adjusted. Only available when the blur type is trials. Editable.

Colour Interpolation Settings



Colour Interpolation button Enable this button to verify that the source is already premultiplied. In this case, the colour interpolation effect must treat the image differently for the effect to work properly. This button is only functional when there is an input matte.

Motion Opacity Curve

Motion Opacity Curve Allows you to control the opacity of the trail by shaping it using the curve.

Home button Reverts to the original view.

Reset box Resets the curve view.

Motion Convert

Use Motion Convert to convert normalized 2D vectos into absolute vectors that you can work with in Flame.



To access the Motion Convert menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).

This node accepts a front clip, and forward and backward vectors. It outputs a result, as well as forward and backward vectors.

Motion Vectors are 2D vectors that represent the displacement in normalized pixel units of a pixel in the current frame to the next frame (forward motion vector), or its displacement from the previous frame (backward motion vector). Motion Vectors can be rendered by a 3D application when dealing with synthetic images, or produced through image analysis when images come from live action footage.

You can attach the forward and backward outputs from the Motion Analysis node or import motion data from another source. See Motion Convert (page 1405).

Motion Convert Menu Settings

Motion Convert Node



Vector Input Type box Select whether the vector inputs are Absolute or Normalized.

Maximum Displacement field Displays the level of motion displacement in the image. Set to the same value that was set in the 3D application.

Blue Channel Magnitude button Enable to use the blue channel as a magnitude multiplication of the maximum displacement value.

Black As No Movement button Enable to set black pixels as no movement in the motion vector. When enabled, you can set a threshold value for near-black pixels.

Threshold field Displays the level of near-black pixels taken into account in the conversion. Editable.

Vector Output Type Displays the absolute vector output type mode.

Forward button Enable to display the forward motion vectors in the Result view.

Forward Arrow colour pot Displays the current colour of forward motion vectors. Editable.

Backward button Enable to display the backward motion vectors in the Result view.

Backward Arrow colour pot Displays the current colour of backward motion vectors. Editable.

Forward Grid field Displays the length of the pixel area used to calculate each forward motion vector. For example, enter 5 to sample an area of 25 pixels.

Backward Grid field Displays the length of the pixel area used to calculate each backward motion vector. For example, enter 5 to sample an area of 25 pixels.

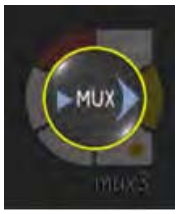
Forward Scaling field Displays the size of forward motion vectors. Editable.

Backward Scaling field Displays the size of backward motion vectors. Editable.

Blend Previous Frame field Displays the level of opacity of the current frame, which is overlaid on the previous frame. Set the value to 100 to display the current frame only.

MUX

Use MUX to make multiple output sockets and to propagate an input to multiple other nodes in the Schematic. You can also use it to clarify both its graphical representation and the connection scheme of nodes within the group.



To access the MUX menu, use:

- The I/O tab in Batch.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front clip, and outputs a result.

MUX Menu Settings



Hide Links Settings

Input button Enable to hide the input link to the MUX node.

Output button Enable to hide all output links to the MUX node.

Timing Offset Settings

Timing Offset field Displays the number of frames by which the timing of the MUX pipeline is offset. Editable.

FX Range Settings

Range Active button Enable to activate the range settings.

Range From field Displays the first impacted frame. Editable.

Range To field Displays the last impacted frame. Editable.

Range Before box Select an option to apply before the set frame range.

Range After box Select an option to apply after the set frame range.

NOTE There are two Ping Pong options included in the Range Before box and Range After box: Ping Pong and Ping Pong+. The Ping Pong repeat mode is inclusive, meaning that the last frame of the sequence is always repeated. For example, a five frame sequence would be: 1-2-3-4-5-5-4-3-2-1-1-2-3-4. The Ping Pong+ repeat mode is exclusive, meaning that the last frame of the sequence is never repeated creating a sequence that is always 1 frame shorter, but that does not create stuttering with a frame repeat.

Freeze Current Frame button Click to output the image of the currently selected frame for the duration of the clip.

Optics

Use Optics to add a glow effect to the clip in the process tree.



To access the Optics menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

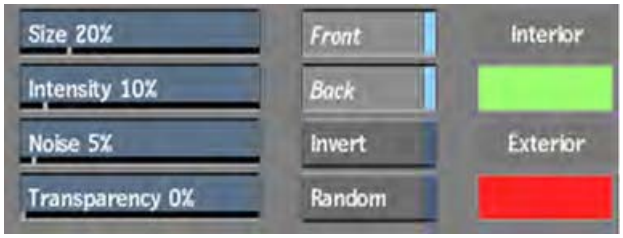
This node accepts front, back, and matte clips, and outputs a result.

The Optics node processes gaps in clips set to No Media based on the input tabs receiving the information.

Input	Result
Front	No media
Back	Black frames
Matte	No media

Optics Menu Settings

Optics Node Settings



Size field Displays the size of the glow effect. Editable.

Intensity field Displays the intensity level of the glow effect. Editable.

Noise field Displays the jitter level in the glow effect. Editable.

Transparency field Displays the transparency level of the front clip. Editable.

Optics Front button Enable to use the front clip.

Optics Back button Enable to use the back clip.

Invert button Enable to invert the matte clip.

Random button Enable to produce a different glow effect.

Interior colour pot Select a colour for the interior of the glow effect.

Exterior Colour pot Select a colour for the exterior of the glow effect.

Optics: Creating Glow Effects

Use Optics to add glow effects around front clips composited over back clips. The shape of the glow is determined by the matte you select when entering Optics.

To add a glow effect to a clip:

1. In the Main menu, click Effects.
2. In the Effects menu, click Optics.
3. Select the front, back, and matte clips, and a destination.

   The Optics menu appears. By default, Front view is enabled to speed up processing.



(a) View box (b) Optics Front and Back buttons

4. From the View box, select Result to view the glow effect.
5. Use the Optics Front and Back buttons to enable or disable the front and back clips. You can also invert the matte by enabling Invert.
6. If necessary, use the Size and Intensity fields to adjust the size and intensity of the glow effect.
7. Adjust the noise factor to make the glow jitter in the processed clip.

   NOTE If you want to create a second processed clip where the glow jitters in a slightly different way from the first one, enable Random before processing the second clip.

8. If necessary, use the Transparency field to adjust the transparency of the front clip.

   NOTE Make sure to enable the Optics Front button.

9. Use the colour pots to change the colour of the glow. Click the colour pot to display the colour picker.

   The interior and exterior glow colours are blended.

   NOTE To reset the glow, click the Reset All button and Confirm.

10. Click process.

    The generated clip appears on the destination reel.

Paint Node

About Paint Node

Paint Node is a system that provides a scalable matte painting, retouching, or restoration workflow in Batch or Batch FX.

Due to its underlying technology, Paint Node automatically scales strokes when changing the resolution, ratio, or bit depth of input clips, or when switching from Full Resolution to Proxy mode. Paint Node supports

“clipless” setups, which can be applied to any image input, while accurately reproducing the sequence of painted strokes.

The Paint Node accepts a front and matte clip as input, and creates a result and output matte clip, respectively. You can paint on the result and output matte, with a selection of brushes in different paint modes.

Paint Node also allows you to connect multiple sources and use them to paint the contents of source images onto the result. This paint operation, applied with the Reveal paint tool, can be used with in-context overlay over the result image. A front and matte clip can be connected as a source by connecting the clips to a source node. The content of source front and source matte input can be used to create brush strokes on the result and output matte.

Paint Node can be accessed from:

- Batch FX. (page 1001)
- Batch, then select a node from the Node bin. (page 1002)
- Modular Keyer. (page 1002)

The following views are available in Paint Node.

Select:	To display:
Front (F1)	The front clip or Batch or Batch FX tree input. The modifications to the front create the result clip.
Matte (F3)	The matte clip or Batch or Batch FX tree input. The modifications to the matte create the output matte.
Source Front (F1 F1)	The source front that is selected in the Sources list.
Source Matte (F3 F3)	The source matte that is selected in the Sources list.
Result (F4)	The result image. Paint can be applied to the result.
Output Matte (F4 F4)	The output matte image. Paint can be applied to the output matte.

Paint Node Menu Settings

Source Controls

Displays the front clip, matte clip, and sources. The Sources list manages the sources attached to the current node. A source is composed of a front source and a matte source. See Using Sources (page 1003) .

Add button Click to add a new source node, and select the source front and source matte. Ctrl-click to add a source node only.

Source Front option box Select Front Lock to use the current frame for the duration of the source front clip. Select Front On to unlock the clip and apply it in its original state.

Source Matte option box Select Matte Off to paint anywhere on the canvas, Matte On to limit painting to areas on the source matte, or Matte Invert to limit painting to areas outside the source matte.

Hide Strokes button Hides the strokes associated with the current source.

Clear Strokes button Clears the strokes associated with the current source.

Current Frame box Select to clear strokes for the Current Frame or the entire Sequence.

Sources List Displays the sources attached to the current node.

Brush Attributes

Sets the size of the brush and other brush attributes affecting how paint is applied to the canvas. See Brush Attributes and Attribute Modes (page 1007).

Size field Set the radius of the brush in pixels. Editable.

Rate field Set the rate at which to apply brush strokes to the canvas. Use a higher percentage value to produce a smoother continuous stroke. Editable.

Fixed Rate button Enable to make the stroke dependent on the speed at which you move the brush.

Pressure field Set the transparency on the pressure and direction of the pen. Editable.

Jitter field Set the level of dispersion. A lower value produces a greater concentration of paint. Editable.

Direction field Set the percentage of a complete rotation to rotate the brush around the Z-axis. Editable.

Roll field Set the percentage of a complete rotation to rotate the brush around the X-axis. Editable.

Brush Attribute Modes

Brush Attribute Modes option box Select an attribute as a reference value. Select a clip to use its luminance as a reference.

Brush Attribute Modes option box Select an attribute as a reference value. Select a clip to use its luminance as a reference.

Brush Attribute Modes option box Select an attribute as a reference value. Select a clip to use its luminance as a reference.

Brush Attribute Modes option box Select an attribute as a reference value. Select a clip to use its luminance as a reference.

Brush Attribute Modes option box Select an attribute as a reference value. Select a clip to use its luminance as a reference.

Brush Attribute Modes option box Select an attribute as a reference value. Select a clip to use its luminance as a reference.

Paint Mode

Sets the type of paint operation that is applied to the brush. Some brushes paint in a colour; others use contents of the result or a source. See Using Paint Modes (page 1016) and Using Blending Modes (page 1020).

Paint Modes box Select the type of effect you want to apply to the brush.

Opacity field Displays the opacity of the brush. Use a lower value to apply a more transparent colour. Editable.

Fade field Displays how quickly the stroke fades when the Fade brush attribute mode is selected. A higher value fades the stroke faster.

Blending Modes box Select the blending operation to apply to the brush colour components.

Paint On box Select to apply strokes to the current frame, from the current frame to the last frame, or to all frames in the sequence.

Scale field Displays the scale value of the reference image. Editable.

Rotate field Displays the angle of rotation to rotate the reference image. Editable.

X Offset field Displays the horizontal coordinate for the Clone offset. Editable.

Y Offset field Displays the vertical coordinate for the Clone offset. Editable.

Overlay Controls

Overlay button Enable to show a reference clip overlaid onto the canvas.

Reference box Select the view with the reference clip you want overlay.

Transparency field Set the transparency percentage of the reference clip. Editable.

Colour Palette and Brushes

Current colour pot Select the current brush colour. Editable.

Preset colour pots Select preset brush colours. Editable.

Brushes Select a default brush profile to set it as the brush stroke.

Matte Controls

Defines the area that can be used to paint. You can paint on the entire canvas or areas delimited by the matte properties. See Restricting Strokes with the Source Matte (page 1006) and Restricting Brush Strokes (page 1014).

Use Matte button Enable to limit brush strokes on the canvas to areas inside the matte.

Invert button Enable to limit brush strokes on the canvas to areas outside the matte.

Both button Enable to paint on the result and output matte simultaneously.

Canvas Controls

Controls clearing and wiping the canvas at the current frame. See Using the Canvas (page 1022).

Clear box Select whether to clear all strokes from the result image and output matte at the current frame or to clear strokes from all frames.

Wipe button Select to apply a uniform colour in a single brush stroke to the entire result image, the front only or the output matte only at the current frame.

Wipe colour pot Select the wipe brush stroke. Editable.

Use Source button Enable to use the current source to wipe over the image.

Rotation field Displays the angle of rotation used to rotate the canvas in the image window. Editable.

Node Setup

Active Enable to activate smooth filtering of pixels for enhanced display.

Pixel Spread

Use Pixel Spread to create a stretching effect from the image edges delimted by a matte. This can be useful to solve keying problems resulting from dark edges around the area to be keyed, for example.



To access the Pixel Spread menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts a front, back and matte clip, as well as a vector map clip, and outputs a result and an outmatte.

Pixel Spread Menu Settings

Type Settings



Spread Type box Select the type of pixel spread distortion effect to apply to the clip.

Spread Mode box Select whether to have the distortion effect work in expansion mode or contraction mode.Active when Spread Type is set to Stretch, Parallax, or Interpolate.

Spread Settings



The following settings are available when Interpolate is selected from the Spread Type box.

Interpolation Softness Amount field Displays the amount of softness applied. Editable.

Edges Shrink Amount field Displays the amount of shrinking applied to the edges. Editable.

Edges Shrink Softness field Displays the softness of the edge. Editable.

Interpolation Source Premultiplied button Enable to verify if the source is already premultiplied, in which case the interpolation effect must treat the image differently for the effect to work properly.



The following settings are available when Parallax is selected from the Spread Type box.

Parallax Amount field Displays the amount of parallax distortion applied to the clip, in pixels. Editable.

Parallax X Bias field Displays the amount of horizontal offset applied to the parallax distortion effect, in pixels. Editable.

Parallax Y Bias field Displays the amount of vertical offset applied to the parallax distortion effect, in pixels. Editable.

Parallax Rotation field Displays the degree of rotation applied to the pixel distortion. Editable.

Edges Width field Displays the width of the distortion edge, in pixels. Editable.

Edges MidPoint field Displays the offset of the distortion edge from the middle. Editable.

Spread Blur Amount field Displays the amount of blur applied to the clip following the direction of the edge. Editable.

Parallax Repeat Mode box Select an option to fill the empty portions of the frame.



The following settings are available when Stretch is selected from the Spread Type box.

Stretch Amount field Displays the amount of stretching distortion applied to the clip, in pixels. Editable.

Edges Width field Displays the width of the distortion edge, in pixels. Editable.

Edges MidPoint field Displays the offset of the distortion edge from the middle. Editable.

Spread Blur Amount field Displays the amount of blur applied to the clip following the direction of the edge. Editable.



The following settings are available when Vector Warp is selected from the Spread Type box.

Motion Distance field Displays the amount of distortion to be applied to the image based on a given motion vector. A negative amount inverts the direction of the spread. Editable.

Motion Threshold field Displays the percentage for the cut-off point below which motion data is not applied to the image. Editable.

Vector X Origin field Displays the starting point of any horizontal motion, which can be used to offset values entered for X Gain. Editable.

Vector Y Origin field Displays the starting point of any vertical motion, which can be used to offset values entered for Y Gain. Editable.

Vector Overlap button Enable to invert the effects of the vector warp.

Red X Gain field Displays the amount of gain used from the red channel to augment the horizontal motion of the distortion effect on the image. Editable.

Red Y Gain field Displays the amount of gain used from the red channel to augment the vertical motion of the distortion effect on the image. Editable.

Green X Gain field Displays the amount of gain used from the green channel to augment the horizontal motion of the distortion effect on the image. Editable.

Green Y Gain field Displays the amount of gain used from the green channel to augment the vertical motion of the distortion effect on the image. Editable.

Blue X Gain field Displays the amount of gain used from the blue channel to augment the horizontal motion of the distortion effect on the image. Editable.

Blue Y Gain field Displays the amount of gain used from the blue channel to augment the vertical motion of the distortion effect on the image. Editable.

Spread Blur Amount field Displays the amount of blur applied to the clip following the direction of the edge. Editable.

Output Settings



Result Output box Select whether to output a combined result or the pixel spread effect only.

Alpha Output box Select Input to output the input matte, or Spread Matte to output the matte generated by the pixel spread.

Pulldown

Use the Pulldown to remove or add pulldown to a clip.



To access the Pulldown menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front clip as input, and outputs a result.

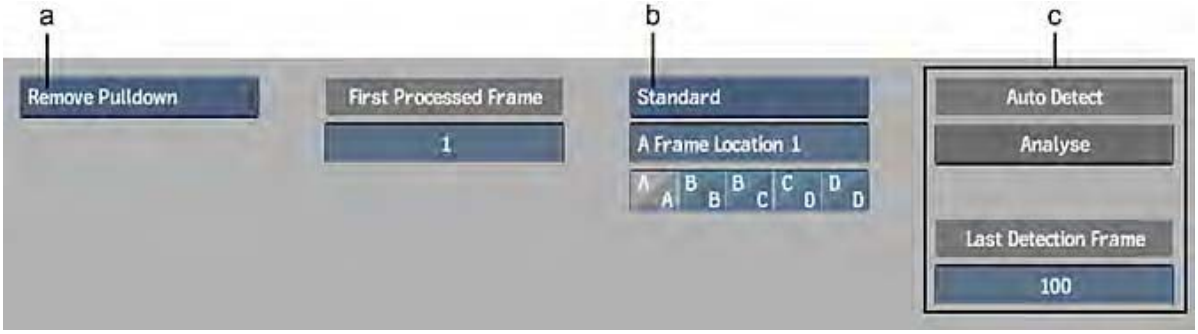
Additionally, the Pulldown node supports the following:

- 2:3 pulldown, also known as standard pulldown
- 2:3:3:2 pulldown, also known as advanced pulldown
- 24-to-25 fps conversion, also known as PAL pulldown

Pulldown Menu Settings

Pulldown Settings

When removing a pulldown, you can analyse the clip to automatically detect the type of pulldown and the A frame. If the analysis fails to determine the type of pulldown or the location of the AA frame, manually set the Pulldown and the A Frame Location options.



(a) Pulldown Mode option box (b) Pulldown Type option box (c) Remove Pulldown options; available when Pulldown Mode is set to Remove Pulldown

Pulldown type option box Select an option to add or remove pulldown from the clip.

First Rendered Frame field Displays the first frame at which output is rendered from the node. Unrendered output does not display any media.Editable.

Pulldown option box Select the type of pulldown process to apply to the clip.

A Frame Location field Displays the value of a reference (AA) frame. The selected frame becomes the frame of reference when adding or removing pulldown frames.Editable.

NOTE The following settings are available when Remove Pulldown is selected from the Pulldown Type option box.

Analyze button Click to determine the type of pulldown used and the A frame.

Last Detection Frame field Displays the value of the last frame of the clip used by the Analyse button. Selecting a subset of a clip speeds up the analysis. Editable.

NOTE When the node renders a transition, preceding and subsequent frames with no media are replaced with black frames.

Quick Composite

Use Quick Composite for one-step compositing of a front and back clip using a matte clip.



To access the Quick Composite menu, use:

- Batch, then select a node from the Node bin.

- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front, back and matte clip, and outputs a result.

You can use the Front and Back fields to increase or decrease the luminance of the front and back clips. The values specify the gain for the clips. Quick Composite multiplies the luminance values in the clips by the gain values you set. The resulting luminance values are clipped at the maximum and minimum values of 255 (white) and 0 (black), respectively.

Gain is expressed as a percentage value. The default value of 100% has no effect on the image since the luminance values are multiplied by 1. To increase the gain, set the value above 100%. To decrease the gain, set the value below 100%.

Quick Composite Menu Settings

Quick Composite Node Settings



Blend Mode box Select an option for blending clips in the composite.

Select:	To:
Blend	Use subtractive blending.
Additive	Use additive blending. Select this option if the front clip has a black background, as with a computer-generated image. This option eliminates the black fringe at the edge of the front image.
FrontMin	Prevent loss of detail from the front image. For example, if the front clip is a human model, this option prevents losing details such as hair in the composite clip.

Front Gain field Displays the value of the luminance of the front clip. Editable.

Back Gain field Displays the value of the luminance of the back clip. Editable.

Invert Matte button Enable to invert the matte.

Recursive Ops

Use Recursive Ops to perform a wide range of recursive, animation-based blending effects. Recursive Ops uses the processed result of the previous frame and blends it with the current frame using a selected blending mode. It features a built-in colour selection, which constrains the accumulation effect within that selection. Additionally, it accepts an external matte input which also conditions the way in which the internal blending occurs.



To access the Recursive Ops menu, use:

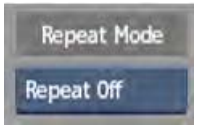
- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts front and matte clips, and outputs a result and outmatte.

Recursive Ops Menu Settings

Regen button Enable to dynamically refresh the image as changes are made to the settings.

Repeat Mode Settings



Repeat Mode options box Select an option to fill the empty portions of the frame.

Range Settings



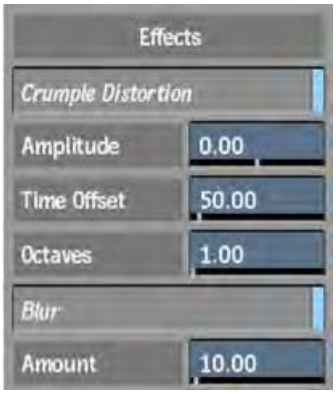
Range From field Displays the beginning of the range of frames that to be impacted by the effect. Editable.

Range To field Displays the end of the range of frames that to be impacted by the effect. Editable.

Range Before box Select an option to be applied before the set frame range. Bypass eliminates the effect, while Cycle repeats the effect.

Range After box Select an option to be applied afer the set frame range. Bypass eliminates the effect, while Cycle repeats the effect.

Effects Settings



Crumple Distortion button Enable to activate the crumple settings.

Amplitude field Displays the amount of distortion. Increase the value to increase the crumple effect. Editable.

Time Offset field Displays the time offset interval of the crumpling. Editable.

Octaves field Displays the number of layers summed in the operation, from 0 to 10. Increase the value to increase the fractal crumple effect. Editable.

Blur button Enable to activate the blur settings.

Amount field Displays the amount of blur applied to the image. Editable.

2D Transform Settings



Active button Enable to activate the 2D Transform settings.

Show Icons button Enable to display the vertex editing tools in the image window.

Position X field Displays the horizontal position of the transformation. Editable.

Position Y field Displays the vertical position of the transformation. Editable.

Centre X field Displays the centre point value of the transformation along the horizontal axis. Editable.

Centre Y field Displays the centre point value of the transformation along the vertical axis. Editable.

Scale X field Displays the horizontal scale factor. Editable.

Scale Y field Displays the vertical scale factor. Editable.

Proportional button Enable to scale X and Y values proportionally.

Rotation field Displays the rotation of the transformation. Editable.

Colour Correction Settings



Activate button Enable to activate the color correction settings.

Clamping box Select a clamping option.

Gain Trackball Adjusts the gain of the input.

Saturation field Displays level of colour purity in the image. Editable.

Gamma field Displays the gamma level. Editable.

Offset field Displays a value that modifies all of the colour parameters. Editable.

Red Gain field Set the percentage of colour values in the red channel. Editable.

Green Gain field Set the percentage of colour values in the green channel. Editable.

Blue Gain field Set the percentage of colour values in the blue channel. Editable.

Luma Gain field Set the percentage of luma gain value to display. Editable.

Proportional button Enable to adjust the gain of the colour values proportionally.

Blending Settings



Blending options box Select a logical operation that can be used to blend the front clip and the result clip.

Transparency field Displays the percentage of blending when the result is composited on the front clip. Editable.

Use Matte button Enable to apply the effect with the areas defined by the matte.

Rendering Settings



Clamp Render box Select a clamping option for colour and luminance values on output in the 16-bit floating point rendering pipeline.

Matte Output Settings



Matte Output options box Select an matte output option. Choose Selective to select the colour you wish to keep.

Selective Tolerance field Displays the tolderance level of the selected colour of the matte output. Editable. Available when Selective is chosen in the Matte Ouput options box.

Selective Colour box Click to enable the crosshair to select a colour in the image to be used as the matte output. Available when Selective is chosen in the Matte Ouput options box.

Regrain

Use Regrain to add grain from the RGB channels of a selected colour in an image.



To access the Regrain menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.

This node accepts front, back, and matte clips as input, and outputs a result.

Regrain Menu Settings

General Settings

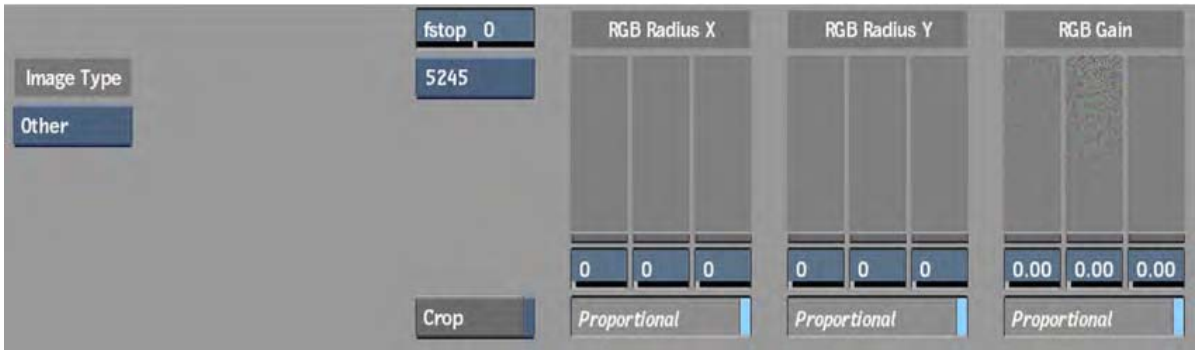


Image Type box Select the type of image data that is input into the node or tool. This determines the type of transformation applied to the input clip.

fstop field Displays the relative exposure offset. Use to compensate for under or over exposure. Editable.

Grain Signature box Select a film stock grain to add to the clip.

Crop box Applies the regrain to a specific region in the clip.

Red X Radius slider Displays the value of the radius on the x-axis for the red channel. Editable.

Red X Radius field Displays the value of the radius on the x-axis for the red channel. Editable.

Green X Radius slider Displays the value of the radius on the x-axis for the green channel. Editable.

Green X Radius field Displays the value of the radius on the x-axis for the green channel. Editable.

Blue X Radius slider Displays the value of the radius on the x-axis for the blue channel. Editable.

Blue X Radius field Displays the value of the radius on the x-axis for the blue channel. Editable.

Proportional RGB Radius X button Enable to adjust the radius on the x-axis of the red, green, and blue channels proportionally.

Red Y Radius slider Displays the value of the radius on the y-axis for the red channel. Editable.

Red Y Radius field Displays the value of the radius on the y-axis for the red channel. Editable.

Green Y Radius slider Displays the value of the radius on the y-axis for the green channel. Editable.

Green Y Radius field Displays the value of the radius on the y-axis for the green channel. Editable.

Blue Y Radius slider Displays the value of the radius on the y-axis for the blue channel. Editable.

Blue Y Radius field Displays the value of the radius on the y-axis for the blue channel. Editable.

Proportional RGB Radius Y button Enable to adjust the radius on the y-axis of the red, green, and blue channels proportionally.

Red Gain slider Displays the grain value in the red channel. Editable.

Red Gain field Displays the grain value in the red channel. Editable.

Green Gain slider Displays the grain value in the green channel. Editable.

Green Gain field Displays the grain value in the green channel. Editable.

Blue Gain slider Displays the grain value in the blue channel. Editable.

Blue Gain field Displays the grain value in the blue channel. Editable.

Proportional RGB Gain button Enable to adjust the grain on the red, green, and blue channels proportionally.

NOTE The following options are available when Mono or Custom are selected from the Grain Signature box.

Curves Display the curves for the luminance and each RGB channel.

Histogram box Select to display the red, green, blue, or luminance histogram in the graph. Select Current Curve to display the histogram for the currently selected Curves Channel.

Curves box Select to display the red, green, blue or all curves in the graph.

Home button Restores the position of panned or zoomed curves to the default setting.

Reset button Resets to default curve settings.

Resize

Use Resize to change a clip's resolution, frame depth, and aspect ratio.



To access the Resize menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.
- The Modular Keyer, then select a node from the Node bin.
- Timeline effects, then choose Resize.

This node accepts a front clip, and outputs a result.

You can also use Resize to pan and scan the destination image relative to the source. You can then select and animate the portion of the source clip that appears as the destination clip, and process to create the result image.

NOTE You can use Resize to output, for example, a HD project to a lower resolution for quick viewing.

Resize Menu Settings

Source Settings

Use the Source settings to set or animate the position and size of the crop box.



Position/Scale Settings

X Position field Displays the horizontal position from the centre of the crop box relative to the centre of the source frame, in pixels. Drag left or right, or click to enter a new X Position value.

Y Position field Displays the vertical position from the centre of the crop box relative to the centre of the source frame, in pixels. Drag left or right, or click to enter a new Y Position value.

X Scale field Displays the horizontal scale of the crop box relative to the Crop Box Width field value, as a percentage. Drag left or right, or click to enter a new X Scale value.

Y Scale field Displays the vertical scale of the crop box relative to the Crop Box Height field value, as a percentage. Drag left or right, or click to enter a new Y Scale value.

Crop Mode box Select an option to determine the scaling behaviour of the crop box while repositioning or rescaling. Use Free to adjust the crop box freely. Use Prop to use the current Crop Box Width and Crop Box Height settings proportionally. Use Source or Destination to use the respective aspect ratio for the crop box.

Source Ratio field Displays the aspect ratio of the crop box in the source frame. Editable.

Crop Options Settings

Crop Box Width field Displays the current width setting of the crop box, in pixels. Editable.

Crop Box Height field Displays the current height setting of the crop box, in pixels. Editable.

Source Width button Uses the width settings of the source frame for the Crop Box Width field.

Source Height button Uses the height settings of the source frame for the Crop Box Height field.

Source Frame button Uses the width and height settings of the source frame for the Crop Box Width and Crop Box Height fields, respectively.

Destination Width button Uses the width settings of the destination frame for the Crop Box Width field.

Destination Height button Uses the height settings of the destination frame for the Crop Box Height field.

Destination Frame button Uses the width and height settings of the destination frame for the Crop Box Width and Crop Box Height fields, respectively.

Crop Box Border colour pot Displays the current colour of the crop box border. Editable.

Crop Box Line Style box Select the line style for the crop box.

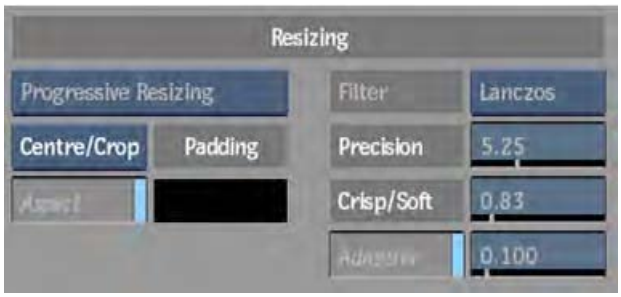
Source Pre-Rendering

Source Pre-Rendering buttons Select a conversion method to use for the resize. Use the Both Fields button when both source and destination formats are interlaced or progressive. Use the Field Merge button to combine the two fields of the source clip. Use the Deinterlace button to select one field, in which case the Source Pre-Rendering Field box is enabled.

Source Pre-Rendering Field box Select a field to use for the resize. Active when the Deinterlace Source Pre-Rendering button is enabled.

Resizing Settings

Use the Resize settings to change the size of a clip.



Resize Field Format box Select an option to determine whether the resize is performed on progressive or interlaced frames, if both source and destination clips are interlaced. If either clip is progressive, the resize operation is always progressive.

Fit Method box Select a fit method to be applied to the selected clip.

Precision field Displays the frequency cut-off point used during resize. Editable.

Crisp/Soft field Displays the amount of blurring used during resize. Editable.

Resize Filter box This option is available when Letterbox, Crop Edges or Fill is selected from the Fit Method box. Select the filter option to determine the quality of the interpolated resize result.

Aspect button This option is available when Letterbox or Crop Edges is selected from the Fit Method box. Enable to use non-square pixel formats. Active when Fit Method is set to Crop Edges or Letterbox.

Destination Padding colour pot Displays the colour used for padding the destination frame when the source image is smaller than the destination. Editable.

Adaptive Deinterlacing button Enable to use data from the adjoining interlaced field to improve the results of the resizing algorithm. This is most effective for clips containing stationary objects.

Adaptive Deinterlacing field Displays the amount of data used from the adjoining field for adaptive interlacing during resize. Active when Adaptive button is enabled.

Destination Settings

Use the Destination settings to define the format of the resized clip.



Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

Width field Displays the custom width resolution of the clip. Editable.

Height field Displays the custom width resolution of the clip. Editable.

Aspect Ratio Presets box Select a standard frame aspect ratio. Select the Set to w:h option to set the clip to use square pixels. Select Custom to define a custom frame aspect ratio in the Aspect Ratio field.

Aspect Ratio field Displays the custom render/output aspect ratio. Editable

Frame Depth box Select the render/output frame depth of clips.

Scan Mode box Select the scan mode of clips.

Crop/Lock Output button Enable to lock the destination resolution to match the crop box. Use this option to animate the resolution of a clip and change it on a per-frame basis. Available when you access Resize settings.

Sparks

Sparks are software plug-ins created by Autodesk or third-party developers.

To access Sparks, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

The Autodesk SparksAPI Reference Guide is available in PDF format. If you are interested in marketing or selling your Sparks, you must apply to the system Sparks program. To request an application, send an e-mail message to *email Sparks Manager* .

Using a Sparks Plug-in

A Sparks plug-in functions in the same way as other commands or modules. Before using a Sparks plug-in, you will have to load it. After loading the plug-ins, you can replace them if all the Sparks buttons are already in use.

To load a Sparks plug-in, do one of the following:

1. From the Tools menu, select the Plugins tab and then click one of the Sparks buttons.
2. Or, from Batch or Batch FX, select the Sparks node and drag it into the schematic.
3. Or, from the Timeline, select a clip and add a Sparks Timeline FX to it.

Once you have loaded a Sparks plug-in using one of these methods, the Sparks file browser will appear.

NOTE An L on a Sparks button indicates that you can load a Sparks plug-in. An E on a Sparks button indicates that there are settings for the plug-in. An S on a Sparks button indicates that you can enter the module using the same media from the previous session.

4 Navigate to *usr/discreet/sparks*.

You can also load Sparks from *usr//discreet/<product home>/sparks*.

5 Select a Sparks plug-in.

You are returned to your previous location. The name of the Sparks plug-in now appears on the selected button, however the L indicating that you can load a Sparks plug-in no longer appears.

To exit the Sparks plug-in file browser without loading a plug-in, click the Exit Sparks Browser button.

To use Sparks plug-ins from the Tools menu, do the following:

1 Click a loaded Sparks button.

The Sparks controls, if any, appear.

2 Enter the parameter values for the selected Sparks plug-in.

3 Select the source clips and the destination reel.

The processed clip appears in the destination reel.

To use Sparks plug-ins from Batch or Batch FX, do the following:

1 Select the Sparks node from the node bin and drag it into the schematic.

The Sparks Plug-in file browser appears.

2 Select a plug-in.

You are returned to the schematic and a Sparks node corresponding to the plug-in that you selected now appears in the schematic.

3 Double-click the Sparks node to access and adjust the settings for the Sparks node.

To use Sparks plug-ins from the Timeline, do the following:

1 Select a clip on the Timeline.

2 Click the FX button and add a Sparks Timeline FX to the clip.

3 Click the Sparks button to enter the Sparks Plug-in file browser.

4 Select a Sparks Plug-in.

You are returned to the Timeline.

The Sparks button in the Effects pipeline now appears with the name of the plug-in that you selected, and it's settings, if any, are available in the quick menu. You can also double-click the Sparks button to enter the full editor, if the Sparks Plug-in has additional settings.

If you want to load a new Sparks plug-in, but all the Sparks buttons are already in use, you can replace an existing Sparks plug-in with the new one. This can be done from the Tools menu, Batch or Batch FXor the Timeline.

To replace a Sparks plug-in, do the following:

1 Press Alt and click the existing Sparks button.

The Sparks file browser appears.

2 Navigate to *usr/discreet/sparks* or *usr//discreet/<product home>/sparks* and select a Sparks plug-in.

You are returned to your previous location, and the name of the new Sparks plug-in appears on the selected button.

Using Sparks in Batch or Batch FX

There are two Sparks nodes that you can access from Batch or Batch FX: the Sparks node and the Sparks Load node.

The Sparks node allows you to load Sparks Plug-ins by accessing the Sparks file browser. The number of inputs and bit depth that a Sparks node accepts depends on the Sparks plug-in. A Sparks node has light grey source tabs, since each Sparks plug-in varies in the type of clip it uses

When working with a Sparks plug-in, missing media in front clips set to No Media can be converted into black frames while they are in use. When missing media is output from this node, it returns to a No Media state, regardless of whether it was set to display differently in the node.

NOTE When you attempt to use a Sparks plug-in, the error message "SPARK IS NOT SUPPORTED" may appear. This means the selected Sparks plug-in cannot be used with Batch or Batch FX. Contact the company that developed the Sparks plug-in for information on obtaining a compatible version.

The Sparks Load node allows you to populate multiple Sparks nodes at the same time into a destination bin. Populating a bin with predefined Sparks nodes saves you time since you do not have to access the Sparks browser each time you want to use one of the preloaded Sparks.

Unlike the Sparks node, the Sparks Load node itself cannot be dragged to the schematic; only the predefined node populated into a bin can be dragged to the schematic.

To create a predefined Sparks node in a bin:

1. Drag the Sparks Load node on top of any tab in the ALL Tools bin except the ALL Nodes tab.
   The Sparks Plug-in file browser appears.
2. Select a Sparks Plug-in or select multiple plug-ins by Ctrl-clicking the Sparks you want to load.
3. Click load.

The selected Sparks are created as their own Sparks nodes in the destination bin. To use one of the preloaded Sparks, drag it from the destination bin to the schematic. You do not have to re-enter the Sparks browser to load a Sparks if you preloaded it into a bin.

For more information on Sparks plug-ins, see Using a Sparks Plug-in (page 1427).

Stabilizer

Stabilizing and Tracking

Use the Stabilizer to remove camera instability and motion jitter, and to track reference points in your clips. You can also use the Stabilizer to produce 2D motion or lock a bilinear surface to the clip's background. With tracking, a point or points on the clip are tracked as they move through the scene. You can then apply the resulting motion path to an object on another layer so that it follows the same path as the object you tracked.

Stabilizing is the inverse of tracking. With stabilizing, the motion path is used to shift the scene so that the point that is tracked remains fixed at one position.

Tracking and stabilizing are often processes of trial and error. It is recommended that you track or stabilize using the default settings. If the tracker box strays from its original point, you can fine-tune the analysis.

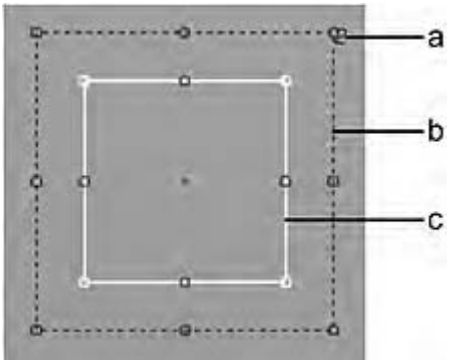
Accessing the Stabilizer

You access the Stabilizer differently, depending on how you want to track or stabilize a clip. For example, when you access the Stabilizer from the Tools tab, you stabilize with one tracker. When you access from Action, you have the option to use two trackers. You need two trackers when the clip you want to stabilize has a camera roll or zoom—the second tracker enables you to track the rotation and zoom of the camera.

Access the Stabilizer from:	To:
The Tools tab	Stabilize.
Action/ Timeline Effects Axis	Track or stabilize.
Keyer/ GMask	Track a garbage mask or the vertices of a GMask.
Distort	Track vertices or the axis of a spline when warping or morphing.
Warper	Track points or an axis of a mesh when warping or morphing.
3D Tracker	Provide a 2D tracking path for 3D manual tracking.
Paint	Track an AutoPaint stroke.
Blur, Glow	Track the center point of the radial blur.
2D Transform	Provide correction for 2D motion, rotation and scaling.

How the Stabilizer Works

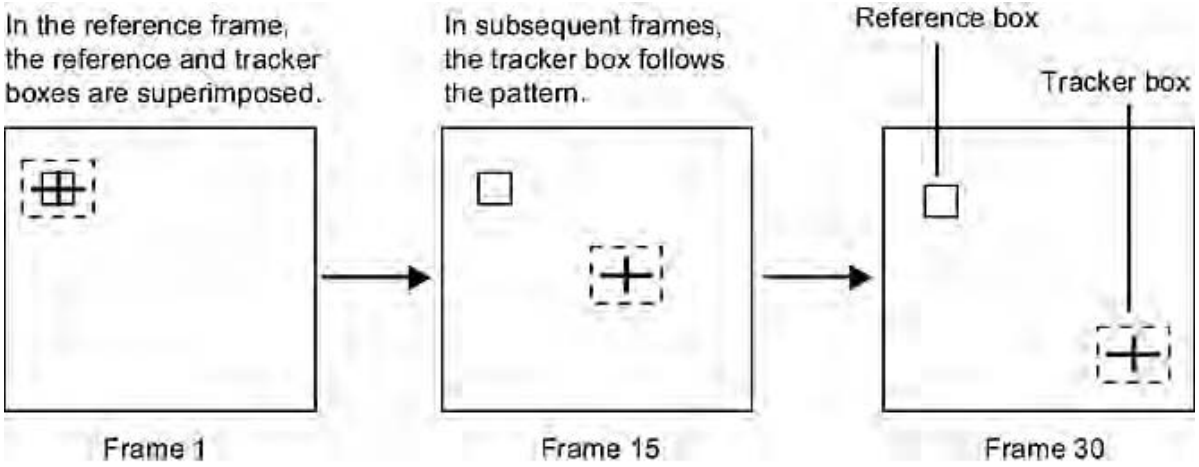
The Stabilizer uses trackers to generate tracking data. Each tracker consists of a solid box, called the *reference box*, and a dashed box, called the *tracker box*. The reference box establishes the reference point (the feature to track or stabilize) in any frame of the sequence. The tracker box indicates to the Stabilizer where to locate the reference point. The tracker box follows the frame-to-frame movement of the reference point.



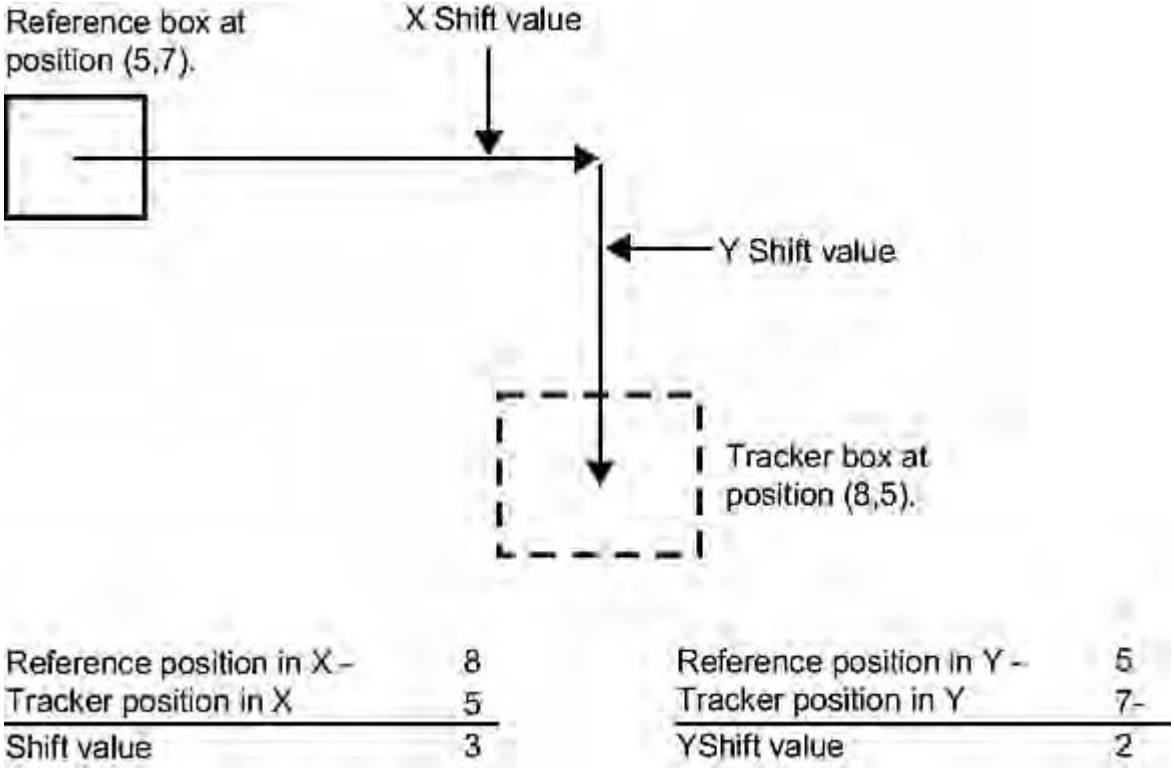
(a) Tracker number (b) Tracker box (c) Reference box

You start by selecting one or more reference points on your clip. Locate the first frame containing the movement to be tracked (the reference frame). In general, the reference frame is the first frame of the sequence. The choice of the reference point depends on whether you are tracking or stabilizing. When tracking, the reference point is a feature you want to track; when stabilizing, the reference point represents the point around which the image is stabilized. See Selecting a Reference Point (page 862) for details. Place the reference box(es) around the selected feature(s).

Once you have set the tracker positions, start the tracking process, also referred to as analyzing the clip. During the analysis, the tracker box associated with each tracker moves as the Stabilizer looks for a pattern that matches the reference in each frame of the clip.



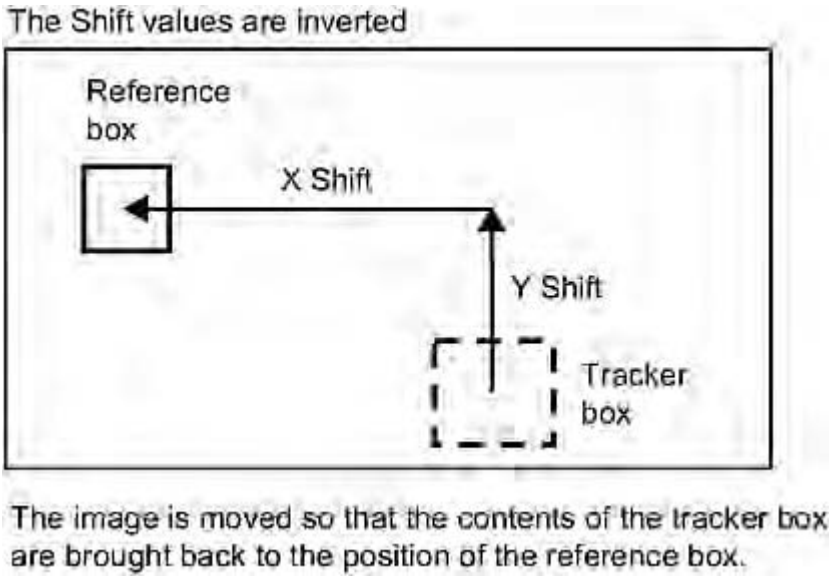
The Stabilizer calculates the difference between the position of the tracker box and the position of the reference box to produce X and Y Shift values. Shift values represent a measurement in pixels and subpixels of how much the reference point has moved.

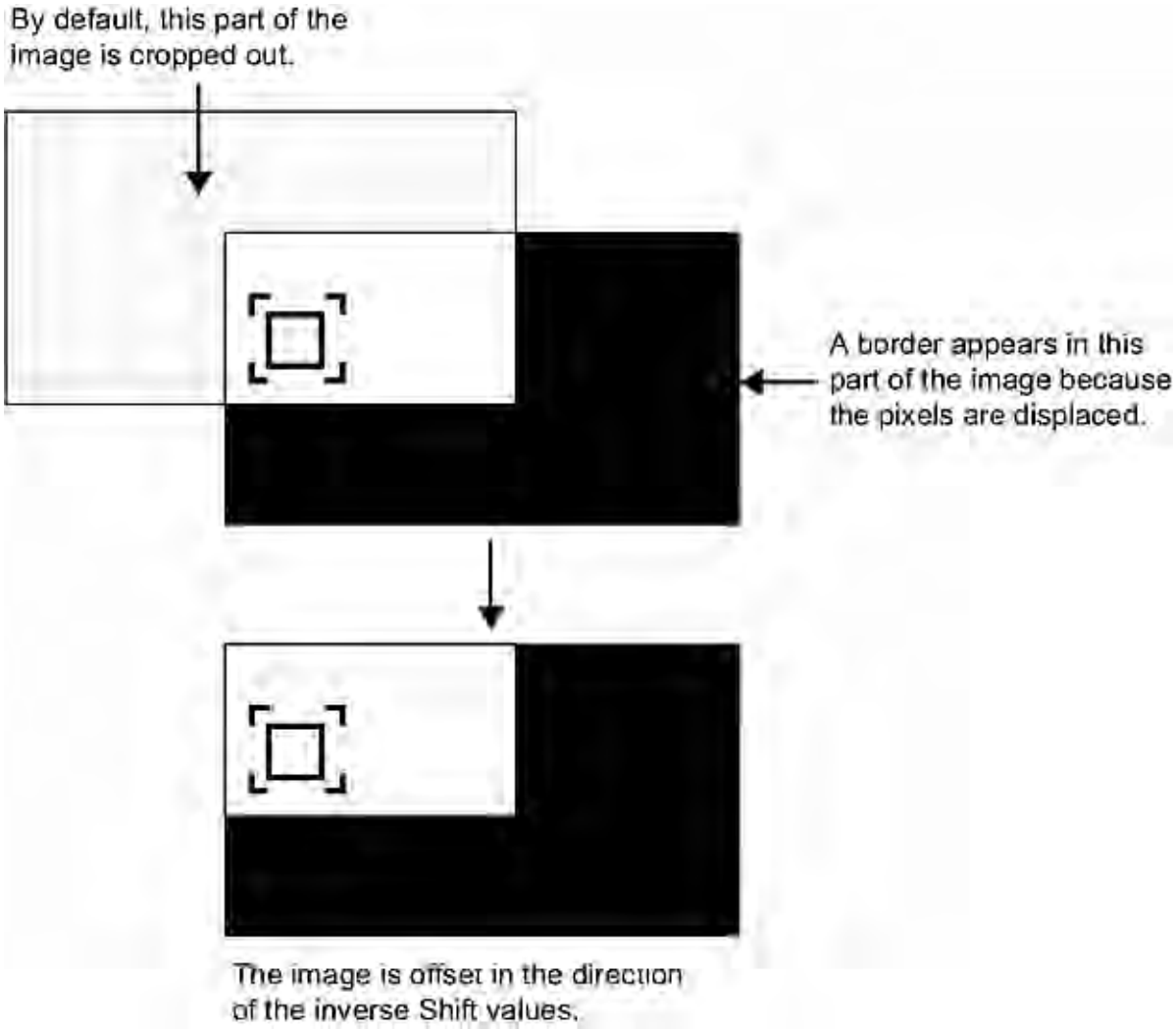


When the analysis is complete, you fine-tune it if a tracker box has strayed from the reference it was supposed to follow. Once you are satisfied with the results, you can apply the data to the clip.

To track, the Stabilizer applies the Shift values "as is." To stabilize, the Stabilizer inverts the X and Y Shift values in each frame of the sequence, and moves the image according to these values. This gives the impression that the reference point stays in the same position throughout the sequence. Because the image is moved

during stabilization, a border appears on one or more edges, which means that you lose some pixels. The following illustrations summarize the process.





Stabilizer Menu Settings

Miscellaneous Buttons

Exit button Exits the Stabilizer tool.

Return button Returns to the previous tool.

Load button Loads a setup.

Save button Saves a setup.

Setup Name field Displays the name of the last saved setup.

Revert button Reverts to the last saved setup.

Step Render field Displays the frame number interval to be rendered (to see intermediate results, for example). Editable.

Scale and Shift Option box Select a method to fill or remove the area where the image has been shifted after stabilization.

Colour pot Displays the colour to fill the area where the image has been shifted after stabilization (available if Shift or Letterbox is chosen in the Scale and Shift Option box). Editable.

Player button Opens the Player to view the rendered clip.

Context button Enable to view the tracking result with all of the Action scene, except for the selected node (and any children of the selected node).

Setup button Opens the Setup menu, where you can specify user interface and tracking preferences.

Animation button Opens the channel editor, where you can animate the various Stabilizer settings.

Expand/Collapse box Select whether to expand or collapse selected animation channels.

View box Select an option to set the view in the image window.

Auto Key button Enable to set a keyframe automatically each time you change a value at any frame.

Set Key button Sets a keyframe at the selected frame.

Delete Key button Deletes the selected keyframe.

Current Frame field Enter a frame number to jump to the corresponding frame.

Duration field Displays the duration of the clip in frames.

Reset box Select an option to reset shift, tracking, reference, or all data.

Reset All button Resets all parameters.

Grid button Accesses the overlay menu.

View button Accesses the viewing settings menu.

Undo button Undoes the last action performed.

Main Stabilizer Menu

Tracker buttons Select the Tracker to work with.

Add Tracker button Click to add a new tracker.

Active button Enable to activate the selected tracker.

Tracker colour pot Displays the colour of the selected tracker. Editable.

When you add a new tracker, the system automatically assigns it a unique colour so that you can easily distinguish between multiple trackers. You can change the colour of an individual tracker or of all the trackers at once.

Tolerance field Displays a value the Stabilizer uses to match reference points from frame to frame and to set keyframes. Editable.

Tracker Selection box Select which trackers are affected when you change a parameter.

Analyze button Click to generate stabilization or tracking data.

Step button Click to analyze a single frame and advance to the next frame.

Direction box Select to analyze forward or backward.

Snap button Click to redefine the reference at the selected frame.

Lock Key button Click to lock the selected keyframe as a point on the tracking path. Click again to unlock.

Delete Key button Click to delete the selected point on the tracking path.

Source option box Select whether to track in Progressive (frame mode) or Interlaced (field mode).

By default, the Stabilizer works in Progressive (frame mode). Select Interlaced from the Source Option box mode when working with interlaced images, or with images that display a lot of field jitter. In Interlaced (field) mode, the Stabilizer sets two keyframes for every frame: one for the even field and one for the odd field, and an asterisk appears in the image viewer's current frame display to indicate the second field for each frame.

Fixed X button Enable to remove vertical jitter.

Fixed Y button Enable to remove horizontal jitter.

Reference X field Displays the position of the reference box along the X axis. Editable.

Reference Y field Displays the position of the reference box along the Y axis. Editable.

Reference Width field Displays the width of the reference box. Editable.

Reference Height field Displays the height of the reference box. Editable.

Fixed Reference button Enable to track the movement of the reference point specified in the reference frame. Disable to update the reference point at each frame during tracking.

Normally, you use the Stabilizer with the Fixed Reference button enabled. Throughout the analysis, the tracker box follows the movement of the reference point that you specified in the reference frame.

You may want to track a pattern that changes considerably from the first frame to the last frame in the clip. For example, the pattern may be rotating or may change size or shape. In this case, disable the Fixed Reference button. The reference point is then updated in each frame. In each frame of the analysis, the Stabilizer looks for the reference point from the previous frame.

NOTE Unless you are in Gang mode, you must set the Fixed Reference button for each active tracker individually.

Tracker X field Displays the position of the tracker box along the X axis. Editable.

Use these options in the Stabilizer menu to determine whether to scale or shift a stabilized image, to track in frame or field mode, and to fix the reference box.

When a clip is stabilized, the image is shifted. You can choose any of the following from the Scale and Shift Option box to fill or remove the area where the image has been shifted.

Select:	To:
Roll	Wrap the image around to fill the area.
Fill	Rescale both dimensions of the image independently to make it fit into the frame. Note that this can change the aspect ratio.
Crop Edges	Rescale the shortest edge of the image to fit into the frame and crop the longest edge. There will not be a black border.
Letterbox	Rescale the longest edge of the image to fit into the frame and fill the rest of the image with a black border.
Shift	Use a colour to fill the area. Use the adjacent colour pot to pick the colour. You will lose texture and will fill the rest of the image with the color you picked.

Tracker Y field Displays the position of the tracker box along the Y axis. Editable.

Tracker Width field Displays the width of the tracker box. Editable.

Tracker Height field Displays the height of the tracker box. Editable.

Import Track button Open the Import Stabilizer menu, where you to import a text file of saved tracking data.

Export Track button Click to open the Export Stabilizer menu, where you to export a text file of saved tracking data.

Shift X field Displays the difference between the position of the reference box (0,0) and the position of the tracker box in the current frame along the X axis. Editable.

Shift Y field Displays the difference between the position of the reference box (0,0) and the position of the tracker box in the current frame along the Y axis. Editable.

Import Shift button Click to open the Import Stabilizer menu to allow you to import a text file of saved shift data.

Shift Copy button Click to copy the selected Shift channel (including the aspect ratio of the clip).

Export Shift button Click to open the Export Stabilizer menu to allow you to export a text file of saved shift data.

Offset X field Displays the offset X axis value. Editable.

Offset Y field Displays the offset Y axis value. Editable.

Setup Menu

Pretracking button Enable to preview the motion path for a specified number of frames. Active when Path is enabled.

Before analysing, you can use the Pretracking option to preview the motion path for a specified number of frames. You can then adjust the tracker position, if necessary, to find the best reference point. The Pretracking option applies only to the selected tracker, regardless of whether you selected Solo, Selected, or Gang in the Tracker Selection box. If you move or resize a tracker with Pretracking enabled, the next frames are analysed.

Pretracking field Displays the number of frames to pretrack. Editable.

Oversampling button Enable to stabilize using a high-quality subpixel shift algorithm. Disable to stabilize using texture memory.

Use Oversampling to toggle high-quality subpixel image adjustment on and off:

- When Oversampling is off, texture memory is used.
- When Oversampling is on, a high-quality subpixel shift algorithm is used. The resulting image is sharp.

After you stabilize a clip with Oversampling on, rotoscope it in Paint to remove the border that appears at the edges. Avoid using Action (or any other component that uses texture memory) to do this because you will lose the benefits of Oversampling.

NOTE You cannot use the Oversampling and Crop Edges options together. If you select Crop Edges, texture memory is used even if Oversampling is on.

Auto Pan button Enable to allow the part of the image that is selected to stay in the image window when zoomed in.

Opacity field Displays the opacity of the reference image. Editable.

The reference image (the image where you placed the reference box) appears in transparency when you select a keyframe. Adjust the opacity of the image to make it more or less transparent. When the opacity is

set to 0, the reference image does not appear. When the opacity is set to 100, the reference image is completely opaque.

Zoom field Displays the magnification factor of the reference box when selected. Editable.

By default, the reference box turns into a magnifying glass when you select it. Increase or reduce the magnification factor in the Zoom box.

Zoom:	Result:
0	No zooming. The crosshair appears in the tracker box after you analyse the clip.
1	No zooming and no crosshair.
2, 3, 4, 5	A magnification factor of 2, 3, 4, and 5, and a crosshair in the tracker box.

Path button Enable to display the tracking path.

The tracking path is the path that the reference point makes as it changes position from frame to frame. By default, the tracking path appears in the image window. You can turn it off by disabling the Path button.

Colour Corrector Input button Enable to display any Colour Corrector settings on the clip.

Linetest button Enable to display the context view at a lower resolution.

Icons button Enable to display the tracker and reference boxes.

The tracker and reference boxes appear in the image window when a tracker is active. You can hide them by disabling the Icons button. This is useful when working with many trackers.

Stereo

Use Stereo to process an anaglyph, interlaced or dual image clip containing one video track from either a left and right eye mono clip, or an existing stereo track directly on the desktop.



To access the Stereo menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- The Modular Keyer, then select a node from the Node bin.

This node accepts a left eye mono clip and a right eye mono clip, and outputs an anaglyph, interlaced or dual image stereo clip. You can add a Stereo node to any part of the pipeline except to the Front and Matte pipes of MBlend nodes and the Matte pipes CBLend nodes.

Stereo Menu Settings

General Settings



Stereo Mode box Select a stereo mode to display the appropriate rendering method options. Different method settings appear based on the type selected.

Select:	To:
Anaglyph	Render a red/cyan clip.
Interlace	Render an interlaced RGB clip.
Dual Image	Render a clip containing adjacent images.



Method box Depending on the selected Stereo Mode, you have different options for modifying the effect.

Anaglyph Settings

Anaglyph Method box Select a method for your anaglyph result. For example, select Dubois to reduce the ghosting between the left and right eyes.

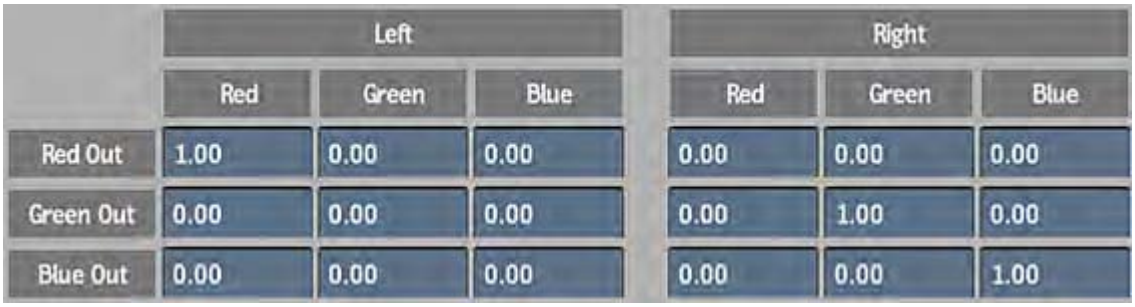
Select:	To:
Custom	Customize the RGB left and right gain factor values. With this option, you can create anaglyph results based on the 3D lenses that will be used. **NOTE** This option is available in Batch or Batch FXand the Modular Keyer.
Dubois	Reduce ghosting between the left and right eyes.
RGB	Create an anaglyph result based on the RGB values.
Mono	Remove the RGB values before creating an anaglyph result. With this option, you will see just the stereo effect.

Interlaced Settings

Interlace Method box Select whether your interlace result outputs the left eye input as field 1 or field 2.

Dual Image Settings

Output the left eye and right eye so that they are adjacent to each other in the same clip, either in a left and right or top and bottom orientation.



	Left			Right		
	Red	Green	Blue	Red	Green	Blue
Red Out	1.00	0.00	0.00	0.00	0.00	0.00
Green Out	0.00	0.00	0.00	0.00	1.00	0.00
Blue Out	0.00	0.00	0.00	0.00	0.00	1.00

Left and Right Custom Gain Factor Fields Displays the gain factor in the colour channel for the left eye and right eye. Enabled when Custom is selected in the Method box.

NOTE This option is available in Batch or Batch FXand the Modular Keyer.

Dual Image Method box Select whether your dual image result outputs the left eye and right eye so that they are adjacent to each other in the same clip, either in a left and right or top and bottom orientation.

Red in Red Output field Displays the red gain factor in the red channel for the Left eye. Editable.

Green in Red Output field Displays the green gain factor in the red channel for the Left eye. Editable.

Blue in Red Output field Displays the blue gain factor in the red channel for the Left eye. Editable.

Red in Green Output field Displays the red gain factor in the green channel for the Left eye. Editable.

Green in Green Output field Displays the green gain factor in the green channel for the Left eye. Editable.

Blue in Green Output field Displays the blue gain factor in the green channel for the Left eye. Editable.

Red in Blue Output field Displays the red gain factor in the blue channel for the Left eye. Editable.

Green in Blue Output field Displays the green gain factor in the blue channel for the Left eye. Editable.

Blue in Blue Output field Displays the blue gain factor in the blue channel for the Left eye. Editable.

Red in Red Output field Displays the red gain factor in the red channel for the Right eye. Editable.

Green in Red Output field Displays the green gain factor in the red channel for the Right eye. Editable.

Blue in Red Output field Displays the blue gain factor in the red channel for the Right eye. Editable.

Red in Green Output field Displays the red gain factor in the green channel for the Right eye. Editable.

Green in Green Output field Displays the green gain factor in the green channel for the Right eye. Editable.

Blue in Green Output field Displays the blue gain factor in the green channel for the Right eye. Editable.

Red in Blue Output field Displays the red gain factor in the blue channel for the Right eye. Editable.

Green in Blue Output field Displays the green gain factor in the blue channel for the Right eye. Editable.

Blue in Blue Output field Displays the blue gain factor in the blue channel for the Right eye. Editable.

Stylize

Use Stylize on an image sequence to create a wide range of visual styles, including painting, printing and sketching. Stylize allows you to build your look by stacking layers with different effects.



Stylize includes a collection of 10 core effects that are stacked and blended together as layers. The core effects fall into four major categories: canvas texture, patterns, colour fillings, and outlines. Each of the core effects has its own settings, which display when an effect is selected:

- Canvas
- Dots
- Hatch Pattern
- Palette Reduction
- Colour Smudge
- Drawing
- Selective Drawing
- Sketched Outlines
- Scribbled Outlines
- Sharp Outlines

Since the possible combinations of core effects are endless, over 80 presets are available to help you get started with Stylize. These presets can also help you learn how Stylize layer stacking works. The presets are divided into a number of categories and are easily added to your scene.

To access the Stylize menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a front, back and matte clip, and outputs a result and an outmatte result.

Stylize Menu Settings

Using Stylize Presets

Use the Stylize presets to create a specific look or to create a starting point to build your effect. Presets are available in 4 categories: Comic, Paint, Print, and Sketch.

Presets button Opens the Presets browser where you can select a preset.

TIP In the browser, switch to Proxies view (press **P**) to see a visual representation of the preset.

Presets dropdown list Select a preset from the dropdown list. Menu settings are changed to reflect the chosen preset.

Setup and General Settings

Clamp Input box Select a clamping option for colour and luminance values on input in the 16-bit floating point rendering pipeline.

Clamp Output box Select a clamping option for colour and luminance values on output in the 16-bit floating point rendering pipeline.

Regen button Enable to dynamically refresh the image as settings are changed.

Layer Table Settings

Use the Layer Table to organize the layers that make up your Stylize effect. The top layer in the Layer Table has the highest priority in the overall effect. Selecting a preset populates the Layer Table with the necessary layers and settings.



Visibility column Displays the visibility of each layer. Click the eye icon to change a layer from visible to invisible or vice-versa.

Name column Displays the name of the layer. Click the Rename button to change the name of a selected layer.

Effect column Displays the effect applied per layer. Use the scroll arrows to change the effect. Different settings appear based on the effect chosen.

Blend column Displays the blend operation per layer. Use the scroll arrows to change the blend value.

Matte column Displays whether a matte is on, off, or inverted per layer. Use the scroll arrows to change the value.

Transparency column Displays the transparency level per layer. Scroll to change the value, or click to display the calculator.

Priority Up button Click to move the selected layer up in priority in the layer list.

Priority Down button Click to move the selected layer down in priority in the layer list.

Solo button Enable to hide all other layers except the selected layer. You can also turn specific layers on or off using the eye icon on the left of each layer.

Seed field Displays the random seed variation value to use in the stylize effect. Editable.

Add button Click to add a new layer to the layer list.

Copy button Click to copy the existing layer to a new layer.

Delete button Click to delete the selected layer from the layer list.

Rename button Click to open the online keyboard to rename the selected layer in the layer list.

Reset button Click to reset all of the attributes associated with the selected layer and returns them to default values.

Background colour pot Displays the background colour used in a blending operation. Click to change the colour.

Canvas Effect Settings

These settings are available when a layer is selected in the Layer Table with Canvas as the effect.



Pattern Type box Select a pattern type to apply to the image.

Transparency field Displays the percentage of transparency applied to the Canvas effect. Editable.

Position X field Displays the position of the pattern along the horizontal axis. Editable.

Position Y field Displays the position of the pattern along the vertical axis. Editable.

Scale X field Displays the change in size of the pattern along the horizontal axis. Editable.

Scale Y field Displays the change in size of the pattern along the vertical axis. Editable.

Proportional button Enable to change the scale fields proportionally.

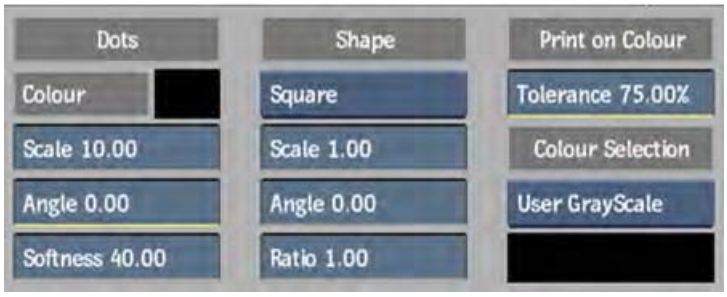
Rotation field Displays the angle of rotation of the pattern along the Z-axis, from its centre point. Editable.

Highlights colour pot Displays the colour applied to the light areas of the Canvas effect. Click to change the colour.

Shadows colour pot Displays the colour applied to the dark areas of the Canvas effect. Click to change the colour.

Dots Effect Settings

These settings are available when a layer is selected in the Layer Table with Dots as the effect.



Effect colour pot Displays the colour applied to the effect. Click to change the colour.

Scale field Displays the change in size of the effect. Editable.

Angle field Displays the change in the angle applied to the effect. Editable.

Softness field Displays the change in softness applied to the effect. Editable.

Shape box Select the shape of the Dots effect.

Scale field Displays the size of the Dots shapes. Editable.

Angle field Displays the change in angle of the Dots shapes. Editable.

Ratio field Displays the change in ratio of the Dots shapes. Editable.

Tolerance field Displays the tolerance level applied to the colour selection of the effect. Editable.

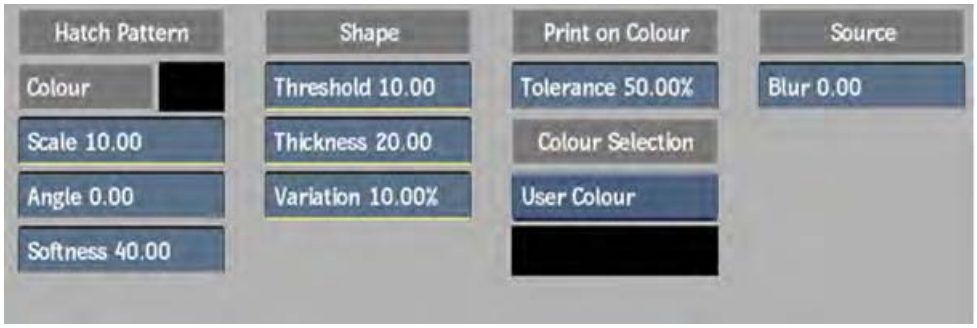
Colour Selection option box Select an option to apply a colour to the image.

Colour Selection colour pot Displays the colour applied to the image. Click to change the colour. Only active if User Colour or User GrayScale is selected.

Blur field Displays the amount of blur applied to the input image before other effects are applied. Editable.

Hatch Pattern Effect Settings

These settings are available when a layer is selected in the Layer Table with Hatch Pattern as the effect.



Effect colour pot Displays the colour applied to the effect. Click to change the colour.

Scale field Displays the change in size of the effect. Editable.

Angle field Displays the change in the angle applied to the effect. Editable.

Softness field Displays the change in softness applied to the effect. Editable.

Threshold field Displays the amount of colour constraint applied to the shape. Editable.

Thickness field Displays the amount of thickness applied to the shape. Editable.

Variation field Displays the percentage of variance applied to the shape. Editable.

Tolerance field Displays the tolerance level applied to the colour selection of the effect. Editable.

Colour Selection option box Select an option to apply a colour to the image.

Colour Selection colour pot Displays the colour applied to the image. Click to change the colour. Only active if User Colour or User GrayScale is selected.

Blur field Displays the amount of blur applied to the input image before other effects are applied. Editable.

Palette Reduction Effect Settings

These settings are available when a layer is selected in the Layer Table with Palette Reduction as the effect.



Colours field Displays the number of colours applied to the Palette Reduction effect. Editable.

Palette box Select between Single Palette to display image stability by using a single colour palette as a reference, and Dynamic Palette to display image transition by using a new colour palette at each frame.

Frame field Displays the frame selected when Single Palette is enabled in the Palette box. Editable.

Softness field Displays the amount of softness applied to the Palette Reduction effect. Editable.

Pattern Type box Select a pattern type to apply to the image.

Amount field Displays the amount of the selected warp pattern that is applied to the image. Editable.

Position X field Displays the position of the pattern along the horizontal axis. Editable.

Position Y field Displays the position of the pattern along the vertical axis. Editable.

Scale X field Displays the change in size of the pattern along the horizontal axis. Editable.

Scale Y field Displays the change in size of the pattern along the vertical axis. Editable.

Proportional button Enable to change the scale fields proportionally.

Rotation field Displays the angle of rotation of the pattern along the Z-axis, from its centre point. Editable.

Blur field Displays the amount of blur applied to the input image before other effects are applied. Editable.

Colour Smudge Effect Settings

These settings are available when a layer is selected in the Layer Table with Colour Smudge as the effect.



Amount field Displays the amount of Colour Smudge effect that is applied to the image. Editable.

Distortion field Displays the amount of distortion applied to the Colour Smudge effect. Editable.

Emboss colour pot Displays the colour applied to the embossing. Click to change the colour.

Amount field Displays the amount of embossing applied to the image. Editable.

Softness field Displays the amount of softness applied to the embossing. Editable.

Angle field Displays the angle applied to the embossing. Editable.

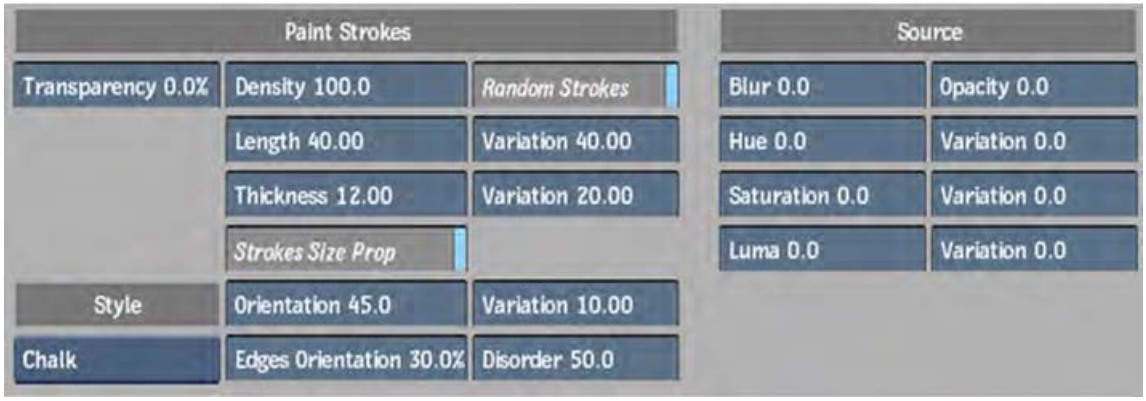
Anti-Aliasing Active button Enable to use anti-aliasing on the edges of the image.

Anti-Aliasing Samples box Select an option to determine the size of the samples. Available when anti-aliasing is enabled.

Anti-Aliasing Softness field Displays the amount of softness applied to the anti-aliasing. Available when anti-aliasing is enabled. Editable.

Drawing Effect Settings

These settings are available when a layer is selected in the Layer Table with Drawing as the effect.



Transparency field Displays the percentage of transparency applied to the paint strokes. Editable.

Style Type box Select the style type of the paint strokes.

Density field Displays the amount of density applied to the paint strokes. Editable.

Length field Displays the length applied to the paint strokes. Editable.

Length Variation field Displays the amount of variance applied to the length of the paint strokes. Editable.

Thickness field Displays the amount of thickness applied to the paint strokes. Editable.

Thickness Variation field Displays the amount of variance applied to the thickness of the paint strokes. Editable.

Paint Each Frame button Enable to apply a random variation in the length and thickness values.

Preserve Stroke Ratio button Enable to allow proportional values to be applied to the length and thickness of paint strokes.

Orientation field Displays the orientation applied to the paint strokes. Editable.

Orientation Variation field Displays the amount of variance applied to the orientation of the paint strokes. Editable.

Edges Orientation field Displays the percentage orientation applied to the edges of the paint strokes. Editable.

Disorder field Displays the amount of random distribution applied to the paint strokes. Editable.

Blur field Displays the amount of blur applied to the input image before other effects are applied. Editable.

Opacity field Displays the amount of opacity applied to the input image before other effects are applied. Editable.

Hue field Displays the amount of hue applied to the input image before other effects are applied. Editable.

Hue Variation field Displays the amount of variance applied to the hue. Editable.

Saturation field Displays the amount of saturation applied to the input image before other effects are applied. Editable.

Saturation Variation field Displays the amount of variance applied to the saturation. Editable.

Luminance field Displays the amount of luminance applied to the input image before other effects are applied. Editable.

Luminance Variation field Displays the amount of variance applied to the luminance. Editable.

Selective Drawing Effect Settings

These settings are available when a layer is selected in the Layer Table with Selective Drawing as the effect.



NOTE Most of the settings in the Paint Strokes section are the same as for the Drawing effect. Selective Drawing also has two extra settings.

Paint Strokes colour pot Displays the colour applied to the paint strokes. Click to change the colour.

Front Mix field Displays the amount of colour from the source image that is visible. Editable

Inclusion colour pot Displays the colour on which the paint strokes are created. Click to change the colour.

Inclusion Tolerance field Displays the percentage of tolerance applied to the selected colour. The higher the tolerance, the more paint strokes are created. Editable.

Exclusion colour pot Displays the colour on which the paint strokes are eliminated. Click to change the colour.

Exclusion Tolerance field Displays the percentage of tolerance applied to the selected colour. The higher the tolerance, the more paint strokes are eliminated. Editable.

Global Colour box Switch between the use of colour selections or grayscale versions of these colour selections.

Global Length field Displays the percentage of length by which semi-transparent strokes are modified. Editable.

Global Softness field Displays the percentage by which the range of transparency of semi-transparent strokes is modified. Editable.

Global Thickness field Displays the percentage by which the range of thickness of semi-transparent strokes is modified. Editable.

NOTE The settings in the Front Controls section are the same as for the Drawing effect.

Sketched Outlines Effect Settings



These settings are available when a layer is selected in the Layer Table with Sketched Outlines as the effect.

Minimum Threshold field Displays the minimum amount of colour restraint applied to the outline edges. Editable.

Maximum Threshold field Displays the maximum amount of colour restraint applied to the outline edges. Editable.

Softness Width field Displays the amount of softness applied to the width of the edges. Editable.

Softness Gain field Displays the amount of softness applied to the gain of the edges. Editable.

Transparency field Displays the percentage of transparency applied to the outlines. Editable.

Outlines colour pot Displays the colour applied to the outlines. Click to change the colour.

Front Mix field Displays the percentage of colour from the source image that is visible. Editable.

Style Type box Select the style type of the outlines.

Density field Displays the amount of density applied to the outlines. Editable.

Paint Each Frame button Enable to apply a random variation in length and thickness values.

Length field Displays the length applied to the outlines. Editable.

Length Variation field Displays the variance applied to the length of the outlines.

Thickness field Displays the amount of thickness applied to the outlines. Editable.

Thickness Variation field Displays the variance applied to the thickness of the outlines.

Preserve Stroke Ratio button Enable to allow proportional values to be applied to the length and thickness for the outlines.

Orientation field Displays the orientation applied to the outlines. Editable

Orientation Variation field Displays the variance applied to the orientation of the outlines.

Follow Edges field Displays the percentage by which the outlines follow their original position. Editable.

NOTE The settings in the Front Controls section are the same as for the Drawing effect.

Scribbled Outlines Effect Settings

The Edge Detect, Softness, and Front Controls settings are the same as for the Sketched Outline effect. Some of the Outlines settings are similar also. The following Outlines settings are specific to the Scribbled Outlines effect.



Smallest Details field Displays the amount by which the smallest outlines will be filtered out. Editable.

Longest Outlines field Displays the amount by which the length of the longest outlines will be increased. Editable.

Straight Lines field Displays the amount of the straightest outlines that are visible. Editable.

Pressure field Displays the variation in the pressure applied to the outlines. Varies with the style of the outline. Editable.

Sharp Outlines Effect Settings

The Edge Detect, Softness, Outlines, and Front Controls settings are the same as for the Sketched Outline effect. The Pattern Transform settings are the same as for the Canvas effect. The following Warp Pattern settings are specific to the Sharp Outlines effect.

Pattern Type box Select a pattern type to apply to the image.

Amount field Displays the amount of warp applied to the image. Editable.

Substance Noise and Splatter

The Substance nodes are procedural noise pattern generators, allowing you to create highly-customizable, repeatable patterns such as cells or concentric shapes to use in your Batch or Batch FX process tree.

Substance Noise node: Use this node to generate a noise pattern based on a chosen preset to use as an input for other nodes.

Substance Splatter node: This node accepts front and matte inputs as the seed that you feed through a chosen preset pattern.

To access the Substance nodes:

1. Drag the Substance Noise node or Substance Splatter node to the Batch or Batch FX schematic.
   The file browser appears, pointing to the default location of the presets:
   - *usr/discreet/<product home>/substance_presets/NOISE* or
   - *usr/discreet/<product home>/substance_presets/SPLATTER*

   TIP Switch to Proxies view to see a visual representation of the presets.

2. Select a preset.

   NOTE The Splatter presets include normal and additive blend versions.

   The preset is automatically loaded into Batch or Batch FX.

3 If you are working with a Substance Splatter node, attach front and matte input clips.

4 Double-click the Substance node in the schematic to access the menu.

Some of the settings in the menu are dependant on the preset you loaded.

5 Change the menu settings, as needed. Enable Regen to update the image as fields are updated.

6 Optional: Enable Force Tile to ensure that your resulting image tiling is repeatable. In this case, the Crop Output and Zoom options are greyed out.



NOTE If your rendering network is set up with GPU-enabled Burn nodes, you can burn Substance clips. See your *Autodesk Burn Installation and User Guide* for installation and configuration information.

To change a Substance preset:

1 Select the Substance Noise or Substance Splatter node in the schematic.

2 Click Change Preset.



The file browser appears, and you can select a different preset to load.

NOTE All menu parameters are reset when a new preset is loaded.

Setting Substance Parameters and Behaviours

Once you have loaded your preset, you can use the Parameters and Behaviours settings in the Substance menu to change and animate the pattern.

Some of the settings in the Parameters tab vary depending on the chosen preset, while other settings, such as Zoom, Random Seed, and Disorder are present in every Substance preset. You can get a quick description of each setting by viewing its tooltip.

NOTE If Force Tile is enabled, the Zoom field is greyed out.



Example of Noise parameters



Example of Splatter parameters

Use the settings in the Behaviours tab to apply preset motion to your pattern, and to help you quickly animate your patterns. For example, you can choose a Flow behaviour and animate your pattern to resemble a flag waving.



The behaviours are divided into a number of categories, each with their own settings.

Organic Pulse Slow pulsing evolution of width and height of the pattern.

Wind Directional force that spins the pattern with intensity.

Flow Directional force that displaces the pattern in different direction.

Oscillation Evolution of size and luminance of the pattern.

Wave Distort Directional warping effect that deforms the pattern with different wavelength warps.

You can adjust the playback rate and timing of your chosen behaviour animation by using the Overall Speed and Time Offset settings (available for all behaviours).

You can also adjust the Overall Speed in the Channel Editor.



Substance Menu Settings

Preset Selection Settings

Preset Name field This locked field displays the name of the current preset.

Change Preset button Click to open the file browser to select a different preset.

Preset Memory Settings

The Memory section of the Noise or Splatter menu allows you to store up to five variations of the current preset. This is useful if you want to compare a certain preset type with different settings and animations.

Use the following preset memory buttons to clear, store, or recall menu parameters. A blue LED indicates that a preset memory is stored in the slot.



(a) Preset stored in this preset memory slot (b) No presets stored in these slots

Clear button Clears the selected preset memory.

Store button Stores the menu parameters in the selected preset memory.

Recall button Recalls the last stored parameters for the selected preset memory.

Preset Memory buttons Use to clear, store, or recall menu parameters. A blue LED indicates that a preset memory is stored in the slot.

Proxy image window Displays a proxy of the stored preset memory.

Output Settings

Use the settings in the Output section to set the size and resolution of clips that are output from the Substance node.



Texture Resolution box Select the resolution of the pattern.

Frame Depth box Select the render/output frame depth of clips.

Crop Output button Enable to crop the output of the pattern by the amounts displayed in the Width and Height fields.

NOTE The crop options are greyed out if the Force Tile button is enabled.

Ratio field Displays the aspect ratio of the pattern clip. The ratio is calculated by W:H to output a square pixel ratio. Non-editable.

Crop Width field Displays the width of the cropped output. Editable.

Crop Height field Displays the height of the cropped output. Editable.

Splatter Pattern Resize Settings

Use the Pattern Resize settings to change the input size and filtering type of a Splatter pattern.

Pattern Resize box Select the input size of the Splatter pattern.

Filter box Select a resize filtering option.

Other Settings

Regen button Enable to automatically update the image as fields are updated.

Force Tile button Enable to ensure that your resulting image tiling is repeatable. In this case, the Crop Output and Zoom options are greyed out.

Behaviours Settings

Behaviour Type box Select the type of preset motion to apply to the pattern.

Overall Speed field Displays the rate at which the animation plays. Editable.

Time Offset field Displays the start point of the animation. With a value of 0, the animation starts at frame 1. With a value of 100, the animation begins as if it has been generating for 99 frames. You cannot animate this field. Editable.

Text

Use Text to make text spin, dance, and change colour over time.



To access the Text menu, use:

- Batch, then select a node from the Node bin.
- Timeline, then use Batch FX (See Creating Batch FX (page 346)).
- Tools, then select from the menu.

This node accepts a background clip. If a background clip is not connected to the Text node, you can select a resolution for the output of the Text node. Additionally, this node outputs a result and an outmatte.

NOTE The Prerender Text preference does not apply when you access Text from Batch or Batch FX.

Text is compromised of layers, paragraphs, and characters. You can create effects such as a text roll of credits, text crawls, bumpers, and a text that moves on a motion path. You can also use logo images in a text roll so that the logo of a sponsor appears in the credit list.

Text Menu Settings

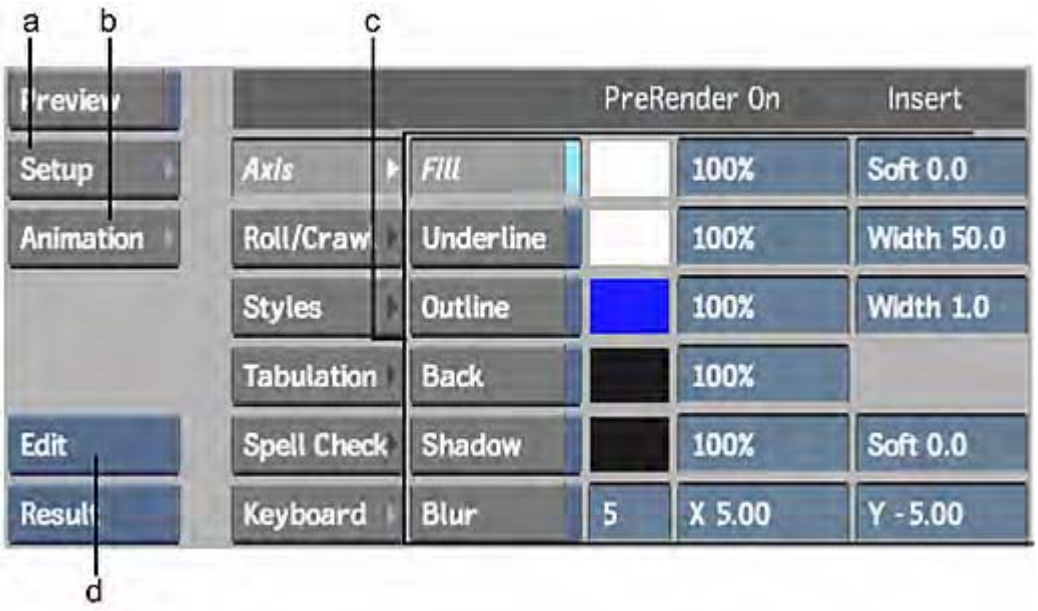
General Settings

Preview button Enable to preview the text effect.

Preview Options box This box is available when you access the Text Menu settings from the Timeline.Select to enable Preview FX or Context, to preview the effect.

Setup button Displays the Text Setup menu.

See Setup and Processing Options (page 1126).



(a) Setup button (b) Animation button (c) Attribute controls (d) Text Mode box

Attribute controls Changes the appearance of text by setting properties such as fill and shadow. See Creating Text Effects (page 1135).

Animation button Creates a text animation by setting different text properties at specific keyframes in a clip.

See Animating Text (page 1144).

Text Mode button Select a text mode to select text, edit text in layers, modify text attributes, or enter text strings.

Applies various text settings to layers, paragraphs, and characters. Use text modes to manipulate layers, add text to a clip, or edit existing text in a clip. See Modifying Layer and Character Properties (page 1135).

Result button Select an option to set the view to either the composited text, the matte of the text, or the original back clip.

Reset box Select an option to reset the selected text layer, the layer axis, the roll/crawl, or the style templates.

Reset All Click to reset the text tool to the default values set in Preferences.

Edit button Opens the Edit menu.

Resolution Presets box Select a resolution for the new clip. Select Custom to specify a non-standard resolution.

Frame Depth box Select the render/output frame depth of clips.

Scan Mode box Select the scan mode of clips.

Width field Displays the custom width resolution of the clip. Editable.

Height field Displays the custom width resolution of the clip. Editable.

Aspect Ratio Presets box Select a standard frame aspect ratio. Select the Set to w:h option to set the clip to use square pixels. Select Custom to define a custom frame aspect ratio in the Aspect Ratio field.

Aspect Ratio field Displays the custom render/output aspect ratio. Editable

Insert Settings



Axis button Opens the Axis menu to set properties for layers and characters.

Roll/Crawl button Open the Roll/Crawl menu to set text motion properties for vertical or horizontal text scrolling.

See Creating Text Rolls and Text Crawls (page 1141).

Styles button Opens the Styles menu to create preset text style formats.

See Defining Styles (page 1150)

Tabulation button Open the Tabulation menu to set text tabs.

See Tabulating Text (page 1140)

Spell Check button Open the Spell Check menu to check text for spelling errors.

See Spell Checking (page 1141)

Keyboard button Opens the on-screen keyboard to enter text.

See Entering Text (page 1131)

Fill button Enable to use fill on the text selection.

Sets fill colour, transparency, and softness. Set fill softness using the Soft field. To increase the softness, enter a value close to the maximum of 100. To decrease the softness, enter a value close to the minimum of -100. Enter 0 for no softness.

Fill colour pot Displays the colour of the fill. Editable.

Fill Transparency field Displays the transparency level of the fill. Editable.

Fill Softness field Displays the softness level of the fill. Editable.

Underline button Enable to underline the text selection.

Underlines the characters using the associated colour, transparency, and width. Click the colour pot to choose a colour from the colour picker. To set underline transparency, enter a percentage in the Transparency field. To make the underline more opaque, enter a value close to the maximum of 100. To make the underline more transparent, enter a value close to the minimum of 0. To set the outline width, enter a value in the Width field.

Underline colour pot Displays the colour of the underline. Editable.

Underline Transparency field Displays the transparency level of the underline. Editable.

Underline Width field Displays the width of the underline. Editable.

Outline button Enable to outline the text selection.

Outlines the characters with a solid colour using the associated colour, transparency, and width. Click the colour pot to choose a colour from the colour picker. To set outline transparency, enter a percentage in the Transparency field. To make the outline more opaque, enter a value close to the maximum of 100. To make the outline more transparent, enter a value close to the minimum of 0.

To set the outline width, enter a value in the Width field. Use anti-aliasing rendering options with outlined text. These options are found in the Text Setup menu. See Setup and Processing Options (page 1126).

Outline colour pot Displays the colour of the outline. Editable.

Outline Transparency field Displays the transparency level of the outline. Editable.

Outline Width field Displays the width of the outline. Editable.

Back button Enable to apply a solid colour background using the associated text layer colour and transparency to the text selection.

Click the colour pot to choose a colour from the colour picker. To set back transparency, enter a percentage in the Transparency field. To make the back more opaque, enter a value close to the maximum of 100. To make the back more transparent, enter a value close to the minimum of 0.

Back colour pot Displays the colour of the background. Editable.

Back Transparency field Displays the transparency level of the background. Editable.

Shadow button Enable to apply a drop shadow to the text selection.

Applies a shadow to text characters using the associated colour, softness, transparency, and Pos X and Pos Y fields. Click the colour pot to choose a colour from the colour picker. To set shadow transparency, enter a percentage in the Transparency field. To make the shadow more opaque, enter a value close to the maximum of 100. To make the shadow more transparent, enter a value close to the minimum of 0. To set shadow softness, use the Soft field. To increase the softness, enter a value close to the maximum of 100. Enter 0 for no softness.

To set the shadow position on the X-axis, enter a value in the X field. A positive value places the shadow to the right and behind the text characters. A negative value places the shadow to the left and in front of the text characters. To set the shadow position on the Y-axis, enter a value in the Y field. A positive value moves the shadow up. A negative value moves the shadow down.

Shadow colour pot Displays the colour of the shadow. Editable.

Shadow Transparency field Displays the transparency level of the shadow. Editable.

Shadow Softness field Displays the softness level of the shadow. Editable.

Shadow X field Displays the shadow position along the X-axis.

Shadow Y field Displays the shadow position along the Y-axis.

Blur Shadow button Enable to apply a blur effect to a drop shadow created with the Shadow button.

The type of blur applied depends on whether Box Blur or Gaussian Blur is selected in the Text Setup menu. Choose a higher value for a greater shadow blur effect. Using Blur Shadow with animated text decreases processing performance.

Blur Shadow field Displays the level of blur applied to the drop shadow. Editable.

Font Settings



Font field Displays the current font. Click to open the font browser to select a different font.

Font Reset button Resets to the default font.

Font Size field Displays the character size for the selected font. Editable.

Font Italic field Displays the angle of italicized text. Positive values make the characters slope to the right. Negative values make them slope to the left.

Font Kerning field Displays the amount of spacing between characters. Positive values increase the spacing. Negative values decrease the spacing.

Paragraph Settings



Justification box Select an alignment option for the paragraph.

Select Left, Right, Centre, or Justified.

Width field Displays the width of the layer. Editable.

Left Margin field Displays the amount of space in the left margin. Editable.

Leading field Displays the space between lines in the selected paragraph. Editable.

Indent field Displays the left indentation value for the first line in the paragraph. Editable.

Separation field Displays the space between the selected paragraph and the one above it. Editable.

Character Channels



Copy Attributes button Copies attribute settings from the selected text character.

Copy Transformations button Copies transformation settings from the selected text character.

Copy All button Copies both attribute and transformation settings from the selected text character.

Paste button Pastes any copied text character channel information.

Text on Path



On Path button Enable to place the selected text on a motion path.

Closed button Enable to close the path by connecting the first and last vertices. Text continues around the shape.

Offset field Displays the position of the text on the path. Animate the Offset value to make the text follow the path for the duration of the clip.

Invert button Reverses the order of the vertices on the path so that the text follows the opposite side of the path.

Text On Path Mode box Select an option to manipulate the selected path spline.

Clear button Resets the text path at the current frame. To reset the path for the entire duration of the clip, click Reset Layer.

Axis Settings



NOTE The Axis settings are available when the Axis button is enabled in the Insert settings.

Axis menu buttons Select whether to display the Layer Axis menu or the Characters Axis menu.

Show Axis button Enable to display the axis in the image window. This button is only available when the Character Axis menu button is selected.

Position X field Moves the layer or selected characters along the X-axis.By default, the value is 0 (for characters). Increase the value to move right and decrease it to move left. Editable .

Y Position field Moves the layer or selected characters along the Y-axis.By default, the value is 0 (for characters). Increase the value to move up and decrease it to move down. Editable .

Lock Position button Enable to lock the position of the text layer. You can lock text in either the Top/Left or Centre position using the box beside this button.

Centre Axis button Moves the axis to the center point of the character. This button is only available when the Character Axis menu button is selected.

X Position Character Axis field Sets the X-axis for the selected characters. Set each letter in a word spinning on its own axis in a different way. This button is only available when the Character Axis menu button is selected.

Using the Axis Pos X field, you set each letter in a word spinning on its own axis in a different way.

Y Position Character Axis field Sets the Y-axis for the selected characters. Set each letter in a word spinning on its own axis in a different way. This button is only available when the Character Axis menu button is selected.

Using the Axis Pos Y field, you set each letter in a word spinning on its own axis in a different way.

X Scale field Scales the layer or selected characters along the X-axis. This value is a percentage, so entering 50 means 50 percent. Use a larger value to increase the scale along the X-axis. Use a negative value to create a mirror image on the X-axis. Enter 0 to make a layer or selected characters disappear. The default is 100.Editable.

Y Scale field Scales the layer or selected characters along the Y-axis. This value is a percentage, so entering 50 means 50 percent. Use a larger value to increase the scale along the Y-axis. Use a negative value to create a mirror image on the Y-axis. Enter 0 to make a layer or selected characters disappear. The default is 100.Editable.

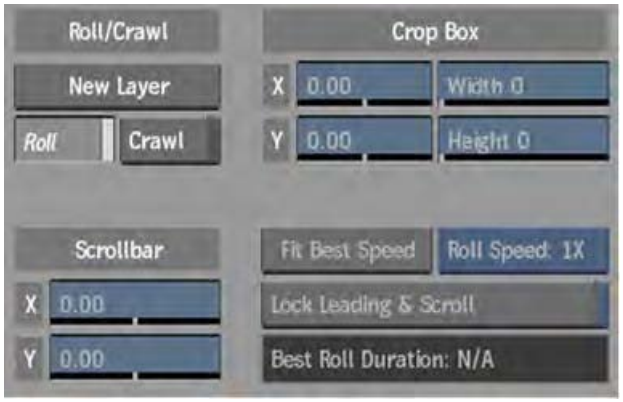
Proportional Scale button Proportionally changes the Scale X and Y values.

X Shear field Shears or slants the layer or selected characters along the X-axis. Use a positive value to slant right and a negative value to slant left. The maximum and minimum values are 60 and -60, respectively. The default value is 0.

Y Shear field Shears or slants the layer or selected characters along the Y-axis. Use a positive value to slant right and a negative value to slant left. The maximum and minimum values are 60 and -60, respectively. The default value is 0.

Rotation field Rotates a layer around its axis or selected characters around their axis.Use a negative value to rotate clockwise and a positive value to rotate counter-clockwise. The default is 0.Editable.

Roll/Crawl Settings



NOTE The Roll/Crawl settings are available when the Roll/Crawl button is enabled in the Insert settings.

Roll/Crawl Settings

New Layer button Creates a layer to contain a text roll or text crawl.

Roll button Enable to create text that scrolls vertically over an image.

Crawl button Enable to create text that scrolls horizontally across an image.

Scrollbar

Scrollbar X field Displays the X position of the roll or crawl layer inside the crop box. The value changes when you move the scrollbar at the right of the text layer for a roll or at the bottom of the layer for a crawl. Editable.

Scrollbar Y field Displays the Y position of the roll or crawl layer inside the crop box. The value changes when you move the scrollbar at the right of the text layer for a roll or at the bottom of the layer for a crawl. Editable.

Fit Best Speed button Creates a broadcast quality text roll based on the number of frames in the clip and the lines of text in the text roll. Enable this button to make corrections to a text roll without altering the speed or duration of the clip.

For NTSC and PAL, broadcast quality is calibrated at four pixels per frame. This rate ensures no flicker in the text roll.

Roll Speed button Select the speed of the text roll, in pixels per second.

Changes the speed of the text roll. For 1X, the speed rate is 120 p/s (pixels/second) in NTSC, and 100 p/s in PAL. For 2X, it is 240 p/s in NTSC, and 200 p/s in PAL, and so on.

Lock Leading & Scroll button Enable to lock the leading of the paragraph text so that you can insert another paragraph into the text roll. If the duration of the clip lengthens after you make modifications, click Fit Best Speed to adjust the length.

Best Roll Duration field Displays the suggested duration for the text roll. The duration is calculated using the start/end position of the roll and the currently selected speed. Non-editable.

Crop Box

Crop Box X field Displays the X position of the crop box inside the image window. The crop box determines the X coordinates of the text roll or text crawl on the clip. Editable.

Crop Box Y field Displays the Y position of the crop box inside the image window. The crop box determines the Y coordinates of the text roll or text crawl on the clip. Editable.

Width field Displays the width at which to crop the text within the text roll or crawl layer. Editable.

Height field Displays the height at which to crop the text within the text roll or crawl layer. Editable.

Styles Settings



NOTE The Styles settings are available when the Styles button is enabled in the Insert settings.

Apply Style 1 button Saves or applies Style 1.

Apply Style 2 button Saves or applies Style 2.

Apply Style 3 button Saves or applies Style 3.

Apply Style 4 button Saves or applies Style 4.

Apply Style 5 button Saves or applies Style 5.

Apply Style 6 button Saves or applies Style 6.

Apply Style 7 button Saves or applies Style 7.

Apply Style 8 button Saves or applies Style 8.

Apply Style 9 button Saves or applies Style 9.

Load Style button Opens the file browser where you can select a saved style to load.

Save Style button Opens the file browser where you can save a style.

Style Mode box Select whether to work with all styles or one style.

Style Save Number box Select the number of the style to save. Available when One Style is selected from the Style Mode box.

Style Name button Sets the name of a defined style.

Style Option box Select an option for creating styles. AutoStyle assigns styles from an existing layer.

Tabulation Settings



NOTE The Tabulation settings are available when the Tabulation button is enabled in the Insert settings.

Add button Adds a tab stop on the text layer ruler. By default, tab stops are set at every 100 pixels.

Delete button Removes the selected tab stop.

Previous button Selects the previous tab stop on the text ruler. Tab stops appear in yellow when selected.

Tab ID field Display the tab number in the current paragraph. When you click Next or Previous, the Tab ID field changes, showing the number of selected tab stops. Editable.

Next button Selects the next tab stop on the text ruler. Tab stops appear in yellow when selected.

Position field Displays the exact tab stop position on the X-axis, in pixels. This value is set in pixels. Editable.

Ruler button Enable to display the tabulation ruler in the text layer.

Justification box Select the text alignment at the tab stop.

Spell Check Settings



NOTE The Spell Check settings are available when the Spell Check button is enabled in the Insert settings.

Spell Check Settings

Language box Select the language for the spell checker.

Check All Layers button Runs the spell checker on the text layers. When the spell checker encounters a misspelled word, it draws a red line through it.

Hide Spelling Errors button Enable to hide the red strike through the line that appears in each misspelled word.

Custom Dictionary button Enable to use your custom dictionary with the spell checker. Disable this button to check all spelling. Disable to check all spelling.

Error Navigation Settings

Previous button Navigates to the previous misspelled word.

Next button Navigates to the next misspelled word.

Add To Custom Dictionary button Adds a word that the spell checker has flagged to the custom dictionary. The next time the spell checker encounters this word, it will be ignored.

Setup and Processing Options

Auto-Softness box Sets the auto-softness and software sampling level.

Auto-Softness mode is set by default and provides a softness equal to the display quality you see in the Text node.

Anti-Aliasing Softness field Displays the softness value of the anti-aliasing sample. Editable.

PreRender button Enable to accelerate rendering on a static layer, such as a text roll (has no effect on text layers with animation).

PreRender has no effect on text layers with animation.

Hardware Anti-aliasing Sample box Select a hardware anti-aliasing sampling level to accelerate edge anti-aliasing with no performance penalty. Available values are dependent on graphics card and project graphic bit depth.

The graphics hardware automatically renders the image at full speed with approximately the equivalent of up to 32 samples of anti-aliasing (depending on your graphics card and project graphic bit depth). Hardware anti-aliasing also gives anti-aliasing during normal interaction instead of only while rendering.

You can combine hardware anti-aliasing level with software anti-aliasing to obtain the desired level of image quality. For example, with hardware anti-aliasing set to 4 samples, and with 4 samples of software anti-aliasing selected, your results should be similar to selecting 16 samples of software anti-aliasing, but with a processing

time much closer to that of 4 samples. You should experiment with different combinations to determine what works best for you.

Motion Update button Enable to update animated text dynamically in the image area when you play the clip.

Controls the playback of animated text. When enabled, animated text is updated dynamically in the image area when you play the clip.

Play Lock button Enable to update the animation settings according to the frame or timebar position as you move through the clip while keeping it locked at the current frame.

Controls the playback of frames and displays the playback of animated clips exclusively. When enabled, the first frame is locked while you play the clip.

Back Clip button Enable to display the back clip in the scene.

If you entered Text with a back clip, you can disable it here.



Crop button Enable to create a custom-sized crop box to define an area in which to render text.

Use this box to define an area in which to render text. When working with a crop box, you view all text, whether it is inside or outside the crop box. You can use the Left, Right, Bottom, and Top fields to set the size of the crop box. You can also drag the vertices at the corners of the crop box in the image window.



Crop Border button Enable to display a coloured border representing the crop area.

You can change the colour by clicking the colour pot.

Crop Border colour pot Displays the colour of the crop border. Editable.

Top Crop field Displays the top boundary value for the crop box. Editable.

Left Crop field Displays the left boundary value for the crop box. Editable.

Bottom Crop field Displays the bottom boundary value for the crop box. Editable.

Right Crop field Displays the right boundary value for the crop box. Editable.

Wireframe colour pot Displays the colour of the text path. Editable.

See Animating Text on a Motion Path (page 1147).



Points colour pot Displays the colour of the vertices on a text path. Editable.

See Animating Text on a Motion Path (page 1147).

Global/Layer Blur box Select whether to apply blurring globally or on a per layer basis.

Global Blur makes all shadows appear together, blurred with a single unique colour, on top of the background of all layers. Layer Blur blurs each layer individually according to its priority—background, blurred shadow, and fill.

When using Global Blur with prerendering, the layers are prerendered once and blurred on the front clip. These layers must not have any animation—they are static. For Layer Blur, there are two possible cases:

- All layers are static, without background. Only fill or shadow can be enabled for all layers—no blur. The layers are prerendered once and blurred on the front clip.
- Auto-Softness On, Progressive Rendering, or All Layers Static. Static layers (up to two) are prerendered once and blurred on the front clip as other layers are rendered.



Shadow Blur box Select whether to use a smooth blur with rounded edges (Gaussian), or a rectangular, rougher edged blur (Box).

Clear Undo Buffer button Resets the Undo entries.

Undo Levels field Displays the number of Undo levels available in the Undo/Redo lists. Editable.

Changes the levels of Undo available in the Undo/Redo lists. The maximum level of undo is 20.



Entering Text

Font Type box Select the type of font to display in the font browser list.

Font Preview window Previews the selected font. Click to enter text.

Exit Load Font button Exits the font browser.

Modifying Layer and Character Properties

Layer Axis box Select an alignment option for the selected text layer.

Select:	To move the axis:
Top/Left	To the upper-left corner of the text layer.
Centre	To the centre point of the text layer.

X Scale field Displays the scale of the text layer along the X axis. Editable.

This value is a percentage, so entering 50 means 50 percent. Use a larger value to increase the scale along the X-axis. Use a negative value to create a mirror image on the X-axis. Enter 0 to make a layer or selected characters disappear. The default is 100.

Y Scale field Displays the scale of the text layer along the Y axis. Editable.

This value is a percentage, so entering 50 means 50 percent. Use a larger value to increase the scale along the Y-axis. Use a negative value to create a mirror image on the Y-axis. Enter 0 to make the layer or selected characters disappear. The default is 100.

Proportional Scale button Enable to proportionally change the X Scale and Y Scale values.

X Shear field Displays the shear or slant value of the text layer along the X axis. Editable.

Use a positive value to slant right and a negative value to slant left. The maximum and minimum values are 60 and -60, respectively. The default is 0.

Y Shear field Displays the shear or slant value of the text layer along the Y axis. Editable. Use a positive value to shear up. A negative value near the minimum value -60 shears down. The maximum and minimum values are 60 and -60, respectively. The default is 0.

Rotation field Displays a rotation value for a layer rotating around its axis. Editable.

Use a negative value to rotate clockwise and a positive value to rotate counter-clockwise. The default is 0.

Warper

Use the Warper to warp a clip or morph from a source clip to a result clip. Warping is free-form distortion of an image; it is a transition effect that matches the morphology of one clip gradually to that of another clip, such as when a human is morphing into an alien.

You can access the Warper Tool through:

- Tools, then select from the menu. (page 1199)

Warper Menu Settings

Miscellaneous Buttons

Exit button Exits the Warper tool.

Load button Loads a setup.

Save button Saves a setup.

Setup Name field Displays the name of the last saved setup.

Revert button Reverts to the last saved setup.

Source option box Select a field rendering mode.

Player button Opens the Player to view the rendered clip.

Step Render field Displays the frame number interval to be rendered (to see intermediate results, for example). Editable.

Preview button Previews the distort effect without rendering.

Setup button Sets playback options, changes display colours, and inputs slip values.

Animation button Opens the channel editor.

View box Select an option to set the view in the image window.

Auto Key button Enable to set a keyframe automatically each time you change a value at any frame.

Reset All button Resets all parameters.

Setup Menu

Decimation field Displays the number of subdivisions for the main patches of the mesh. Editable.

Decimation is the number of subdivisions for the main patches of the mesh (not the small subdivided elements). The default decimation value is 10, which means that each patch is divided into 10 by 10 elements. You should increase this value when:

- You want to use smoother curves along the edges of the patches.
- A patch has been greatly subdivided. When the Decimation value is too low, the subdivided sections will appear to "swim" instead of locking to points and splines.
- A patch is twisted extensively. When the Decimation value is too low, the interior of the patch may appear to fold or break up unexpectedly. Increase the value to smooth the interior of the patch.

Keep the following points in mind when you are adjusting the decimation value:

- Increasing the Decimation value will cause interaction and processing speed to decrease dramatically.
- The Decimation value remains in effect during final processing.
- There is no relationship between the Decimation value and the TexPrec value. The Decimation value defines the precision of the geometry, whereas the TexPrec value affects the precision of the image that is mapped to this geometry.

Samples field Displays the anti-aliasing factor (the number of samples per pixel during final rendering). Editable.

The Samples field specifies the anti-aliasing factor. This number specifies the number of samples per pixel during final processing.

Texture Repeat button Enable to allow pixels to repeat when a source mesh goes outside of the borders of an image.

Texture Quality box Select a texture rendering option.

Provides three texture rendering options.

Select:	To:
High Quality	Use high-quality hardware texture rendering. This option always uses the highest level of precision. High Quality tiles images according to the available texture memory. This option

Select:	To:
	is the default and is the recommended rendering option for film images or any image that is greater than the available texture memory.
Low Quality	Use low-quality hardware texture rendering, specifically during interactive rendering mode or when you want to render an image quickly. This option uses the precision level that you set in the Precision field.
Pixel Rendering	Use polygon rendering.

Click Preview to hide the mesh and preview the resulting frame.

Texture Precision field Displays the speed/quality ratio for interaction, as well as the rendering quality when Low Quality is selected as the rendering option. Editable.

The higher the value, the higher the quality of the display during interaction, and the higher the quality of final processing when rendering under Low Quality.

Motion Update button Enable to use the same values for properties such as position, rotation, and colour in the scene. Disable to manually copy keyframe values from one frame to another.

When disabled, animated objects do not move, but keep the position of their current value. Disable Motion Update when you want to copy keyframe values from one frame to another.

Play Lock button Enable to update the animation settings according to the frame or timebar position as you move through the clip while keeping it locked at the current frame.

Front Slip field Displays the number of frames to slip the front clip. Editable.

Back Slip field Displays the number of frames to slip the back clip. Editable.

Front Matte Slip field Displays the number of frames to slip the front matte clip. Editable.

Back Matte Slip field Displays the number of frames to slip the back matte clip. Editable.

Icons button Enable to display splines, tangents, and vertices in the image.

Spline colour pot Displays the colour of splines in the image. Editable.

Tangent colour pot Displays the colour of tangents in the image. Editable.

Vertex colour pot Displays the colour of vertices in the image. Editable.

Shape Interpolation box Select an interpolation type.

Main Menu

Add Mesh button Enable to add a mesh or patch to the image.

Xcells field Displays the number of patches along the X axis. Editable.

Ycells field Displays the number of patches along the Y axis. Editable.

Proportional button Enable to keep the XCells and YCells fields proportional.

Delete Mesh button Deletes the selected mesh.

Front Clip Setup box Select an option for using the front clip.

Back Clip Setup box Select an option for using the back clip.

Matte Clip Setup box Select an option for using the matte clip.

Back Matte Clip Setup box Select an option for using the back matte clip.

Background Clip Setup box Select an option for using the background clip.

Warp button Displays the warping controls.

Morph button Displays the morphing controls.

Front Source button Enable to create a front source mesh.

Show Front Source button Enable to display the front source mesh in the image.

Front Source colour pot Displays the colour for the front source mesh. Editable.

Front Destination button Enable to create a front destination mesh.

Show Front Destination button Enable to display the front destination mesh in the image.

Front Destination colour pot Displays the colour for the front destination mesh. Editable.

Back Source button Enable to create a back source mesh in a morph.

Show Back Source button Enable to display the back source mesh in the image.

Back Source colour pot Displays the colour for the back source mesh. Editable.

Back Destination button Enable to create a back destination mesh in a morph.

Show Back Destination button Enable to display the back destination mesh in the image.

Back Destination colour pot Displays the colour for the back destination mesh. Editable.

Source Interpolation button Enable to use source interpolation in a morph.

Show Source Interpolation button Enable to display source interpolation in the image.

Front Source Mapping box Select the area that the front source is mapped to.

Front Destination Mapping box Select the area that the front destination is mapped to.

Back Source Mapping box Select the area that the back source is mapped to.

Back Destination Mapping box Select the area that the back destination is mapped to.

X Position field Displays the position of the control points along the X axis. Editable.

Y Position field Displays the position of the control points along the Y axis. Editable.

Z Position field Displays the position of the control points along the Z axis. Editable.

Curves button Enable to activate the interpolation and dissolve curves.

Dissolve field Displays the percentage change in the colour of the pixels at the current frame. Editable.

Interpolate field Displays the percentage change in the position of the pixels at the current frame. Editable.

Move button Enable to move a vertex, an edge, or a tangent handle.

Move Mode box Select an option to change the position, size, and orientation of part or all of the mesh.

Subdivide button Enable to divide patches in a mesh into smaller subpatches and split the edges into smaller segments.

Auto button Enable to reset broken tangent handles.

Select button Enable to select multiple control points to translate, scale, or rotate all or part of the mesh.

Split button Enable to add a vertex along the edge of a patch.

Copy button Copies the selected mesh.

Paste button Pastes the copied mesh.

Freehand button Enable to perform intricate mesh modelling by clicking to create a freehand mesh.

Break button Enable to break tangent handles.

Delete button Enable to delete edges or vertices of the mesh.

Magnet button Enable to use the Magnet tool to define the shape of the warped image.

Magnet Mode box Select a transformation type to use when Magnet is enabled.

Magnet Curve Editor Defines the weighted polarity from the centre to the edge of the magnet.

Magnet Curve Home button Resets the position of the magnet curve after panning.

Magnet Curve Reset button Resets the Magnet Curve Editor.

Magnet Curve Edit Mode box Select an option to modify the magnet curve.

Magnet Load button Loads a custom magnet setup.

Magnet Save button Saves a custom magnet setup.

Stabilizer button Opens the Stabilizer menu to track movement in the clip to help in creating meshes.

Stabilizer Clip box Select the clip to be loaded into the stabilizer.

Tracking Mode box Select whether to track the axis or vertices of the clip.

Tracking Mesh box Select whether to apply the tracking data to the source or destination mesh.

Animating Keyframes

26

You can build animations simply by editing the values of parameters in any effect editor or menu. When you animate a parameter, you create an animation channel that you can use to control the values of the parameter over time.

To use auto key to build animations automatically in the editor for an effect or tool:

1. Enable Auto Key.
2. Go to the frame where you want to add the keyframe.
3. Change the value of the parameter. Enter a new value in the field using the calculator, or click and drag left or right to change the value.

   A keyframe is added to the animation channel for the parameter you edited.

   NOTE When auto key is enabled, you create keyframes whenever you change the value for a parameter. You also change the value of any parameter that you have already set.

To set keyframes manually in the editor for an effect or tool:

1. Disable Auto Key.
2. Go to the frame where you want to add the keyframe.
3. Change the value of the property. Enter a new value in the field using the calculator, or click and drag left or right to change the value.
4. Right-click the field and select Set Keyframe (Current Value).

To delete keyframes:

1. Go to the frame that contains the keyframe you want to delete.
2. Right-click the parameter that contains the keyframe and select Delete Keyframe.

   NOTE A yellow bar appears under values that are keyframed at the current frame.

To reset keyframe values:

1. Right-click the parameter and select one of the following options.

Select:	To:
Reset (Default Value)	Reset the value of the keyframe to the original value.

Select:	To:
Reset Channel (Current Value)	Reset the value of the entire channel to the current value.
Reset Channel (Default Value)	Reset the entire channel to the original value.

Keyframe Indicator Reference

A number of colours are used in numeric fields to indicate that keyframes are set for a parameter.

Indicator	Example
An asterisk in a field indicates that a value has changed from the default, but a keyframe is not set. Once a keyframe is set, the asterisk disappears.	
A blue line under a numeric field indicates that one or more keyframes are present on this channel, but not at the current frame.	
A yellow line under a numeric field indicates that a keyframe is present on this channel at this point in time.	
A dotted yellow line under a numeric field indicates that this channel is linked to another channel using an expression. In this case, the numeric value is also greyed out, as the value can not be modified. The name of the linking channel is also displayed as part of the tooltip.	

Animating Keyframes in the Animation Channel Editor

The Animation Channel Editor offers more control over your animation than is offered when editing the values of parameters in an effect or tool editor. Use the Animation Channel Editor to fine-tune changes across all animation channels available for the effect or tool you are working on.

To display the Channel Editor:

1 Click Animation.



The Animation Channel Editor is displayed in the bottom area of the window.

To display the Animation Channel Editor in a viewport:

1. Click the View box.
2. Select one of the following from the Animation menu:

Select:	To display:
Channels	Animation curves.
Tracks	Animation channels as tracks. This view can help you visualize and align keyframes.
Info	A table displaying keyframe values in text format.

Panning and Zooming in the Animation Channel Editor

To center the Animation Channel Editor on your work, click one of the following:

- Frame: to center the display on the currently selected channel.
- Frame All: to center the display on all the animation channels that have at least one keyframe set on them.

NOTE These buttons are located at the bottom left corner of the Animation Channel Editor.

To pan the Animation Channel Editor:

1. From the Edit Mode box (below the Keyframe tab), select Pan.
2. Click and drag the cursor to pan the view.

To zoom the Animation Channel Editor:

1. From the Edit Mode box (below the Keyframe tab), select Rect Zoom.
2. Click and drag a rectangle on the area you want zoom.

NOTE You can zoom x and y independently. Use the Zoom command from the Edit Mode box. Click and drag up or down to zoom the y axis. Click and drag left or right to zoom the x axis.

Adding Keyframes to Channels in the Animation Channel Editor

To add keyframes automatically:

1. In the Animation Channel Editor, enable Auto Key.
2. Select the channel you want to add keyframes to.
3. Click the value next to the channel name to open the calculator.
4. Enter a value and click enter to close the calculator.
   A keyframe is added to the channel you selected.

To set keyframes manually:

1. In the Animation Channel Editor, disable Auto Key.
2. Select the channel or channels you want to add keyframes to.
3. Drag the positioner to the frame where you want to set the keyframe.
4. Click Set Key.



The current values for the selected channels are set at the current frame.

TIP It can be useful to set keys across multiple channels using the Track Editor since this view focuses on channels, and their composing keyframes, in time. To view the Track Editor, select Track from the Channel View box.

To add keyframes at different locations on the animation channel:

1. From the Edit Mode box (just below the Keyframe tab), select Add Points.
2. Click the spot on the channel where you want to add the keyframe.

NOTE You can snap the keyframes you add to a frame (rather than in between frames). From the Curve Functions box, select FrmSnap.

To insert a keyframe:

1. Select the channel or channels you want to add keyframes to.
2. Drag the positioner to the frame where you want to set the keyframe.

3 Click Insert Key. The Insert Key button behaves differently depending on the current frame:
   - If the current frame is the last keyframe, a new keyframe is created <*duration*> number of frames after the last keyframe (*duration* is the value in the Duration field). The current frame advances to the new keyframe.
   - If the current frame is a keyframe other than the last keyframe, a new keyframe is created <*duration*> number of frames after the present keyframe. Other keyframes are moved by the same duration causing a ripple effect.
   - If the current frame is between keyframes, Insert acts the same as the Set button: a new keyframe is created and the duration is unchanged.

Changing Keyframe Values in the Animation Channel Editor

You can change keyframe values in the Channel or Track View of the Animation Channel Editor.

To gesturally change keyframe values:

1 In the Animation Channel Editor, select the channel that includes the keyframes you want to change.
2 Click Frame to view the channel.
3 From the Edit Mode box (below the Keyframe tab), select Select.
4 Select the keyframes. To select multiple keyframes, press Ctrl+drag to create a rectangular selection.

   NOTE To select keyframes in only the selected channel, press Ctrl+F and drag a rectangular selection around the keyframes you want to select.

5 Drag the keyframes to the new values. Press Alt to snap the frame value (vertical axis) to the nearest integer. Press Ctrl+Alt to snap to the nearest multiple of ten.

NOTE Enable Track Snap on the Filter tab to snap keyframes to frames (not in-between frames). Or, press Alt+Z to snap when Track Snap is disabled.

To set a specific value for a keyframe in the Animation Channel Editor:

1 Select the Channel that has the keyframe you want to change.
2 Set the positioner to the frame where the keyframe is located.
3 Click the value next to the channel name and enter a new value in the calculator. Or click the number and drag to change its value gesturally.

Copying and Pasting Channels or Keyframes in the Animation Channel Editor

You can copy and paste animation curves or keyframes between channels.

You can copy and paste folders provided the folder to which you are pasting has the same number of channels and folders as the folder you copied.

For example, in Action, you can copy the Scale folder from an axis and paste it into the Scale folder of another axis because they have the same channels (X scale, Y scale, for example). Trying to copy and paste the Camera folder to the Scale folder will not work because the two folders do not contain the same number and kinds of channels.

NOTE If an expression is applied to the selected channel, the expression is copied instead.

To copy and paste a selection of keyframes:

1. Select keyframes on a channel, and click Copy.

   TIP Keyframes do not have to follow one another to be copied and pasted to another channel.

2. Drag the positioner to a new frame.
3. Select the channel and the frame where you want to paste the keyframes, then click Paste.

   NOTE With the Select tool active, you can also `Shift+spacebar`-click a channel to paste keyframes starting from the location of the cursor. For example, `Shift+spacebar`-click frame 10 on a channel to paste the copied keyframes into the channel starting at frame 10.

   The keyframe values for the selected channels are pasted and set in the current frame.

To copy and paste a curve:

1. From the Channel View box in the Keyframe tab, select Channels. The Channel View box is to the left of the Insert Key button.
2. Select the channel containing the animation curve.

   TIP Select a folder to copy all the channels in a folder.

3. Click Copy.
4. Select the channel where you want to paste the copied curve.
5. Click Paste.
   The channel values you copied are applied to the selected channel or folder.

Changing the Duration of an Animation in the Animation Channel Editor

You can change the duration of an animation by scaling its channel horizontally in the Channel Editor. This allows you to change the number of frames it takes for an animation to play through, without having to move each keyframe manually.

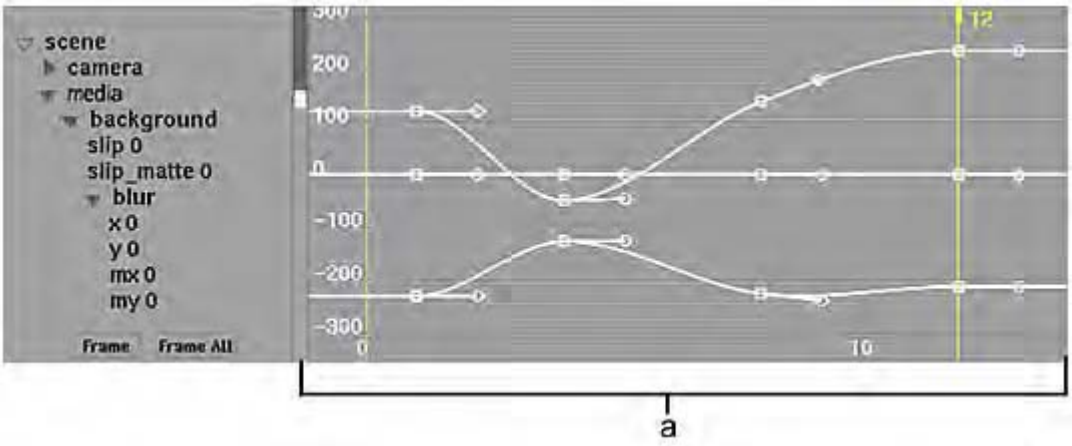
There are three ways to change the duration of an animation in the Animation Channel Editor:

- Use the X Scale tool to scale the channel using any keyframe as the origin of the scale.
- In the Tracks view, Drag the track handle to scales the track proportionally.
- Using the Duration field.

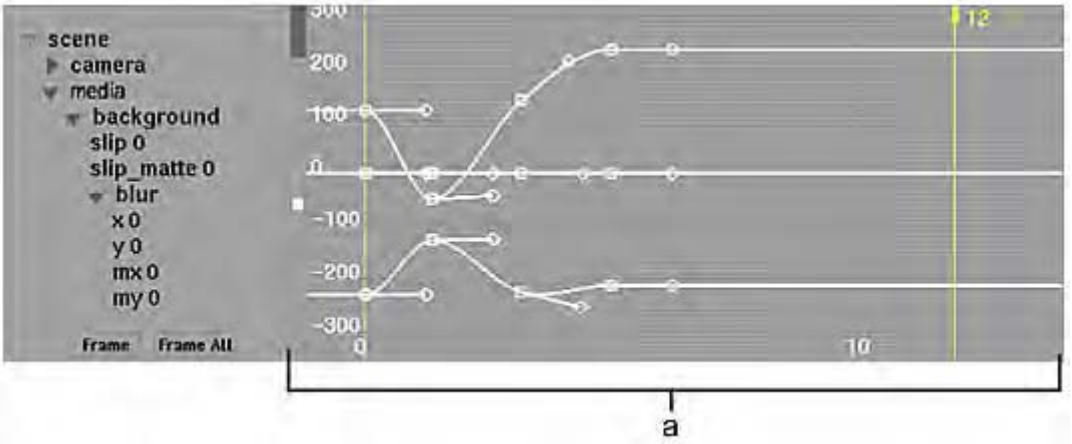
When changing the length of channels, you may want to adjust the number of frames of the clip in the Total Frames field. For example, if you shorten the length of all the clip's channels, you can then shorten the clips' frames accordingly. Or, if you scale the channels longer than the duration of the clip, you can then add frames to account for this.

To change the duration of an animation in the Channel View:

1. From the Edit Mode box, select X Scale.
2. Select the keyframes or channels you want to scale.
   You may have to click Frame to view the curve.
3. Click the keyframe that will act as the centre of the X Scale. Drag left to decrease the value or drag right to increase the value of the other keyframes along the X-axis, relative to the selected keyframe.



(a) Original animation curve



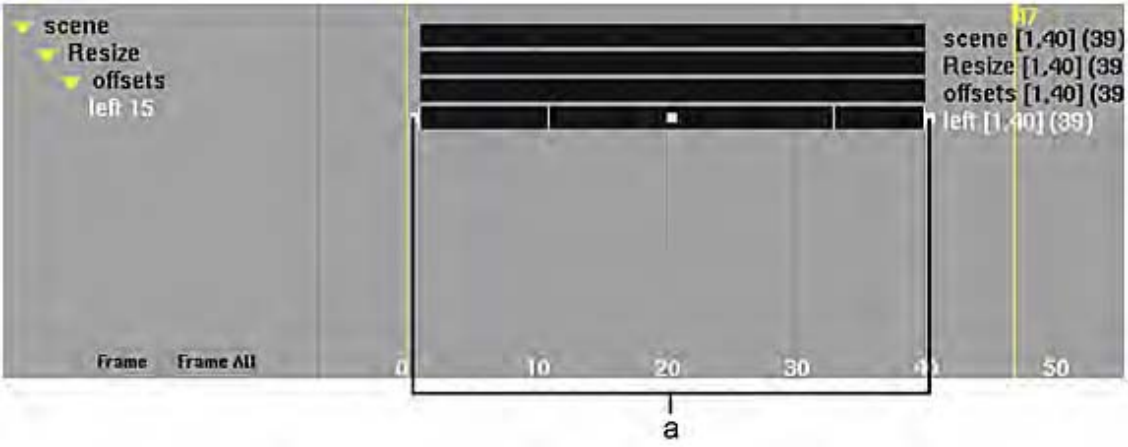
(a) XScale used to compress curves

To scale a channel in the Track Editor:

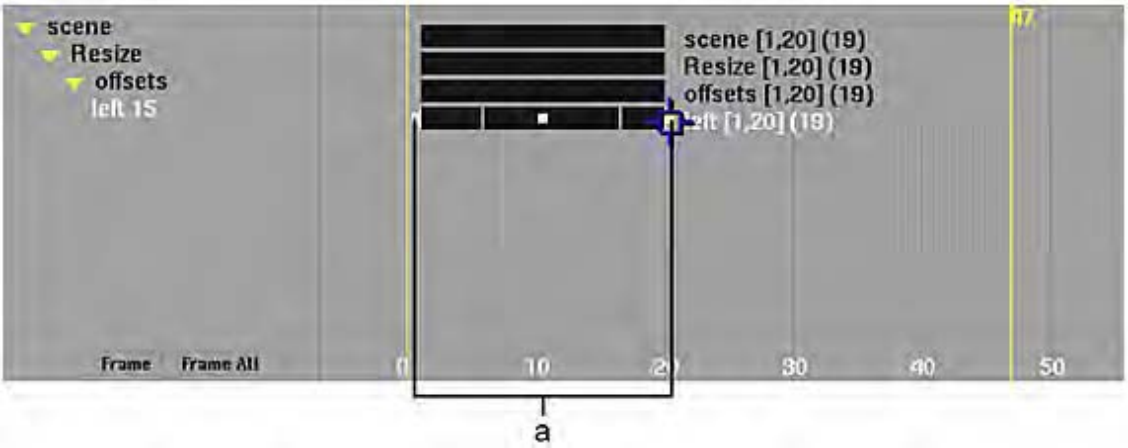
1. From the Channel View box, select Tracks.
2. From the Edit Mode box, select Select.
3. Select the channels you want to scale.
4. Drag the last handle to the left to decrease the duration, or to the right to increase the duration.

   NOTE To avoid adding keyframes between frames when scaling, press Alt-Z while scaling. If you have already scaled the track, Select Frame Snap from the Curve Functions box to snap keyframes in the selected channel to the closest frame numbers.

   The track scales as you drag the handle. All keyframes are repositioned proportionally in time.



(a) Original animation track



(a) Dragging handle used to compress a track from 40 to 20 frames

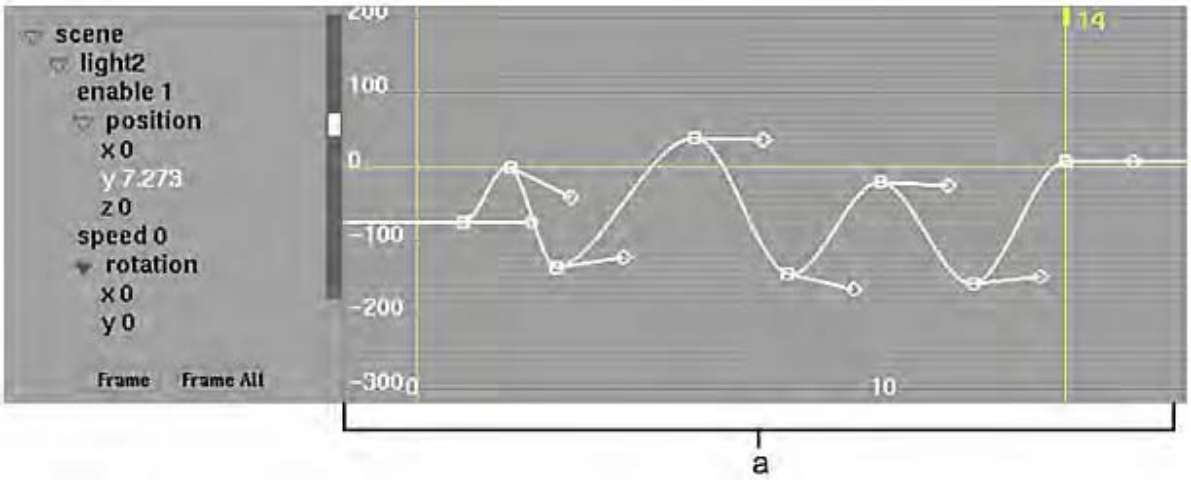
TIP You can move the handle of a folder to scale all channels in the folder. For example, moving the handle of the Scene folder (the top level folder) causes all the folders and channels in the hierarchy to be scaled by the same amount.

To change the number of frames between the current and next keyframe:

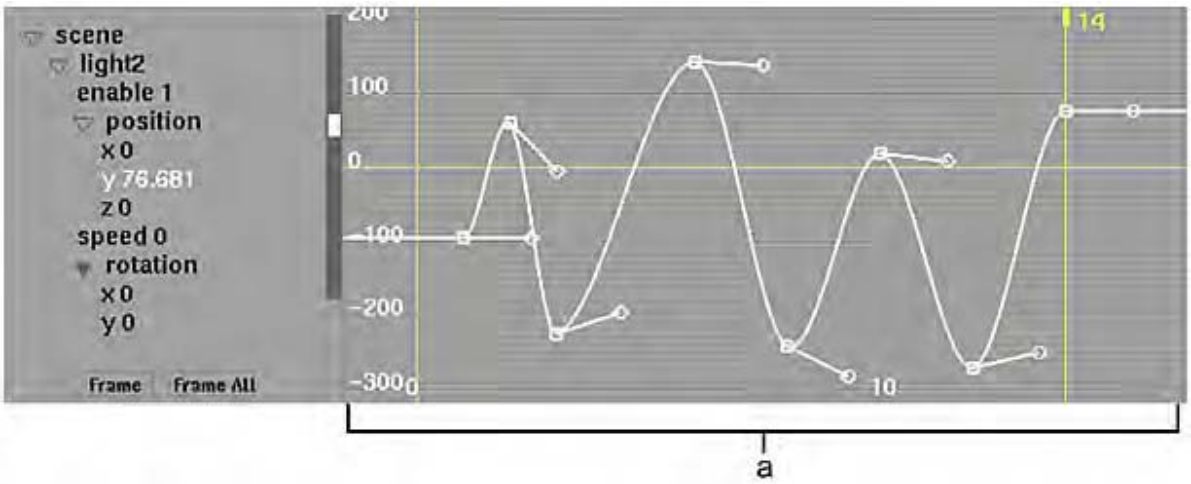
1. Select a keyframe.
2. In the Duration field, which is next to the Insert Key button in the Animation Channel Editor, enter a value for the number of frames between the selected keyframe and the next keyframe.

Changing the Values of Keyframes Proportionally in the Animation Channel Editor

In the Animation Channel Editor, you can change the vertical scale of an animation curve or a selection of keyframes and change keyframe values for a channel or a group of keyframes while maintaining each keyframe's relative frame position.



(a) Original animation curve



(a) YScale used to scale the curve based on a selected keyframe

To change the values of keyframes proportionally:

1. In the Animation Channel Editor, select a channel or group of keyframes.

   You can select more than one channel or a group of keyframes from more than one animation curve.
2. From the Edit Mode box, select Y Scale.
3. Click the keyframe that will determine the centre of the Y Scale.
4. Drag up to increase the value or down to decrease the value of the other keyframes relative to the selected keyframe.

   YScale works differently if you select more than one curve. For the other selected curves, the number of the frame that you select is verified in all animation curves. If the animation curve has a keyframe at the selected frame, this keyframe is used as the centre of the Yscale. This keyframe then sets the proportional scale calculation. If the animation curve does not have a keyframe at the selected frame, the curve remains unscaled.

Offsetting Select Keyframes or Channels in the Animation Channel Editor

You can offset a curve or a group of keyframes on the horizontal or vertical axis.

To offset a channel:

1. Select the channel or keyframes to offset.
2. From the Curve Functions box in the Keyframe tab, select Translate X to offset the channel horizontally or Translate Y to offset vertically.
3. Enter a value in the Curve Value field.

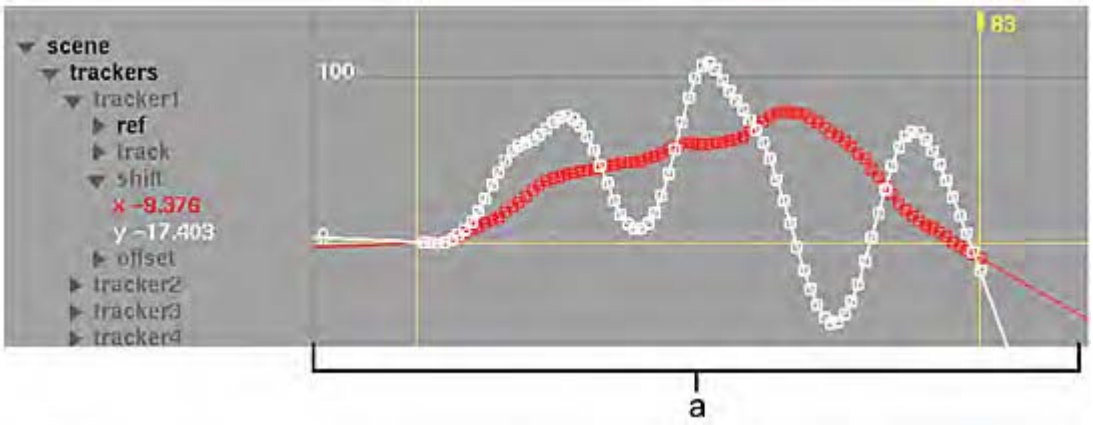
   A negative value offsets the channel or group of keyframes to the left (Translate X) or down (Translate Y). A positive value offsets to the right (Translate X) or up (Translate Y). The value in the Curve Value field produced by the drag returns to zero after release.

   The selected channel or keyframes are offset. The coordinates of each selected keyframe, relative to each other, remain unchanged. Translating along the Y-axis is only visible in the Animation Curve window.

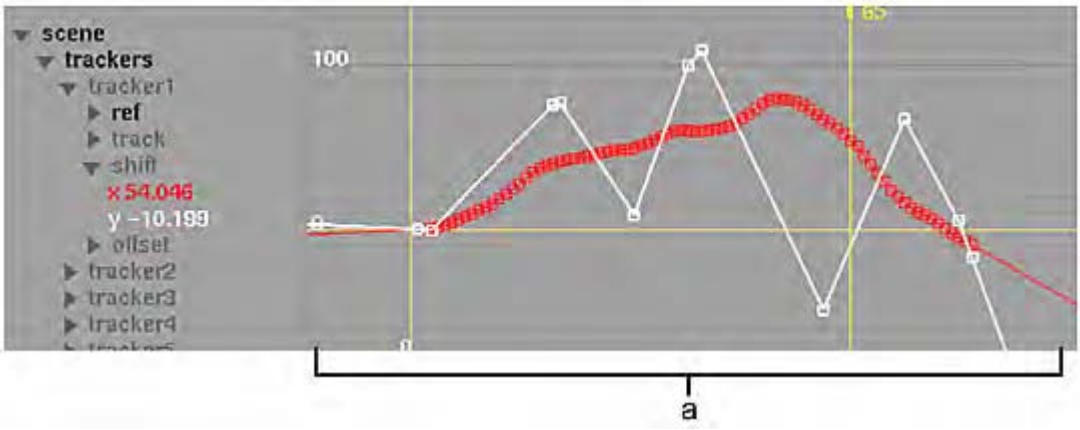
Simplifying the Number of Keyframes in the Animation Channel Editor

You can change the number of keyframes in your animation curve or in a group of keyframes. Using the Simplify function, the peaks and valleys in the selection are analysed and keyframes are generated with a duration between them as specified in the Curve Value field. A large value such as 20 or 30 decreases the number of keyframes between peaks and valleys, whereas a small value such as 1 or 2 increases the number of keyframes. The ideal simplify value varies between animation curves.

For example, the following figure shows an animation curve created using the Stabilizer. Simplify is applied with a value of 20. In sections where the slope of the curve is fairly constant in one direction, keyframes are removed between the peaks and valleys. You can adjust the simplified animation curve more easily because there are fewer keyframes to modify.



(a) The Y curve before applying Simplify has a keyframe at each frame



(a) The Y curve after applying Simplify with a simplification value of 20.

To simplify a channel or a group of keyframes:

1. Select a channel or group of keyframes.
2. From the Curve Functions box in the Keyframe tab, select Simplify.
3. In the Curve Value field, enter a simplification value.

   Use a large number to simplify the curve or group of keyframes. Use a small number to increase the number of keyframes. The value in the Curve Value field produced by the drag returns to zero after release.

Baking Keyframes

You can transform a curve or a group of keyframes by using the Bake function. This can be useful if you want to change an extrapolated cycle curve into a normal curve, with keyframes created along the cycle. You can also use the Bake function to remove the dependence of a curve linked to another curve with an expression.

To bake a cycled curve:

1. Select a curve that has a cycle extrapolation applied to it.
2. Set the current frame to the last frame that you want to be baked.
3. From the Curve Functions box in the Keyframe tab, select Bake.
4. In the Curve Value field, enter a bake value.

   Use a large number to simplify the curve or group of keyframes. Use a small number to increase the number of keyframes. The value in the Curve Value field returns to zero after you have finished baking.

Swapping Animation Curves

You can swap animation curves between two channels. Swap works in conjunction with Copy or Cut. You can also swap single keyframes but not a group of keyframes.

To swap two channels:

1. Select the first channel to swap.
2. Click Cut or Copy.

The selection is copied to the clipboard.

3 Select the second channel to swap.

4 From the Curve Functions box in the Keyframe tab, select Swap.
The selected channel is swapped with the channel in the clipboard.

5 Select the first channel and click Paste.
The two channels are swapped.

Flipping Channels and Keyframes

Select Negate in the Curve Functions box to flip a channel or a group of keyframes vertically. Select Reverse to flip a channel or a group of keyframes horizontally. Typically, you reverse or flip keyframes so that you can create symmetry with an animation. For example, use Reverse on an image that is warping from a sphere to a flat plane so that it then goes from a flat plane back to a sphere again. Perform these functions in the Animation Curve window.

To negate or reverse a channel or group of keyframes:

1 In the Animation Channel Editor, select a channel or group of keyframes.

2 From the Curve Functions box in the Keyframe tab, select Negate or Reverse.
The selection is flipped horizontally (negate) or vertically (reverse).

Removing Jitter from an Animation Curve

Select Jitter in the Curve Functions box to remove jitter from an animation curve or within a group of keyframes. Use the Curve Value field to specify the Over value. As a general rule, start with a large Over value over *n* frames to remove slow jitter, and a small Over value to remove fast jitter. The Jitter option is typically used on shift data in the Stabilizer to remove jitter while keeping camera movement. See Removing Jitter While Keeping Overall Motion (page 875).

To remove jitter:

1 Select a channel or group of keyframes.

2 In the Curve Functions box in the Keyframe tab, select Jitter.

3 In the Curve Value field, specify the Over value.

Applying an Average

Select Average in the Curve Functions box to remove jitter from a jittery animation track. While the Jitter option is used for removing jitter caused by trackers in the Stabilizer, the Average option is used to remove jitter from an animation track. Enter the number of keyframes used to calculate the average in the Curve Value field.

You can change the timing of a motion path animation without changing the motion path spline by adding a keyframe to the speed curve.

To average a channel:

1 Select a channel or group of keyframes.

2 In the Curve Functions box in the Keyframe tab, select Average.

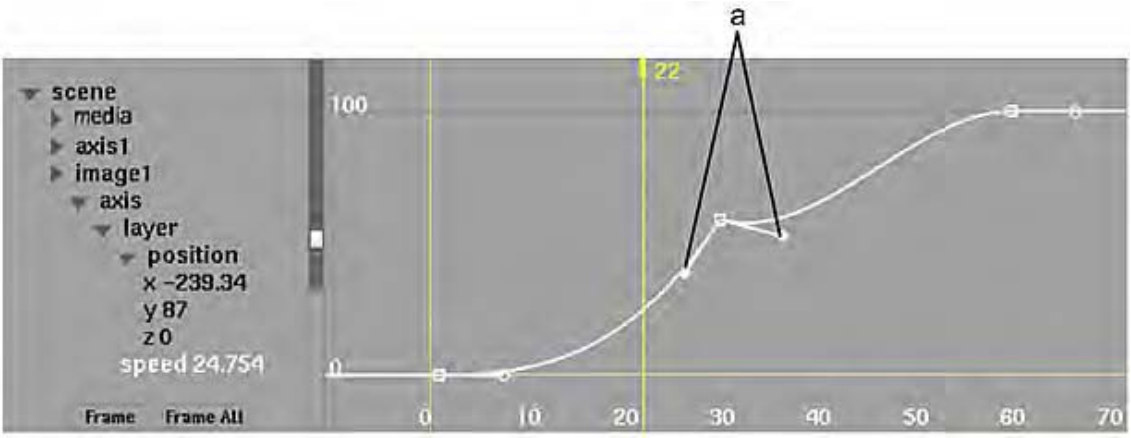
3 In the Curve Value field, specify a value.

The jitter is removed from the spline animation.

4 From the Edit Mode box, select Break.

5 Click a keyframe on the speed channel.

The keyframe breaks into two tangents.



(a) Keyframe tangents

6 From the Edit Mode box, select Move.

7 Adjust the tangents.

In this example, the speed curve makes the apple accelerate rapidly until it reaches frame 29, where it stops and continues moving slowly.

You can also apply the preceding procedure to a group of keyframes.

Modifying Tangents to Reshape the Animation Curve

To modify the tangents in the Animation Channel Editor.

1 From the Tangents box, select Auto.

2 From the Interpolation box, select an interpolation type. Only animation curves that use Bézier, Hermite, or Natural interpolation have keyframes with tangent handles.

3 From the Edit Mode box, select Select and move the tangent handles.

As you move the handles, the slope on each side of the keyframe is modified independently.

NOTE You must break tangent handles to move them independently. From the Edit Mode box, select Break Tangents and click the tangents you want to break.

To reset tangents to their default:

1 From the Edit Mode box, select Auto Tangent.

2 Click the handle of the tangent you want to reset.

Changing the Value of a Tangent Handle by a Specific Amount

You can modify the shape of an animation curve by rotating its keyframe tangent handles to control the smoothness and speed of the animation.

To rotate a keyframe's tangent handles:

1. Select a channel or group of keyframes.
2. From the Curve Functions box in the Keyframe tab, select Tangent R to rotate each right tangent handle or Tangent L to rotate each left handle.

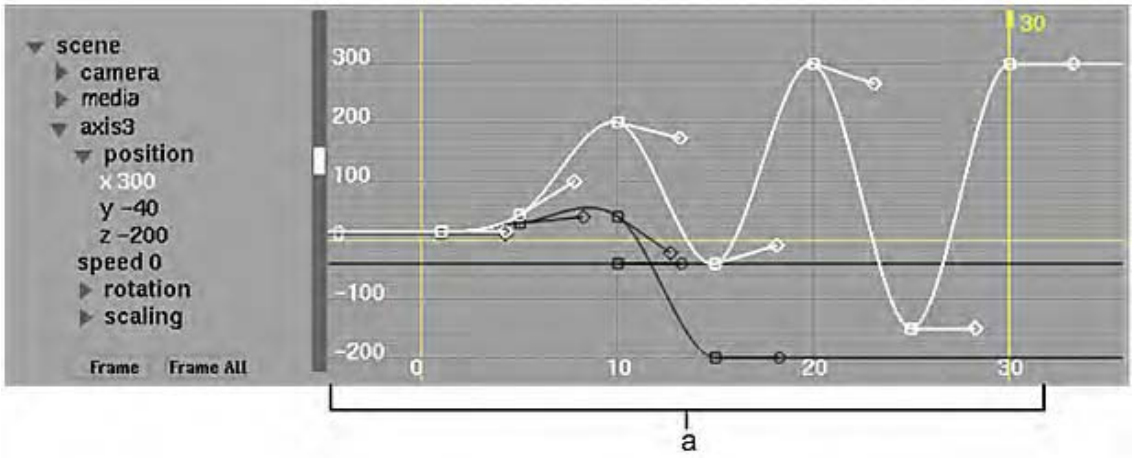
   NOTE Tangent R and Tangent L only work on curves and keyframes that use Bézier, Hermite, or Natural interpolation. Select your option from the Interpolation box.

3. Enter a value in the Curve Value field.

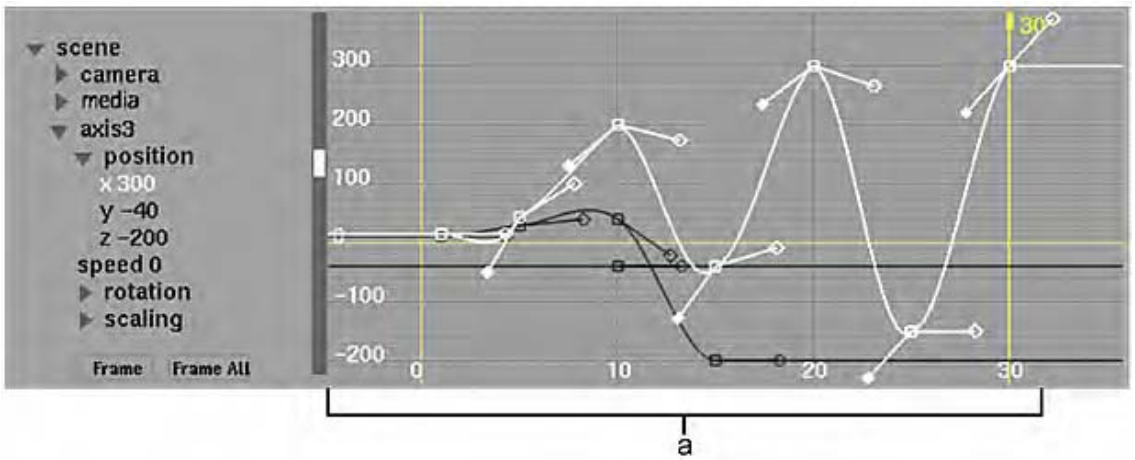
   A negative value rotates each tangent counter-clockwise. A positive value rotates each tangent clockwise.

   The tangent handle first breaks into two and then the handles are rotated.

   The following example shows the results of applying Tangent L with a rotation value of 36 to the keyframes of an entire curve. No keyframes were broken before Tangent L was applied.



(a) The Hermite curve before applying Tangent L. Note that none of the keyframes are broken.



(a) The same Hermite curve after applying Tangent L of 36. A break is applied to all keyframes before the tangents are rotated.

Setting Display Preferences in the Animation Channel Editor

Use the options in the Filter tab to control the display preferences in the Animation Channel Editor. The Animation Channel Editor display preferences can help you focus on the channels you are working on in the editor when many channels are displayed.

You can go in and out of the same tool in one session, and your display preferences are remembered. But if you restart, the your display preferences are not remembered. You must save and load your display preferences manually to access them when you restart the application.

To set display preferences:

1 In the Animation Channel Editor, select the Filter tab.



2 Select the type of channels you want to appear in the filtered view, animated or selected.

Select:	**To:**
Animated	Display channels that are animated.
Selected	Display channels that are selected.

3 Select how you want to expand or collapse channels that appear in the filtered view.

Select:	**To:**
Exp & Col	Expand animated or selected channels and collapse all others.
Expand Only	Expand animated and selected channels. All previously expanded channels remain selected.
No Expand	Not expand animated or selected channels.

4 Select how you want to display folders that contain channels in the filtered view.

Select:	**To:**
Hide All	Show only the animated or selected properties, for example, the X and Y values of an axis. All other properties in the same parent folder are hidden. All other folders are also hidden.

Select:	To:
Hide Group	Show only the animated or selected channel, its parent folder, and all channels in the same group, where a group includes all folders in the same parent folder. All other parent folders and groups are hidden. **NOTE** If multiple channels are animated or selected, all corresponding parent folders and groups are shown.
Show All	Show all animated or selected channels.

5 From the Auto Frame box, select an option for framing your view of the Animation Channel Editor.

If you select All, Animated or Selected, your view of the Animation Channel Editor changes as you change your channel selection.

Select:	To frame:
All	All channels. This option is the same behavior as clicking the Frame All button in the Animation Channel Editor.
Animated	All animated channels.
Selected	All selected channels. This option is the same behavior as clicking the Frame button in the Animation Channel Editor.
None	No channels. This option enables the Frame and Frame All buttons in the Animation controls.

6 Enable Track Snap to force key frames to snap to whole frame numbers as you drag them.

7 Enable Filter Tangent to display tangents only on selected channels. This can help you cleanup the display.

To save display preferences:

1 Click the User tab.



2 Click Save Prefs.

To load display preferences:

1. Open the Animation Channel Editor from the tool where you saved your display preferences in.
2. On the User tab, click Load Prefs.
3. Click the Filter tab. Display preferences you saved are now activated on the Filter tab.

Using Selection Sets to Control the Channels and Folders You Want to Display

With a selection set, you can control the channels and folders you want to display in the Animation Channel Editor.

Selection sets can be particularly helpful if you access the Animation Channel Editor from a complicated Batch or Batch FX schematic, where all the selected nodes are displayed in the editor. With a selection set you can limit the number of channels or folders displayed to just those you are working on.

Selection sets are saved and available throughout a session. However, they are not saved between sessions. This means that you do not have access to the selection sets you create after you restart the application.

To create a selection set:

1. Select the User tab in the Animation Channel Editor.
2. Enable Filter.
3. Type a name in the Selection Set Name field and press `Enter`.

   The selection set is created and selected in the Selection Set box. Notice that the Define button is also enabled and the channels and folders in the Animation Channel Editor are colored red.



4. Select the channels or folders you want to in your selection set. If you are displaying the Batch or Batch FX schematic, you can select the nodes that you want in your selection set.

   Notice that channels and folders in Animation Channel Editor turn green as you make selections. This helps you identify the channels and folders in the selection set.



5 Do one of the following:
- Click Use Sel to save the channels and folders you selected to a selection set.
- Click Add Sel To, to add the selection to an existing selection set.

6 Disable Define.

To switch between selection sets:

1 Select the User tab.
2 Select a selection set from the Selection Set box.

To modify a selection set:

1 Select a selection set from the Selection Set box.
2 Enable Define.
3 Select the channels and folders you want to display in the selection set.
4 Click Add Sel To. The selection is added to the selection set and the selection set is saved.
5 Disable Define.

Generating Keyframes Based on Audio Analysis

In the Channel Editor, you can use an audio clip to automatically generate keyframes on any video or audio channel based on the analysis of the audio.

To map keyframes based on audio analysis:

1 In the Channel Editor, select the Audio tab.



NOTE The Audio menu has a number of submenus with various settings. You may not need to use every setting to analyse your audio clip. See the sections after this procedure for an explanation of each setting.

2 From the Media Source box, select whether you want to analyse the current audio clip (Clip) or import an audio clip (File).



If there is more than one audio clip associated with your choice (for example, Front, Back, or Matte), you can choose which one to use in the Name box. If only one audio clip is available, the name of the clip appears in a locked field.

If you select File, click Import to open the Import Audio browser, and select an audio file to import.

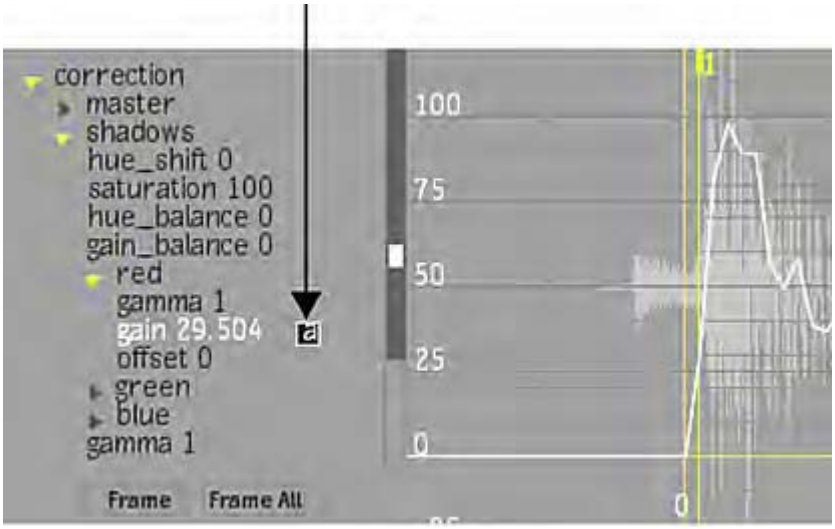
3 In the Channel hierarchy, select the channel or channels you want to link the audio to.

4 Use the settings in the Media submenu to prepare the audio track you want to use. See Audio Keyframe Settings (page 1490).

5 From the Mapping submenu, enable Link To Channel.



The letter 'a' appears next to the channel in the Channel hierarchy, indicating that this channel is linked to the audio.



6 Use the settings in the Mapping submenu to determine how the audio track is mapped to the selected channel. Keyframes are adjusted automatically as changes are made to the settings.

7 Use the settings in the Filtering submenu to filter the exact frequencies you want to analyse. Keyframes are adjusted automatically as changes are made to the settings.

8 Disable Link To Channel or exit the module when you are satisfied with the analysis. The generated keyframes are automatically baked to the animation curve.

You can use different audio tracking settings on different channels. Disable Link To Channel and select a different channel in the hierarchy and restart the mapping process.

NOTE If you want to be able to tweak the settings, you can save the analysis settings in an audio tracker setup.

Audio Keyframe Settings

The animation controls Audio tab has a number of submenus with numerous settings to help you work with your audio clips.

Common Settings

There are a few common settings found in the Audio submenus.

Play button Plays the audio media.

Stop button Stops the audio playback.

Reset box Available in the Media, Mapping, and Filtering submenus. In the Setup submenu, only Reset All is available.Select whether to reset the current Audio submenu, or all of the Audio settings.

Media Submenu



(a) Media Source box

Media Source box Select whether to analyse the current audio clip (audio attached to the front, back, or matte clips, for example), or the imported audio file.

Import button Opens the import audio browser to load a file.

Clip Name box If multiple audio clips are available, select which clip to use as source media. If only one audio clip is available, the name of the clip appears in a locked field.

W+ button Displays the selected media's waveform in the Channel Editor.

W- button Hides the waveform.

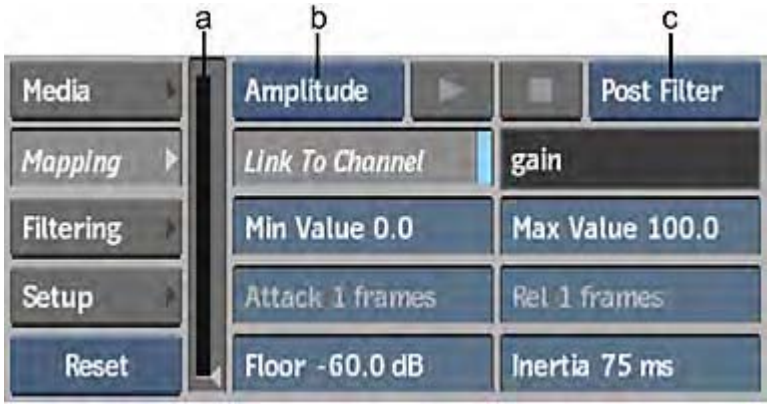
Track box Select which audio track to use as source media.

Slip field Displays the amount of frames to slip the media. Editable.

Handles field Displays the amount of handles (frames) to analyse before and after the current shot duration. Editable.

For example, with a shot of 100 frames, if you enter 30 in this field, the total number of frames becomes 160 (30 frames before and 30 frames after the original 100 frames).

Mapping Submenu



(a) Audio Level Indicator (b) Tracking Mode box (c) Listening Mode box

Tracking Mode box Select whether to use an Amplitude or Transient tracking mode. In Amplitude tracking mode, a keyframe is generated at each frame, following the envelope of the audio signal. In Transient tracking mode, a keyframe is generated for each audio transient (sudden rise in amplitude), provided that the signal rises above the specified threshold.

Listening Mode box Choose between Pre Filter (audio as it was imported) and Post Filter (audio with any changes made) monitoring when playing back audio media. Does not affect the analysis.

Link to Channel button Enable to map the tracked audio to the selected animation channel. An 'a' appears in the channel list next to the name of the linked channel. Multiple channels can be selected simultaneously.

Channel Name field This locked field displays the name of the selected channel being linked to the tracked audio. Multiple Channels is displayed if more than one channel is selected.

Min Value field Displays the minimum value at which keyframes can be set in the linked channel. Editable.

Max Value field Displays the maximum values at which keyframes can be set in the linked channel. Editable.

NOTE The minimum value can be higher than the maximum value to get an inverse mapping of the tracking mode in the animation channel.

Attack field Displays the number of frames needed for the channel to reach the Max Value before a transient. Available only in Transient tracking mode. Editable.

Release field Displays the number of frames needed for the channel to reach the Min Value after a transient. Available only in Transient tracking mode. Editable.

Threshold field (Not shown). Displays the value at which a keyframe is generated each time the signal rises above it. Available only in Transient tracking mode. Editable.

TIP Start with a high threshold, and lower it until all required peaks are detected.

Floor field Displays the minimum level of the audio media to be considered for the analysis (typically used to remove analysis noise between audio transients). Available only in Amplitude tracking mode. Editable.

Inertia field Displays the rate at which the signal decreases after a transient. Editable.

In Transient tracking mode, the Release field can be increased to avoid the generation of several consecutive keyframes when the transient is not clearly defined in the input signal. Conversely, if the Release field is set to 0 ms, each transient rising above the threshold generates a keyframe, even if each occurs within a few milliseconds of another.

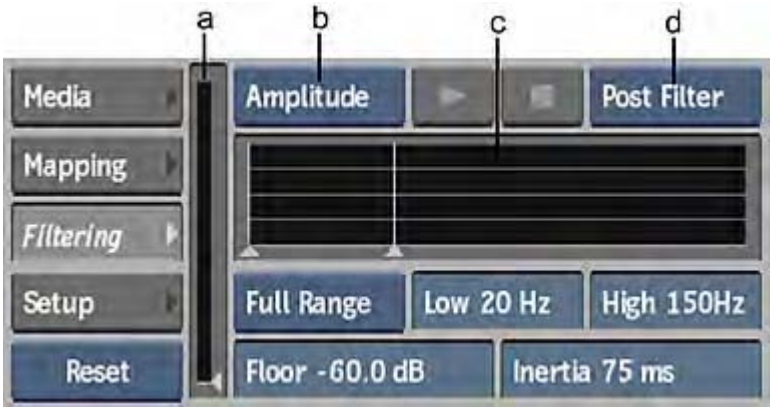
Audio Level Indicator A visual representation of the audio level and the current Threshold or Floor value.

Audio Level Indicator A visual representation of the audio level and the current Threshold or Floor value.

Reset box Select Reset to reset the Mapping submenu settings only. Select Reset All to reset all of the Audio tab settings.

NOTE Some of the settings in the Mapping submenu are repeated in the Filtering submenu so that you do not to have to switch tabs to change the settings. The same settings are reflected in both submenus.

Filtering Submenu



(a) Audio Level Indicator (b) Tracking Mode box (c) Frequency Graph (d) Listening Mode box

Frequency Graph A visual frequency representation of the audio signal being tracked. The audio display is always Pre Filter.

Frequency Range Preset box Select a frequency range preset to determine the cutoff frequencies for Low and High filters.

Low field Displays the lowest frequency of the input signal used in the analysis (also represented by a triangle under the frequency graph). Use the slider or field to adjust. Editable.

High field Displays the highest frequency of the input signal used in the analysis (also represented by a triangle under the frequency graph). Use the slider or field to adjust. Editable.

NOTE Some of the settings in the Filtering submenu are repeated in the Mapping submenu so that you do not to have to switch tabs to change the settings. The same settings are reflected in both submenus.

Setup Submenu



Load button Click to load a saved audio mapping setup.

Save button Click to save an audio mapping setup.

Setup Name field This locked field displays the name of the loaded setup

Animation Menu Settings

The Animation menu has some common settings, and a number of settings specific to certain operations, organized in submenus. For specific Audio submenu settings, see Audio Keyframe Settings (page 1490).

Common Settings

Regen button Enable to automatically update the scene when an animation change is made.

Channel View box Select a view of Channels, Tracks, or Info to control and coordinate your animation.

Auto Select button Enable to select the channel automatically when you modify a channel value in the field for an effect or tool.

Set Key button Sets the current values for the selected channels in the current frame (when Auto Key is disabled).

Delete Key button Deletes the selected keyframes or curve.

Keep button Deletes all keyframes except the current keyframe.

Cut button Cuts the selected keyframes or curve.

Copy button Copies the selected keyframes or curve.

Paste button Pastes the selected keyframes or curve.

Link button Creates an expression by linking one channel to another.

Expressions button Applies an expression to one or more channels.

Animation Reset box Select to reset the selected channel or all channels.

Keyframe Submenu

Curve Functions box Select an operation to apply to your animation.

Curve Value field Displays the value for the selection in the Curve Functions box. Editable.

Insert Key button Inserts a keyframe. Its behaviour changes depending on where the keyframe is inserted.

Duration field Displays the number of frames between keyframes. Editable.

Absolute/Relative box Select whether the value in the Duration field is absolute or relative to the present keyframe value.

Tangents box Select the behaviour of the keyframe tangents to help refine the shape of an animation curve between keyframes.

Tangent Mode	Tangent Behaviour
Auto	Tangents are fixed in an horizontal position to prevent the curve from over- or under-shooting keyframes. Moving a keyframe higher or lower than its neighboring keyframes alters their tangents so the curve remains as uniform as possible. The handles of tangents set to Auto are empty circles. Manually moving a keyframe's tangent voids this setting for that tangent. It will now perform as a Fixed tangent.
Fixed	Tangents are fixed in their current position so that moving a keyframe does not alter its tangents nor those of its neighboring keyframes. The handles of tangents set manually are filled circles.

Tangent Mode	Tangent Behaviour
Smooth	Legacy setting to ensure compatibility with previous versions of a setup. Tangents are not locked like when using the Auto setting: they are affected by the movement of adjacent key-frames. There are no other benefit from using this setting.

Interpolation box Select an interpolation type to define the shape of an animation curve between keyframes.

Extrapolation box Select an extrapolation type to define the shape of an animation curve before the first keyframe and after the last keyframe of the curve.

User Submenu

Filter button Enable to create and name a selection set (the channels, folders, and folder elements that you want to display as part of a set).

Selection Set box Select a user-defined selection set. Select <new user> to create a selection set.

Name field Enter a name for the selection set. Editable.

Define button Enable to display channels in red and green to indicate the channels contained in the current user-defined selection set.

Delete button Removes the currently selected selection set from the Selection Set box.

Use Selection button Displays the current selection set in the Channel Editor and at the same time saves the current selection set.

Add Selection To button Adds the current selection to the active selection set.

Load Filter Preferences button Loads the filter preferences last saved for the module.

Save Filter Preferences button Saves the current filter settings as preferences for the module.

Filter Submenu

Animated button Enable to display channels that have animation keyframes. Use in conjunction with the Animated Expansion and Animated Show/Hide boxes to further control the content displayed.

Selected button Enable to display only channels that are selected in the Channel Editor. Use in conjunction with the Selected Expansion and Selected Show/Hide boxes to further control the content displayed.

Animated Expansion box Select an expansion option for animated folders.

Selected Expansion box Select an expansion option for selected folders.

Animated Show/Hide box Select to display or hide animated folders.

Selected Show/Hide box Select to display or hide selected folders.

Auto Frame box Select an option to frame channels automatically. If None is selected, the Frame and Frame All buttons are enabled.

Track Snap button Enable to force keyframes to snap to frames when dragging tracks by their left or right handles.

Filter Tangents button Enable to display only the tangent handles of selected channels.

Advanced Animation: About Expressions

Use expressions to apply animation dynamically to one or more channels. For example, use expressions to easily simulate real-world forces such as gravity, momentum, and centrifugal force. Expressions save time since you can animate one channel, and then refer other channels to the first channel so that the other channels automatically behave in relation to the first one. You can use arithmetic operators, such as addition and multiplication, as well as predefined functions to precisely control the behaviour of an animation.

Expressions are available through all modules that use the Channel Editor.

Working with Expressions in the Channel Editor

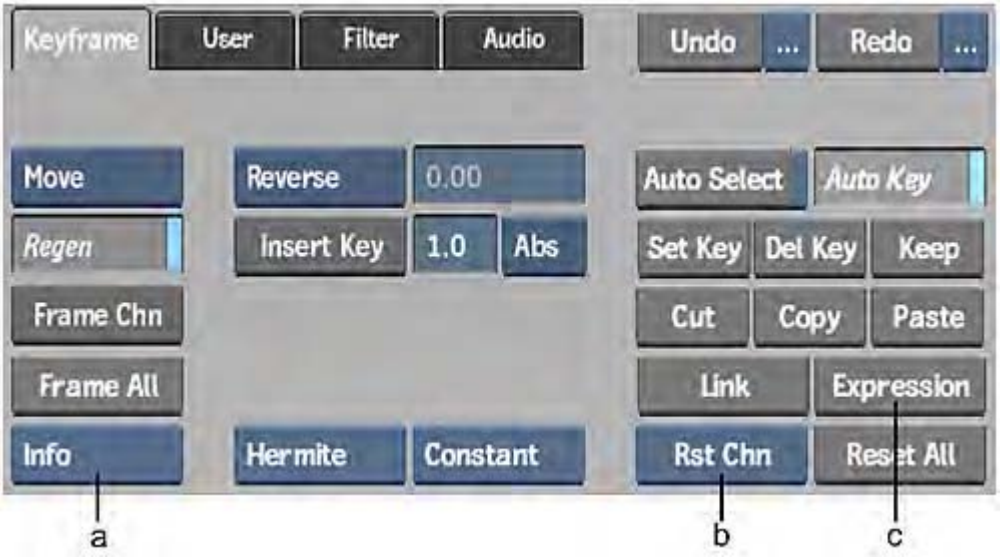
You can apply, modify and view expressions using the Channel Editor. You can use the copy and link functionality to quickly apply basic expressions to multiple channels.

WARNING Expressions override interpolation modes and previously set keyframes for a selected channel.

To apply an expression to a channel:

1 In any module menu, click Animation.

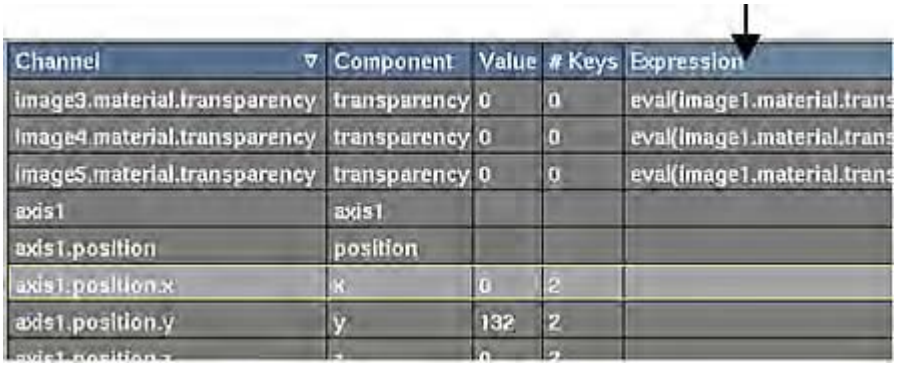
The Animation controls appear.



(a) Channel View box (b) Reset Channel button (c) Expressions button

2 From the Channel View box, select Info.

Channel information appears in a tabular grid for easier viewing of expressions and channel details. From Info view, you can click the Expressions column heading to sort by expressions and view them more easily.



3 Select the channel to which you want to apply an expression.

4 In the Animation controls, click Expression.

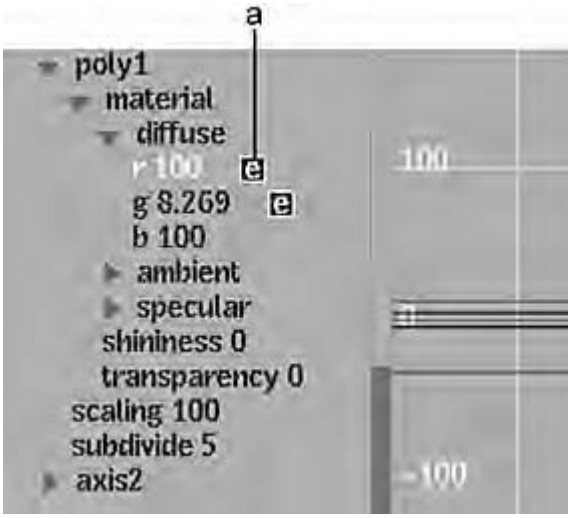
The Expression field appears below the Channel Editor.



(a) Expression field

5 Enter an expression according to the guidelines described in Operator Reference (page 1503) and Function Reference (page 1505) and press `Enter`.

The letter 'e' appears next to the channel in the channel hierarchy, indicating that this channel contains an expression.



(a) Expression indicator

NOTE Entering an invalid expression will result in the message “Error: Expression: parse error” and the entered expression will not be applied to the channel.

To modify an expression:

1 Click Animation to display the Animation controls.

2 In the channel hierarchy or Info view, select the expression that you want to edit.

3 In the Animation controls, click Expression.
The Expression field appears below the Channel Editor.

4 Modify the expression in the field and press Enter.
The modified expression appears in the table.

TIP When you are in the Expression field, you can press Up one or more times to retrieve its previous contents. This can be useful if you want to correct an invalid expression that you recently entered.

To remove an expression:

1 Click Animation to display the Animation controls.

2 In the channel hierarchy or Info view, select the channel with the expression that you want to remove.

3 Click Rst Chn to remove the expression.
The channel is reset.

Linking Channels

You can create an expression by linking the behaviour of one channel to another using Copy and Link. You can link different types of channels together. For example, make the scaling of a layer affect the rotation of another layer, or blur a layer by linking it to the position of an axis. Any change that occurs in the position of the axis is reflected dynamically in the layer blur.

To create an expression by linking one channel to another:

1 Click Animation to display the Animation controls.

2 From the Channel View box, select Info.



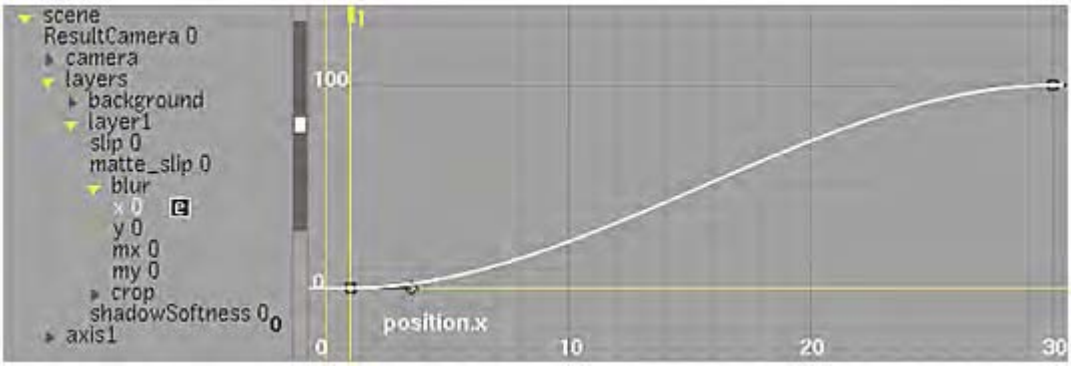
(a) Channel View box (b) Link button (c) Copy button

TIP Enable the Hierarchy View button  to better view the channel hierarchy.

3 Select a source channel from which to link and click Copy.
For example, copy the x position of axis1 and link it to the x blur value of a layer.

4 Select the destination channel and click Link.

The x blur value of layer1 is linked to the x position value of axis41, so as the position increases, so does the blur. This is shown in the following illustration.



Copying Nodes with Expressions

When working in Action, Keyer or Garbage Masks, if you copy a branch in Schematic view, or append to a setup by loading a second setup, any nodes with the same name as existing ones are renamed with unique names. Likewise, if expressions are associated with those nodes, the expressions are replicated and the associated channels are renamed to correspond with the new node names. This preserves the integrity of the expression within the copied branch or appended setup.

If you copy a portion of a branch that has expressions associated with it, the expressions related to the copied portion of the branch will be associated with the new copy. Expressions related to uncopied portions of the branch will maintain their association with the original branch.

Cascading Expressions

You can copy a single expression to multiple channels that build on the result in the previous channel using the *SelectionOrder* and *CascadeChannel* keywords. These keywords increment the channel number or name as follows:

- Each instance of SelectionOrder increments the channel number.
- Each instance of CascadeChannel replaces the occurrence of the channel name with the name of the preceding channel, relative to the order in which the channels are selected.

In general, experiment with the SelectionOrder and CascadeChannel keywords to determine which results are optimal for your animation.

To apply cascading expressions using SelectionOrder:

1 Click Animation to display the Animation controls.

2 From the Channel View box, select Info.

3 Select multiple channels to which you want to apply the cascaded expression, in the desired order.

As you select each channel, the selection order is shown next to the channel name in Info view.

TIP To select a range of channels, click the first channel that you want to include in the range. Then click the last channel to include in the range while holding `Shift`. To add to a range of files, hold `Ctrl` or the stylus button while selecting the channels from the Channel Editor.

4 Enter an expression that includes the keyword SelectionOrder.

Every instance of the keyword SelectionOrder will be substituted in your expression by a different number for each channel, which corresponds to the order in which you selected them.

To animate four channels using SelectionOrder:

1 In Action, add four axes named axis1, axis2, axis3 and axis4.
2 Animate axis1.position by creating a few keyframes.
3 Select the channels axis2.position, axis3.position and axis4.position in this order.
4 Enter the following expression: eval(axis1.position, frame - 5 * SelectionOrder).

The following expressions are applied:

- axis1.position: no expression
- axis2.position: eval(axis1.position, frame - 5 * 1)
- axis3.position: eval(axis1.position, frame - 5 * 2)
- axis4.position: eval(axis1.position, frame - 5 * 3)

To apply cascading expressions using CascadeChannel:

1 Click Animation to display the Animation controls.
2 From the Channel View box, select Info.
3 Apply an expression to a base channel to which the other channels will relate.
4 Select multiple channels, starting with the base channel, to which you want to apply the cascaded expression, in the desired order.
5 Enter an expression that includes the keyword CascadeChannel.

The base channel (the first channel selected) will remain unmodified. In the expressions for the other selected channels, every instance of the keyword CascadeChannel will be substituted by the name of the channel that preceded it, in the order in which you selected them.

To animate four axes using CascadeChannel:

1 In Action, add four axes named axis1, axis2, axis3 and axis4.
2 Animate axis1.position by creating a few keyframes.
3 Select, in this order, the channels axis1.position, axis2.position, axis3.position and axis4.position.
4 Enter the following expression: eval(CascadeChannel, frame - 5).

The following are applied:

- axis1.position: no expression
- axis2.position: eval(axis1.position, frame - 5)
- axis3.position: eval(axis2.position, frame - 5)
- axis4.position: eval(axis3.position, frame - 5)

NOTE This results in the same animation as the SelectionOrder example given above.

Expression Content

An expression is composed of numeric values, constants, channel references, or combinations of these used with arithmetic operators. Expressions are calculated from left to right, according to a specific order defined in Operator Precedence (page 1504).

For example, the following expression contains channel references and multiplies the transparency value of image1 by 2 to affect the transparency of image2 across all keyframes.

Channel	Expression
image2.material.transparency	image1.material.transparency* 2

Vectors

Certain channels such as Position, Rotation, Scale, and Shear are vector based since they represent 3D space and include X-, Y-, and Z-axes. A vector contains three elements that represent the individual components of the channel. Vectors are written using the convention (x, y, z) where x, y, and z represent separate scalar values.

Expressions can either affect the components of a vector individually or collectively. For example, the following expression sets the individual rotation for the x, y, and z axes to 30°, 45° and 90°, respectively.

Channel	Expression
axis1.rotation	(30, 45, 90)

The following expression sets the rotation for axis2 twice that of axis1, affecting the vector collectively.

Channel	Expression
axis2.rotation	axis1.rotation* 2

Functions

Flame Premium includes many predefined functions that can be used to perform calculations in an expression. You pass function-specific values, called *arguments*, and they return another value back to the expression that called it. A function call in an expression begins with the function name, followed by an opening parenthesis, the arguments for the function separated by commas, and finally a closing parenthesis.

NOTE Function names are case-sensitive.

Arguments for functions can be either scalar values or vectors. When you use a function, make sure that you pass it the correct type of parameter. See Function Reference (page 1505) for information on the arguments and return values for each function.

You can nest function calls by using the return value of a function as one of the arguments of another function. When a nested function is used as an argument, it must return a value that conforms to the type and range that the argument requires.

You can define your own functions and use them in your expressions just as you would with any of the predefined functions. See Defining Your Own Functions (page 1526).

Examples

The following expression uses the noise function to create a random positioning effect for axis1.

Channel	Expression
axis1.position.x	noise(frame)*5

The following expression uses the eval function to make the position of axis2 the same as that of axis1, but delayed by 10 frames.

Channel	Expression
axis2.position	eval(axis1.position, frame - 10)

The following expression uses the eval function to make the animation of axis3 the same as that of axis1, but at half the speed.

Channel	Expression
axis3	eval(axis1, frame / 2)

Keywords and Constants

Use the keyword *frame* in an expression to get the value of the current frame number in your animation. This allows you to create an animation by specifying how a value changes over a sequence of frames.

Use the constant *PI* instead of the literal numeric value of π (3.1416...).

NOTE Keywords and constant names are case-sensitive.

Select:	To display:
PI	3.1416...
e	Euler's number (2.7182818284...)
phi	Golden Ratio (1.6180339887...)
c	Speed of light in vacuum (299792458 m/sec)
freefall	Standard free fall acceleration near the earth's surface (9.80665 m/sec^2)
echarge	Elementary charge (1.602176487e-19 Coulomb)

Simplified Expressions

A reference to another channel does not have to be fully qualified when it is at the same hierarchical level as the channel to which the expression is being applied. For example, the following simplified expressions

are valid when applied to the specified channel. The fully qualified versions of the expressions are also provided.

Channel	Simplified Expression	Fully Qualified Expression
axis1.position.y	x + 50	axis1.position.x + 50
axis1.rotation	position / 2	axis1.position / 2
image1.material.diffuse	specular * 3	image1.material.specular * 3

References to other channels also work when referring to the siblings of any parent. For example, consider the following structure.



You can apply the expression position.x to axis.rotation.x because position is a sibling of rotation, and rotation is a parent of rotation.x.

Operator Reference

Operators specify a mathematical or logical calculation to be performed between various elements of an expression.

Arithmetic Operators

Use the following arithmetic operators to perform basic mathematical operations.

Operator	Description
+	Addition
-	Subtraction
*	Multiplication
/	Division

Operator	Description
%	Modulo (this is a binary operator)
(*x, y, z*)	Vector where *x, y, and z* are scalar values

Comparison Operators

Use the following operators to compare two values with each other. When two values are compared using these operators, the result is 1 if the comparison is true and 0 if the comparison is false.

Operator	Description
==	Equal to
!=	Not equal to
<	Less than
<=	Less than or equal to
>	Greater than
>=	Greater than or equal to
&&	Boolean AND
||	Boolean OR
!	Boolean NOT

Operator Precedence

When you combine several operators in a single expression, the operations are performed in the following order.

Order	Operator	Description
1	!	Boolean NOT
2	-	Negation (as in -1)
3	*, / and %	Multiplication, division, and modulus
4	+ and -	Addition and subtraction

Order	Operator	Description
5	< , <=, >, and >=	Comparison
6	== and !=	Equivalence
7	&&	Boolean AND
8	||	Boolean OR

When operators with the same precedence are encountered, operators are evaluated from left to right. However, when part of a formula is enclosed in parentheses, it is evaluated first.

Function Reference

You can easily create very complex expressions by using the available functions. This function reference provides detailed information about each function. Function arguments in square brackets are optional. However, if you give a value to an optional argument, you must also give a value to every optional argument before that one.

Animation Functions

Use the following functions to animate channels in a general way.

NOTE Arguments shown in square brackets are optional. For example, in the syntax of the align function, the options [AxisToAlign] and [BankingAngle] are optional.

align

Returns a rotation vector such that a designated axis of an object is aligned with the direction of the object's movement. You can also bank the rotation around the axis. The result should usually be assigned to a rotation channel.

Syntax:	align(PosToFollow, [AxisToAlign], [BankingAngle])
Arguments:	■ *PosToFollow* is the vector representing the channel to align on, typically an animated position. ■ *AxisToAlign* is the vector representing the axis with which to align, by default the X-axis (1,0,0). ■ *BankingAngle* is the angle in radians by which to rotate the result about the AxisToAlign (performs banking). If the BankingAngle argument is specified, the AxisToAlign argument must also be specified.
Example:	align(axis.position, (0,0,1), frame * PI / 8) returns a rotation vector that points the object's Z-axis in the direction of its motion while the rest of the axis rotates along this axis.

lookat

Returns a rotation vector based on an object's position that points it towards a second moving object. The result should usually be assigned to a rotation channel.

Syntax:	lookat(TargetPos, ObserverPos, [AlignVector], [UpVector])
Arguments:	■ TargetPos is the vector of an object's position channel that you want the rotation vector to point towards. ■ ObserverPos is the vector of the position channel from which you are looking. ■ AlignVector is the vector of the direction that you want to have looking at the target, by default the Z-axis (0,0,1). ■ UpVector is the vector of the direction that you want to be pointing upwards, by default the Y-axis (0,1,0). The UpVector should be set to a different vector than the AlignVector. If the UpVector argument is specified, the AlignVector argument must also be specified.
Examples:	■ lookat(followed_axis.position, follows_axis.position) returns the rotation vector required so that follows_axis points its Z-axis towards followed_axis. ■ lookat(followed_axis.position, follows_axis.position, (0,1,0),(1,0,0)) returns the rotation vector required so that follows_axis points its Y-axis towards followed_axis, with its X-axis pointing upwards.

eval

Returns the value of a given expression at another point in time.

Syntax:	eval(Expression, FrameNumber)
Arguments:	■ Expression is the expression to be evaluated. This can be any valid channel value. ■ FrameNumber is the frame to simulate when evaluating the given expression.
Examples:	■ eval(axis1.position.x, 5) returns the value of axis1.position.xat frame 5. ■ eval(axis1.position, frame - 10) returns the axis1.positionvector at 10 frames behind the current frame. ■ eval(axis1, frame / 2) returns the entire axis1 channel at half the normal speed.

if

Returns one of two values based on the result of a conditional test. You can nest multiple functions inside each other to handle multiple results.

Syntax:	if(Condition, TrueValue, FalseValue)
Arguments:	■ Condition is any channel or expression. When comparison operators are used, a true expression evaluates to 1 and a false expression evaluates to 0. See Comparison Operators (page 1504).

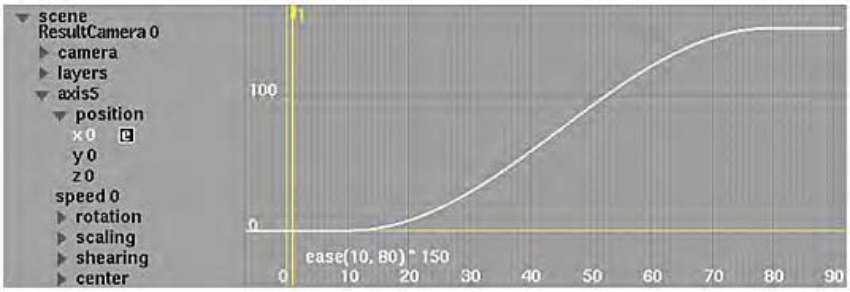
	■ TrueValue is the value to be returned for any non-zero result. ■ FalseValue is the value to be returned if Condition evaluates to 0.

Examples:	■ if(axis.position.y >= 100, 5, -5) returns 5 when axis.position.yis greater than or equal to 100, and -5 otherwise.
	■ if(frame < 10 || frame >20, 100, 200) returns 100 when the current frame is less than 10 or above 20, and 200 otherwise.
	■ if(axis.position.x, 6, 7) returns 6 when axis.position.x is something other than 0, and 7 when it is 0.
	■ if(axis.position.x == 100 && axis.position.y != 200,8, 9) returns 8 when axis.position.x is 100 and axis.position.y is not 200, and 9 otherwise.
	■ The next four examples all equivalently return 5 when neither axis.position.x nor axis.position.y are greater than or equal to 0, and -5 otherwise.
	
	if(!(axis.position.x =>0 || axis.position.y =>0), 5, -5)
	if(!(axis.position.x =>0) && !(axis.position.y =>0), 5, -5)
	if(axis.position.x < 0 && axis.position.y < 0, 5, -5)
	if(axis.position.x < 0, if(axis.position.y < 0, 5, -5), -5)

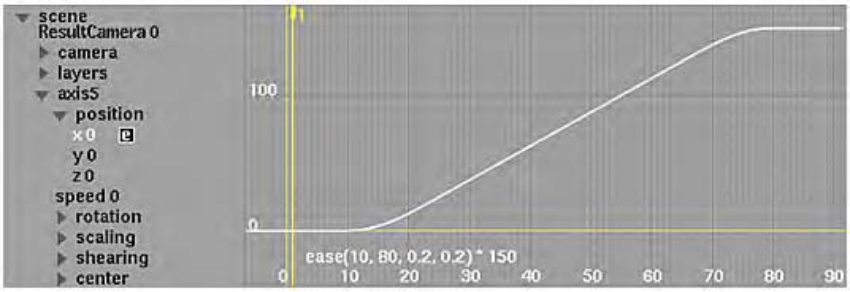
ease

Returns a number from 0 to 1 representing a smooth S-curve transition between a given range of frames. All frames before the start frame are assigned 0 and all frames after the end frame are assigned 1. A start weight and end weight specify how the S-curve is formed.

Syntax:	ease(StartFrame, EndFrame, [StartWeight], [EndWeight])
Arguments:	■ StartFrame and EndFrame are the frames at which the transition starts and ends respectively. ■ StartWeight and EndWeight are numbers that specify how curvature is distributed at the start and end of the curve respectively. To get an S-curve, their sum should be less than 1, otherwise a square curve will result. In addition, if either of these two arguments are specified, the other must also be specified.
Examples:	■ ease(10, 50) * 100 returns a standard S-curve from 0 to 100between frames 10 and 50. ■ 80 - ease(1, 40) * 20 returns a standard S-curve from 80 to60 between frames 1 and 40. ■ ease(1, 30, 0.2, 0.2) * 50 + 10 returns a tight S-curve from10 to 60 between frames 1 and 30. ■ ease(10, 80) * 150 yields the following curve:



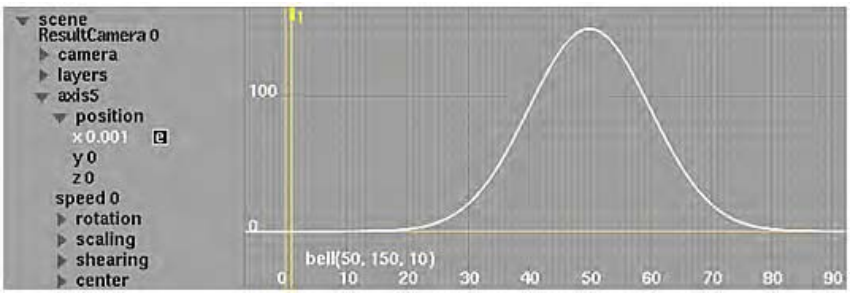
- ease(10, 80, 0.2, 0.2) * 150 yields the following curve:



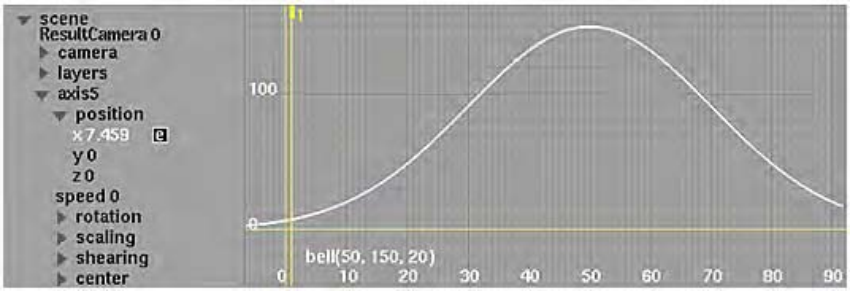
bell

Returns a set of values representing a bell (normal distribution) curve over time. You can specify the centre frame number and height and width characteristics of the curve.

Syntax:	bell(CentreFrame, Height, Width)
Arguments:	■ CentreFrame is the frame number at which the bell curve will reach its maximum height. ■ Height is the maximum height of the bell curve. ■ Width specifies the horizontal distribution of the curve.
Examples:	■ bell(10, 100, 3) returns a bell curve centred around frame10 with a maximum height of 100 and a width distribution of 3. ■ bell(50, 150, 10) yields the following curve:



- bell(50, 150, 20) yields the following curve:



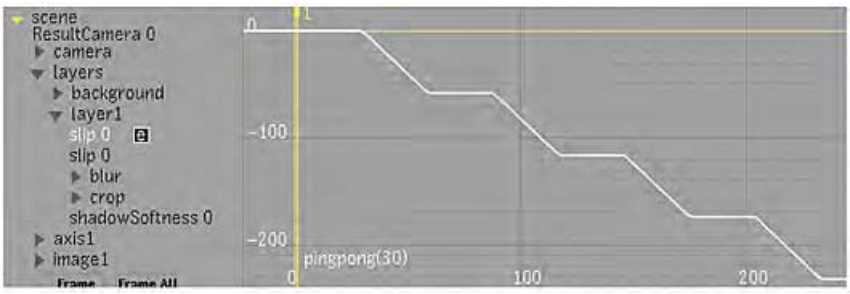
Slip Functions

Use the following functions to slip the starting frame of a clip backwards or forwards using the slip channel. In effect, the displayed frame in a layer's clip will be the animation's current frame number plus the slip value. For example, a slip value of -15 holds the clip at the first frame and repeats it 15 times before the clip begins. In this way, you can create a freeze frame effect with one layer while animating another layer. A slip value of 10 begins the clip at frame 11. See Modifying Surfaces (page 458).

NOTE Slip functions are actually implemented as user-defined functions in the functions file. See Defining Your Own Functions (page 1526).

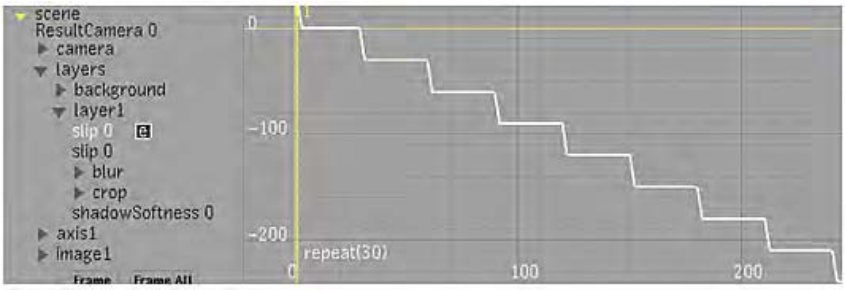
pingpong

Returns a slip value that makes a clip continuously cycle forwards and backwards. The result should usually be assigned to the slip channel of a layer.

Syntax:	pingpong(length)
Arguments:	■ length is the number of frames to display before starting to play backwards. This is usually the length of the original clip.
Examples:	■ pingpong(10) returns a slip value that makes a clip continuously play in a forward and backward cycle from frame 1 to frame 10. ■ pingpong(30) yields the following curve: 

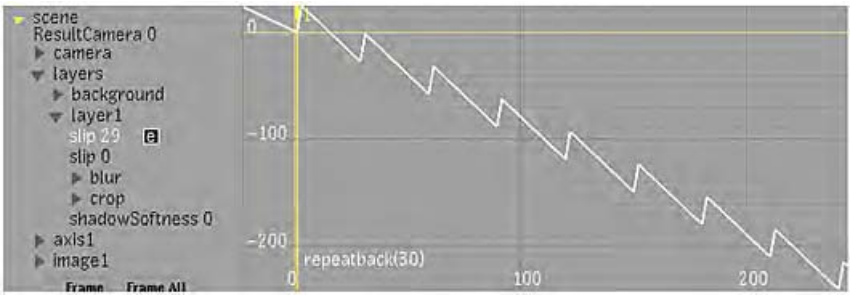
repeat

Returns a slip value that makes a clip continuously repeat forwards. The result should usually be assigned to the slip channel of a layer.

Syntax:	repeat(length)
Arguments:	■ length is the number of frames to display before repeating from the beginning. This is usually the length of the original clip.
Examples:	■ repeat(10) returns a slip value that makes a clip repeat forwards from frame 1 to frame 10. ■ repeat(30) yields the following curve: 

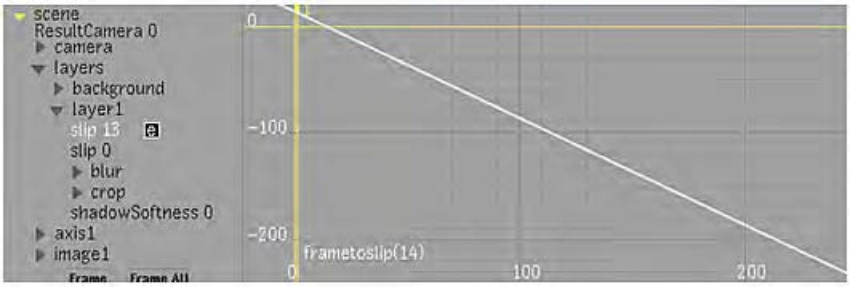
repeatback

Returns a slip value that makes a clip continuously repeat backwards. The result should usually be assigned to the slip channel of a layer.

Syntax:	repeatback(length)
Arguments:	■ length is the number of frames from the beginning to display backwards before repeating. This is usually the length of the original clip.
Examples:	■ repeatback(10) returns a slip value that makes a clip repeat backwards from frame 10 to frame 1. ■ repeatback(30) yields the following curve: 

frametoslip

Returns a slip value that makes a clip display at a given timebar frame. If the given frame number is constant, the resulting clips will freeze at that frame. The result should usually be assigned to the slip channel of a layer.

Syntax:	frametoslip(TimebarFrame)
Arguments:	■ TimebarFrame is the timebar frame at which to display the clip.
Examples:	■ frametoslip(10) returns a slip value that makes a clip freeze at frame 10. ■ frametoslip(frame / 2) returns a slip value that makes a clisplay at half the normal speed. ■ frametoslip(14) yields the following curve: 

Simple Mathematical Functions

The following functions are useful for performing various simple mathematical calculations.

abs

Returns the absolute value of a given number. The absolute value is the positive value of any number.

Syntax:	abs(Number)
Arguments:	■ Number is the number of which you want the absolute value.
Examples:	■ abs(3) returns 3. ■ abs(-3) returns 3.

sign

Returns the sign of a given number. The sign function returns 1 if the number is above or equal to zero, and returns -1 if less than zero.

Syntax:	sign(Number)

Arguments:	■ Number is the number of which you want the sign.
Examples:	■ sign(5) returns 1. ■ sign(0) returns 1. ■ sign(-0.001) returns -1.

pow

Returns a number raised to the power of an exponent.

Syntax:	pow(Number,Power)
Arguments:	■ Number is the base number to be raised. ■ Power is the exponent to which the base number is raised.
Examples:	■ pow(3, 2) returns 9. ■ pow(4, 3) returns 64. ■ pow(5, 0) returns 1. ■ pow(-2.5, 4) returns 39.0625.

sqrt

Returns the square root of a given number.

Syntax:	sqrt(Number)
Arguments:	■ Number is the non-negative number of which you want the square root.
Examples:	■ sqrt(25) returns 5. ■ sqrt(abs(-25)) returns 5. ■ sqrt(0) returns 0.

max

Returns the larger of two given numbers.

Syntax:	max(Number1, Number2)
Arguments:	■ Number1 and Number2 are numbers of which you want to find the maximum value.
Examples:	■ max(5.9, 8.1) returns 8.1. ■ max(-14, -32) returns -14.

	■ max(axis1.position.x, axis2.position.x) returns the larger of axis1.position.x or axis2.position.x.

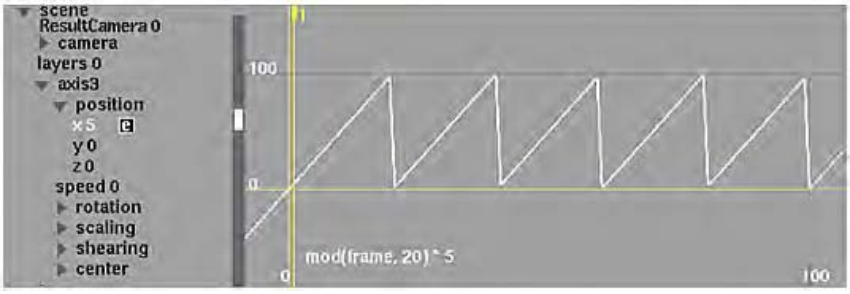
min

Returns the smaller of two given numbers.

Syntax:	min(Number1, Number2)
Arguments:	■ Number1 and Number2 are numbers of which you want to find the minimum value.
Examples:	■ min(5.9, 8.1) returns 5.9. ■ min(-14, -32) returns -32. ■ min(axis1.position.x, axis2.position.x) returns the smaller of axis1.position.x or axis2.position.x.

mod

Returns the integer remainder from dividing one number by another. This function is useful for repeating an animation every given number of frames.

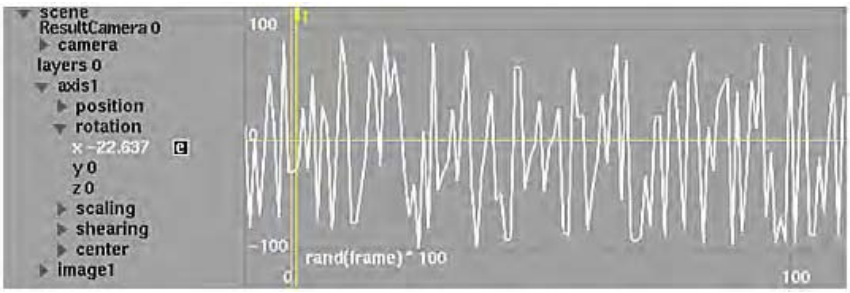
Syntax:	mod(Number, Divisor)
Arguments:	■ Number is the number to divide. ■ Divisor is the number by which you want to divide Number.
Examples:	■ mod(8, 3) returns 2 because 8 divided by 3 is 2 with 2 as the remainder. ■ mod(-8, 3) returns -2. ■ mod(8, -3) returns 2. ■ mod(-8, -3) returns -2. ■ mod(7.5, 2.25) returns 0.75. ■ mod(frame, 20) * 5 yields the following curve: 

Random Number Functions

Use the following functions to create curves based on random values.

rand

Returns a random value between -1 and 1, based on the value used as an argument, called the *seed*. Using the same seed in the rand function for multiple channels produces the exact same results. To create completely random results, use the true rand function.

Syntax:	rand(Seed)
Arguments:	■ Seed is the value used to generate the random return value. This is usually set to a changing value such as the current frame number.
Examples:	■ rand(frame % 10) * 50 + 50 returns a set of random values between0 and 100 that repeats every 10 frames. ■ rand(frame) * 100 returns a random value between -100 and 100for every frame in the animation. The following curve shows the result of this function: 

truerand

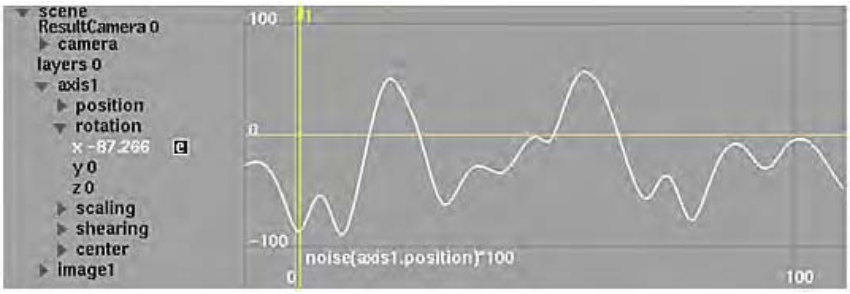
Returns a truly random value between two given numbers. The sequence of returned values will constantly change, never reproducing past results.

Syntax:	truerand(Low, High)
Arguments:	■ Low and High are the upper and lower bounds, respectively, of the random number to generate.
Example:	■ truerand(-5.5,10.8) returns a truly random value between -5.5and 10.8.

noise

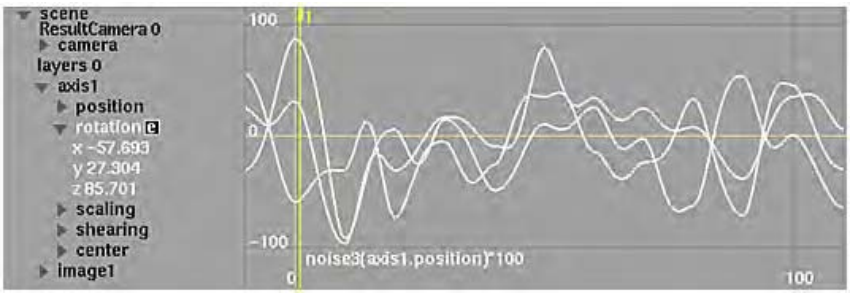
Returns a random value between -1 and 1, based on a given vector. If the parameter varies smoothly, this function will return a continuously changing value that also varies smoothly.

Syntax:	noise(Position)

Arguments:	■ Position is a vector used as a seed for the returned random value.
Examples:	■ noise(frame) * 5 returns a continuous random value between-5 and 5. ■ (noise(axis1.position) + 1) / 2 * 100 returns a continuous random value between 0 and 100. ■ noise(axis1.position) * 100 yields the following curve: 

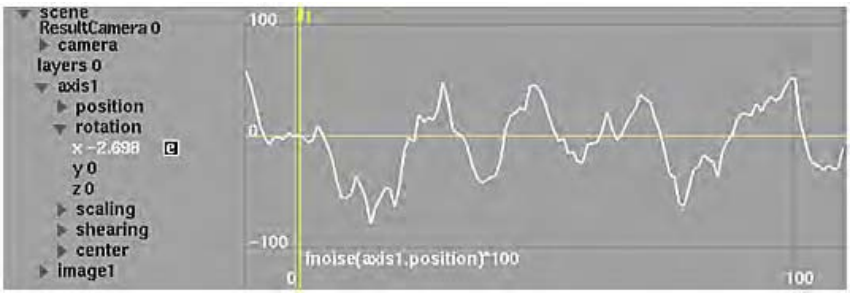
noise3

Returns a random vector for all elements in a vector between -1 and 1. If the parameter varies smoothly, this function will return a vector of continuously changing values that also vary smoothly.

Syntax:	noise3(Position)
Arguments:	■ Position is a vector used as a seed for the returned random vector.
Examples:	■ noise3(frame) * 5 returns a vector of continuous random values between -5 and 5. ■ (noise3(axis1.position) + 1) / 2 * 100 returns a vector of continuous random values between 0 and 100. ■ noise3(axis1.position) * 100 returns the following three curves: 

fnoise

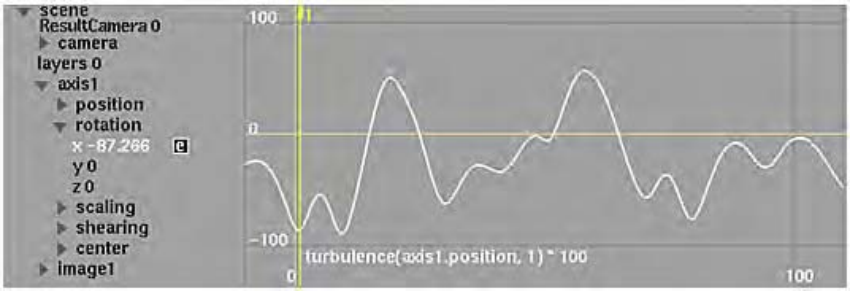
Returns a random value between -1 and 1, based on a given vector. If the parameter varies smoothly, this function will return a continuously changing value that also varies smoothly to a fractal pattern. This function is similar to the noise function, but it applies a fractal pattern to the result.

Syntax:	fnoise(Position)
Arguments:	■ Position is a vector used as a seed for the returned random value.
Examples:	■ fnoise(frame) * 5 returns a continuous random value between-5 and 5. ■ (fnoise(axis1.position) + 1) / 2 * 100 returns a continuous random value between 0 and 100. ■ fnoise(axis1.position) * 100 yields the following curve: 

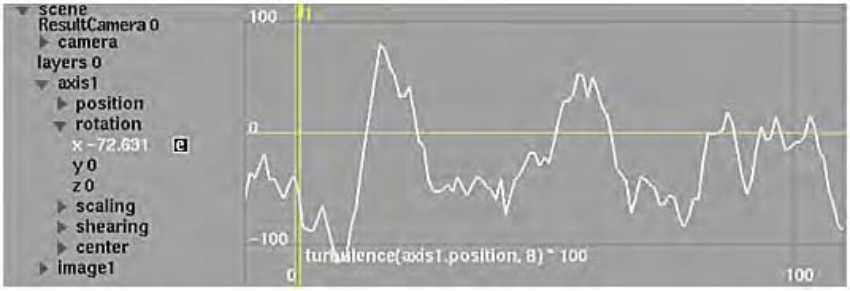
turbulence

Returns a random value between -1 and 1, based on a given vector and with the ability to control the level of smoothness for the resulting curve.

Syntax:	turbulence(Position, Level)
Arguments:	■ Position is a vector used as a seed for the returned random value. ■ Level is a positive integer where the greater the value, the greater the jitter applied to the resulting curve. If this argument is set to a value less than 1, the level will be treated as if 1 were used. If it is set to a number with a fractional component, the number will be rounded down to the nearest integer.
Examples:	■ turbulence(axis1.position, 1) * 100 yields the following curve:



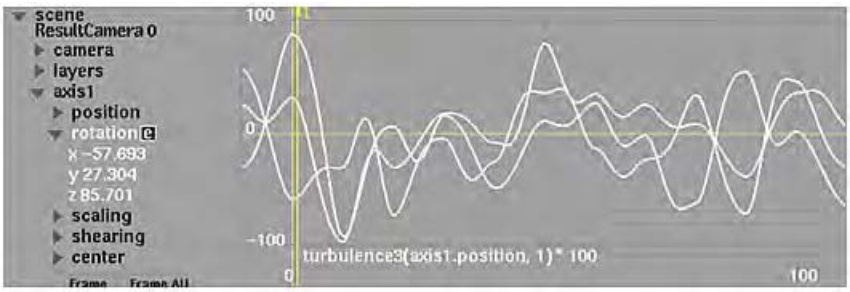
- turbulence(axis1.position, 8) * 100 yields the following curve:



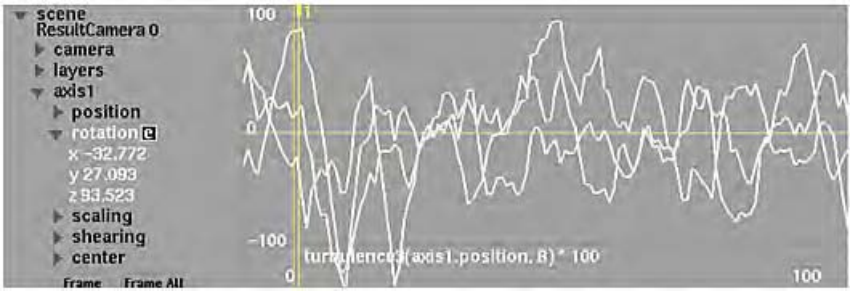
turbulence3

Returns a random vector for all elements in a vector between -1 and 1, along with the ability to control the smoothness of the resulting curve. If the position parameter varies smoothly, this function will return a vector of continuously changing values that also vary smoothly.

Syntax:	turbulence3(Position, Level)
Arguments:	■ Position is a vector used as a seed for the returned random vector. ■ Level is a positive integer used to control the level of jitter of the resulting curve. If this argument is set to a value less than 1, the level will be treated as if 1 were used. If it is set to a number with a fractional component, the number will be rounded down to the nearest integer.
Examples:	■ turbulence3(axis1.position, 1) * 100 returns the following three curves:



- turbulence3(axis1.position, 8) * 100 returns the following three curves:



Rounding Functions

The following functions are useful for performing various rounding calculations.

NOTE Although you can use these rounding functions as you would any other predefined function, they are actually implemented as user-defined functions in the functions file provided with the Flame Premium installation. See Defining Your Own Functions (page 1526).

round

Returns a number rounded to the nearest integer.

Syntax:	round(Number)
Arguments:	■ Number is the number to round.
Examples:	■ round(2.8) returns 3. ■ round(-2.8) returns -3. ■ round(2.3) returns 2.

ceil

Rounds a number up to the next integer value regardless of its value.

Syntax:	ceil(Number)
Arguments:	■ Number is the number to round up.
Examples:	■ ceil(2.8) returns 3. ■ ceil(-2.8) returns -2. ■ ceil(4) returns 4.

floor

Rounds a number down to the nearest integer regardless of its value.

Syntax:	floor(Number)
Arguments:	■ Number is the number to round down.
Examples:	■ floor(2.3) returns 2. ■ floor(-2.3) returns -3. ■ floor(4) returns 4.

trunc

Returns the integer value of a number by truncating its fractional part.

Syntax:	trunc(Number)
Arguments:	■ Number is the number that you want to truncate.
Examples:	■ trunc(3.8) returns 3. ■ trunc(-3.8) returns -3. ■ trunc(PI) returns 3.

Trigonometric Functions

The following functions are useful for working with angles and performing various trigonometric calculations.

degrees

Converts angle units from radians into degrees.

Syntax:	degrees(Angle)
Arguments:	■ Angle is the angle in radians that you want to convert.
Examples:	■ degrees(PI) returns 180. ■ degrees(PI/2) returns 90.

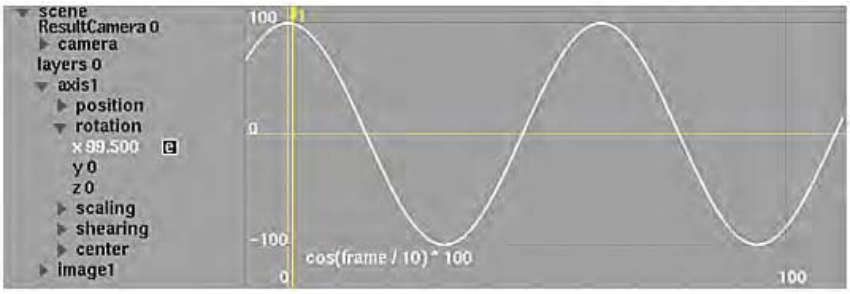
radians

Converts angle units from degrees into radians.

Syntax:	radians(Angle)
Arguments:	■ Angle is the angle in degrees that you want to convert.
Example:	■ radians(225) returns 3.927 (5*PI/4).

cos

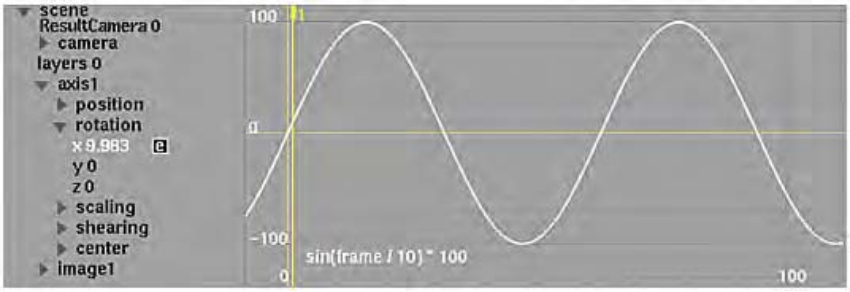
Returns the cosine of a given angle.

Syntax:	cos(Angle)
Arguments:	■ Angle is the angle in radians of which you want the cosine.
Examples:	■ cos(0) returns 1. ■ cos(PI / 3) returns 0.5. ■ cos(frame / 10) * 100 yields the following curve: 

sin

Returns the sine of a given angle.

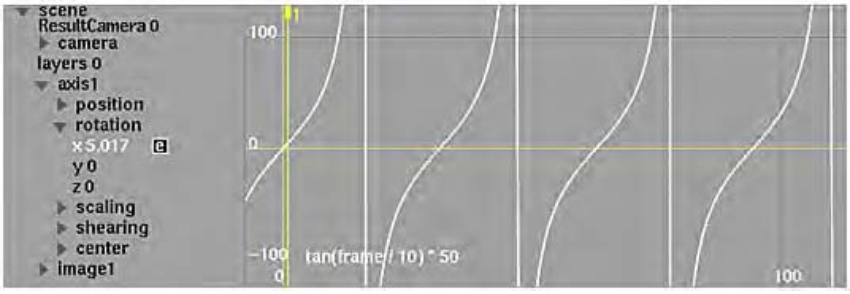
Syntax:	sin(Angle)
Arguments:	■ Angle is the angle in radians of which you want the sine.
Examples:	■ sin(0) returns 0. ■ sin(PI / 6) returns 0.5. ■ sin(frame / 10) * 100 yields the following curve:



tan

Returns the tangent of a given angle.

Syntax:	tan(Angle)
Arguments:	■ Angle is the angle in radians of which you want the tangent.
Examples:	■ tan(0) returns 0. ■ tan(PI / 4) returns 1. ■ tan(PI / 3) returns 1.7321. ■ tan(frame / 10) * 50 yields the following curve:



acos

Returns the arccosine—the inverse function of the cosine—of a given number. The returned angle is given in radians within the range 0 to PI.

Syntax:	acos(Number)
Arguments:	■ Number is the cosine of the angle you want and must be between-1 and 1.
Examples:	■ acos(0.5) returns 1.0472 (PI/3 radians). ■ degrees(acos(0.5)) returns 60.

asin

Returns the arcsine—the inverse function of the sine—of a given number. The returned angle is given in radians within the range -PI/2 to PI/2.

Syntax:	asin(Number)
Arguments:	■ Number is the sine of the angle you want and must be between-1 and 1.
Examples:	■ asin(0.5) returns 0.5236 (PI/6 radians). ■ degrees(asin(0.5)) returns 30.

atan

Returns the arctangent—the inverse function of the tangent— of a given number. The returned angle is given in radians within the range -PI/2 to PI/2.

Syntax:	atan(Number)
Arguments:	■ Number is the tangent of the angle you want.
Examples:	■ atan(1) returns 0.7854 (PI/4 radians). ■ degrees(atan(1)) returns 45.

atan2

Returns the arctangent of y/x, using the signs of both arguments to determine the quadrant of the return value. The arctangent is the angle from the origin to the vector (x,y). The returned angle is given in radians within the range -PI to PI.

Syntax:	atan2(x, y)
Arguments:	■ x and y are the components of the vector to be used in the function.
Examples:	■ atan2(1, 1) returns 0.7854 (PI/4 radians). ■ atan2(-1, -1) returns -2.3562 (-3*PI/4 radians). ■ atan2(1, 0) returns 1.5708 (PI/2 radians). ■ degrees(atan2(1,1)) returns 45.

Vector Functions

Use the following functions for performing various vector-related operations.

length

Returns the euclidean length (magnitude) of a given vector. The euclidian length is equivalent to the expression sqrt(pow(Vector.x, 2) + pow(Vector.y, 2) + pow(Vector.z, 2)).

Syntax:	length(Vector)
Arguments:	■ Vector is the vector of which you want the euclidean length.
Examples:	■ length((2, 0, 0)) returns 2. ■ length((1, 1, 0)) returns 1.4142. ■ length((-1, -1, -1)) returns 1.7321. ■ length(axis1.position - axis2.position) returns the distance between axis1 and axis2.

dot

Returns the scalar dot-product of two given vectors. The *dot-product* is the product of the lengths of two vectors and the cosine of the angle between them. If the two vectors are at a right angle (90 degrees), their dot-product is 0.

If the product of their lengths equals 1 and they point in opposite directions (180 degrees), their dot-product is -1. The dot-product is equivalent to the expression V1.x * V2.x + V1.y * V2.y + V1.z * V2.z.

Syntax:	dot(V1, V2)
Arguments:	■ V1 and V2 are the vectors of which you want the dot-product.
Examples:	■ dot((1, 1, 0), (0, 0, 1)) returns 0. ■ dot((2, 0, 0), (0.5, 0, 0)) returns 1. ■ dot((0, 2, 0), (0, -0.5, 0)) returns -1. ■ dot((2, 0, 1), (4, 5, 5)) returns 13.

cross

Returns the vector cross-product of two given vectors. The cross-product is the vector perpendicular to the plane containing the two vectors. In effect, there will be a right angle between the returned vector and the first given vector, as well as a right angle between the returned vector and the second given vector. The length of the resulting vector is equal to the product of the two vectors and the sine of the angle between them. The cross-product is equivalent to the vector (V1.y * V2.z - V1.z *V2.y, V1.z * V2.x - V1.x * V2.z, V1.x * V2.y - V1.y * V2.x).

Syntax:	cross(V1, V2)
Arguments:	■ V1 and V2 are the vectors of which you want the cross-product.
Examples:	■ cross((1, 0, 0), (0, 1, 0)) returns (0, 0, 1).

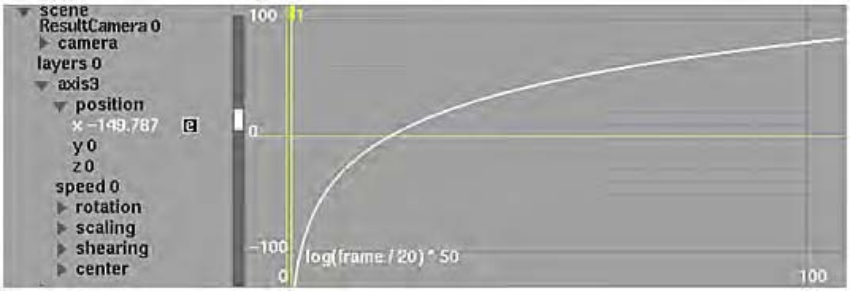
	■ cross((1, 1, 0), (0, 1, 1)) returns (1, -1, 1). ■ cross((2, 0, 0), (0, 0.5, 0)) returns (0, 0, 1).

Logarithmic Functions

Use the following functions for performing various logarithmic calculations.

log

Returns the natural logarithm of a given number. The log function is the inverse of the exp function.

Syntax:	log(Number)
Arguments:	■ Number is the positive number of which you want the natural logarithm.
Examples:	■ log(1) returns 0. ■ log(2) returns 0.6931. ■ log(exp(5)) returns 5. ■ log(256) / log(2) returns 8. ■ log(frame / 20) * 50 yields the following curve: 

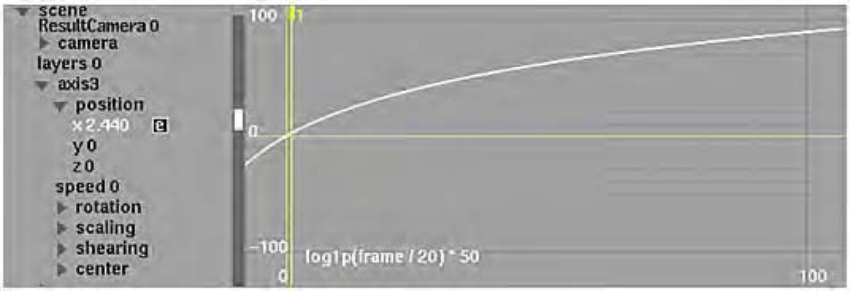
log10

Returns the base-10 logarithm of a given number.

Syntax:	log10(Number)
Arguments:	■ Number is the positive number of which you want the base-10logarithm.
Examples:	■ log10(1) returns 0. ■ log10(10) returns 1. ■ log10(100) returns 2.

log1p

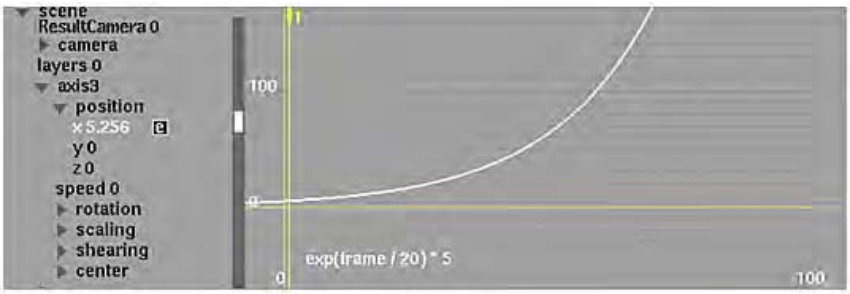
Returns the natural logarithm of 1 plus a given number. The log1p function is the inverse of the expm1 function.

Syntax:	log1p(Number)
Arguments:	■ Number is the positive number less 1 of which you want the natural logarithm.
Examples:	■ log(0) returns 0. ■ log(1) returns 0.6931. ■ log1p(expm1(5)) returns 5. ■ log1p(frame) returns the equivalent of log(1+frame). ■ log1p(frame / 20) * 50 yields the following curve: 

exp

Returns the constant *e* (the base of the natural logarithm) raised to the power of a given number. The exp function is the inverse of the log function.

Syntax:	exp(Number)
Arguments:	■ Number is the exponent applied to the base *e*.
Examples:	■ exp(0) returns 1. ■ exp(1) returns 2.7182. ■ exp(2) returns 7.3890. ■ exp(log(5)) returns 5. ■ exp(frame / 20) * 5 yields the following curve:



expm1

Returns the constant *e* (the base of the natural logarithm) raised to the power of a given number, minus 1. The expm1 function is the inverse of the log1p function.

Syntax:	expm1(Number)
Arguments:	■ Number is the exponent applied to the base *e*.
Examples:	■ expm1(0) returns 0. ■ expm1(1) returns 1.7182. ■ expm1(2) returns 6.3890. ■ expm1(log1p(5)) returns 5. ■ expm1(frame) returns the equivalent of exp(frame) - 1.

Defining Your Own Functions

Flame Premium allows you to define your own functions and use them like any other predefined function. By defining functions in terms of existing functions, you can simplify the creation of complex expressions in your animations.

NOTE You should have a good understanding of how to use functions in expressions before you define your own functions.

The Functions File

There are two text files in which you can enter user-defined functions:

- Project-specific file: */usr/discreet/project/<user_name>/expressions/userfun.expressions*
- Template file: */usr/discreet/<product_name>/expressions/userfun.expressions*

To define a function for your current project, add it to the project-specific file. Modifying this file will affect only that project and not any other existing or future projects.

If you decide that you want to use the function in future projects, copy it from the project-specific file to the template file. When you create a project, the template file is copied to your project directory as the

project-specific file for that new project. This functionality allows you to modify user-defined functions in new projects without affecting the behaviour of your animations in older projects.

To edit the functions file:

1. Open a command window.
2. Use the mouse to position the cursor in the shell and type one of the following commands, depending on which file you want to edit.

Enter:	To:
nedit /usr/discreet/project/<user_name>/expressions/user-fun.expressions	Edit the project-specific file.
nedit /usr/discreet/<product_name>/expressions/userfun.expressions	Edit the template file.

   The functions file opens in a shell and you can edit it as you would any other text file.

Defining a Function

A function is composed of a function name, with the list of arguments, and the expression that forms its definition. You start a function by its name, followed by an opening parenthesis, the arguments for the function separated by commas, and finally a closing parenthesis. The following rules apply for function and argument names:

- The first character of a function name must be a letter, and subsequent characters can only contain letters or numbers. For example, sin100 is a valid function name, although 100sin is not.
- The first character of an argument must be a dollar sign ($), the second character must be a letter, and subsequent characters can only contain letters or numbers. For example, $arg1 is a valid argument name, although $1 and arg3 are not.
- Function names and arguments are case sensitive, meaning that lowercase and uppercase make a difference.

The function name and its definition are separated by a colon (:). The definition itself can be any valid expression, except that channel names are not accepted since user-defined expressions are not related to any particular setup. In the definition, the arguments can appear, complete with the dollar sign preceding it. Finally, the definition must end with a semicolon (;) and it can span several lines.

NOTE Errors in the functions file will be reported when it is first read. This file is read every time that an unknown function is used in an expression.

Comments

You can include comments in the functions file by starting the line with a number sign (#). The comments extend through the end of the line on which they appear and are ignored by Flame Premium when the user-defined functions file is interpreted. Comments are useful for describing the purpose of a function and adding any notes you feel are pertinent. They can be introduced anywhere in the file, except for inside of function definitions that span several lines.

Sample Function Definitions

The following sample functions are found in the functions file that is installed with Flame Premium.

- This function creates a sine curve for a given channel that oscillates between 0 and 100. The sin100 function takes one argument, named $arg1, that it uses in conjunction with the predefined sin function. The return value for sin100 is the sine of $arg1 times 50, plus 50.

  sin100($arg1) : sin($arg1)*50 + 50;

- This function creates a sine curve based on the specified frequency and amplitude. The sinfreqamp function takes three arguments, named $pos, $freq and $amp.

  sinfreqamp($pos,$freq,$amp) : sin($pos*$freq)*$amp;

- This function provides an approximation of the speed, or more precisely, the derivative (rate of change), of a given channel. The speed function takes one argument, named $channelName.

  speed($channelName) :eval($channelName,frame+0.1) -

  eval($channelName,frame-0.1))/0.2;

NOTE For information on the other example functions in the functions file, see Slip Functions (page 1509) and Rounding Functions (page 1518).

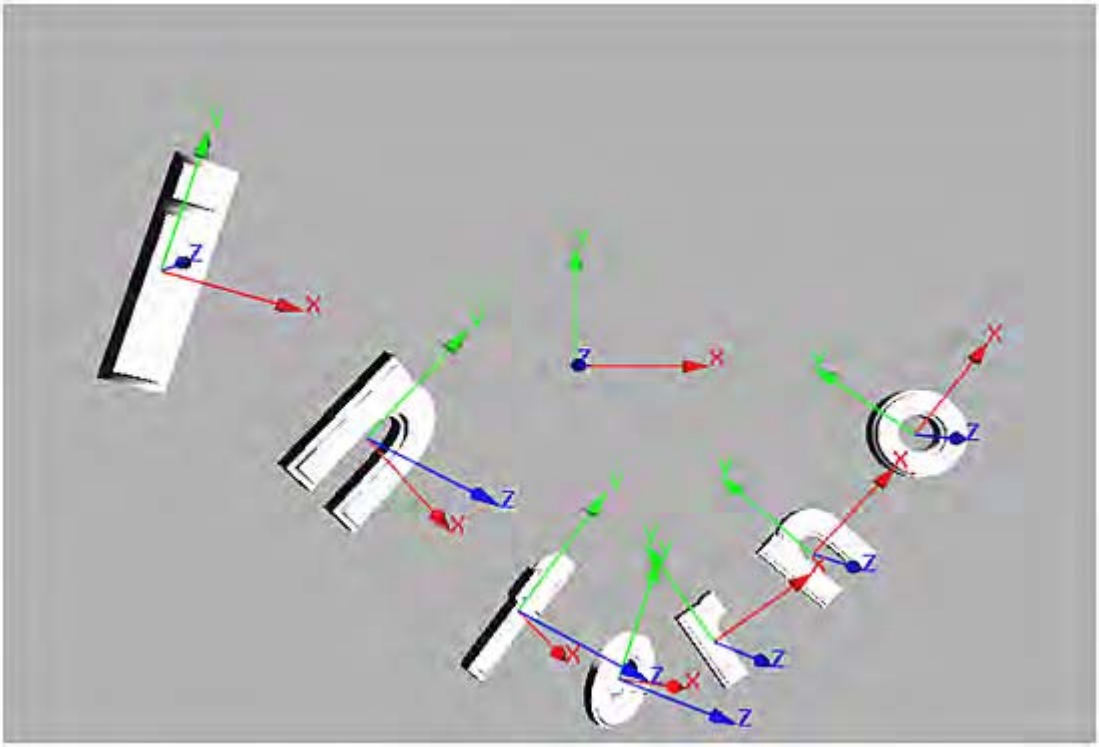
Sample Expression Setups

A few Action setups with sample expressions are provided in the directory */usr/discreet/<product_name>/examples/action*.

To load an setup file from Action accessed from the Tools tab, click the Load button in the Action menu. To load an Action setup from Batch or Batch FX, select the Action node in the schematic, then click Node Prefs and Load Node.

Animated Text

To see this example, load the example setup named *expressions_text* into Action. This example involves multiple text objects that move along a path, with each letter following slightly behind the last.



Each letter's position and rotation are created through the expressions in the following folder. The letter 'o' rotates its X-axis towards the direction of its movement. The align function is used to generate the rotation vector.

Each preceding letter sets its position and rotation an increasing number of frames behind the letter 'o'. The channel dummy_axis.position.x is used here to store a value determining the distance between each letter. The eval function is used to determine the values of the o_axis.position and o_axis.rotation channels at different points in time.

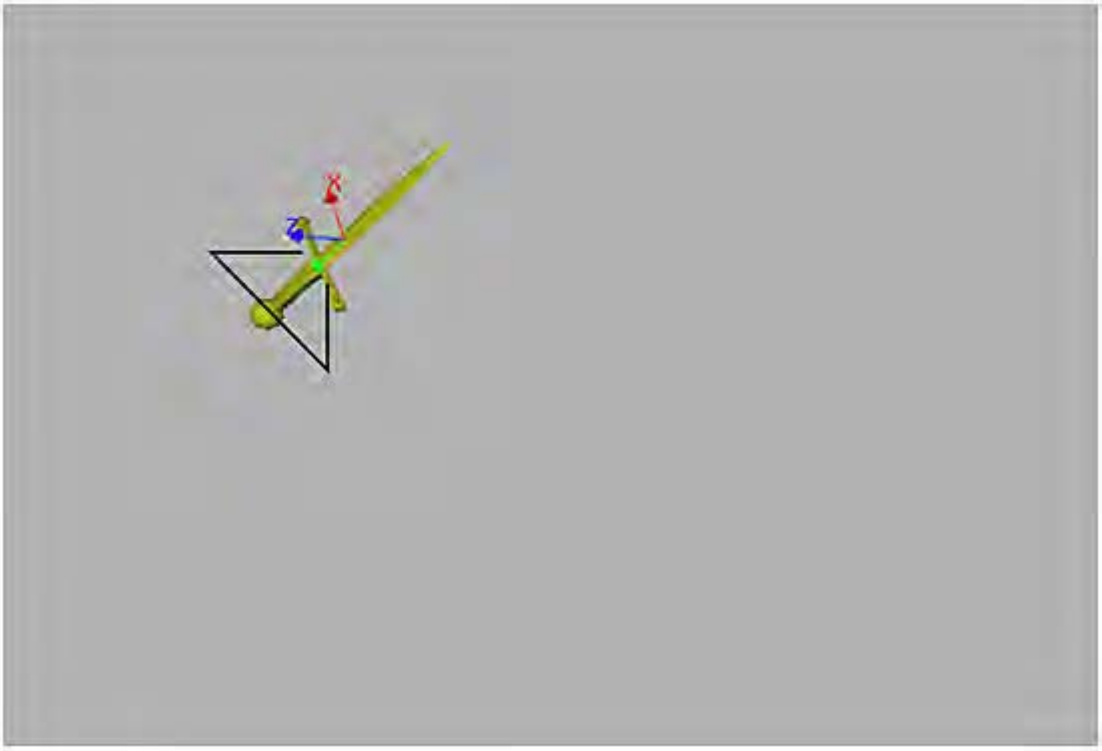
NOTE A series of expressions that differ only by a numeric value, such as in this example, can be entered quickly using the SelectionOrder keyword. See Cascading Expressions (page 1499).

Channel	Expression
o_axis.rotation	align(o_axis.position)
n2_axis.position	eval(o_axis.position, frame - dummy_axis.position.x * 1)
r_axis.position	eval(o_axis.position, frame - dummy_axis.position.x * 2)
e_axis.position	eval(o_axis.position, frame - dummy_axis.position.x * 3)
f_axis.position	eval(o_axis.position, frame - dummy_axis.position.x * 4)
n_axis.position	eval(o_axis.position, frame - dummy_axis.position.x * 5)
i_axis.position	eval(o_axis.position, frame - dummy_axis.position.x * 6)

Channel	Expression
n2_axis.rotation	eval(o_axis.position, frame - dummy_axis.position.x * 1)
r_axis.rotation	eval(o_axis.position, frame - dummy_axis.position.x * 2)
e_axis.rotation	eval(o_axis.position, frame - dummy_axis.position.x * 3)
f_axis.rotation	eval(o_axis.position, frame - dummy_axis.position.x * 4)
n_axis.rotation	eval(o_axis.position, frame - dummy_axis.position.x * 5)
i_axis.rotation	eval(o_axis.position, frame - dummy_axis.position.x * 6)

Spinning Dagger

To see this example, load the setup named *expressions_spinning_dagger* into Action. This example features a dagger moving in an arc. The dagger's Y-axis is aligned with the direction of its movement and the rest of the dagger spins around its axis.

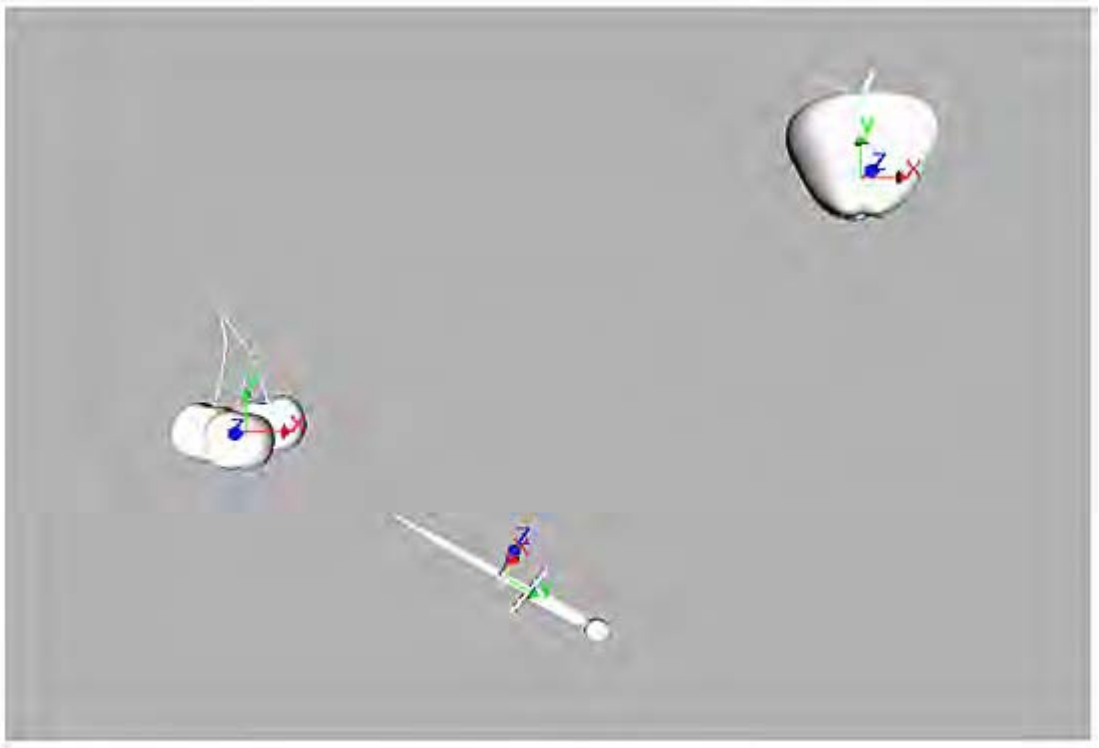


The dagger's rotation is created through the dagger's rotation of its negative Y-axis towards the direction of its movement and the rest of the dagger spins one full rotation around this same axis every 32 frames. The align function is used to generate the rotation vector.

Channel	Expression
dagger_axis.rotation	align(dagger_axis.position, (0,-1,0), frame * 2 * PI / 32)

Dagger, Apple, and Cherry

To see this example, load the setup named *expressions_dagger_apple_cherries* into Action. This animation points a dagger at either a moving apple or moving cherries, whichever is closer in position at the time.

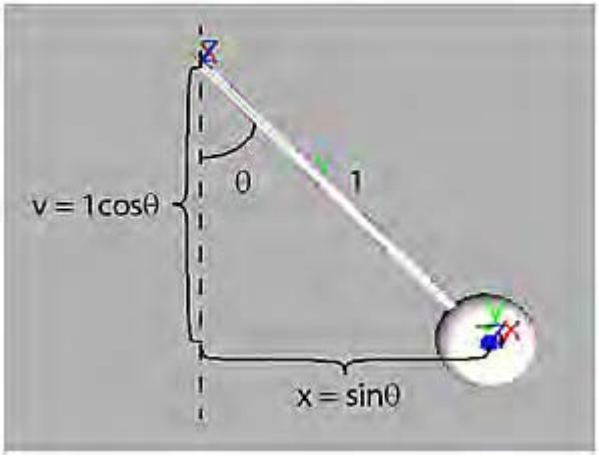


The dagger's rotation is created by the dagger's position as the dagger points its negative Y-axis in the direction of either the apple or the cherries, whichever happens to be closest. The length function is used to find the distance between the dagger and the other objects, while the if function is used to determine which is the shorter distance. The lookat function is used to generate the rotation vector.

Channel	Expression
dagger_axis.rotation	if(length(cherries_axis.position - dagger_axis.position) <length(apple_axis.position - dagger_axis.position), lookat(cherries_axis.position, dagger_axis.position, (0,-1,0), (1,0,1)), lookat(apple_axis.position, dagger_axis.position, (0,-1,0), (1,0,1)))

Dampened Pendulum

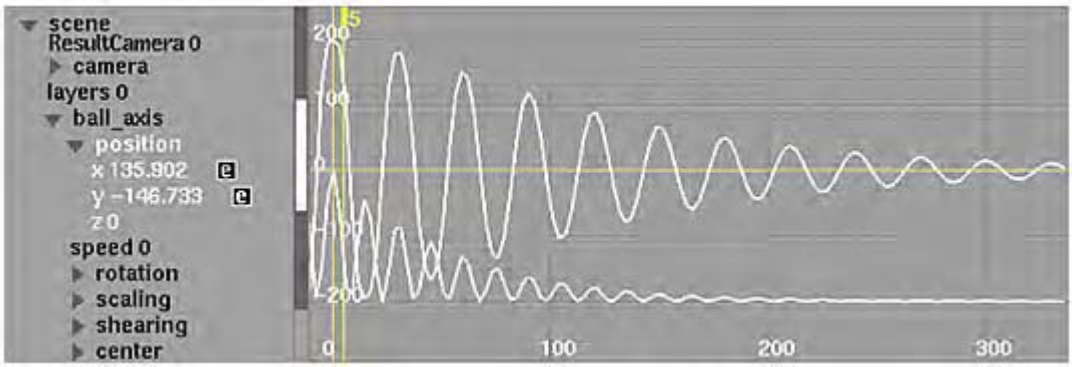
To see this example, load the setup named *expressions_dampened_pendulum* into Action. This examples involves a pendulum (a ball attached to a rope) that swings freely back and forth under the force of gravity until it comes to a stop.



The ball oscillates in a dampened harmonic motion that is created by animating the position of the ball and the rotation of the rope. The ball's x position is defined as the sine of the swing angle between the rope and centre axis, multiplied by the length of the rope. This angle is initially PI/2 and decreases logarithmically. In addition, the ball oscillates with a period of 30 frames. The sin function is used to calculate the sine of the swing angle and the exp function is used to create the logarithmic dampening effect. The cos function is used to create the oscillation.

The ball's y position is defined as the negative cosine of the swing angle between the rope and centre axis, multiplied by the length of the rope. This expression is formed similarly to that of the ball_axis.position.x channel. The rope rotates such that its Y-axis is always pointing toward the centre of the ball. The lookat function is used to generate the rotation vector.

Channel	Expression
ball_axis.position.x	sin(PI / 2 / exp(frame / 100) * cos(frame * 2 * PI / 30)) *200
ball_axis.position.y	-cos(PI / 2 / exp(frame / 100) * cos(frame * 2 * PI / 30))* 200
rope_axis.rotation	lookat(ball_axis.position, rope_axis.position, (0,1,0), (0,0,1))



Modifying Clips with Clip History

27

To create clip history while rendering:

1. In the Preferences menu, click the General tab.
2. Under Default Rendering Options, select Render & Create History in the Render box.

   NOTE In Batch and Tools editors, you also have this option in the Render box, but by setting it in the Preferences menu, you can make sure that history is enabled for all renders (including tools that render a result directly in the workspace without entering the editor).

To view clip history in the timeline:

1. Select a source or sequence clip with history.
2. From the Timeline View Mode box, select History.

   The view changes to a history view of the selected clip. Batch FX and Timeline FX applied to a clip are included in its history. From the history view you can see and edit all of the effects or tools applied to a clip. Double-click effects to re-enter the editor, or use the right-click context menu for other editing options.



   TIP To quickly enter the history view, double-click the "H" icon on a clip on the desktop. To bypass history view, and quickly re-enter the last tool used on a desktop clip with history, press `Shift` and double-click the “H” icon on the clip.

To modify clip history through History View:

1. Display the clip history of a clip.
2. Double-click a node (in schematic view) or an entry (in list view) to open the corresponding tool or Timeline FX editor.
3. Edit the settings as needed.
4. If you loaded the clip into a tool editor, do one of the following:
   - To save your changes, select Return from the Exit Options box.
   - To exit without saving your changes, select Cancel from the Exit Options box, and then confirm.

5 If you loaded the clip into a Timeline FX editor, click Exit.

To display History View in a Timeline FX editor:

1 Load a clip with clip history into a Timeline FX editor.

2 Click History.

The clip history appears with the Timeline FX selected. You can display the history in Schematic or List View.

To replace sources in a clip's history:

1 Display the history of a clip.

2 Do one of the following.

Do this:	To swap:
Double-click a source	The original source with another.
Press `Ctrl` and double-click a tool node	All of the sources used in a tool.

The cursor becomes a coloured crosshair to indicate what type of source you can select.

3 Click the source clips you want to swap in to the clip history. The sources you swap must be of the same resolution and bit depth.

NOTE If the new source is longer than the original source, the new source is shortened to fit the duration of the original source. If the new source is shorter that the original, the last frame of the new source is repeated to the end of the segment.

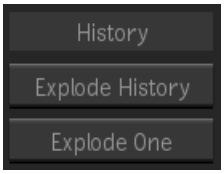
The sources are swapped and the Render button becomes active.

4 Click Render to render the swapped sources.

To explode clip history in Batch or Batch FX:

1 Double-click a clip with history in the Batch or Batch FX schematic to view its clip settings.

2 In the History section, select Explode History to expand the entire clip history into the process tree, or Explode One to expand only the most recent step of the clip's history.



If the input clip to the last node also contains history, you can also expand the history for that clip.

NOTE These history options are not available for Batch or Batch FX clips with non-supported effects (the clip appears with a lowercase "h" next to its name in the schematic). In this case, if you double-click the clip again, you are returned to the timeline view. You can switch to History view to see its clip history.

About Clip History

Use clip history to track and update operations you have applied to a clip. The Timeline History view expands a clip, providing a view of your operations. You can also use the clip history in Batch or Batch FX as an

access point for modifying clips. Clips with history have a "H" icon when displayed on the desktop. In the Batch or Batch FX schematic, a [H] appears beside the clip name under the clip proxy.



Desktop clip with history



Batch clip with history

Certain tools, including those that do not have an equivalent Batch node, such as Warper or Reverse, appear with a lowercase "h" icon when rendered. In this case, you can view the clip's history in the History view, but not make changes to that tool, or any upstream tools in the pipeline. To edit these effects, you need to rebuild the effect in the tool originally used to create it. You can use the Match button in the timeline History view to help you locate the clip to its source in the workspace.

Clips that have a clip history contain sources and intermediates. Sources are any of the original clips used to build an effect, for example, clips captured using an EDL. Intermediates are clips that are created when building a clip that uses multiple effects or operations.

A simple way of using clip history is to load a rendered clip back into the last editor that was used to create the clip. For instance, you load a clip into the Colour Corrector, modify the colour, and render a new clip. You decide that you want to change the colour slightly, so you press `Shift` and double-click the “H” icon of the new clip to go back into the Colour Corrector with the settings restored. You tweak the values slightly, and then render another clip (which has its own clip history).

If multiple tools were used to create a clip, you can load the result into a Batch setup, explode the history, and make modifications at any point in the process tree. For instance, you may have created a composite by bringing a colour-corrected back layer into the Keyer with a front and key-in layer that may also have been modified (for example, with a Flip). By loading the resulting clip into Batch, you can modify it at any point in the process tree, and then process a new clip.

NOTE You can also use a Create BFX clip (page 385) instead of, or in addition to, clip history for added flexibility in reusing or versioning complex pipelines.

Clip History Tips

- In the Batch or Batch FX preferences Rendering tab, you can enable Fill with Intermediates in the Cache section to so that each node's cache is filled with the intermediate clips contained in the history. Filling a history tree's cache increases interaction speed and avoids the need to reprocess effects that were already rendered. If an intermediate clip corresponding to a step in the history is deleted, the cache for that particular node is not filled.
- If you modify and then render an intermediate that is part of a clip history, all the clip history elements after the rendered event are also rendered. Additionally, all instances of the intermediate used in the clip history are modified. If the same intermediate appears in another clip history, however, it is not modified — once you modify an intermediate in a clip history, any copies you made of the intermediate in other clip histories are discrete.
- If you have a Paint or Stabilizer element in your clip history, all modules that precede the Paint or Stabilizer module will not be editable. You can, however, modify the elements before the Paint or Stabilizer element by selecting the element immediately preceding the Paint or Stabilizer element in the history, and then using Match to create a clip with the same history in the Source Area. You can then use the result clip from the match. The history is expandable and editable.

- The duration of a clip created in the Paint tool is not retained. For example, enter the Paint tool with None selected, and then create a scene that is 30 frames. Then go into a tool such as Action and make some changes. When you view the clip history and click the Paint tool element, the clip will contain one frame only.
- When you enter a tool from clip history, in and out points mark the duration in the source clip that was used in the clip history (unless the entire clip was used). If you then make a modification and process the result, only the duration of the clip marked by in and out points is rendered, unless you park the cursor before the in point. If the cursor is parked before the in point, media is added to the head frames, starting from the cursor position.
- Note the following when accessing the history of Action clips:
  - When accessing the history of stereo output, you can change either the left or right eye output or both.
  - When accessing the history of clips containing multiple outputs, you can change the output enabled in the Action Output List but you can only have one output enabled.
  - You cannot change an Output mode to Stereo if the previous output was not stereo.

Clip History Reference

Timeline History View

History View box Select a list or schematic representation of the clip's history.

History box Select to display the history of the selected source or sequence clip.

Render button Click to render sources that were replaced through the clip's history.

Match button Click to match the clip to its source in the workspace.

History View horizontal scroll bar Drag the horizontal scroll bar to pan the schematic.

History View vertical scroll bar Drag the vertical scroll bar to pan the schematic up or down.

Home button Returns the schematic to its default position.

Current Timecode field Displays the timecode of the current position. Editable.

Clip History Media Management Options

These settings can be found in the Timeline Gear menu's Commit section.

Delete History button Click to delete intermediates from selected timeline elements. If no elements are selected, intermediates for the entire edit sequence are deleted. Use when you want to free up disk space or shorten the time it takes to perform a wire transfer or archive clips with clip history.

Commit History button Click to commit a clip with history. When committing a clip, you delete the clip history without removing the clip or element's soft properties.

Clip History Preferences

These history-related preferences can be found in **Preferences ➤ General**.

Render box Select whether you want to include history and/or Burn when rendering.

Keep History Mode box Select whether Sources or Intermediates and Sources appear on the workspace when working with clip history.

Batch or Batch FX History Settings

These history options are available in the clip settings of a clip with history.

Explode History button Click to expand the entire clip history into the rendering tree.

Explode One button Click to expand only the most recent step of the clip's history. If the input clip to the last node also contains history, you can also expand the history for that clip.

The following setting is available in the Batch or BFX Prefs Rendering section, under Cache.

Fill with Intermediates button Enable to fill each node's cache with the intermediate clips contained in the history. If a clip was rendered in proxy mode, the history is not cached.

Colour Management 28

Colour management enables the desired colour values to be represented as accurately as possible throughout the finishing process. It is different from colour correction in that its intention is not to change colours but instead to preserve the perceived colour in different situations.

NOTE Colour management is performed with the LUT Editor and other controls that are labelled "LUT", but other operations including colour transforms and gamma functions are supported in addition to look-up tables.

Colour Management Workflow

Colour management works by applying colour transforms and other operations at various stages of the pipeline. Typically, you use colour management when:

- Importing media. Images from different sources (such as digital cameras, scanned film, 3D renders, and matte paintings) often use different colour spaces and encodings, and they should be converted to a single working colour space.
- Outputting media. Different output formats require different encodings depending on the medium and expected viewing conditions.

You might also want to apply colour transforms to convert into and out of a specific colour space in order to perform a particular operation. For example, operations like compositing or lens blur work best in a linear colour space, while operations like tracking and edge detection work best in video or log colour space.

In addition, colour management is used to display images in Flame Premium to ensure that the colours on the monitor and projector match what will be displayed on the final output as closely as possible. For example, if you are working in the ACES colour space, you can apply the ACES_to_current-monitor transform.

Applying Colour Management to Clips

Use the LUT Editor to apply colour management to clips. This changes the underlying colour values of the clips' pixels. You should do this when you want the changes to flow though the pipeline to the final output. If you want to apply colour management to the display only, see Applying Colour Management to Displayed Images (page 1556) instead.

Accessing the LUT Editor

You can access the LUT Editor from Batch, from Batch FX, from the Tools tab, from the Import menu and from the Export menu.

To access the LUT Editor from the Import menu, see Importing File-Based Media (page 51).

To access the LUT Editor from the Export menu, see Exporting Media to Files (page 52).

Accessing the LUT Editor in Batch or Batch FX

You can use the LUT Editor from the LUT Editor, Read File, Write File and Render nodes in Batch and the BFX Output node in Batch FX to import or create a LUT or colour transform.

To access the LUT Editor using the LUT Editor node:

1. Drag the LUT Editor node to the schematic.
2. Parent the output of another node to the front input socket of the LUT Editor node.



3. Select the LUT Editor node to view its menu.

To access the LUT Editor as an integrated part of clip input and output processes

1. Select the BFX Output node in Batch FX or drag the Read File, Write File, or Render node to the schematic.
2. Set up the node so the LUT you create is applied to the clip that you want to convert:
   - If you selected the BFX output node in Batch FX, double click the node to display the LUT Editor menu.
   - If you dragged the Read File to the schematic, use the file browser to locate the clip that you want to import.
   - If you dragged a Write File or Render node to the Batch schematic, parent outputs from the process tree you are working on to the input socket(s) of the node.
3. Select the clip.
4. Select the RGB LUT or Matte LUT menu.



5. Enable Active to apply the current LUT settings to the front clip.



NOTE Active is always enabled if the bit-depth needs to be changed.

The LUT Editor menu is displayed.

TIP The RGB LUT button in the Basic menu of the Write File and Export nodes is linked to the Active button in the RGB LUT menu. Toggling either button enables the LUT.

Accessing the LUT Editor from the Tools Tab

1. Click the Tools tab.
2. From the Tools menu, select the Look tab.
3. Click Lut Editor.

   The cursor turns red, prompting you to select a front clip.
4. Select a clip from the Viewing panel, the Desktop or the Media panel by clicking it.

   The cursor turns white, prompting you to select a destination for the result.
5. Click the Viewing panel, a Desktop reel or the Media panel to select the destination.

   The LUT Editor menu is displayed.

Specifying a Colour Management Operator

To apply colour management to a clip, you must first specify the type of operation to perform in the LUT Editor (page 1539). You can:

- Apply a preset LUT and then modify its values.
- Choose an operator type. Depending on the type you choose, you can either import a baked operator or modify its values.

Applying a Preset

1. Select an option from the Presets box in the LUT Editor.
2. Confirm that you want to replace the existing settings.

   The Conversion LUT Type and the field values are all updated with preset conversion parameters. In Batch or Batch FX, a note is added with a description of the preset's usage.
3. If desired, modify the curve values as described in Modifying Basic Conversion LUT Curves (page 1544) and Advanced Editing of the Conversion LUT Curve (page 1551).

TIP To make your own preset available, save a setup to the /lut/presets subfolder of the application home directory. This does not work with 3D LUTs.

Choosing an Operator Type

1 Select an output bit-depth option from the Destination box in the LUT Editor.



2 Select an option from the LUT Type box.



Select:	To:
Colour Transform	Apply a colour transform based on the Academy/ASC colour transform XML format, as well as several other supported LUT and transform formats.
PhotoMap	Begin with a default curve for tone mapping.
3D LUT	Import a 3D LUT. Note that 3D LUTs are not available for floating-point values.
1D LUT	Import a 1D LUT. Note that some 1D LUTs are intended for integer values and others are intended for floating-point values.

Select:	To:
EXR Display	Begin with a default curve for tone mapping.
Gamma	Begin with a default curve for modifying linear data.
Lin to Log	Begin with a default curve for converting linear or video data to logarithmic data.
Log to Lin	Begin with a default curve for converting logarithmic data to linear or video data.

3 Do one of the following:

- If you chose 1D LUT or 3D LUT, import a LUT operator.
- If you chose Colour Transform, you can import an colour transform operator. Alternatively, you can define a custom colour transform — see Building Custom Colour Transforms (page 1543).
- If you chose any other option, you can modify the curve values as described in Modifying Basic Conversion LUT Curves (page 1544) and Advanced Editing of the Conversion LUT Curve (page 1551).

Importing an Operator

1 Click Import.

The file browser appears.

2 Locate and select the file that you want to import. See Colour Management Files and Locations (page 1562).

When importing a colour transform, see Autodesk Colour Transform Types (page 1561) for a description of the available categories.

You are returned to the LUT Editor. The LUT name appears in the field next to the Import button.

- When you import a 1D LUT, the conversion curves appear in the LUT Editor. You can modify them only with the Advanced settings.
- When you import a colour transform, information about the transform and the operations it contains appears in the LUT Editor.

Building Custom Colour Transforms

You can build a chain of colour transforms using the Custom mode of the LUT Editor (page 1539). The transforms are applied in order from beginning to end. For example, you can build a colour transform that first modifies the gamma and then changes the primaries.



1 In the LUT Editor, set the LUT type box to Colour Transform.

2 Enable the Custom button.

3 Click Add to insert an empty row.

4 Click in the Type column and select a type:

- **Shared:** The location for shared custom colour transforms (available to all applications that use Autodesk Colour Management).
- **Project:** The project transform folder (saved and archived with the project).
- **Autodesk:** The default location for preset colour transforms supplied with the application. See Autodesk Colour Transform Types (page 1561).
- **Import:** Browse for colour transforms on your file system. You can select native .ctf format files, as well as several other supported LUT and transform formats.

5 After you've selected a colour transform type, click in the Transform column and select a transform.

Information about the transform and the operations it contains appears in the LUT Editor.

6 To add more transforms to the end of the chain, repeat steps 3 to 5.

You can also:

- Click in the Type or Transform columns again to change a transform in the chain.
- Select a row and click Delete to remove a transform from the chain.
- Use the Up or Down buttons to reorder the chain. Moving a transform up applies it earlier in the chain, and moving it down applies it later.
- Use the visibility icon to mute a transform in the chain.

TIP You can create subfolders in the Shared or Project locations to define custom colour transform types.

Modifying Basic Conversion LUT Curves

NOTE The procedures in this section apply only to Gamma, Lin to Log, and Log to Lin operators.

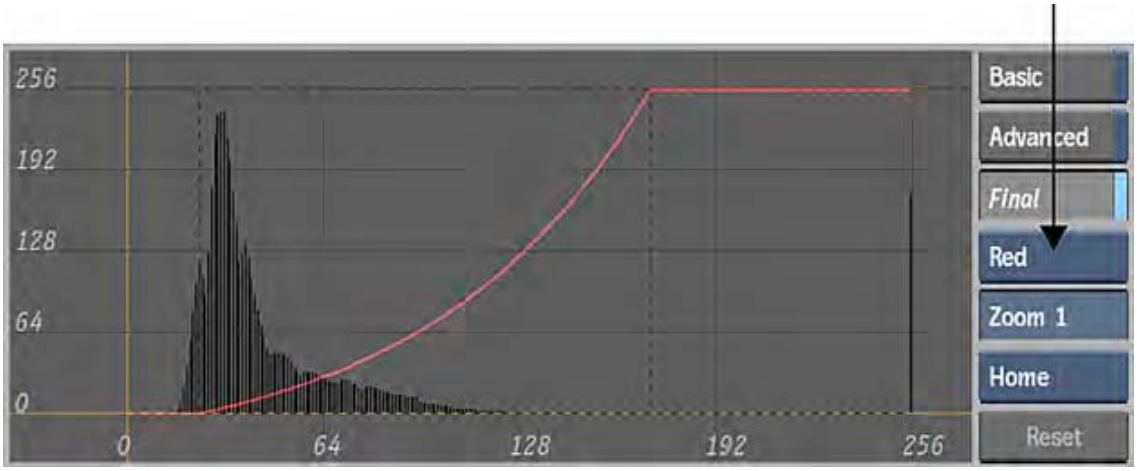
Once you define a basic LUT type in the LUT Editor (page 1539), you can modify:

- The reference white and reference black values
- The highlights and shadows
- The gamma of the conversion curve
- The gamma of the incoming film negative
- The degree to which the shoulder at the upper end of the conversion curve is softened

When you modify these values, the basic LUT curves are updated to reflect the changes. For all procedures, you can monitor the R, G, and B channels independently while modifying the LUT.

To monitor the R, G, and B channels independently:

1 From the Edit Curve box, select Red, Green, or Blue to monitor the red, green, or blue channels independently.



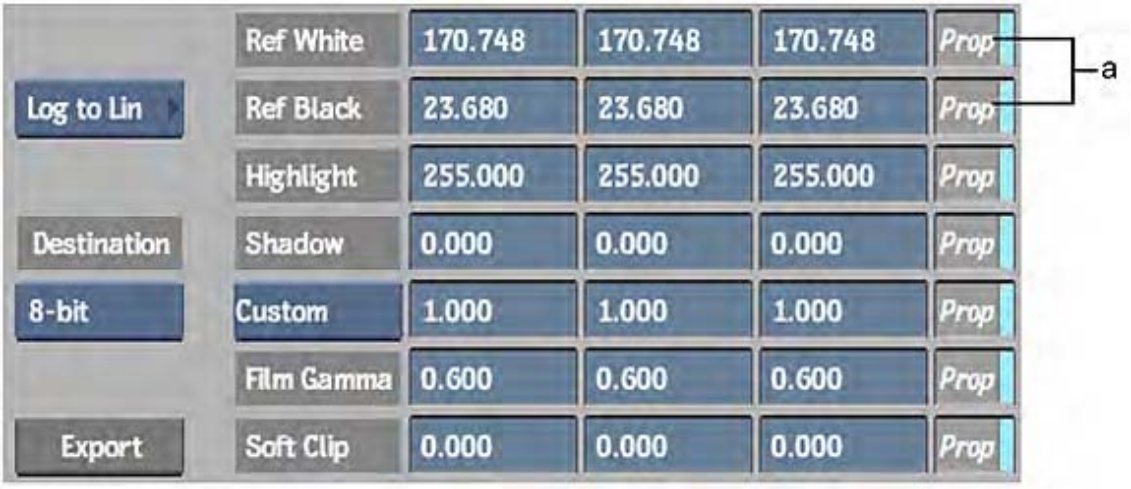
Modifying Reference White and Reference Black Values

The reference white and reference black values define the intensity levels at which incoming pixels are considered white or black respectively. For example, selecting a logarithmic-to-linear basic conversion LUT curve sets the reference white to 685 and the reference black to 95. (These have been shown to be good values for Kodak™ film stock.) As a result, source pixels with values from 685 and 1023 will produce white pixels, while those from 0 and 95 will be set to black.

Reference white and black are affected by the values assigned to highlights and shadows. White pixels are mapped to the value set for highlights, while black pixels are mapped to the value set for shadows. See Modifying Highlights and Shadows (page 1546).

To modify reference white or black values using the Ref White and Ref Black fields:

1 Do one of the following:
   - To set proportional R, G, and B values, enable Prop and then enter the value in one of the (left-to-right) R, G, and B Ref White and Ref Black fields.
   - To set independent R, G, or B values, disable Prop and then enter the value in the corresponding (left-to-right) R, G, or B Ref White and Ref Black fields.

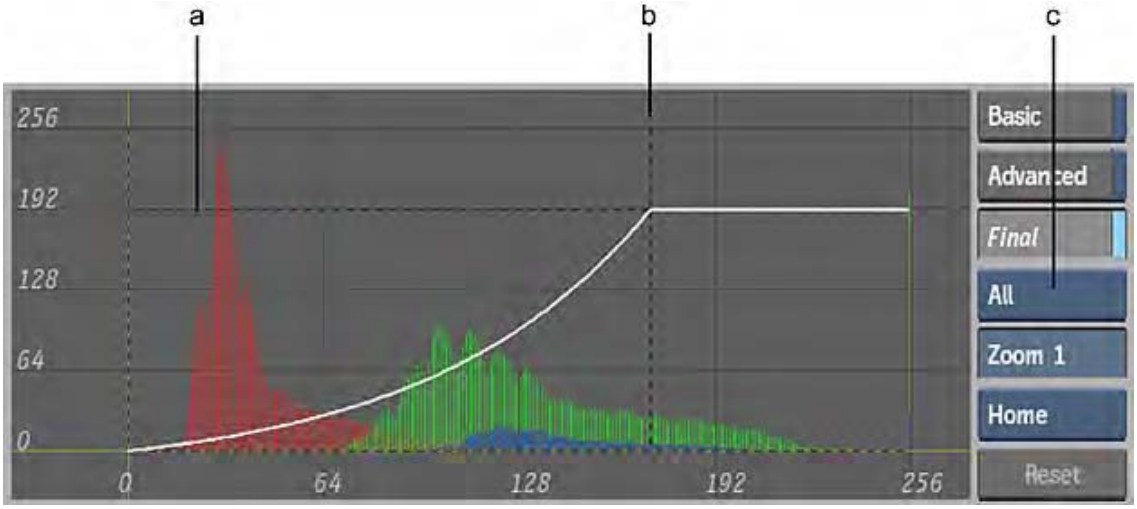


(a) Ref White/Black fields and Prop (Proportional) buttons

To modify reference white or black values using the LUT Editor graph:

1 Do one of the following:

- To set proportional R, G, and B values, select All from the Edit Curve box, and then drag the vertical reference white or reference black line left or right.
- To set independent R, G, or B values, select Red, Green, or Blue from the Edit Curve box, and then drag the vertical reference white or reference black line left or right.



(a) Highlight line (b) Reference White line (c) Edit Curve box

TIP If you already entered independent values and want to modify the R, G, and B values proportionally, you must drag the reference white or reference black line that corresponds to the corresponding channel. To view the selected R, G, or B luma values as a greyscale luma image in the Import Image menu, enable the Exclusive button.

Modifying Highlights and Shadows

Highlight and shadow values define the maximum and minimum values permitted for each channel. All values above the highlight are clamped down to it. Similarly, all values below the shadow are clamped up.

To modify highlight and shadow values using the Highlights and Shadows fields:

1 Do one of the following:
- To set proportional R, G, and B values, enable Prop and enter a value in one of the (left-to-right) R, G, and B Highlights and Shadows fields.
- To set independent R, G, or B values, disable Prop and enter the value in the corresponding (left-to-right) R, G, or B Highlights and Shadows fields.



(a) Highlight/Shadow fields

To modify highlight or shadow values using the LUT Editor graph:

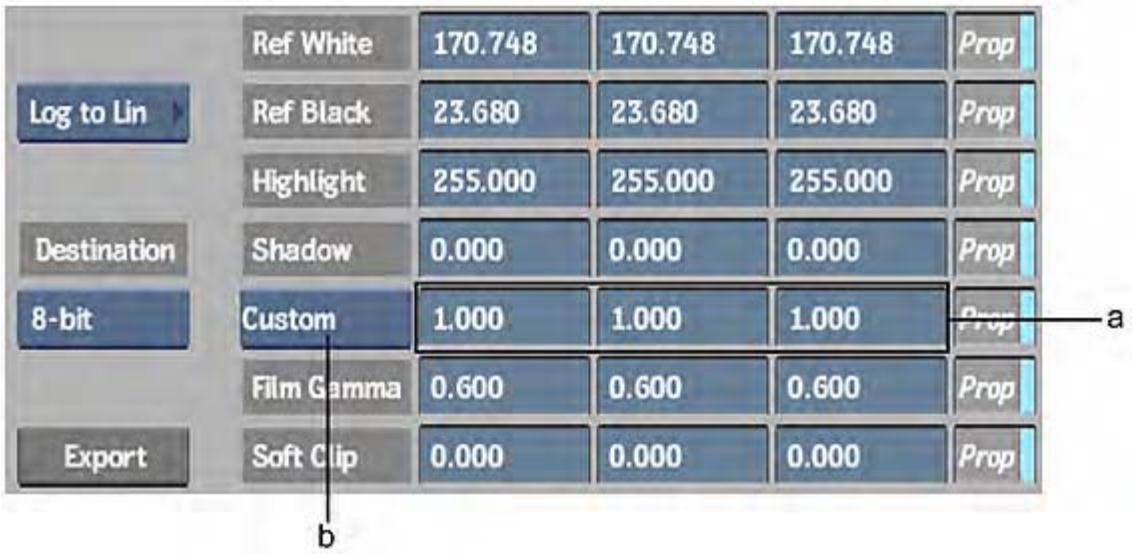
1 Do one of the following:
- To set proportional R, G, and B values, select All from the Edit Curve box, and then drag the horizontal highlight or shadow line up or down.
- To set independent R, G, or B values, select Red, Green, or Blue from the Edit Curve box, and then drag the horizontal highlight or shadow line up or down.

Modifying the Gamma of the Conversion Curve

Set the gamma correction value to correspond to the display gamma you set your system to use. You can select a preset from the Gamma Correction box or enter a custom value.

To modify the gamma correction curve using the Gamma Correction fields:

1 Select an option from the Gamma Correction box.



(a) Gamma Correction fields (b) Gamma Correction box

Select:	To:
Video Display	Set video gamma correction presets.
Custom	Enter custom gamma values.

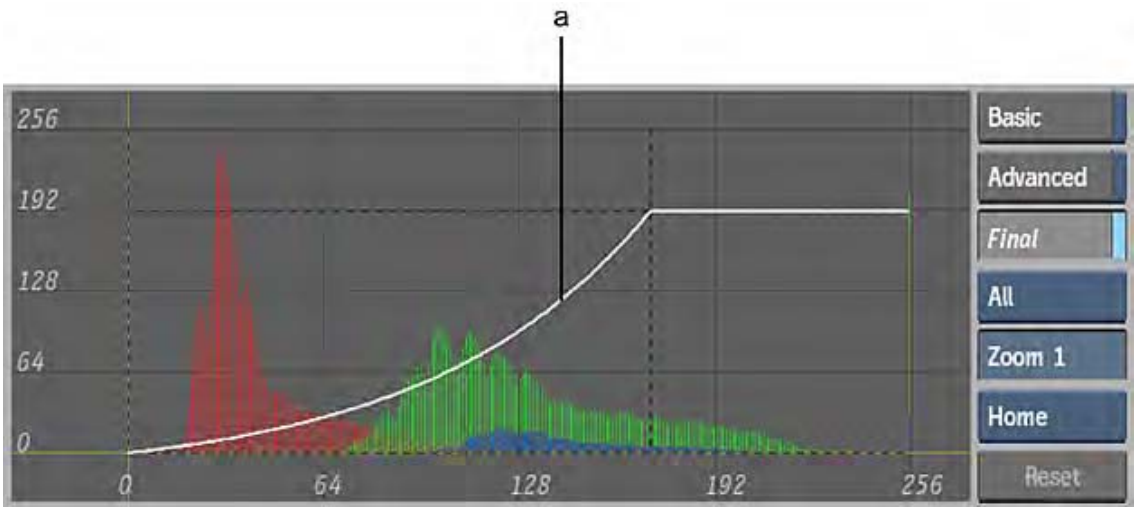
2 If you selected Custom, do one of the following:

- To set proportional custom R, G, and B gamma correction values, enable Prop, and then enter the value in one of the (left-to-right) R, G, and B Gamma Correction fields.
- To set independent custom R, G, or B gamma correction values, disable Prop, and then enter the value in the corresponding (left-to-right) R, G, or B Gamma Correction field.

To modify the gamma correction curve using the LUT Editor graph:

1 Do one of the following:

- To modify the combined R, G, and B gamma correction curves proportionally, select All from the Edit Curve box, and then drag the gamma correction curve left or right.
- To set independent R, G, or B values, select Red, Green, or Blue from the Edit Curve box, and then drag the gamma correction curve left or right.



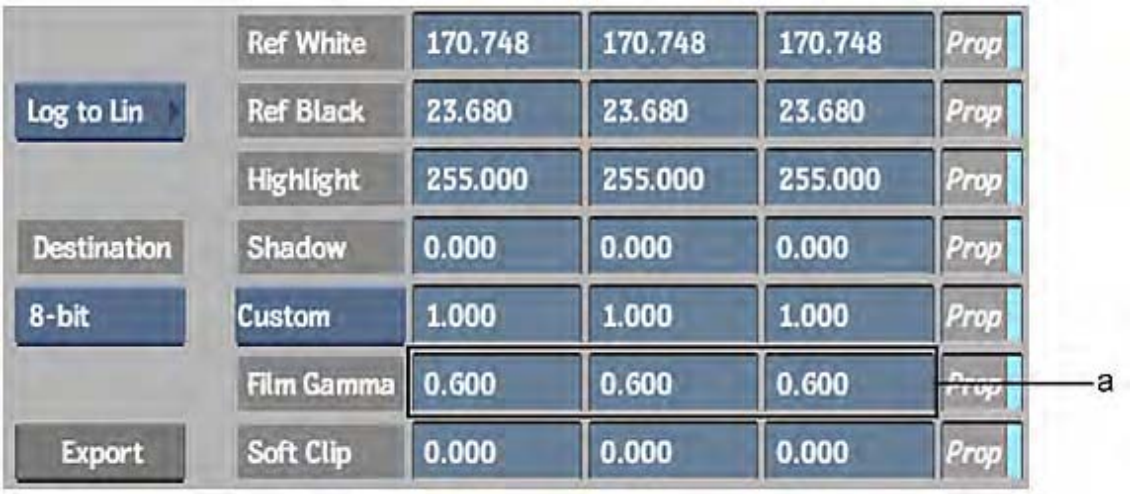
(a) Gamma correction curve

Modifying the Gamma of the Incoming Film Negative

The values in the Film Gamma fields affect the gamma correction curve inversely to the Gamma Correction fields. This is because the gamma correction performed using these controls is based on the gamma of the original film negative. Typically, the gamma of a film negative is 0.6. Only adjust this value if your film negative is of a non-standard gamma. You should not have to modify this value by very much.

To modify the gamma correction curve using the Film Gamma fields:

1. Do one of the following:
   - To set proportional custom R, G, and B gamma correction values, enable Prop and then enter the value in one of the (left-to-right) R, G, and B Film Gamma fields.
   - To set independent custom R, G, or B gamma correction values, disable Prop and then enter the value in the corresponding (left-to-right) R, G, or B Film Gamma field.



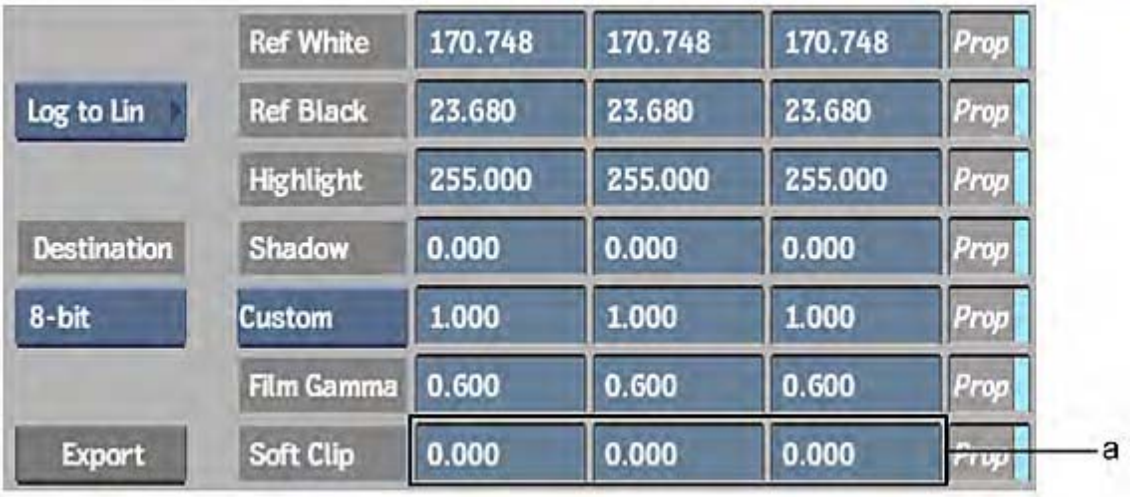
(a) Film Gamma fields

Adjusting the Softclip Values

You can soften the shoulder of the gamma correction curve by adjusting the Softclip values. When you soften the shoulder of the conversion curve, you soften the transition of colours toward the reference white value. This results in less harsh highlights in your clip.

To soften the shoulder of the gamma correction curve using the Softclip fields:

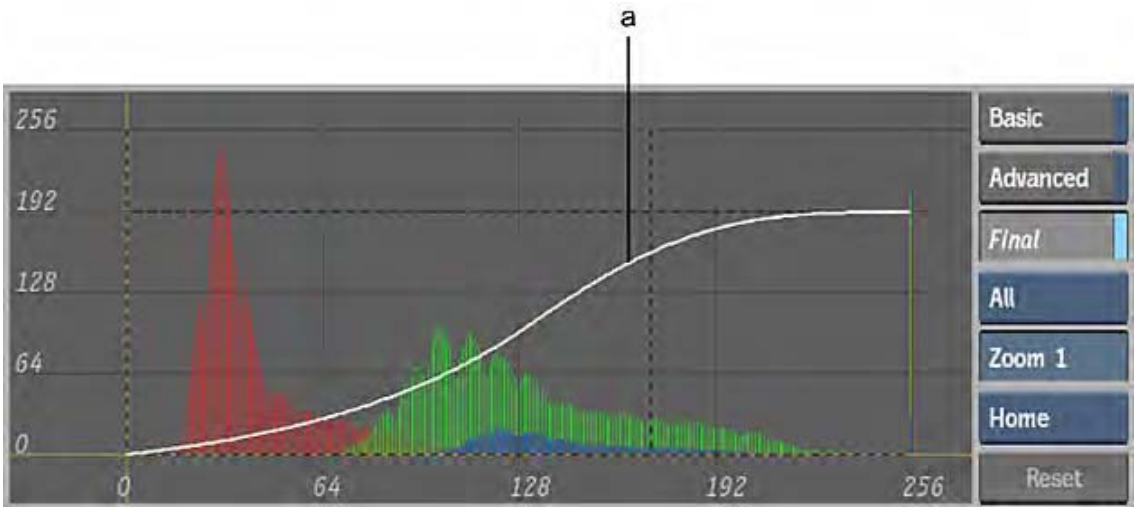
1. Do one of the following:
   - To set proportional custom R, G, and B gamma correction values, enable Prop, and then enter the value in one of the (left-to-right) R, G, and B Softclip fields.
   - To set independent custom R, G, or B gamma correction values, disable Prop, and then enter the value in the corresponding (left-to-right) R, G, or B Softclip field.



(a) Soft Clip fields

To soften the shoulder of the gamma correction curve using the LUT Editor graph:

1. Do one of the following:
   - To soften the shoulder of the combined R, G, and B gamma correction curves proportionally, select All from the Edit Curve box, and then `Shift`-drag the gamma correction curve left or right.
   - To soften the shoulder of the R, G, or B curves separately, select Red, Green, or Blue from the Edit Curve box, and then `Shift`-drag the gamma correction curve left or right.



(a) Softened shoulder

Advanced Editing of the Conversion LUT Curve

NOTE The procedure in this section does not apply to colour transforms or 3D LUTs.

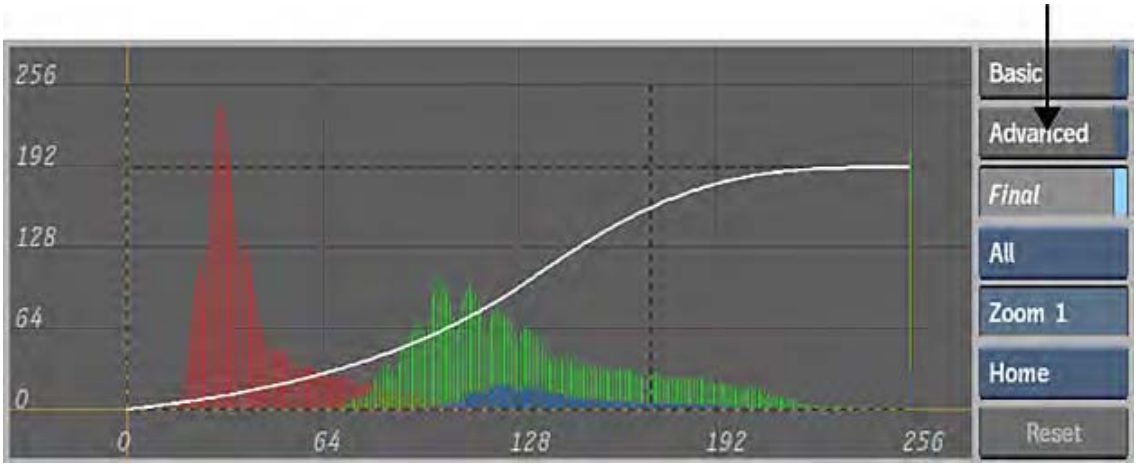
After you modify the basic conversion curves in the LUT Editor (page 1539), you may want to further refine the final result. To do so, you can use advanced editing tools. With advanced editing, you modify additional curves for the red, green, and blue channels to fine-tune the final conversion LUT and bring out or hide detail in specific areas. These curves have editable points, adding flexibility in how they affect each channel in the final conversion LUT.

You can modify the shape of each advanced editing curve using control points. Modifications you make to these curves influence the curve of corresponding channels in the final conversion LUT. For example, you can add points to the advanced editing curve for the red channel, and by dragging the points, affect the slope of the red channel's final conversion LUT curve.

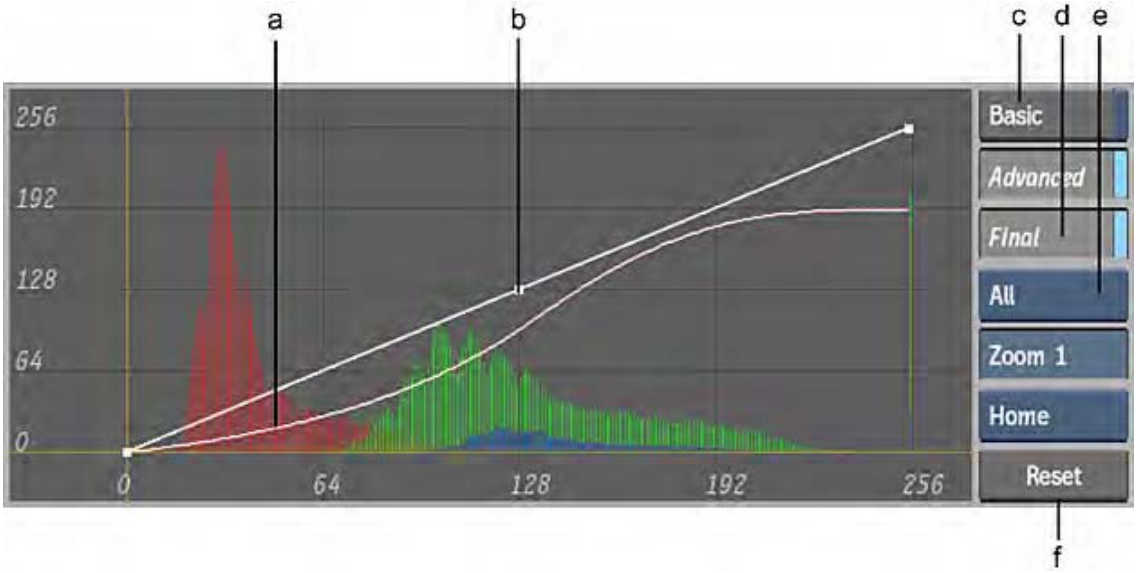
The modifications you make to the basic conversion LUT curves and the advanced editing curves are independent. The final LUT that you generate, however, is a single curve (or set of R, G, and B curves) that consists of basic LUT curves and adjustments you make with advanced editing curves.

To perform advanced editing of the conversion LUT curve:

1 In the LUT Editor, click Advanced.



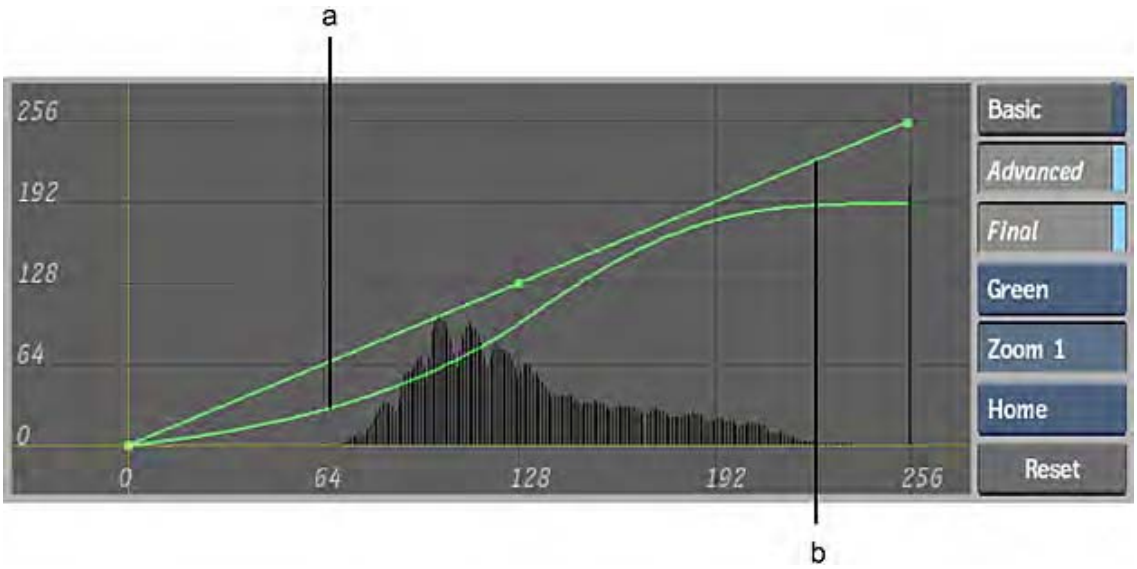
Additional advanced editing curves appear for the red, green, and blue channels. RGB curves use blending to distinguish overlapping curves. For example, when all RGB curves overlap, a white curve is produced, whereas green and blue overlapping curves produce a cyan curve.



(a) Red, blue and green overlapping advanced editing curves (b) Control point (c) Basic button (d) Final button (e) Edit Curve box (f) Reset button

2 To adjust the R, G, and B curves more easily, select one of the channels from the Edit Curve box.

The corresponding channel's basic and advanced editing curves are displayed. The histogram only displays values for the selected channel.



(a) Color channel selected in Edit Curves box (b) Color channel's advanced editing curve

3 To modify the advanced editing curves, use the Tools box.

Select:	To:
Add	Add control points to an advanced editing curve. With Add selected, click either the red, green, or blue advanced editing curve.
Delete	Delete control points from an advanced editing curve. With Delete selected, click a point on the red, green or blue advanced editing curve.
Move	Move control points. With Move selected, drag the points, or press Control-drag to select multiple points. **TIP** Since the advanced editing curves use B-spline interpolation, the control points you manipulate will not necessarily rest on the curve itself. Points have a weighted influence on the curve depending on the position of the other points on the curve.
Zoom	Zoom in on the curves. With Zoom selected, drag over the curves right or left to zoom in or out.
Rect Zoom	Zoom in on a section of the curves. With Rect Zoom selected, drag a selection box to zoom in on the area of the curves contained by the selection box.
Pan	Pan the curves. With Pan selected, drag over the curves to pan the curves in any direction.

Dragging control points of advanced editing curves affects the basic curves and the colour effect is immediately reflected in the clip.

4 To view the basic and final conversion LUT curves, toggle the Basic LUT and Final LUT buttons.

5 To reset the advanced editing curves only, click Reset.

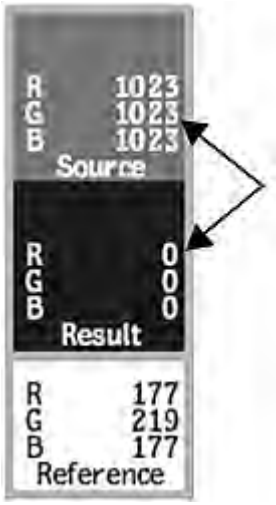
Comparing Colour Values Using Colour Patches

NOTE The procedures in this section do not apply to 3D LUTs.

While using the LUT Editor (page 1539), you may want to sample image channel values in the source, result, and reference clips to compare values of similar colours, or even corresponding pixels. The colour patches appear to the right of the LUT Editor graph.

To sample colours in the LUT Editor accessed from the Import Image menu:

1 Click the Source or Result Colour patch.



The cursor changes to a colour picker.

2 Click the image.

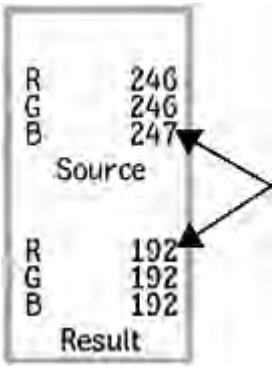
Both source and result colours are sampled simultaneously. The RGB values for the selected pixel are displayed, and the sampling boxes take on the corresponding colour.

3 To sample a set of reference values, click the Reference sampling box and use the colour picker to sample the image.

The values are taken from either the result or reference image, depending on which clip display option is currently selected.

To use colour patches in the LUT Editor menu in Batch orBatch FX:

1 Click the Source or Result sampling box.



The cursor changes to a colour picker.

2 Click the image.

Both source and result colours are sampled simultaneously. The RGB values for the selected pixel are displayed, and the sampling boxes take on the corresponding colour.

Exporting LUTs and Colour Transforms

Exporting a LUT or colour transform from the LUT Editor (page 1539) is different from saving a setup:

- PhotoMap, EXR Display, Gamma, Lin to Log, and Log to Lin are exported as baked 1D LUTs (*.lut* files). When imported, you can modify their curves only with the Advanced settings.
- A chain of colour transforms is exported as a single *.ctf* file that losslessly preserves each component operation. When imported, you cannot modify it.

Exporting a LUT

1 Access the LUT Editor.

2 Click Export.

The file browser appears.

3 Type a file name for the LUT in the Export field.

4 If you want to generate an inverse LUT with the *_inv* suffix appended to the file name, make sure that the Generate Inverse Lut option is active.

5 Click Export.

The LUT is exported to the specified location.

Exporting a Colour Transform

1 Access the LUT Editor.

2 Click Export.

The file browser appears.

3 Type a file name in the Export field. The *.ctf* extension is added automatically.

4 If desired, type a nickname for the colour transform in the Nickname field. The nickname is displayed in the metadata panel of the LUT editor, and is also shown in the Viewer Colour Management pop-up (in the lower left of image windows) when loaded in your LUT preferences.



5 If desired, type a description for the colour transform. The description is displayed in the metadata panel of the LUT editor.

6 Select the destination location using the location shortcut under the EXIT Export LUT button:

- **Shared** is a useful location for storing transforms that can be used by other Autodesk applications on the same workstation that use Autodesk Colour Management.
- **Project** (/usr/discreet/project/<project name>/synColor/transforms) is a useful location for storing the transforms that are specific to the current project, for example, for archiving.

- Saving transforms to the **Autodesk** location is not recommended.

7 Click Export.

The colour transform is exported to the specified location.

Applying Colour Management to Displayed Images

When you apply colour management to displayed images, the colours on the monitor or projector are affected but the underlying colour values of the clips' pixels are not changed. You should do this when you want to preview images in different colour spaces. For example, if you are working in the ACES colour space, then you can apply the ACES_to_current-monitor transform to display the images.

Configuring Flame Premium to Use LUTs and Colour Transforms for Display

Before you can apply colour management to displayed images, you must add the LUTs and colour transforms that you want to use to your preferences. This makes them available for use in image windows.

Making 1D LUTs Available for Display

1 Click Flame > Preferences, and select LUT.

2 On the 1D LUT tab at the right, click in a slot.

3 Do one of the following:

- Select Gamma in the LUT Type box, and set a value.
- Select LUT File in the LUT Type box, then click Import and select a 1D LUT file.

Making 3D LUTs and Colour Transforms Available for Display

1 Click Flame > Preferences, and select LUT.

2 On the 3D LUT tab at the right, click in a slot.

3 Click Import and select a native 3D LUT (.3dl) or Colour Transform (.ctf) file. You can also select files in several other supported LUT and transform formats.

Configuring the Colour Transform Aliases

Some colour transforms include references to one or more of several aliases. You can configure which colour transforms are used in place of these aliases. This lets you change the value of the aliases while still using the same colour transform files. For example, you can:

- Use the same colour transforms for display on different computers with different monitors.
- Change the default look applied by a colour transform without changing the transform file itself.

To set the colour transform aliases:

1 Click Flame > Preferences, and select LUT.

2 Set the transform to use for each of the following:

- **Broadcast Monitor Transform**. This is the transform that gets used when another transform includes a reference to the broadcastMonitor alias, as well as when another transform includes a

reference to the currentMonitor alias for images displayed on the broadcast monitor or projector. Click **Browse**, select **Autodesk** at the lower left, and choose one of the tranforms in the display/ folder that convert from CIE-XYZ to whatever is appropriate for your broadcast display, or use one of your own transforms.

- **Graphics Monitor Transform**. This is the transform that gets used when another transform includes a reference to the graphicsMonitor alias, as well as when another transform includes a reference to the currentMonitor alias for images displayed on the local monitor. Click **Browse**, select **Autodesk** at the lower left, and choose one of the tranforms in the display/ folder that convert from CIE-XYZ to whatever is appropriate for your local display, or use one of your own transforms.
- **Default Look Transform**. This is the transform that gets used when another transform includes a reference to the defaultLook alias. It should be set to the transform that you are using to re-create the look used on set.

Applying 1D LUTs to the Monitor

Applying a Gamma or 1D LUT to the monitor affects all images displayed on the workstation in Flame. Any 3D LUTs or colour transforms that are being used to display specific viewports become deactivated automatically. The last-used 1D LUT is displayed at the bottom right of the Player or Viewport, and is highlighted when the LUT is in use.



NOTE You can also apply a 1D LUT to the monitor from the LUT preferences.

Applying 1D LUTs to the Monitor Using the View Menu

1. Do one of the following:
   - From the Player, select Show Viewing Settings from the Options box.
   - From Src-Seq or Triptych, select Edit Viewing Settings from the Options box.
   - From any other view, such a Tools, Batch or Batch FX effect, open the View menu.
2. Enable Use 1D LUT.

   The 1D LUT displayed in the 1D LUT List box is applied to the display.
3. To change 1D LUTs, select an option from the list in the Monitor LUT List box.



Applying 1D LUTs to the Monitor Using the Keyboard

Press:	To do:
`Control+Shift+1`	Apply the first 1D LUT defined in the 1D LUT list.
`Control+Shift+[2-9]`	Apply the second through ninth 1D LUT defined in the 1D LUT list.
`Control+Shift+0`	Apply the tenth 1D LUT defined in the 1D LUT list.
`Control+Shift+~`	Toggle the last-used 1D LUT on and off.

Applying 3D LUTs and Colour Transforms to Viewports

You can apply 3D LUTs and colour transforms to individual viewports, and use different ones in different viewports. Any 1D LUT that is being used on the monitor becomes deactivated automatically. The file name of the last-used 3D LUT or colour transform is displayed at the bottom right of the viewport, and is highlighted when in use.



The nicknames of the available colour transforms also appear on the Viewer Colour Management pop-up menu, and can be used quickly apply a colour transform to the viewport.

NOTE 3D LUTs are assumed to require colour values in log space. Depending on your display mode, a linear-to-log or video-to-log conversion is automatically applied.

NOTE 3D LUTs and color transforms are not applied to every view. For example, Sparks uses the basic Image Data Type, while other views and modules such as Text, the MediaHub preview, desktop reels, and thumbnails have no color management at all. If necessary, you can apply a 1D LUT to the whole interface to affect the colors displayed in these views.

Applying Colour Transforms to a Viewport Using the Viewer Colour Management Pop-up Menu

1 Click on the Image Display pop-up menu in the bottom left of an image viewport.



2 Select a colour transform.

NOTE 3D LUTs are not available on the Image Display pop-up menu.

Applying 3D LUTs and Colour Transforms to a Viewport Using the View Menu

1. Do one of the following:
   - From the Player, select Show Viewing Settings from the Options box.
   - From Src-Seq or Triptych, select Edit Viewing Settings from the Options box.
   - From any other view, such a Tools, Batch or Batch FX effect, open the View menu.
2. If multiple image views are displayed, click in the one you want to affect.
3. Enable Use 3D LUT.
   The 3D LUT or colour transform displayed in the 3D LUT List box is applied to the display.
4. To change the 3D LUT or colour transform, select an option from the list in the Monitor LUT List box.



5. If multiple image views are displayed, repeat steps 2 to 4 to affect other views.

Applying 3D LUTs and Colour Transforms to a Viewport Using the Keyboard

1. If multiple image views are displayed, click in the one you want to affect.
2. Press one of the following keys.

Press:	To apply:
`Alt+Shift+1`	The first 3D LUT or colour transform defined in the 3D LUT & Colour Transform list.
`Alt+Shift+[2-9]`	The second through ninth 3D LUT or colour transform defined in the 3D LUT & Colour Transform list.
`Alt+Shift+0`	The tenth 3D LUT or colour transform defined in the 3D LUT & Colour Transform list.
`Alt+Shift+~`	Toggle the last-used 3D LUT or colour transform on and off.

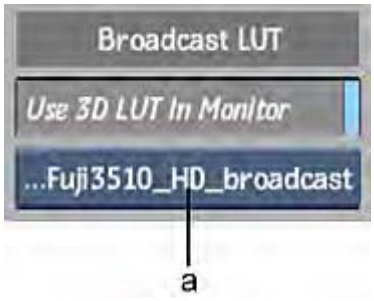
Applying a 3D LUT or Colour Transform to the Projector or Broadcast Monitor

You can apply a different 3D LUT or colour transform to the projector or broadcast monitor than the one used for display on the workstation.

1. Click Flame > Preferences, and select Broadcast.

2 Under Broadcast LUT, enable Use 3D LUT In Monitor.

3 To change the 3D LUT or colour transform, select an option from the list in the 3D LUT List box.



(a) 3D LUT List box

Modifying Exposure and Contrast with Display Transforms

If you apply a 1D or 3D LUT for display, then you can modify the exposure and contrast in the usual way.

However if you apply a colour transform to an image view or the broadcast monitor, then you cannot adjust the exposure and contrast interactively. The exception is when there is a dynamic ExposureContrast operator in the CTF file. In that case, the exposure and contrast values defined in the CTF file are not used. Instead, the values set in the Image Display options are used.

Controlling the Look in Viewports

When some transforms are applied as viewing transforms in a viewport, you can toggle portions of the transform on and off interactively. This can be helpful when working with looks.

Looks are basic colour grades that are created by a cinematographer to establish a visual mood for a scene during principal photography. However, the look must not be baked into the raw footage or visual effects because it is only the starting point for the final grading. Therefore, it is useful to be able to toggle the look on and off during finishing.

The Look on/off switch is available when:

- The current viewing transform in a viewport contains a reference to the Default Look Transform set in LUT Preferences.
- The current viewing transform is an ASC colour decision list (.cdl) file.
- The current viewing transform is a custom Colour Transform (.ctf) file that contains one or more operators with the LOOK_SWITCH dynamic parameter.



You can click directly on the Look switch at the bottom of the viewport to toggle the look on and off. Alternatively, you can use the Enable Look Transform button in the Image Display Viewer settings.

Setting Up the Default Look

1. In LUT Preferences, specify a transform as the Default Look Transform. You can select a transform file in any supported format.
2. In LUT Preferences, add one or more viewing transforms that reference the default look to the 3D LUT and Colour Transform list. The transforms that reference the default look include the following:
   - The transforms in the RRT+ODT directory of the Autodesk location are available for displaying ACES material on various devices.
   - The ADX10_to_current-monitor and ADX16_to_current-monitor transforms in the film\ADX\ directory are available for displaying scanned film on the system or broadcast monitor.

   NOTE The default_look transform in the misc\ directory simply applies the default look transform, and the default_look-ACESproxy transform applies the default look transform in the ACESproxy color space that is typically used on set (first converting from the ACES color space and then back again). These transforms are not intended to be used directly as viewing transforms, but should be used as building blocks for creating your own custom chain of transforms.
3. Display an image in a viewport, and apply a viewing transform.

   The Look switch appears at the bottom of the viewport.

Autodesk Colour Transform Types

The colour transforms available in the Autodesk location are CTF files based on the Academy/ASC XML colour transform format. They include several categories for different purposes.

Category	Description
RRT+ODT	Various combinations of a reference rendering transform (RRT) followed by an output device transform (ODT) designed for output from the ACES colour space.
bit-depth	Converts between various integer and floating-point value formats.
camera	Converts from various digital-camera native formats to other colour spaces such as ACES.
display	Converts colours in a connection space to and from output spaces appropriate for various display devices.
film	Converts scanned-film colour spaces to and from ACES.
gamma	Applies or removes various common gamma values, including those for sRGB and Rec709.
interchange	Converts colours in a connection space to and from spaces that are commonly used for file input and output, such as AdobeRGB and sRGB.
levels	Various functions to clamp or adjust colour values.

Category	Description
misc	Various transforms that may be useful for debugging or other purposes.
primaries	Converts between linear colour spaces that use different primary colours.
tone-map	Converts from scene-linear to video colour spaces.
whitepoint	Converts the white point of linear colour spaces.

Colour Management Files and Locations

The colour management files consist of 1D LUTs, 3D LUTs, and colour transforms. You can use files created in other applications as long as they are in a compatible format.

You can use the location shortcut button at the lower left of file browsers to switch between the standard file locations.



You can use files located anywhere on your file system, but it is recommended to store your custom LUTs and colour transforms in either the **Project** or **Shared** location.

1D LUTs

All 1D LUT files have the *.lut* extension. They are stored in the **Project** (/usr/discreet/project/<project name>/lut) location.

3D LUTs

All standard 3D LUT files have the *.3dl* extension; encrypted 3D LUTs have the *.e3d* extension.

The 3D LUTs installed with Flame Premium are in the **Lustre Colour** (/usr/discreet/Lustre_Color/lut/Lustre_Color_3DLUTs) location. Use the **Project** (/usr/discreet/project/<project name>/lut) location to store 3D LUTs specific to a particular project.

Colour Transforms

Colour transforms are based on the Academy/ASC XML colour transform format and have the *.ctf* extension. They are stored in the following locations:

- **Shared** (/opt/Autodesk/Synergy/SynColor/Shared/transforms by default) is a useful location for storing transforms that can be used by other applications that use Autodesk Colour Management.
- **Project** (/usr/discreet/project/<project name>/synColor/transforms) is a useful location for storing the transforms that are specific to the current project, for example, for archiving.
- **Autodesk** (/opt/Autodesk/Synergy/SynColor/<version>/transforms) contains the transforms installed with Flame Premium. See Autodesk Colour Transform Types (page 1561).

Supported Colour Transform File Formats

In addition to the native .ctf format, applications that use Autodesk Color Management can read several LUT and transform file formats.

Supported formats include:

.lut Autodesk legacy 1D LUT.

.3dl Autodesk legacy 3D LUT.

.cdl ASC color decision list.

ASC .cdl files are loaded as ASC_CDL operators. The style used by the colour transform operator is determined by the SYNCOLOR_DEFAULT_CDL_STYLE environment variable. This can be one of the following values: `v1.2_Fwd`, `v1.2_Rev`, `noClampFwd`, or `noClampRev`. If this environment variable is undefined, the default is `v1.2_Fwd`. For more information about these styles, see ASC_CDL (page 1594).

When a .cdl file is applied to the display, a Look switch is added that allows you to toggle the transform on and off.

.ccc ASC color correction collection.

The first CDL defined in the file is loaded as an ASC_CDL operator. The style is also determined by the SYNCOLOR_DEFAULT_CDL_STYLE environment variable.

.csp Cinespace

.icc, **.icm**, **.pf** ICC monitor profile. The PCS-to-monitor-RGB transform defined in the profile is loaded.

.cube Iridas Cube

.itx Iridas itx

.look Iridas Look (no mask support)

.m3d, **.mga** Pandora

.spi1d Imageworks 1D LUT

.spi3d Imageworks 3D LUT

.spimtx Imageworks matrix

.vf Nuke

When imported, these formats are converted to native operators and can be exported as .ctf files.

Autodesk CTF File Format Version 1.3

There are a wide variety of colour transform (also known as "LUT") formats used in the media and entertainment industry. This creates workflow problems since you may receive a LUT which is not supported by one or more of the applications you use. Furthermore, most of these LUT formats are based on out-dated technology and lack the quality, flexibility, and metadata needed to meet today's requirements.

To address these problems, the technology committees of the Academy of Motion Picture Arts and Sciences and the American Society of Cinematographers convened a group of experts to design a common transform format that could become an industry standard.

Autodesk has chosen to adopt this format as the native format for our colour management technology and we provide a large collection of transforms in this format with our applications (e.g., to support the Academy Color Encoding System).

This guide provides information about the Academy/ASC format to help you author your own colour transforms. In addition it defines several Autodesk-specific extensions which you may use when authoring colour transforms solely for use in Autodesk applications. All Autodesk extensions are marked with the version in which they were introduced or updated. All other elements and attributes are part of the original Academy/ASC format.

Anatomy of a CTF File

The transform format allows the author to define an arbitrary chain of color operators (also called processing nodes). The set of available operator types includes 1D LUTs, 3D LUTs, matrices, and others defined below. The format is written in XML and is therefore human readable.

Here is an example:

```
<?xml version="1.0" encoding="UTF-8"?>
<ProcessList id="7401489a-9e4d-441d-a8b7-795739801f4e" name="ACES to HD"
version="1.2">
    <Info>
        <Copyright>Copyright 2013 Autodesk, Inc.  All rights
reserved.</Copyright>
        <Release>2013.3.pr27</Release>
    </Info>
   <Description>ACES Reference Rendering Transform + HD video Output Device
 Transform</Description>
    <Description>aces_dev version 0.1.1</Description>
    <InputDescriptor>Academy Color Encoding Specification
(ACES)</InputDescriptor>
    <OutputDescriptor>HD (Rec 709) video</OutputDescriptor>
    <LUT1D halfDomain="true" inBitDepth="32f" outBitDepth="32f"
rawHalfs="true">
        <Array dim="65536 1">
    0
    1
    1

    <!-- 65533 lines omitted -->

        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 0.9259999990  0.0379999988  0.0359999985
 0.1299999952  0.8000000119  0.0700000003
```

```
-0.0350000001 -0.0350000001  1.0700000525
        </Array>
    </Matrix>
    <LUT1D halfDomain="true" inBitDepth="32f" outBitDepth="32f"
rawHalfs="true">
        <Array dim="65536 1">
    0
    0
    0

    <!-- 65533 lines omitted -->

       </Array>
    </LUT1D>
    <LUT3D inBitDepth="32f" interpolation="tetrahedral" outBitDepth="32f">
        <Array dim="48 48 48 3">
 0.000669299  0.000704187  0.000766901
 0.001848371  0.001297010  0.010368987
 0.002599007  0.001144927  0.023227118

        </Array>
    </LUT3D>
    <Gamma inBitDepth="32f" outBitDepth="32f" style="moncurveFwd">
        <GammaParams gamma="2.600000000000000" offset="0.050000" />
    </Gamma>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 3.2409698963 -1.5373831987 -0.4986107647
-0.9692436457  1.8759675026  0.0415550582
 0.0556300804 -0.2039769590  1.0569715500
        </Array>
    </Matrix>
    <Range inBitDepth="32f" outBitDepth="32f">
        <minInValue>  0.0000</minInValue>
        <maxInValue>  1.0000</maxInValue>
        <minOutValue>  0.0000</minOutValue>
        <maxOutValue>  1.0000</maxOutValue>
    </Range>
    <Gamma inBitDepth="32f" outBitDepth="12i" style="basicRev">
        <GammaParams gamma="2.400000000000000" />
    </Gamma>
</ProcessList>
```

CTF Syntax Summary

Here is a brief overview of the syntax of a CTF file.

In the list below, **?** means "0 or 1 occurrence", ***** means "0 or more occurrences" , and **+** means "1 or more occurrences". If not otherwise marked, a single occurence of an element is required.

ProcessList (page 1567) Attributes: id, name Contains:

Info (page 1568) ? Contains:

Copyright (page 1569) ?

Release (page 1570) ?

Description (page 1571) *

InputDescriptor (page 1572) ?

OutputDescriptor (page 1572) ?

OperatorNode + This is a virtual element — replace it by one of the substitutions below (LUT1D, LUT3D, etc.). Attributes: id, name, inBitDepth, outBitDepth, bypass Contains:

Description (page 1571) *

DynamicParameter (page 1601) ? (up to 3 for ExposureContrast) Attributes: param

Substitute each OperatorNode by any one of the following:

LUT1D (page 1574) Inherited attributes: id, name, inBitDepth, outBitDepth Attributes: interpolation, halfDomain, rawHalfs Contains:

Array (page 1576) Attributes: dim

LUT3D (page 1577) Inherited attributes: id, name, inBitDepth, outBitDepth Attributes: interpolation Contains:

Array (page 1577) Attributes: dim

Matrix (page 1579) Inherited attributes: id, name, inBitDepth, outBitDepth Contains:

Array (page 1580) Attributes: dim

Range (page 1582) Inherited attributes: id, name, inBitDepth, outBitDepth Contains:

minInValue (page 1584) ?

maxInValue (page 1584) ?

minOutValue (page 1585) ?

maxOutValue (page 1585) ?

Gamma (page 1586) Inherited attributes: id, name, inBitDepth, outBitDepth Attributes: style Contains:

GammaParams (page 1588) Attributes: channel, gamma, offset

ExposureContrast (page 1589) Inherited attributes: id, name, inBitDepth, outBitDepth Contains:

ECParams (page 1590) Attributes: exposure, contrast, pivot

Log (page 1591) Inherited attributes: id, name, inBitDepth, outBitDepth Attributes: style Contains:

LogParams (page 1592) 1 or 3 Attributes: channel, gamma, refWhite, refBlack, highlight, shadow

ASC_CDL (page 1594) Inherited attributes: id, name, inBitDepth, outBitDepth Contains:

SOPNode (page 1595) Contains:

Slope (page 1596)

Offset (page 1597)

Power (page 1597)

SatNode (page 1598) Contains:

Saturation (page 1599)

Reference (page 1599) attributes: alias, path, basepath

XML Declaration

The XML declaration is optional but recommended. It indicates to the XML parser the version of XML and character encoding used.

If it is present, the XML declaration must be the first line of the file.

Example

```
<?xml version="1.0" encoding="UTF-8"?>
```

ProcessList

The ProcessList element is the root element of a CTF file. It must contain at least one operator element.

Contains

In order:

- Info (page 1568). Optional, no more than one.
- Description (page 1571). Optional, any number.
- InputDescriptor (page 1572). Optional, no more than one.
- OutputDescriptor (page 1572). Optional, no more than one.
- At least one of LUT1D (page 1574), LUT3D (page 1577), Matrix (page 1579), Range (page 1582), Gamma (page 1586), ExposureContrast (page 1589), or Reference (page 1599) in any combination in the order in which they are to be applied.

  NOTE The inBitDepth attribute of each operator must match the outBitDepth attribute of the previous operator in the chain. See Common Operator Attributes (page 1573).

Contained By

- no parent

Attributes

id A unique identifier (required).

name A nickname for the entire color transform (optional). This is used for display and selection in applications that support Autodesk Color Management.

version Indicates the presence of Autodesk extensions to the Academy/ASC XML color transform format. Files with a value higher than that supported by the application are rejected.

inverseOf The id attribute from another CTF file. It identities this transform as the inverse of the other one. This information is useful to the color management system to detect the case where a transform and then its inverse are applied, so the transform pair may be optimized out. It is an Autodesk extension to the Academy/ASC XML color transform format.

Example

```
<ProcessList id="7214fa79-6c42-48b9-b164-3d0c5e543fe0" version="1.2">
    <Description>F65 Raw integer code values to ACES (tungsten
illuminant).</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>Academy Color Encoding Specification
(ACES)</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
```

```
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.0110238791 -0.1362526119  0.1252287328
 0.1011994481  0.9562196136 -0.0574190766
 0.0600766540 -0.1010185331  1.0409418344
        </Array>
    </Matrix>
</ProcessList>
```

Metadata Elements

The metadata elements provide information about a color transformation that can be displayed in information panels in applications.

Info

The Info element is a container for the Copyright and Release elements.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.2.

Contains

In any order:

- Copyright (page 1569). Optional, no more than one.
- Release (page 1570). Optional, no more than one.

Contained By

- ProcessList (page 1567)

Attributes

None.

Example

```
<ProcessList id="f8d905db-4ab8-4af6-8465-0db81f7e993c" version="1.2">
    <Info>
        <Copyright>Copyright 2012 Autodesk, Inc.  All rights
reserved.</Copyright>
        <Release>2013.1.pr20</Release>
    </Info>
    <Description>F65 Raw integer code values to scene-linear floats with
Rec 709 primaries.</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>scene-linear Rec 709 primaries</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
```

```
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.8779151440 -0.7941687703 -0.0837463662
-0.1768069863  1.3509995937 -0.1741926372
-0.0262011271 -0.1484222561  1.1746233702
        </Array>
    </Matrix>
</ProcessList>
```

Copyright

The Copyright element is intended to contain a copyright notice covering the CTF file.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.2.

Contains

- Text string. Use `>` for >, `<` for <, and `&` for &.

Contained By

- Info (page 1568)

Attributes

None.

Example

```
<ProcessList id="f8d905db-4ab8-4af6-8465-0db81f7e993c" version="1.2">
    <Info>
        <Copyright>Copyright 2012 Autodesk, Inc.  All rights reserved.</Copyright>

        <Release>2013.1.pr20</Release>
    </Info>
    <Description>F65 Raw integer code values to scene-linear floats with
Rec 709 primaries.</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>scene-linear Rec 709 primaries</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.8779151440 -0.7941687703 -0.0837463662
-0.1768069863  1.3509995937 -0.1741926372
-0.0262011271 -0.1484222561  1.1746233702
```

```
        </Array>
    </Matrix>
</ProcessList>
```

Release

The Release element is intended to contain the version of the SynColor library corresponding to an Autodesk-supplied CTF file.

It is an Autodesk extension to the Academy/ASC XML color transform format. You should not add this element to color transforms that you author yourself.

Introduced

Version 1.2.

Contains

- Text string. Use `>` for >, `<` for <, and `&` for &.

Contained By

- Info (page 1568)

Attributes

None.

Example

```
<ProcessList id="f8d905db-4ab8-4af6-8465-0db81f7e993c" version="1.2">
    <Info>
        <Copyright>Copyright 2012 Autodesk, Inc.  All rights
reserved.</Copyright>
        <Release>2013.1.pr20</Release>
    </Info>
    <Description>F65 Raw integer code values to scene-linear floats with
Rec 709 primaries.</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>scene-linear Rec 709 primaries</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.8779151440 -0.7941687703 -0.0837463662
-0.1768069863  1.3509995937 -0.1741926372
-0.0262011271 -0.1484222561  1.1746233702
        </Array>
    </Matrix>
</ProcessList>
```

Description

The Description element allows you to add information about the function or usage of a ProcessList or operator element. It is displayed in the metadata panel of color transforms in products that support Autodesk Color Management.

Contains

- Text string. Use `>` for >, `<` for <, and `&` for &.

Contained By

- ProcessList (page 1567)
- LUT1D (page 1574)
- LUT3D (page 1577)
- Matrix (page 1579)
- Range (page 1582)
- ExposureContrast (page 1589)
- Gamma (page 1586)
- Reference (page 1599)

Attributes

None.

Example

```
<ProcessList id="f8d905db-4ab8-4af6-8465-0db81f7e993c" version="1.2">
    <Info>
        <Copyright>Copyright 2012 Autodesk, Inc.  All rights
reserved.</Copyright>
        <Release>2013.1.pr20</Release>
    </Info>
    <Description>F65 Raw integer code values to scene-linear floats with Rec 709
primaries.</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>scene-linear Rec 709 primaries</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.8779151440 -0.7941687703 -0.0837463662
-0.1768069863  1.3509995937 -0.1741926372
-0.0262011271 -0.1484222561  1.1746233702
        </Array>
    </Matrix>
</ProcessList>
```

InputDescriptor

The InputDescriptor element allows you to add a comment describing the intended input color space and encoding for a color transform. It is does not affect the processing, but is displayed in the metadata panel of color transforms in products that support Autodesk Color Management.

Contains

- Text string. Use `>` for >, `<` for <, and `&` for &.

Contained By

- ProcessList (page 1567)

Attributes

None.

Example

Example

```
<ProcessList id="f8d905db-4ab8-4af6-8465-0db81f7e993c" version="1.2">
    <Info>
        <Copyright>Copyright 2012 Autodesk, Inc.  All rights
reserved.</Copyright>
        <Release>2013.1.pr20</Release>
    </Info>
    <Description>F65 Raw integer code values to scene-linear floats with
Rec 709 primaries.</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>scene-linear Rec 709 primaries</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.8779151440 -0.7941687703 -0.0837463662
-0.1768069863  1.3509995937 -0.1741926372
-0.0262011271 -0.1484222561  1.1746233702
        </Array>
    </Matrix>
</ProcessList>
```

OutputDescriptor

The OutputDescriptor element allows you to add a comment describing the output color space and encoding for a color transform. It is does not affect the processing, but is displayed in the metadata panel of color transforms in products that support Autodesk Color Management.

Contains

- Text string. Use `>` for >, `<` for <, and `&` for &.

Contained By

- ProcessList (page 1567)

Attributes

None.

Example

Example

```
<ProcessList id="f8d905db-4ab8-4af6-8465-0db81f7e993c" version="1.2">
    <Info>
        <Copyright>Copyright 2012 Autodesk, Inc.  All rights
reserved.</Copyright>
        <Release>2013.1.pr20</Release>
    </Info>
    <Description>F65 Raw integer code values to scene-linear floats with
Rec 709 primaries.</Description>
    <InputDescriptor>16-bit integer F65 Raw</InputDescriptor>
    <OutputDescriptor>scene-linear Rec 709 primaries</OutputDescriptor>
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 1.8779151440 -0.7941687703 -0.0837463662
-0.1768069863  1.3509995937 -0.1741926372
-0.0262011271 -0.1484222561  1.1746233702
        </Array>
    </Matrix>
</ProcessList>
```

Operator Elements

The operator elements specify the operations to be applied to the color data. You can have as many as you like in a ProcessList element, in the order in which they are to be applied.

NOTE The inBitDepth attribute of each operator must match the outBitDepth attribute of the previous operator in the chain. See Common Operator Attributes (page 1573).

Common Operator Attributes

Several attributes are common to all of the color transform operator elements.

id A unique identifier for the element. Optional.

name A friendly name for the element. Optional.

bypass Introduced in version 1.3. Specifies that the operator should not be applied during color processing. Optional. If this attribute exists, its value must be `"true"`. By using this attribute in combination with a DynamicParameter (page 1601) element that has its param attribute set to `"LOOK_SWITCH"`, you can define operators that will never affect values rendered to file but that can still be toggled on and off for display.

inBitDepth The input color data type expected by the operator. Required.

outBitDepth The output color data type produced by the operator. Required.

NOTE The inBitDepth attribute of each operator must match the outBitDepth attribute of the previous operator in the chain.

Supported Values for inBitDepth and outBitDepth

Value	Meaning
`"8i"`	8-bit integer
`"10i"`	10-bit integer
`"12i"`	12-bit integer
`"16i"`	16-bit integer
`"16f"`	16-bit floating point (half-float)
`"32f"`	32-bit floating point (single precision)

About Bit Depth

The values specified for inBitDepth and outBitDepth attributes do not affect the quantization of color values. All processing is performed using 32-bit floating-point values.

The inBitDepth and outBitDepth attributes affect only the format of the numbers used in the CTF file, such as the values used in an Array, Matrix, or Range element. For example, if you find it convenient to specify values as integers in the range [0, 1023] then you can use an inBitDepth of `"10i"`. You can still specify values outside this range, and those values will be used in the intermediate computations.

LUT1D

The LUT1D element specifies a 1D LUT to apply.

Contains

- Description (page 1571). Optional, any number.
- Array (LUT1D) (page 1576). Required, one only.
- DynamicParameter (page 1601). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

NOTE The IndexMap element is not supported. The most efficient way to achieve the same result is to precede the operator with another LUT1D element, or to combine both LUTs into a single one.

Contained By

- ProcessList (page 1567)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1573).

interpolation Optional. The only currently supported value is `"linear"`.

halfDomain Optional. If this attribute is present, its value must be `"true"`. In this case, the input domain is all possible 16-bit floating-point values, and there must be exactly 65536 entries in the Array element. For example, the unsigned integer 15360 has the same bit-pattern (0011110000000000) as the half-float value 1.0, so the 15360th entry (zero-indexed) in the Array element is the output value corresponding to an input value of 1.0.

rawHalfs Optional. If this attribute is present, its value must be `"true"`. In this case, the values in the array should be 16-bit floating point values expressed as unsigned 16-bit integers representing the equivalent bit pattern. For example, to represent the value 1.0, you would enter the integer `15360` in the Array element because it has the same bit-pattern. This allows you to specify the exact half-float values you want without relying on conversion from decimal text strings.

Example

1D LUT

```
<ProcessList id="8a52d5fb-a903-4805-8bae-24f7553bfb70" version="1.2">
    <LUT1D inBitDepth="10i" outBitDepth="32f">
        <Array dim="1024 1">
-0.014279292
-0.014160193
-0.014040368
<!-- 1021 lines omitted -->

        </Array>
    </LUT1D>
</ProcessList>
```

3×1D LUT

```
<ProcessList id="a76dbe2e-e610-49a6-8c3b-5962375a8b4a" version="1.2">
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
 </ProcessList>
```

Array (LUT1D)

The Array element specifies an array of values for a LUT or matrix. When it is contained by a LUT1D element, the format of its contents is different than when it is contained by LUT3D or Matrix elements.

Contains

- A series of numeric values specifying the output values of the LUT. The corresponding input values are implied by the inBitDepth (page 1573) of the containing element. For integers, the first value corresponds to an input of 0 and the last to an input of $(2^{bitdepth} - 1)$. For floating-point numbers, the first value corresponds to an input of 0.0 and the last to an input of 1.0. Input values outside these domains are clamped.

  The intervals between the input values are uniform and depend on the number of entries. For example, if there are 5 floating-point entries, they correspond to inputs of 0.0, 0.25, 0.5, 0.75, and 1.0.

  For a 1D LUT, there is one value per entry which is used for all color channels. For a 3×1D LUT, there are three values per entry specifying the separate R, G, and B output values.

Contained By

- LUT1D (page 1574)

Attributes

dim Two integers representing the dimensions of the array. The first value is the number of entries in the contents of the Array element. The second value is number of values per entry: 1 for a single value applied to all channels or 3 for for separate values applied to the R, G, and B channels respectively.

The dim attribute is required, and its value must match the number of entries actually present.

Example

1D LUT

```
<ProcessList id="8a52d5fb-a903-4805-8bae-24f7553bfb70" version="1.2">
    <LUT1D inBitDepth="10i" outBitDepth="32f">
        <Array dim="1024 1">
-0.014279292
-0.014160193
-0.014040368
<!-- 1021 values omitted -->

        </Array>
    </LUT1D>
</ProcessList>
```

3×1D LUT

```
<ProcessList id="a76dbe2e-e610-49a6-8c3b-5962375a8b4a" version="1.2">
    <LUT1D inBitDepth="16i" outBitDepth="32f">
        <Array dim="2 3">
 -0.092903227  -0.092903227  -0.092903227
 11.798709869  11.798709869  11.798709869
        </Array>
    </LUT1D>
 </ProcessList>
```

See Also

- Array (LUT3D) (page 1577)
- Array (Matrix) (page 1580)

LUT3D

The LUT3D element specifies a 3D LUT to apply.

Contains

- Description (page 1571). Optional, any number.
- Array (LUT3D) (page 1577). Required, one only.
- DynamicParameter (page 1601). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

NOTE The IndexMap element is not supported. The most efficient way to achieve the same result is to precede the operator with a LUT1D element.

Contained By

- ProcessList (page 1567)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1573).

interpolation Supported values are `"trilinear"` and `"tetrahedral"`.

This attribute is optional. If it is not specified, the default is `"trilinear"`.

Example

```
<ProcessList id="b5b90615-573b-4959-a838-f35e0e588ac2" version="1.2">
    <LUT3D inBitDepth="32f" interpolation="tetrahedral" outBitDepth="32f">
        <Array dim="48 48 48 3">
 0.000669299  0.000704187  0.000766901
 0.001848371  0.001297010  0.010368987
 0.002599007  0.001144927  0.023227118
<!-- 48*48*48-3 lines omitted -->

        </Array>
    </LUT3D>
</ProcessList>
```

Array (LUT3D)

The Array element specifies an array of values for a LUT or matrix. When it is contained by a LUT3D element, the format of its contents is different than when it is contained by LUT1D or Matrix elements.

Contains

- A series of numeric values specifying the output values of the LUT. The corresponding input values are implied by the inBitDepth (page 1573) of the containing element and the number of entries (uniform distribution).

  For a 3D LUT, each entry has three values specifying the separate R, G, and B output values. The entries are in order from minimum to maximum with the index of the last column changing fastest and the index of the first column changing slowest. For example, the order of entries for a 2×2×2 cube is:

Entry	R	G	B
1	0	0	0
2	0	0	1
3	0	1	0
4	0	1	1
5	0	0	0
6	0	0	1
7	0	1	0
8	0	1	1

Contained By

- LUT3D (page 1577)

Attributes

dim Four integers representing the dimensions of a 3D cube followed by followed by the number of components per entry.

Only 3D LUTs with the same dimension in all three channels are supported, so the first three values must be the same. The fourth value must be 3.

The dim attribute is required, and its value must match the number of entries actually present.

Example

```
<ProcessList id="b5b90615-573b-4959-a838-f35e0e588ac2" version="1.2">
    <LUT3D inBitDepth="32f" interpolation="tetrahedral" outBitDepth="32f">
        <Array dim="48 48 48 3">
 0.000669299  0.000704187  0.000766901
 0.001848371  0.001297010  0.010368987
 0.002599007  0.001144927  0.023227118

<!-- 48*48*48-3 lines omitted -->

        </Array>
```

```
    </LUT3D>
</ProcessList>
```

See Also

- Array (LUT1D) (page 1576)
- Array (Matrix) (page 1580)

Matrix

The Matrix element specifies a matrix for multiplying color values, and optionally applying an offset.

Contains

- Description (page 1571). Optional, any number.
- Array (Matrix) (page 1580). Required, one only.
- DynamicParameter (page 1601). Optional, no more than one. The param attribute must be set to "DEFAULT_LOOK".

Contained By

- ProcessList (page 1567)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1573).

Example

3x3 Matrix

```
<ProcessList id="af6a6c73-aae9-4be6-8051-a796bc480b1c" version="1.2">
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 0.7841929793  0.0904410034  0.1253670007
 0.0445200019  1.0391299725 -0.0836500004
 0.0358299986 -0.3302420080  1.2944129705
        </Array>
    </Matrix>
</ProcessList>
```

4x4 Matrix

```
<ProcessList id="d6783d48-dd96-4c71-99af-f3be0cc9392e" version="1.2">
    <Matrix inBitDepth="32f" outBitDepth="10i">
        <Array dim="4 4 3">
499.99996948   0.00000000   0.00000000  94.99999237
  0.00000000 499.99996948   0.00000000  94.99999237
  0.00000000   0.00000000 499.99996948  94.99999237
  0.00000000   0.00000000   0.00000000   1.00000000
        </Array>
    </Matrix>
 </ProcessList>
```

Array (Matrix)

The Array element specifies an array of values for a LUT or matrix. When it is contained by a Matrix element, the format of its contents is different than when it is contained by LUT1D or LUT3D elements.

Updated

Version 1.3.

Contains

- A series of numeric values specifying the elements of the matrix.

  In a Matrix, the values specify the entries of the matrix which are serialized row by row from top to bottom and from left to right. The scaling of the array values depends on the input and output bit-depths. Color values are treated as column vectors for the purpose of multiplication.

  The scaling of the array values depends on the input and output bit-depths. For example if the input depth is 10i and the output depth is 12i, then a matrix containing entries of 4095/1023 along the main diagonal and entries of 0 everywhere else is the identity transform.

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} \\ a_{21} & a_{22} & a_{23} \\ a_{31} & a_{32} & a_{33} \end{bmatrix} \begin{bmatrix} R_{in} \\ G_{in} \\ B_{in} \end{bmatrix} = \begin{bmatrix} R_{out} \\ G_{out} \\ B_{out} \end{bmatrix}$$

$$\begin{aligned} R_{out} &= a_{11}R_{in} + a_{12}G_{in} + a_{13}B_{in} \\ G_{out} &= a_{21}R_{in} + a_{22}G_{in} + a_{23}B_{in} \\ B_{out} &= a_{31}R_{in} + a_{32}G_{in} + a_{33}B_{in} \end{aligned}$$

3×3 Matrix Multiplication

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} & offset_R \\ a_{21} & a_{22} & a_{23} & offset_G \\ a_{31} & a_{32} & a_{33} & offset_B \end{bmatrix} \begin{bmatrix} R_{in} \\ G_{in} \\ B_{in} \\ 1.0 \end{bmatrix} = \begin{bmatrix} R_{out} \\ G_{out} \\ B_{out} \end{bmatrix}$$

$$\begin{aligned} R_{out} &= a_{11}R_{in} + a_{12}G_{in} + a_{13}B_{in} + offset_R \\ G_{out} &= a_{21}R_{in} + a_{22}G_{in} + a_{23}B_{in} + offset_G \\ B_{out} &= a_{31}R_{in} + a_{32}G_{in} + a_{33}B_{in} + offset_B \end{aligned}$$

3×4 Matrix Multiplication

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} & a_{14} \\ a_{21} & a_{22} & a_{23} & a_{24} \\ a_{31} & a_{32} & a_{33} & a_{34} \\ a_{41} & a_{42} & a_{43} & a_{44} \end{bmatrix} \begin{bmatrix} R_{in} \\ G_{in} \\ B_{in} \\ A_{in} \end{bmatrix} = \begin{bmatrix} R_{out} \\ G_{out} \\ B_{out} \\ A_{out} \end{bmatrix}$$

$$R_{out} = a_{11}R_{in} + a_{12}G_{in} + a_{13}B_{in} + a_{14}A_{in}$$
$$G_{out} = a_{21}R_{in} + a_{22}G_{in} + a_{23}B_{in} + a_{24}A_{in}$$
$$B_{out} = a_{31}R_{in} + a_{32}G_{in} + a_{33}B_{in} + a_{34}A_{in}$$
$$A_{out} = a_{41}R_{in} + a_{42}G_{in} + a_{43}B_{in} + a_{44}A_{in}$$

4×4 Matrix Multiplication

$$\begin{bmatrix} a_{11} & a_{12} & a_{13} & a_{14} & offset_R \\ a_{21} & a_{22} & a_{23} & a_{24} & offset_G \\ a_{31} & a_{32} & a_{33} & a_{34} & offset_B \\ a_{41} & a_{42} & a_{43} & a_{44} & offset_A \end{bmatrix} \begin{bmatrix} R_{in} \\ G_{in} \\ B_{in} \\ A_{in} \\ 1.0 \end{bmatrix} = \begin{bmatrix} R_{out} \\ G_{out} \\ B_{out} \\ A_{out} \end{bmatrix}$$

$$R_{out} = a_{11}R_{in} + a_{12}G_{in} + a_{13}B_{in} + a_{14}A_{in} + offset_R$$
$$G_{out} = a_{21}R_{in} + a_{22}G_{in} + a_{23}B_{in} + a_{24}A_{in} + offset_G$$
$$B_{out} = a_{31}R_{in} + a_{32}G_{in} + a_{33}B_{in} + a_{34}A_{in} + offset_B$$
$$A_{out} = a_{41}R_{in} + a_{42}G_{in} + a_{43}B_{in} + a_{44}A_{in} + offset_A$$

4×5 Matrix Multiplication

Contained By

- Matrix (page 1579)

Attributes

dim Specifies the size of the matrix and the number of channels to operate on.

Value	Meaning
`"3 3 3"`	A 3×3 matrix operating on RGB values.
`"3 4 3"`	A 3×4 matrix operating on RGB values. Introduced in version 1.3.
`"4 4 4"`	A 4×4 matrix operating on RGBA values. Introduced in version 1.3.

Value	Meaning
"4 5 4"	A 4×5 matrix operating on RGBA values. Introduced in version 1.3.
"4 4 3"	A 4×4 matrix operating on RGB values. If you use this option, the bottom row of the matrix should be [0 0 0 1]. This option is valid only if the version attribute of the ProcessList element is "1.2". For versions 1.3 and later, use "3 4 3" instead.

Example

3x3 Matrix Operating on RGB Values

```
<ProcessList id="af6a6c73-aae9-4be6-8051-a796bc480b1c" version="1.2">
    <Matrix inBitDepth="32f" outBitDepth="32f">
        <Array dim="3 3 3">
 0.7841929793  0.0904410034  0.1253670007
 0.0445200019  1.0391299725 -0.0836500004
 0.0358299986 -0.3302420080  1.2944129705
        </Array>
    </Matrix>
</ProcessList>
```

3x4 Matrix Operating on RGB Values

```
<ProcessList id="53d366de-e200-476f-b3fd-ed1ca7044197" version="1.3">
    <Matrix inBitDepth="32f" outBitDepth="10i">
        <Array dim="3 4 3">
499.99996948   0.00000000   0.00000000  94.99999237
  0.00000000 499.99996948   0.00000000  94.99999237
  0.00000000   0.00000000 499.99996948  94.99999237
        </Array>
    </Matrix>
 </ProcessList>
```

See Also

- Array (LUT1D) (page 1576)
- Array (LUT3D) (page 1577)

Range

The Range element maps the input domain to the output range by scaling and offsetting values.

If a minInValue is present, then minOutValue must also be present and the result is clamped at the low end. Similarly, if maxInValue is present, then maxOutValue must also be present and the result is clamped at the high end. If none of minInValue, minOutValue, maxInValue, or maxOutValue are present, then the Range operator performs only bit-depth conversion.

The scaling of minInValue and maxInValue depends on the input bit-depth, and the scaling of minOutValue and maxOutValue depends the output bit-depth.

If both minimum and maximum values are specified, then the formula for Range is:

$$out = \min\left(\max\left(in\left(\frac{max_{out} - min_{out}}{max_{in} - min_{in}}\right) + min_{out} - min_{in}\left(\frac{max_{out} - min_{out}}{max_{in} - min_{in}}\right),\ min_{out}\right),\ max_{out}\right)$$

If only minimum values are specified, the formula is:

$$out = \max\left(in\left(\frac{\text{size}(outBitDepth)}{\text{size}(inBitDepth)}\right) + min_{out} - min_{in}\left(\frac{\text{size}(outBitDepth)}{\text{size}(inBitDepth)}\right),\ min_{out}\right)$$

If only maximum values are specified, the formula is:

$$out = \min\left(in\left(\frac{\text{size}(outBitDepth)}{\text{size}(inBitDepth)}\right) + max_{out} - max_{in}\left(\frac{\text{size}(outBitDepth)}{\text{size}(inBitDepth)}\right),\ max_{out}\right)$$

Where:

$$size(\text{int}) = 2^{bitDepth} - 1$$

$$size(\text{float}) = 1.0$$

Contains

- Description (page 1571). Optional, any number.
- minInValue (page 1584). Optional, no more than one.
- minOutValue (page 1585). Optional, no more than one.
- maxInValue (page 1584). Optional, no more than one.
- maxOutValue (page 1585). Optional, no more than one.
- DynamicParameter (page 1601). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

Contained By

- ProcessList (page 1567)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1573).

Example

```
<ProcessList id="7f5b1560-fec4-4603-aa8d-c55bf5e22280" version="1.2">
    <Description>10-bit full range to SMPTE (legal) range.</Description>
    <InputDescriptor>generic RGB</InputDescriptor>
    <OutputDescriptor>generic RGB</OutputDescriptor>
    <Range inBitDepth="10i" outBitDepth="10i">
        <minInValue>  0.0000</minInValue>
        <maxInValue>1023.0000</maxInValue>
        <minOutValue> 64.0000</minOutValue>
        <maxOutValue>940.0000</maxOutValue>
    </Range>
</ProcessList>
```

minInValue

The minValue element specifies the minimum input value for a Range element.

Contains

- A single numeric value used for all channels.

Contained By

- Range (page 1582)

Attributes

None.

Example

```
<ProcessList id="7f5b1560-fec4-4603-aa8d-c55bf5e22280" version="1.2">
    <Description>10-bit full range to SMPTE (legal) range.</Description>
    <InputDescriptor>generic RGB</InputDescriptor>
    <OutputDescriptor>generic RGB</OutputDescriptor>
    <Range inBitDepth="10i" outBitDepth="10i">
        <minInValue>  0.0000</minInValue>
        <maxInValue>1023.0000</maxInValue>
        <minOutValue>  64.0000</minOutValue>
        <maxOutValue>940.0000</maxOutValue>
    </Range>
</ProcessList>
```

maxInValue

The maxInValue element specifies the maximum input value for a Range element.

Contains

- A single numeric value used for all channels.

Contained By

- Range (page 1582)

Attributes

None.

Example

```
<ProcessList id="7f5b1560-fec4-4603-aa8d-c55bf5e22280" version="1.2">
    <Description>10-bit full range to SMPTE (legal) range.</Description>
    <InputDescriptor>generic RGB</InputDescriptor>
    <OutputDescriptor>generic RGB</OutputDescriptor>
    <Range inBitDepth="10i" outBitDepth="10i">
        <minInValue>  0.0000</minInValue>
```

```
        <maxInValue>1023.0000</maxInValue>
        <minOutValue> 64.0000</minOutValue>
        <maxOutValue>940.0000</maxOutValue>
    </Range>
</ProcessList>
```

minOutValue

The minOutValue element specifies the minimum output value for a Range element.

Contains

- A single numeric value used for all channels.

Contained By

- Range (page 1582)

Attributes

None.

Example

```
<ProcessList id="7f5b1560-fec4-4603-aa8d-c55bf5e22280" version="1.2">
    <Description>10-bit full range to SMPTE (legal) range.</Description>
    <InputDescriptor>generic RGB</InputDescriptor>
    <OutputDescriptor>generic RGB</OutputDescriptor>
    <Range inBitDepth="10i" outBitDepth="10i">
        <minInValue>  0.0000</minInValue>
        <maxInValue>1023.0000</maxInValue>
        <minOutValue> 64.0000</minOutValue>
        <maxOutValue>940.0000</maxOutValue>
    </Range>
</ProcessList>
```

maxOutValue

The maxOutValue element specifies the maximum output value for a Range element.

Contains

- A single numeric value used for all channels.

Contained By

- Range (page 1582)

Attributes

None.

Example

```
<ProcessList id="7f5b1560-fec4-4603-aa8d-c55bf5e22280" version="1.2">
    <Description>10-bit full range to SMPTE (legal) range.</Description>
    <InputDescriptor>generic RGB</InputDescriptor>
    <OutputDescriptor>generic RGB</OutputDescriptor>
    <Range inBitDepth="10i" outBitDepth="10i">
        <minInValue>  0.0000</minInValue>
        <maxInValue>1023.0000</maxInValue>
        <minOutValue> 64.0000</minOutValue>
        <maxOutValue>940.0000</maxOutValue>
    </Range>
</ProcessList>
```

Gamma

The Gamma element applies a gamma correction to color values.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.2.

Contains

- Description (page 1571). Optional, any number.
- GammaParams (page 1588). Required, maximum three total.
- DynamicParameter (page 1601). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

Contained By

- ProcessList (page 1567)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1573).

style Specifies the type of power law curve. Required.

NOTE The following equations assume that the input and output bit-depths are floating-point. For integers, the values are normalized to [0.0, 1.0]. In other words, the gamma and offset values stored in the CTF file do not depend on the input and output bit depths.

Value	Meaning
`"basicFwd"`	Applies the power law with the gamma value specified in the GammaParams element. The GammaParams element must not have an offset attribute. $$out = (\max(0, in))^{gamma}$$

Value	Meaning
"basicRev"	Applies the power law with the inverse of gamma value. The GammaParams element must not have an offset attribute. $$out = (\max(0, in))^{\frac{1}{gamma}}$$
"moncurve-Fwd"	Applies a function that has a linear segment near the origin (black) and a power-law segment at higher values. At the junction, the curve is continuous in noth value and tangent. This type of curve is used in the Rec. 709, sRGB, and CIE L* equations. The GammaParams element must have an offset attribute. $$\gamma = \text{gamma} \quad k = \text{offset} \quad x = \text{in} \quad y = \text{out}$$ $$x \leq \frac{k}{\gamma - 1} \qquad y = \frac{\gamma - 1}{k}\left(\frac{k\gamma}{(\gamma - 1)(1 + k)}\right)^{\gamma} x$$ $$x > \frac{k}{\gamma - 1} \qquad y = \left(\max\left(0, \left(\frac{x + k}{1 + k}\right)\right)\right)^{\gamma}$$
"mon-curveRev"	Applies a function that has a linear segment near the origin (black) and an inverse power-law segment at higher values. The GammaParams element must have an offset attribute. $$\gamma = \text{gamma} \quad k = \text{offset} \quad x = \text{in} \quad y = \text{out}$$ $$x \leq \left(\frac{k\gamma}{(\gamma - 1)(1 + k)}\right)^{\gamma} \qquad y = \left(\frac{\gamma - 1}{k}\right)^{\gamma - 1}\left(\frac{1 + k}{\gamma}\right)^{\gamma} x$$ $$x > \left(\frac{k\gamma}{(\gamma - 1)(1 + k)}\right)^{\gamma} \qquad y = \left(\max(0, x)\right)^{\frac{1}{\gamma}}(1 + k) - k$$

Example

Without Offset

```
<ProcessList id="9e999646-3e76-4374-814c-e4c46c6438de" version="1.2">
    </Matrix>
    <Gamma inBitDepth="32f" outBitDepth="12i" style="basicRev">
        <GammaParams gamma="2.400000000000000" />
    </Gamma>
</ProcessList>
```

With Offset

```
<ProcessList id="1b86aac2-1fdb-4dd8-bc6b-54f3551b9bd4" version="1.2">
    <Gamma inBitDepth="32f" outBitDepth="32f" style="moncurveFwd">
        <GammaParams gamma="2.600000000000000" offset="0.050000" />
```

```
    </Gamma>
</ProcessList>
```

GammaParams

The GammaParams element defines the gamma value and optional offset for a Gamma element.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.2.

Contains

No content.

Contained By

- Gamma (page 1586)

Attributes

channel The color channel to apply gamma correction to. Possible values are "R", "G", and "B". Optional.

If this attribute is not specified, the gamma correction is applied to all channels.

gamma The power value to use. Valid values range from 0.1 to 10.0 inclusive. Required.

offset The offset to use. Valid values range from 10^{-6} to 0.9 inclusive. Optional.

If it is used, the Gamma element's style attribute should be set to "moncurveFwd" or "moncurveRev". Othewise, the style attribuute should be "basicFwd" or "basicRev".

Example

Without Offset

```
<ProcessList id="9e999646-3e76-4374-814c-e4c46c6438de" version="1.2">
    </Matrix>
    <Gamma inBitDepth="32f" outBitDepth="12i" style="basicRev">
        <GammaParams gamma="2.400000000000000" />
    </Gamma>
</ProcessList>
```

With Offset

```
<ProcessList id="1b86aac2-1fdb-4dd8-bc6b-54f3551b9bd4" version="1.2">
    <Gamma inBitDepth="32f" outBitDepth="32f" style="moncurveFwd">
        <GammaParams gamma="2.600000000000000" offset="0.050000" />
    </Gamma>
</ProcessList>
```

Separate Channels

```
<ProcessList id="id">
    <Gamma inBitDepth="32f" outBitDepth="10i" style="basicRev">
```

```
        <GammaParams channel="R" gamma="2.4" />
    <GammaParams channel="G" gamma="2.35" />
    <GammaParams channel="B" gamma="2.2" />
    </Gamma>
</ProcessList>
```

ExposureContrast

The ExposureContrast element simulates adjusting the exposure of an image, and also modifies the contrast between light and dark colors.

It is an Autodesk extension to the Academy/ASC XML color transform format.

If a color transform is used for display, then depending on the application users can still adjust the exposure and contrast interactively as long as the CTF file contains at least one ExposureContrast element with DynamicParameter sub-elements. In this case, the corresponding parameter values defined in the ECParams sub-element are ignored for display only, and instead the values set in the application are applied at each point in the transform chain that an ExposureContrast element with DynamicParameter sub-elements occurs.

Introduced

Version 1.2.

Contains

- Description (page 1571). Optional, any number.
- ECParams (page 1590). Required, one only.
- DynamicParameter (page 1601). Optional, up to 3. Each param attribute must be set to a different value.

Contained By

- ProcessList (page 1567)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1573).

style Declares the input data type. Required.

NOTE The following equations assume that the input and output bit-depths are floating-point. For integers, the values are normalized to [0.0, 1.0]. In other words, the exposure offset, contrast level, and pivot values stored in the CTF file do not depend on the input and output bit depths.

Value	Meaning
`"linear"`	$out = pivot \times \max\left(0,\ \frac{2^{exposure}\, in}{pivot}\right)^{contrast}$
`"video"`	$out = pivot^{\frac{1}{1.83}} \times \left(\max\left(0,\ in \times \left(\frac{2^{exposure}}{pivot}\right)^{\frac{1}{1.83}}\right)\right)^{contrast}$

Value	Meaning
"log"	$out = \left(in + exposure \frac{0.6}{2.046}\log_{10}(2) - \left(\frac{0.6}{2.046}\log_{10}\left(\frac{pivot}{0.9}\right) + \frac{685}{1023}\right)\right) contrast + \left(\frac{0.6}{2.046}\log_{10}\left(\frac{pivot}{0.9}\right) + \frac{685}{1023}\right)$

Example

```
<ProcessList id="561c5aba-8709-477c-8211-7ec4a6fe8820">
   <ExposureContrast inBitDepth="16f" outBitDepth="32f" style="linear">
      <ECParams exposure="0.65" contrast="1.2" pivot="1" />
      <DynamicParameter param="EXPOSURE" />
      <DynamicParameter param="CONTRAST" />
   </ExposureContrast>
</ProcessList>
```

ECParams

The ECParams element specifies the parameters for an ExposureContrast element.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.2.

Contains

No content.

Contained By

- ExposureContrast (page 1589)

Attributes

exposure A single decimal value representing the exposure offset in stops. A value of 0.0 corresponds to no exposure offset. Required.

contrast A single decimal value representing the contrast level. A value of 1.0 corresponds to no contrast adjustment. Required.

pivot A single decimal value representing the color value used as a pivot for adjusting contrast. A value of 0.18 will pivot around mid-gray. Required.

Example

```
<ProcessList id="561c5aba-8709-477c-8211-7ec4a6fe8820">
   <ExposureContrast inBitDepth="16f" outBitDepth="32f" style="linear">
      <ECParams exposure="0.65" contrast="1.2" pivot="1" />
      <DynamicParameter param="EXPOSURE" />
      <DynamicParameter param="CONTRAST" />
   </ExposureContrast>
</ProcessList>
```

Log

The Log operator applies a logarithmic or anti-logarithmic function to convert between linear and Cineon-style encodings.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

- Description (page 1571). Optional, any number.
- LogParams (page 1592). Required if the style attribute is `"logToLin"` or `"linToLog"`. There should be one occurence for values applied to all RGB channels, or three occurences for different values applied to R, G, and B separately.
- DynamicParameter (page 1601). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

Contained By

ProcessList (page 1567)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1573).

style Specifies the formula to use for the conversion. Required.

NOTE The following equations assume that the input and output bit-depths are floating-point. For integers, the values are normalized to [0.0, 1.0]. In other words, the gamma, reference white, reference black, highlight, and shadow values stored in the CTF file do not depend on the input and output bit depths.

In the following equations:

$$\text{FLOAT_MIN} = 1.1754943508222875 \times 10^{-38}$$

$$linearRefBlack = 10^{\min\left(-0.0001,\ \frac{(refBlack - refWhite) \times 0.002}{gamma}\right)}$$

$$gain = \frac{highlight - shadow}{1 - linearRefBlack}$$

Value	Meaning
`"log10"`	$out = \log_{10}(\max(in,\ \text{FLOAT_MIN}))$
`"log2"`	$out = \log_{2}(\max(in,\ \text{FLOAT_MIN}))$

Value	Meaning
"antiLog10"	$out = 10^{in}$
"antiLog2"	$out = 2^{in}$
"logToLin"	$relativeExposure = 10^{\frac{(1023 \times in - refWhite) \times 0.002}{gamma}}$ $out = (relativeExposure - linearRefBlack) \times gain + shadow$
"linToLog"	$relativeExposure = linearRefBlack + \frac{in - shadow}{gain}$ $out = \frac{refWhite + \log_{10}(\max(\text{FLOAT_MIN}, relativeExposure)) \times \frac{gamma}{0.002}}{1023}$

Example

All Channels

```
<ProcessList id="19510ea6-69a0-4d3d-bc75-d0b36d458f62" name="Log op 1"
version="1.3">
     <Log inBitDepth="16f" outBitDepth="12i" style="linToLog">
        <LogParams gamma="0.6" refWhite="685" refBlack="95" highlight="1.0"
 shadow="0.0005"/>
    </Log>
</ProcessList>
```

Separate Channels

```
<ProcessList id="19510ea6-69a0-4d3d-bc75-d0b36d458f62" name="Log op 1"
version="1.3">
     <Log inBitDepth="16f" outBitDepth="12i" style="linToLog">
        <LogParams channel="R" gamma="0.5" refWhite="685" refBlack="95"
highlight="1.0" shadow="0.0005"/>
        <LogParams channel="G" gamma="0.6" refWhite="685" refBlack="95"
highlight="1.0" shadow="0.0005"/>
        <LogParams channel="B" gamma="0.65" refWhite="685" refBlack="95"
highlight="1.0" shadow="0.0005"/>
    </Log>
</ProcessList>
```

LogParams

The LogParams element defines the parameter values used by a Log element.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

No content.

Contained By

Log (page 1591)

Attributes

channel Specifies the channel to apply the parameter values to. Possible values are `"R"`, `"G"`, and `"B"`.

Optional. If it is omitted, the values are applied to each of the RGB channles.

gamma Combined film and video gamma (contrast in log space). A typical value is 0.6.

refWhite Value in log space that maps to highlight in linear space. Specified on a 10-bit scale (e.g. 685 is typical).

refBlack Value in log space that maps to shadow in linear space. Specified on a 10-bit scale (e.g. 95 is typical).

highlight Value in linear space that maps to refWhite in log space. Specified on a floating-point scale (e.g. 1.0 is typical). Note that log values above refWhite are not clamped and will map to values above highlight in linear space.

shadow Value in linear space that maps to refBlack in log space. Specified on a floating-point scale (e.g. 0.0 is typical). Note that log values below refBlack are not clamped and will map to values below shadow in linear space (possibly even negative).

Example

All Channels

```
<ProcessList id="19510ea6-69a0-4d3d-bc75-d0b36d458f62" name="Log op 1"
version="1.3">
     <Log inBitDepth="16f" outBitDepth="12i" style="linToLog">
          <LogParams gamma="0.6" refWhite="685" refBlack="95" highlight="1.0"
shadow="0.0005"/>
     </Log>
</ProcessList>
```

Separate Channels

```
<ProcessList id="19510ea6-69a0-4d3d-bc75-d0b36d458f62" name="Log op 1"
version="1.3">
     <Log inBitDepth="16f" outBitDepth="12i" style="linToLog">
          <LogParams channel="R" gamma="0.5" refWhite="685" refBlack="95"
highlight="1.0" shadow="0.0005"/>
     <LogParams channel="G" gamma="0.6" refWhite="685" refBlack="95"
highlight="1.0" shadow="0.0005"/>
     <LogParams channel="B" gamma="0.65" refWhite="685" refBlack="95"
highlight="1.0" shadow="0.0005"/>
     </Log>
</ProcessList>
```

ASC_CDL

The ASC_CDL operator represents a color transform in the American Society of Cinematographers' Color Decision List format (*.ccc or *.cdl).

An ASC CDL transform consists of a slope-offset-power (SOP) function applied individually to each of the RGB channels, followed by a saturation function applied equally to all channels.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

- Description (page 1571). Optional, any number.
- SOPNode (page 1595). Optional, no more than one. If absent, no slope-offset-power function is applied.
- SatNode (page 1598). Optional, no more than one. If absent, no saturation function is applied.
- DynamicParameter (page 1601). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

Contained By

- ProcessList (page 1567)

Attributes

id, name, bypass, inBitDepth, outBitDepth See Common Operator Attributes (page 1573).

style Determines the formula applied by the operator.

NOTE The equations assume that the input and output bit-depths are floating-point. For integers, the values are normalized to [0.0, 1.0]. In other words, the slope, offset, power, and saturation values stored in the CTF file do not depend on the input and output bit depths.

NOTE The luma weights assume that the image uses the primaries specified by Rec. 709 (for example, HD and sRGB).

Value	Meaning
`"v1.2_Fwd"`	$out_{SOP} = in_{SAT} = (\text{clamp}_{[0,1]}(in \times slope + offset))^{power}$ $luma = 0.2126 \times in_{SAT_R} + 0.7152 \times in_{SAT_G} + 0.0722 \times in_{SAT_B}$ $out = \text{clamp}_{[0,1]}(luma + saturation \times (in_{SAT} - luma))$
`"v1.2_Rev"`	The inverse of `"v1.2_Fwd"`.
`"noClampFwd"`	The same as `"v1.2_Fwd"`, but without the two clamp functions. If $(in \times slope + offset) < 0$

Value	Meaning
	then no power function is applied.
"noClampRev"	The inverse of "noClampFwd".

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
    <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

SOPNode

The SOPNode element defines the slope-offset-power function of an ASC CDL color transform.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

- Slope (page 1596). Optional, no more than one. If absent, the default is 1.0 for all channels, that is, no gain is applied.
- Offset (page 1597). Optional, no more than one. If absent, the default is 0.0 for all channels, that is, no offset is applied.
- Power (page 1597). Optional, no more than one. If absent, the default is 1.0 for all channels, that is, no gamma is applied.

Contained By

- ASC_CDL (page 1594)

Attributes

None.

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
   <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

Slope

The Offset element contains the RGB slope values of the slope-offset-power function of an ASC CDL color transform.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

Three decimal values representing the R, G, and B slope (gain) values.

Contained By

- SOPNode (page 1595)

Attributes

None.

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
   <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

Offset

The Offset element contains the RGB offset values of the slope-offset-power function of an ASC CDL color transform.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

Three decimal values representing the R, G, and B offset values.

Contained By

- SOPNode (page 1595)

Attributes

None.

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
   <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

Power

The Offset element contains the RGB power values of the slope-offset-power function of an ASC CDL color transform.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

Three decimal values representing the R, G, and B power (gamma) values.

Contained By

- SOPNode (page 1595)

Attributes

None.

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
   <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

SatNode

The SatNode element defines the saturation function of an ASC CDL color transform.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

- Saturation (page 1599). Required, one only.

Contained By

- ASC_CDL (page 1594)

Attributes

None.

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
   <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
```

```
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

Saturation

The Saturation element defines the value used by the saturation function of an ASC CDL color transform.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.3.

Contains

A single decimal value applied to all color channels.

Contained By

- SatNode (page 1598)

Attributes

None.

Example

```
<ProcessList version="1.3" id="7b42c76a-8c7a-4142-8b6e-1c467084075a">
   <ASC_CDL id="cc01234" inBitDepth="16f" outBitDepth="16f" style="v1.2_Fwd">

        <Description>scene 1 exterior look</Description>
        <SOPNode>
            <Slope>1.000000  1.000000  0.900000</Slope>
            <Offset>-0.030000  -0.020000  0.000000</Offset>
            <Power>1.2500000  1.000000  1.000000</Power>
        </SOPNode>
        <SatNode>
            <Saturation>1.700000</Saturation>
        </SatNode>
    </ASC_CDL>
</ProcessList>
```

Reference

The Reference element refers to another CTF file by path or by alias. The external file is imported into the corresponding location of the ProcessList.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Introduced

Version 1.2.

Contains

- Description (page 1571). Optional, any number.
- DynamicParameter (page 1601). Optional, no more than one. The param attribute must be set to `"DEFAULT_LOOK"`.

Contained By

- ProcessList (page 1567)

Attributes

It is necessary to specify either a path or an alias attribute, but not both.

id, **name**, **bypass**, **inBitDepth**, **outBitDepth** See Common Operator Attributes (page 1573).

Note that the values specified here override the values in the external file.

alias A short name for the path and file name of another CTF file. The alias must be defined in a Ref (page 1606) element in the SynColor configuration file (page 1602). By default, the following aliases are available, and can be changed from the user interface of some applications that support Autodesk Color Management:

- The `"graphicsMonitor"` alias should refer to the appropriate transform for display on the workstation monitor.
- The `"broadcastMonitor"` alias should refer to the appropriate transform for display on the broadcast monitor.
- The `"currentMonitor"` alias gets resolved to either the value of the `"graphicsMonitor"` alias or the value of the `"broadcastMonitor"` alias depending on where an image is displayed.
- The `"defaultLook"` alias should refer to the transform that re-creates the look used on set.

You can create additional aliases as desired.

path The path and file name of another CTF file. The path is relative to basePath if that attribute is defined; otherwise, the path is absolute.

basePath The base location for a relative path.

Value	Meaning
Autodesk	The installation location of the color transforms supplied with Autodesk applications. The exact path is resolved based on the version of Autodesk Color Management used by the running application.
Shared	The shared location for custom color transforms for any application that uses Autodesk Color Management. The location is set in the SharedHome (page 1604) of the SynColor configuration file (page 1602).

Example

```
<ProcessList id="561c5aba-8709-477c-8211-7ec4a6fe8820">
    <Reference basePath="Autodesk" inBitDepth="32f" outBitDepth="32f"
path="RRT+ODT/ACES_to_CIE-XYZ_v0.1.1.ctf" />
    <Reference alias="graphicsMonitor" inBitDepth="32f" outBitDepth="8i" />

    <Reference inBitDepth="8i" outBitDepth="8i" path="lut1d_example.xml"
basePath="Shared" />
    <Reference inBitDepth="8i" outBitDepth="8i" path="/zeus/toto/toto.xml"
 />
</ProcessList>
```

DynamicParameter

The DynamicParameter element signals to a compatible application that a parameter is dynamic. If the transform is used for display, the parameter value defined in the CTF file is ignored and the value set in the application is used instead.

It is an Autodesk extension to the Academy/ASC XML color transform format.

Updated

Version 1.3.

Contains

No content.

Contained By

- Any operator element (LOOK_SWITCH)
- ExposureContrast (page 1589) (EXPOSURE, CONTRAST).

Attributes

param Specifies the parameter that will be overridden by the application when the CTF file is applied to the display.

Value	Meaning
`"EXPOSURE"`	Allows the exposure value to be overridden when the transform is used for display in applications that support this ability. This value can be used only when the DynamicParameter element is a child of an ExposureContrast (page 1589) element.
`"CONTRAST"`	Allows the contrast value to be overridden when the transform is used for display in applications that support this ability. This value can be used only when the DynamicParameter element is a child of an ExposureContrast (page 1589) element.

Value	Meaning
"LOOK_SWITCH"	Allows the operator's effect to be toggled on and off when the transform is used for display in applications that support this ability. This can be used in conjunction with the bypass (page 1573) attribute of the parent element to create operators that will never affect values rendered to file but that can still be toggled on and off for display. Introduced in version 1.3.

Example

```
<ProcessList id="561c5aba-8709-477c-8211-7ec4a6fe8820">
    <ExposureContrast inBitDepth="16f" outBitDepth="16f" style="linear">
        <ECParams contrast="1.0000" exposure="0.0000" pivot="0.1800" />
        <DynamicParameter param="EXPOSURE" /><DynamicParameter
param="CONTRAST" />
    </ExposureContrast>
    <Reference basePath="Autodesk" inBitDepth="16f" outBitDepth="16f"
path="misc/default_look-ACESproxy.ctf" bypass="true">
        <DynamicParameter param="LOOK_SWITCH" />
    </Reference>
    <Reference basePath="Autodesk" inBitDepth="16f" outBitDepth="16f"
path="RRT+ODT/ACES_to_CIE-XYZ_v0.1.1.ctf" />
    <Reference alias="currentMonitor" inBitDepth="16f" outBitDepth="10i" />
<</ProcessList>
```

Comments

You can include standard XML comments in CTF files.

However if you want the information to appear in the metadata panel of Autodesk applications, you can use Description (page 1571) elements instead of comments.

XML comments begin with:

```
<!--
```

and end with:

```
-->
```

Comments cannot appear inside tags (between < and >). In addition, comments cannot be nested.

SynColor Configuration File

The SynColor configuration file is an XML file that controls how Autodesk Color Management works for all applications on the same workstation.

Some applications might read this file only on start-up. You may need to restart the application after modifying this file.

On Linux, the file path and name is /opt/Autodesk/Synergy/SynColor/<version>/synColorConfig.xml.

On Mac OS X, the file path and name is /Applications/Autodesk/Synergy/SynColor/<version>/synColorConfig.xml.

Anatomy of the SynColor Configuration File

This section shows a sample of a SynColor configuration file.

```
<?xml (page 1566) version="1.0" encoding="UTF-8"?>
<SynColorConfig (page 1603) version="1.0">
   <SharedHome (page 1604)
dir="/opt/Autodesk/Synergy/SynColor/Shared/transforms"/>
   <ReferenceTable (page 1605)>
      <Ref (page 1606) alias="broadcastMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/broadcast/CIE-XYZ_to_HD-video.ctf"
 />
      <Ref alias="defaultLook"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/misc/identity.ctf"
/>
      <Ref alias="graphicsMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/Eizo/CIE-XYZ_to_CG245W.ctf"
 />
   </ReferenceTable>
</SynColorConfig>
```

SynColorConfig

The SynColorConfig element is the root element of the SynColor configuration file.

Contains

In any order:

- SharedHome (page 1604). Required, one only.
- ReferenceTable (page 1605). Required, one only.

Contained By

- no parent

Attributes

version The version of the SynColor configuration file format.

Example

```
<?xml version="1.0" encoding="UTF-8"?>
<SynColorConfig version="1.0">
   <SharedHome dir="/opt/Autodesk/Synergy/SynColor/Shared/transforms"/>
   <ReferenceTable>
      <Ref alias="broadcastMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/broadcast/CIE-XYZ_to_HD-video.ctf"
 />
      <Ref alias="defaultLook"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/misc/identity.ctf"
/>
      <Ref alias="graphicsMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/Eizo/CIE-XYZ_to_CG245W.ctf"
 />
```

```
  </ReferenceTable>
</SynColorConfig>
```

AutoConfigure

The AutoConfigure element determines whether certain options get automatically configured when you start an application that supports Autodesk Color Management.

Currently this works only for Autodesk Smoke on Mac OS X.

Contains

- No content.

Contained By

- SynColorConfig (page 1603)

Attributes

graphicsMonitor Uses the ICC profile set in your operating system's preferences as the graphicsMonitor alias.

Example

```
<?xml version="1.0" encoding="UTF-8"?>
<SynColorConfig version="1.0">
   <AutoConfigure graphicsMonitor="true" />
   <SharedHome dir="/Applications/Autodesk/Synergy/SynColor/Shared/transforms"
 />
   <ReferenceTable>
      <Ref alias="broadcastMonitor"
path="/Applications/Autodesk/Synergy/SynColor/2013.3/transforms/display/broadcast/CIE-XYZ_to_HD-video.ctf"
 />
      <Ref alias="defaultLook"
path="/Applications/Autodesk/Synergy/SynColor/2013.3/transforms/misc/identity.ctf"
 />
      <Ref alias="graphicsMonitor"
path="/Library/ColorSync/Profiles/Displays/Color
LCD-00000610-0000-9CCF-0000-0000042733C0.icc" />
   </ReferenceTable>
</SynColorConfig>
```

SharedHome

The SharedHome element specifies the location of the Shared directory. This is a useful location for storing your own custom transforms that can be used by other applications that use Autodesk Colour Management installed on the same computer.

Some applications allow you to set this value in their preferences.

Contains

- No content.

Contained By

- SynColorConfig (page 1603)

Attributes

dir Full path to the shared directory. This can be on the local or a remote computer.

Example

```
<?xml version="1.0" encoding="UTF-8"?>
<SynColorConfig version="1.0">
   <SharedHome dir="/opt/Autodesk/Synergy/SynColor/Shared/transforms"/>
   <ReferenceTable>
      <Ref alias="broadcastMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/broadcast/CIE-XYZ_to_HD-video.ctf"
 />
      <Ref alias="defaultLook"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/misc/identity.ctf"
/>
      <Ref alias="graphicsMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/Eizo/CIE-XYZ_to_CG245W.ctf"
 />
   </ReferenceTable>
</SynColorConfig>
```

ReferenceTable

The ReferenceTable element is a container for Ref elements defining aliases, or short names for CTF files.

Contains

- Ref (page 1606). Optional, any number.

  However, if there are no Ref elements to define the following aliases, then some Autodesk-supplied transforms will not work:
  - The `graphicsMonitor` alias should set to the appropriate transform for display on the workstation monitor.
  - The `broadcastMonitor` alias should set to the appropriate transform for display on the broadcast monitor.
  - The `defaultLook` alias should be set to the transform that you are using to re-create the look used on set.

  Users can define additional aliases for other transform files as they wish.

Contained By

- SynColorConfig (page 1603)

Attributes

None.

Example

```
<?xml version="1.0" encoding="UTF-8"?>
<SynColorConfig version="1.0">
   <SharedHome dir="/opt/Autodesk/Synergy/SynColor/Shared/transforms"/>
   <ReferenceTable>
      <Ref alias="broadcastMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/broadcast/CIE-XYZ_to_HD-video.ctf"
 />
      <Ref alias="defaultLook"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/misc/identity.ctf"
/>
      <Ref alias="graphicsMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/Eizo/CIE-XYZ_to_CG245W.ctf"
 />
   </ReferenceTable>
</SynColorConfig>
```

Ref

The Ref element defines an alias for a CTF file so that it can be easily referenced in other CTF files.

Some applications allow users to set certain aliases, like graphicsMonitor, in their preferences.

When an alias is changed, references that use it might not get updated until the application is restarted.

Contains

- No content.

Contained By

- ReferenceTable (page 1605)

Attributes

alias A unique name for the alias. Required.

path The path and file name of the corresponding CTF file. Required.

basePath A base path that is prefixed to the path attribute value to fully resolve the file or location. Optional.

If this attribute is used, then the path attribute is interpreted as relative to the base path. Otherwise, the path attribute is interpreted as an absolute path.

Example

```
<?xml version="1.0" encoding="UTF-8"?>
<SynColorConfig version="1.0">
   <SharedHome dir="/opt/Autodesk/Synergy/SynColor/Shared/transforms"/>
   <ReferenceTable>
      <Ref alias="broadcastMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/display/broadcast/CIE-XYZ_to_HD-video.ctf"
 />
      <Ref alias="defaultLook"
path="/opt/Autodesk/Synergy/SynColor/2013.3/transforms/misc/identity.ctf"
/>
```

```
        <Ref alias="graphicsMonitor"
path="/opt/Autodesk/Synergy/SynColor/2013.1/transforms/display/Eizo/CIE-XYZ_to_CG245W.ctf"/>

    </ReferenceTable>
</SynColorConfig>
```

Tangent Element Control Surface 29

The Tangent Element control surface is designed to improve interactivity when working in Flame Premium. Adding to the colour grading functions for use with Lustre, the Tangent Elements panel supports navigation and interaction with the Workspace, the thumbnails, the reels, the players and the timeline and Timeline FX, as well as the Colour Corrector and the Colour Warper in Flame Premium.

NOTE Because colour grading tasks are particularly well suited for use with the Tangent Element control surface, the Flame Premium Grading application has expanded functionality. For more information, see Tangent Element Control Surface (page 2377).

Installing the Control Surface

The first time you connect the Tangent Element Control Surface to your Flame Premium station, it automatically connects to your software and populates all the menus.

If the Control Surface does not automatically connect, see the Tangent Troubleshooting (page 1654) section.

The Modes

The basic Tangent Element workflow is mode based. The panel automatically sets itself to the appropriate mode, based on where you are in the application. Alternatively, you can manually set the panel to the mode of your choice and the application's user interface follows. For example, accessing the Colour Corrector from the user interface automatically sets the panel to the appropriate Colour Corrector mode (Master, Shadows, Midtone, Highlights). And setting the CC Master mode on the panel, opens the Colour Corrector in Master mode in the user interface.

The following modes are supported by the Tangent Element Control Surface in Flame Premium:

Media Panel Focus Mode

This mode focuses on the Media panel and enables you to scrub the selected clip, as well as navigate the current reel within the Media panel, select, scrub and play clips in the players. Additionally, you can perform insert, overwrite and replace edits in the timeline, from the Media panel.

Thumbnail Focus Mode

This mode focuses on the Viewing panel in Thumbnails view and enables you to scrub the selected clip, as well as navigate through and select, scrub and play clips in the players. You can also perform insert, overwrite and replace edits in the timeline.

Reels Focus Mode

This mode focuses on the Viewing panel in Reels view and enables you to scrub the selected clip, as well as navigate the reels and select, scrub and play clips in the players. Additionally, you can perform insert, overwrite and replace edits in the timeline.

Timeline Focus Mode

This mode focuses on the timeline and enables you to navigate the timeline, its layers and tracks. This mode also supports panning and zooming the timeline via the trackball and wheel. In addition to the ability to perform insert, overwrite and replace edits, you can also apply dissolves, wipes and axis as well as add Timeline FX. All of the Timeline FX are supported.

NOTE The Tangent panel only supports the Timeline FX Colour Corrector and Colour Warper Editors for input parameters.

Player Modes

When you bring up a player, either using the Tangent panel or from the user interface, the panel enters one of the player modes, based on the selected player. There are four player modes:

- 1Up Player
- Src-Seq Player
- Triptych Player
- Full Screen Player

From the Player modes, you can play and navigate through clips and sequences, drop markers on clips and toggle the different players.

Colour Corrector Modes

These modes focus on the Timeline FX Colour Corrector and enable you to control all of the creative settings within the Colour Corrector, including the Colour Corrector Quick Bar.

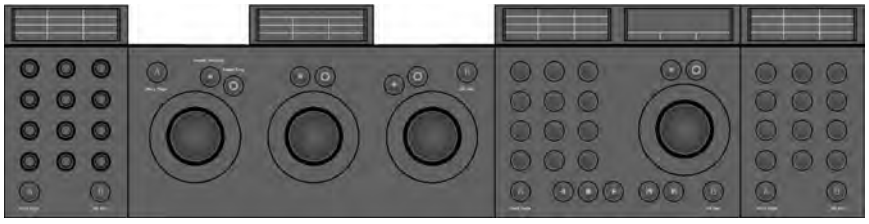
NOTE The Colour Corrector Batch node and the Colour Corrector Tool in the Tools tab are not supported.

Colour Warper Mode

This mode focuses on the Timeline FX Colour Warper and enables you to control all of the creative settings within the Colour Warper, including the Colour Warper Quick Bar.

NOTE The Colour Warper Batch node and the Colour Warper Tool in the Tools tab are not supported.

Using the Modules



The Tangent Element Control Surface is composed of four modules.

The layout of the displays of each module mirrors the functions of the knobs/buttons of each module.

KB module (Knobs)

The KB module is a knob-based interface for user input parameters.

The values of each parameter assigned to the knobs can be reset by pressing the knob.

A button: Cycle through the different pages of functions mapped to the module.

B button:

- Alt mode: Press to enable the alternate functions mapped across the entire panel.
- Precise mode: Hold to enable Precise Mode.

NOTE You can press the B button twice and lock Alt mode.

TK module (Trackballs)

The TK module is also an interface for user input parameters. The trackballs and rings provide an intuitive way of precisely modifying parameters and are especially well suited for grading, among other things.

Trackballs: You can modify parameters by spinning the trackballs left / right or up / down based on the current mode.

Rings: You can modify parameters by turning the rings left / right.

The values of the parameters assigned to the trackballs and the rings can be reset by pressing the Reset button above them.

A button: Cycle through the different pages of functions mapped to the module.

B button:

- Alt mode: Press to enable the alternate functions mapped across the entire panel.
- Precise mode: Hold to enable Precise Mode.

NOTE You can press the B button twice and lock Alt mode.

MF module (Multi-Function)

The MF module enables you to select the mode you want to work in and provides a transport section, which allows navigation via the trackball and ring as well.

Trackball: You can modify parameters by spinning the trackball left / right or up / down based on the current mode.

Ring: You can modify parameters by turning the ring left / right.

The values of the parameters assigned to the trackballs and the rings can be reset by pressing the Reset button above them.

A button: Cycle through the different pages of functions mapped to the module.

B button:

- Alt mode: Press to enable the alternate functions mapped across the entire panel.
- Precise mode: Hold to enable Precise Mode.

NOTE You can press the B button twice and lock Alt mode.

Button sequences for alternate functions on the MF module

- **Next Element Button + B Button:** Move the positioner forward to the next keyframe in your timeline .
- **Previous Element Button + B Button:** Move the positioner backward to the previous keyframe in your timeline.
- **Stop Button + Forward Play button:** Move the positioner forward by one frame in your timeline.
- **Stop Button + Backward Play button:** Move the positioner backward by one frame in your timeline.

BT module (Buttons)

The BT module is dedicated to providing quick access to the editing operations and to the Timeline FX.

A button: Cycle through the different pages of functions mapped to the module.

B button:

- Alt mode: Press to enable the alternate functions mapped across the entire panel.
- Precise mode: Hold to enable Precise Mode.

NOTE You can press the B button twice and lock Alt mode.

Control Surface Layouts

Media Panel Focus Layout

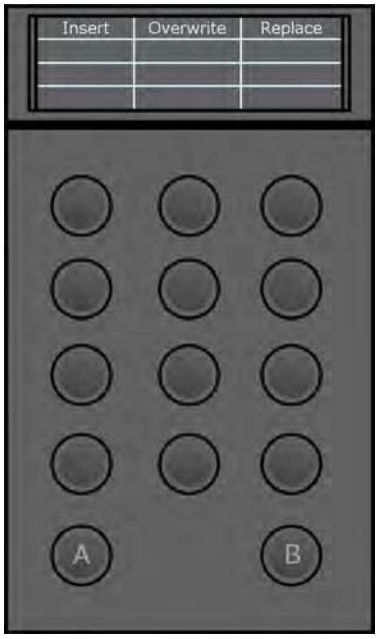
Module 3 (MF)



- **MediaPanl:** Enable Media Panel Focus mode.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.

- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Go to 1Up player mode and backward play the currently selected clip.
- **Backward Play + Stop buttons:** Move the clip displayed in the player one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the clip displayed in the player one frame forward.
- **Forward Play button:** Go to 1Up player mode and forward play the currently selected clip.
- **Previous button:** Move the selection in the current reel within the Media panel down by one increment.
- **Next button:** Move the selection in the current reel with in the Media panel up by one increment.

Module 4 (BT)



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.

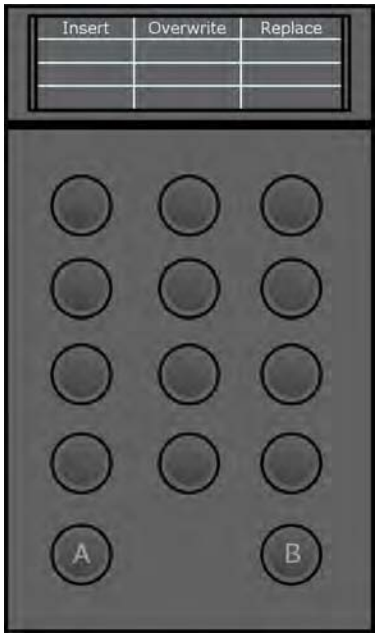
Thumbnail Focus Layout

Module 3 (MF)



- **Thumbnail:** Enable Thumbnail Focus mode.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Go to full screen player mode and backward play the currently selected clip.
- **Backward Play + Stop buttons:** Move the currently selected thumbnail one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the currently selected thumbnail one frame forward.
- **Forward Play button:** Go to full screen player mode and forward play the currently selected clip.
- **Previous button:** Select the previous thumbnail.
- **Next button:** Select the next thumbnail.

Module 4 (BT)



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.

Reels Focus Layout

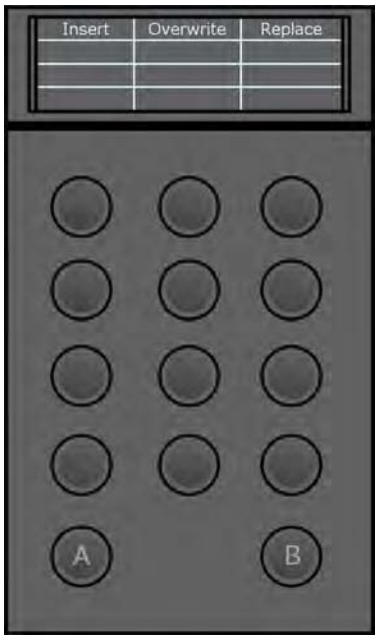
Module 3 (MF)



- **Reels:** Enable Reels Focus mode.

- **Up:**Move the selection in the Media panel up by one increment.
- **Down:** Move the selection in the Media panel down by one increment.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Go to the full screen player mode and backward play the currently selected clip.
- **Backward Play + Stop buttons:** Move the currently selected thumbnail one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the currently selected thumbnail one frame forward.
- **Forward Play button:** Go to the full screen player mode and forward play the currently selected clip.
- **Previous button:** Select the previous thumbnail.
- **Next button:** Select the next thumbnail.

Module 4 (BT)



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.

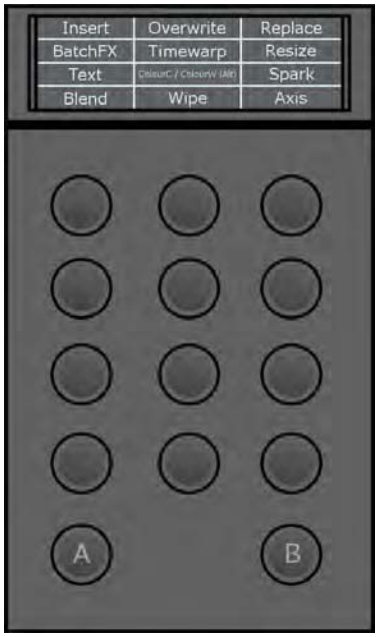
Timeline Focus Layout

Module 3 (MF)



- **Timeline:** Enable Timeline Focus mode.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip/sequence in the timeline backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip/sequence in the timeline backward or forward.
- **Backward Play button:** Go to the 1Up player mode and backward play the currently selected clip.
- **Backward Play + Stop buttons:** Move the timeline positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the timeline positioner one frame forward.
- **Forward Play button:** Go to the 1Up player mode and backward play the currently selected clip.
- **Previous button:** Move the timeline positioner to the previous transition.
- **Next button:** Move the timeline positioner to the next transition.

Module 4 (BT) - Page 1 of 2



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.
- **BatchFX:** Single press to apply a BatchFX to the currently selected timeline segment(s). You are taken into Batch.

  NOTE If the selected timeline segment already has a BatchFX applied to it, double pressing the button takes to into Batch for editing.

- **Timewarp:** Single press to apply a Timewarp Timeline FX to the currently selected timeline segment. Double press to enter the Timewarp editor.
- **Resize:** Single press to apply a Resize Timeline FX to the currently selected timeline segment. Double press to enter the Resize editor.
- **Text:** Single press to apply a Text Timeline FX to the currently selected timeline segment. Double press to enter the Text editor.
- **ColourC:** Single press to apply a Colour Corrector Timeline FX to the currently selected timeline segment. Double press to enter the Colour Corrector editor.

  NOTE The Tangent panel only supports the Timeline FX Colour Corrector and Colour Warper Editors for input parameters.

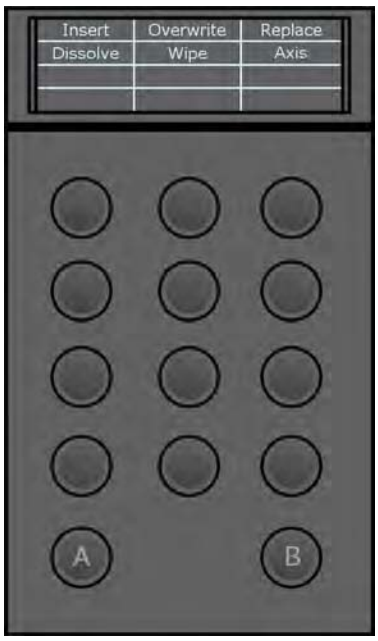
- **ColourW (Alt):** Single press (while holding Alt) to apply a Colour Warper Timeline FX to the currently selected timeline segment. Double press (while holding Alt) to enter the Colour Warper editor.

  NOTE The Tangent panel only supports the Timeline FX Colour Corrector and Colour Warper Editors for input parameters.

- **Spark:** Press to apply a placeholder Spark. Click the Load Spark button (L), with your mouse or pen, to open the Sparks Browser and navigate to the appropriate Spark.
- **Blend:** Single press to apply a Blend Timeline FX to the currently selected timeline segment. Double press to enter the Blend editor.

- **Wipe:** Single press to apply a Wipe Timeline FX to the currently selected timeline segment. Double press to enter the Wipe editor.
- **Axis:** Single press to apply a Axis Timeline FX to the currently selected timeline segment. Double press to enter the Axis editor.

Module 4 (BT) - Page 2 of 2



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.
- **Dissolve:** Single press to apply a Dissolve video transition to the currently selected timeline segment(s). Double press to enter the Dissolve editor.
- **Wipe:** Single press to apply a Wipe video transition to the currently selected timeline segment(s). Double press to enter the Wipe editor.
- **Axis:** Single press to apply an Axis video transition to the currently selected timeline segment(s). Double press to enter the Axis editor.

Player Focus Layouts

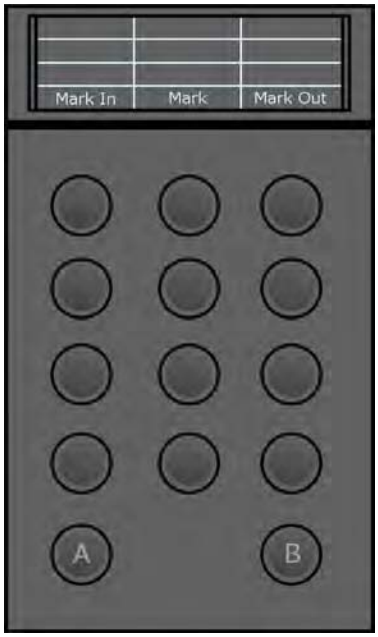
1Up Player Focus Layout

Module 3 (MF)



- **Player:** Enable 1Up Player Focus mode. Press again to toggle between Source Player and Sequence Player.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Play the current clip backward.
- **Backward Play + Stop buttons:** Move the player head one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the player head one frame backward.
- **Forward Play button:** Play the current clip backward.
- **Previous button:** Got to the previous transition.
- **Previous button (Alt):** Go to In mark.
- **Next button:** Go to the next transition.
- **Next button (Alt):** Got to Out mark.

Module 4 (BT)



- **Mark In:** Drop an In point marker.
- **Mark:** Drop a navigation marker.
- **Mark Out:** Drop an Out point marker.

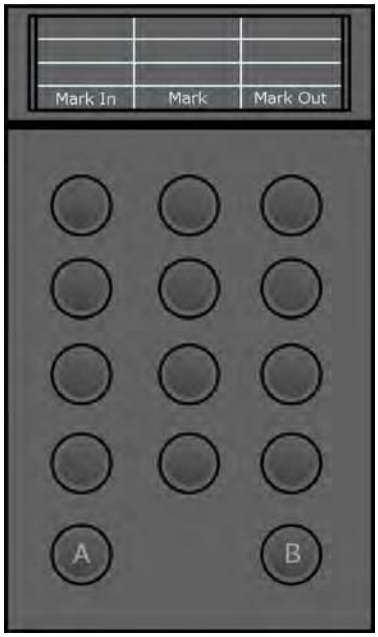
Src-Seq Player Focus Layout

Module 3 (MF)



- **Src-Seq:** Enable Src-Seq Player Focus mode. Press again to toggle between Source focus and Sequence focus.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Play the current clip backward.
- **Backward Play + Stop buttons:** Move the player head one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the player head one frame backward.
- **Forward Play button:** Play the current clip backward.
- **Previous button:** Got to the previous transition.
- **Previous button (Alt):** Go to In mark.
- **Next button:** Go to the next transition.
- **Next button (Alt):** Got to Out mark.

Module 4 (BT)



- **Mark In:** Drop an In point marker.
- **Mark:** Drop a navigation marker.
- **Mark Out:** Drop an Out point marker.

Triptych Player Focus Layout

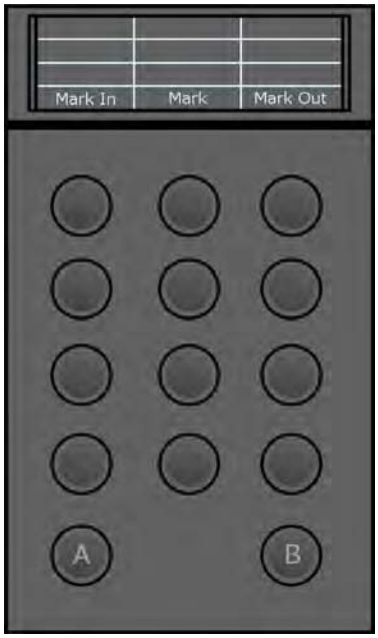
Module 3 (MF)



- **Triptych:** Enable Triptych Player Focus mode. Press again to toggle the Player focus.

- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Play the current clip backward.
- **Backward Play + Stop buttons:** Move the player head one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the player head one frame backward.
- **Forward Play button:** Play the current clip backward.
- **Previous button:** Got to the previous transition.
- **Previous button (Alt):** Go to In mark.
- **Next button:** Go to the next transition.
- **Next button (Alt):** Got to Out mark.

Module 4 (BT)



- **Mark In:** Drop an In point marker.
- **Mark:** Drop a navigation marker.
- **Mark Out:** Drop an Out point marker.

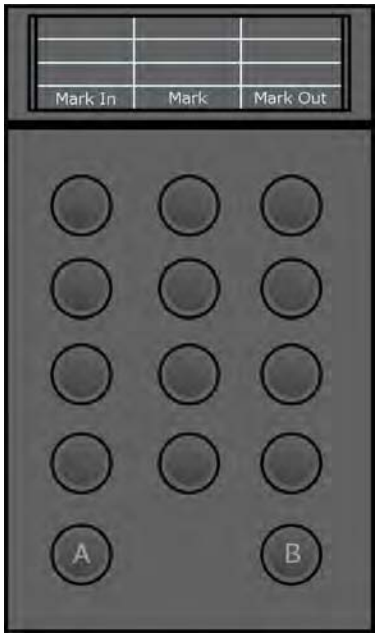
Full Screen Player Focus Layout

Module 3 (MF)



- **FullPlayr:** Enable Full Screen Player Focus mode.
- **QuickScrol (Trackball):** Spin the ball along the X axis to quickly scroll the currently selected clip backward or forward.
- **Scroll (Ring):** Turn the ring to scroll the selected clip backward or forward.
- **Backward Play button:** Play the current clip backward.
- **Backward Play + Stop buttons:** Move the player head one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the player head one frame backward.
- **Forward Play button:** Play the current clip backward.
- **Previous button:** Got to the previous transition.
- **Previous button (Alt):** Go to In mark.
- **Next button:** Go to the next transition.
- **Next button (Alt):** Got to Out mark.

Module 4 (BT)

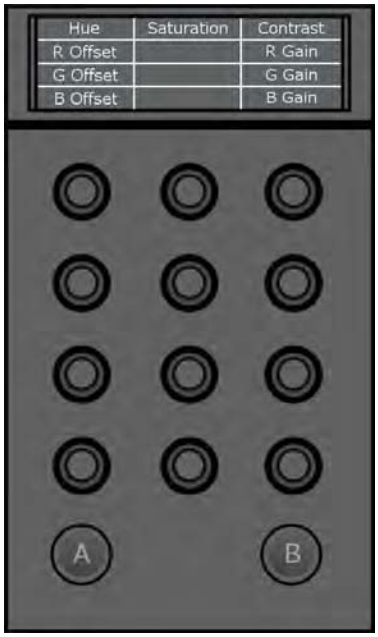


- **Mark In:** Drop an In point marker.
- **Mark:** Drop a navigation marker.
- **Mark Out:** Drop an Out point marker.

Colour Correction Focus Layouts

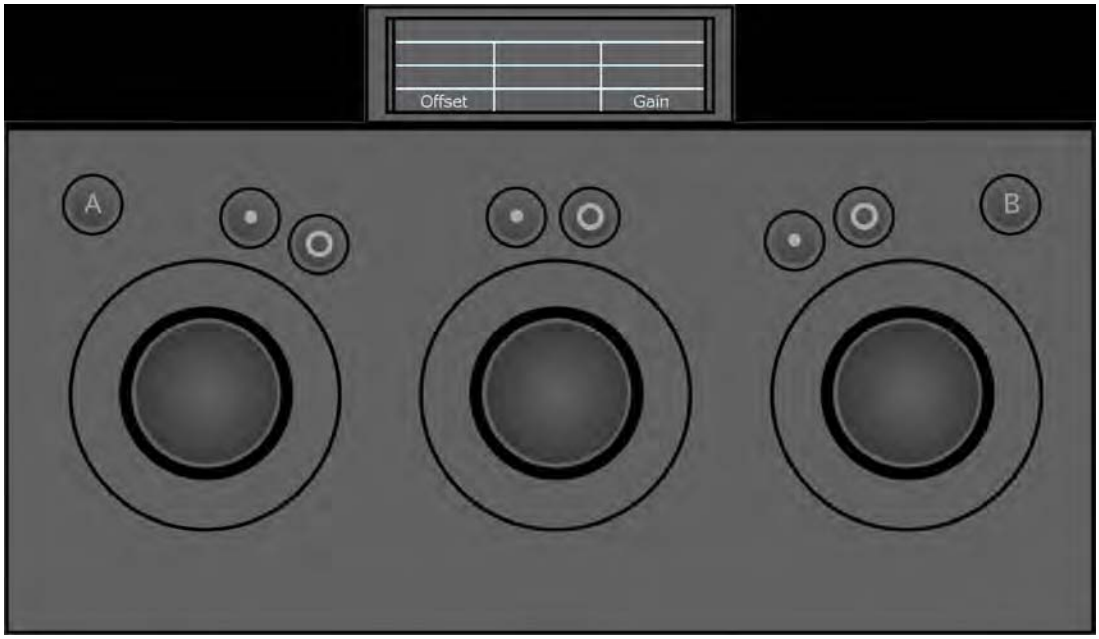
Colour Corrector Quick Bar Focus Layout

Module 1 (KB)



- **Hue:** Adjust the RGB hue value.
- **Saturation:** Adjust the RGB saturation value.
- **Contrast:** Adjust the RGB contrast value.
- **R Offset:** Adjust the Red Offset value.
- **R Gain:** Adjust the Red Gain value.
- **G Offset:** Adjust the Green Offset value.
- **G Gain:** Adjust the Green Gain value.
- **B Offset:** Adjust the Blue Offset value.
- **B Gain:** Adjust the Blue Gain value.

Module 2 (TK)



- **Offset:** Adjust the Master Offset value.
- **Gain:** Adjust the Master Gain value.

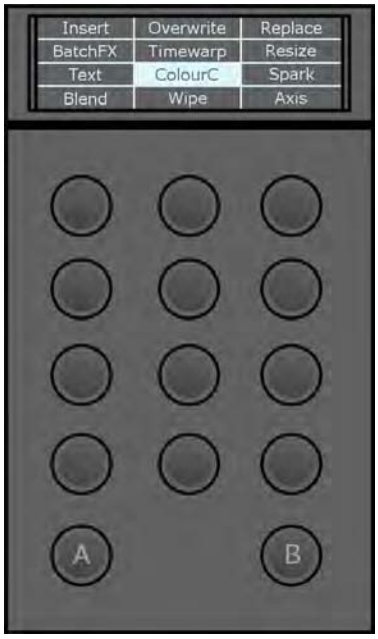
Module 3 (MF)



- **Timeline:** Enable Timeline Focus mode.
- **Pan Tmln (Trackball):** Spin the ball along the X axis to zoom the timeline.
- **Zoom Tmln (Trackball + Alt):** Spin the ball along the X axis to pan the the timeline left and right.
- **ScrolTmln (Ring):** Turn the ring to scroll the timeline backward or forward.

- **Backward Play button:** Go to the 1Up player mode and backward play the currently selected clip.
- **Backward Play + Stop buttons:** Move the timeline positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the timeline positioner one frame forward.
- **Forward Play button:** Go to the 1Up player mode and backward play the currently selected clip.
- **Previous button:** Move the timeline positioner to the previous transition.
- **Next button:** Move the timeline positioner to the next transition.

Module 4 (BT) - Page 1 of 2



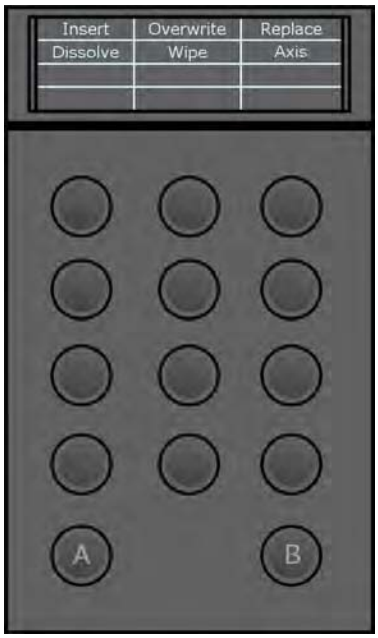
- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.
- **BatchFX:** Single press to apply a BatchFX to the currently selected timeline segment(s). You are taken into Batch.

  NOTE If the selected timeline segment already has a BatchFX applied to it, double pressing the button takes you to Batch for editing. Press and hold to mute the effect.

- **Timewarp:** Single press to apply a Timewarp Timeline FX to the currently selected timeline segment. Double press to enter the Timewarp editor. Press and hold to mute the effect.
- **Resize:** Single press to apply a Resize Timeline FX to the currently selected timeline segment. Double press to enter the Resize editor. Press and hold to mute the effect.
- **Text:** Single press to apply a Text Timeline FX to the currently selected timeline segment. Double press to enter the Text editor. Press and hold to mute the effect.
- **ColourC:** Single press to apply a Colour Corrector Timeline FX to the currently selected timeline segment. Double press to enter the Colour Corrector editor. Press and hold to mute the effect.
- **ColourW (Alt):** Single press (while holding Alt) to apply a Colour Warper Timeline FX to the currently selected timeline segment. Double press (while holding Alt) to enter the Colour Warper editor. Press and hold to mute the effect.

- **Spark:** Single press to apply an empty Spark Timeline FX to the currently selected timeline segment. The Quick Editor is displayed and you can select the spark to be applied. If a spark has already been applied to the segment, double press to enter the Sprak Editor. Press and hold to mute the effect.
- **Blend:** Single press to apply a Blend Timeline FX to the currently selected timeline segment. Double press to enter the Blend editor. Press and hold to mute the effect.
- **Wipe:** Single press to apply a Wipe Timeline FX to the currently selected timeline segment. Double press to enter the Wipe editor. Press and hold to mute the effect.
- **Axis:** Single press to apply a Axis Timeline FX to the currently selected timeline segment. Double press to enter the Axis editor. Press and hold to mute the effect.

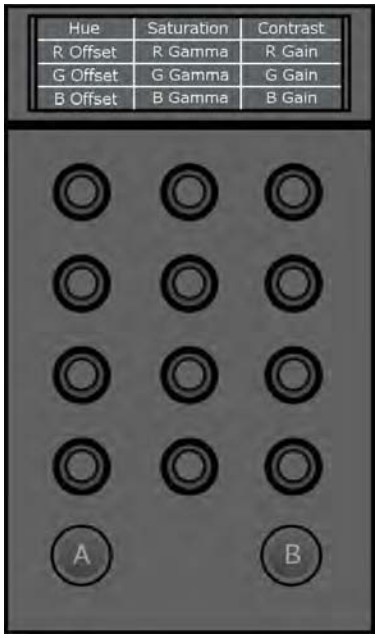
Module 4 (BT) - Page 2 of 2



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.
- **Dissolve:** Single press to apply a Dissolve video transition to the currently selected timeline segment(s). Double press to enter the Dissolve editor.
- **Wipe:** Single press to apply a Wipe video transition to the currently selected timeline segment(s). Double press to enter the Wipe editor.
- **Axis:** Single press to apply an Axis video transition to the currently selected timeline segment(s). Double press to enter the Axis editor.

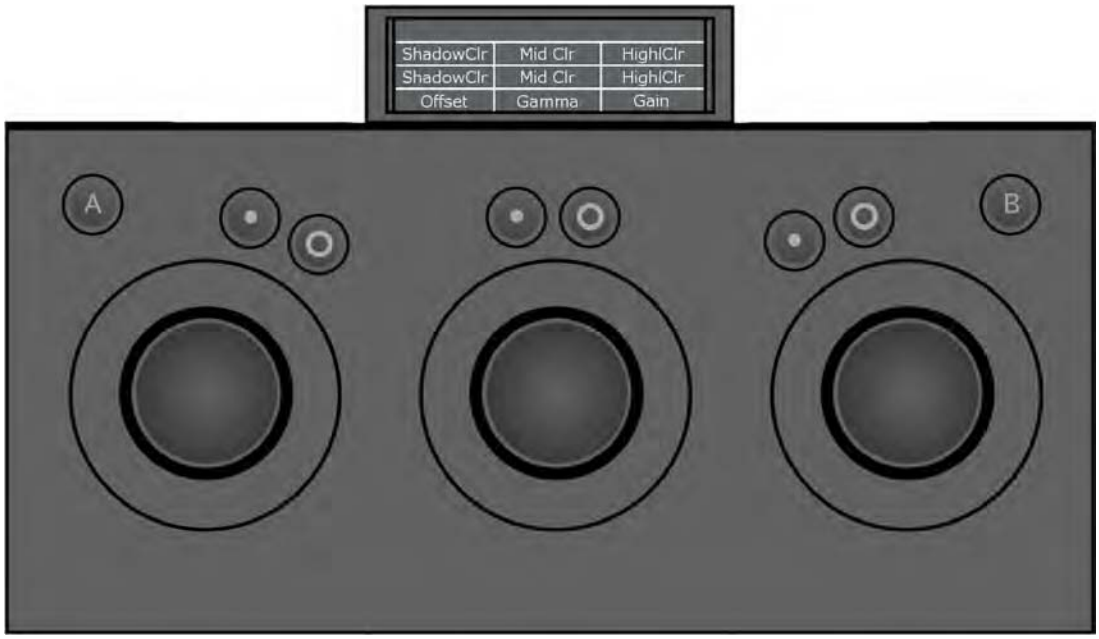
Colour Corrector Master Focus Layout

Module 1 (KB)



- **Hue:** Adjust the hue value across the entire spectrum.
- **Saturation:** Adjust the saturation value across the entire spectrum.
- **Contrast:** Adjust the contrast value across the entire spectrum..
- **R Offset:** Adjust the Red Offset value.
- **R Gamma:** Adjust the Red Gamma value.
- **R Gain:** Adjust the Red Gain value.
- **G Offset:** Adjust the Green Offset value.
- **G Gamma:** Adjust the Green Gamma value.
- **G Gain:** Adjust the Green Gain value.
- **B Offset:** Adjust the Blue Offset value.
- **B Gamma:** Adjust the Blue Gamma value.
- **B Gain:** Adjust the Blue Gain value.

Module 2 (TK)



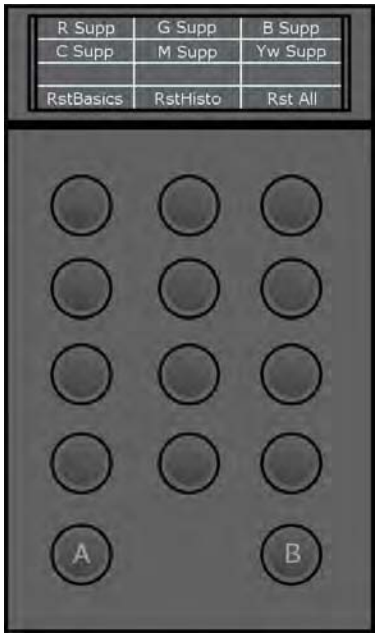
- **ShadowClr:** Move the trackball to adjust the colour of the shadows.
- **Mid Clr:** Move the trackball to adjust the colour of the midtones.
- **HighClr:** Move the trackball to adjust the colour of the highlights.
- **Offset:** Turn the ring to adjust the RGB offset value.
- **Gamma:** Turn the ring to adjust the RGB gamma value.
- **Gain:** Turn the ring to adjust the RGB gain value.

Module 3 (MF)



- **Exit:** Exit the current menu.
- **Redo:** Reapply the last operation performed after an Undo operation.
- **Undo:** Undo the last operation performed.
- **Pan (Trackball + Alt):** Spin the ball along the X axis to pan the current shot left and right.
- **Scroll (Ring):** Turn the ring to scroll the current shot backward or forward.
- **Zoom (Ring + Alt):** Turn the ring to zoom the current shot in and out
- **Backward Play button:** Play the current shot backward. Results may vary according to the number of unrendered effects on the shot.
- **Backward Play + Stop buttons:** Move the positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the positioner one frame forward.
- **Forward Play button:** Play the current shot forward. Results may vary according to the number of unrendered effects on the shot.
- **Previous button:** Go to the start of the current segment.
- **Previous button (Alt):** Go to the previous keyframe.
- **Next button:** Go to the end of the current segment.
- **Next button (Alt):** Go to the next keyframe.

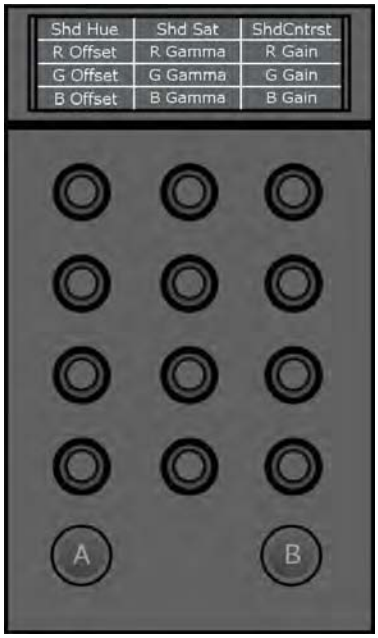
Module 4 (BT) - Page 6 of 6



- **R Supp:** Adjust the Red suppression value.
- **G Supp:** Adjust the Green suppression value.
- **B Supp:** Adjust the Blue suppression value.
- **C Supp:** Adjust the Cyan suppression value.
- **M Supp:** Adjust the Magenta suppression value.
- **Yw Supp:** Adjust the Yellow suppression value.
- **RstBasics:** Reset the basic Colour Corrector parameter values.
- **RstHisto:** Reset the Histogram values.
- **Rst All:** Reset all parameter values.

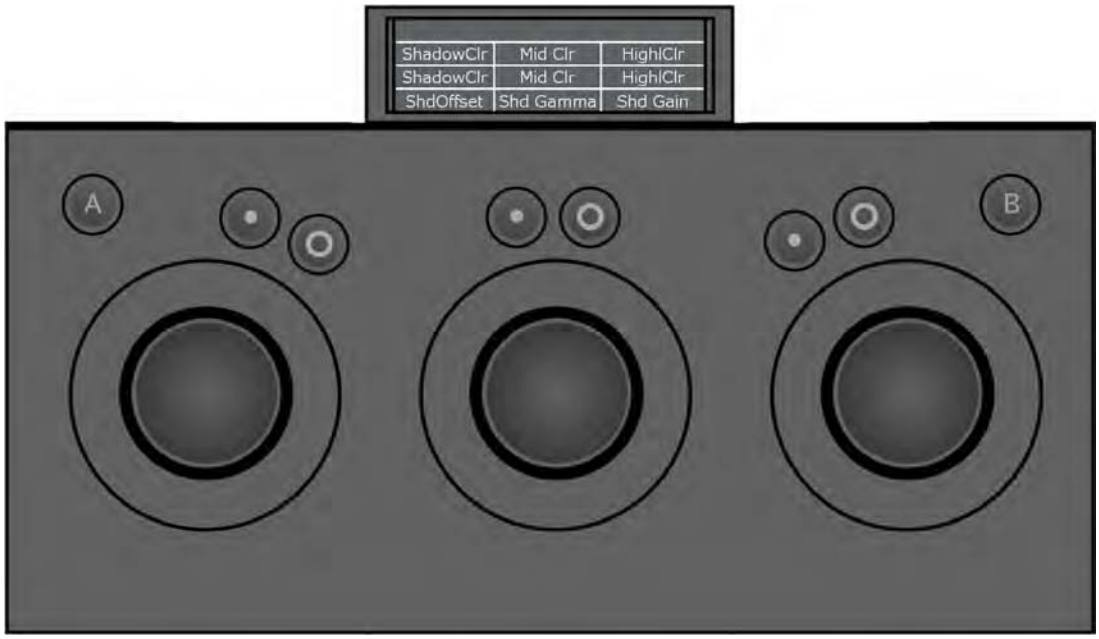
Colour Corrector Shadows Focus Layout

Module 1 (KB)



- **Shd Hue:** Adjust the hue value of the shadows.
- **Shd Sat:** Adjust the saturation value of the shadows.
- **ShdCntrst:** Adjust the contrast value of the shadows.
- **R Offset:** Adjust the Red Offset value.
- **R Gamma:** Adjust the Red Gamma value.
- **R Gain:** Adjust the Red Gain value.
- **G Offset:** Adjust the Green Offset value.
- **G Gamma:** Adjust the Green Gamma value.
- **G Gain:** Adjust the Green Gain value.
- **B Offset:** Adjust the Blue Offset value.
- **B Gamma:** Adjust the Blue Gamma value.
- **B Gain:** Adjust the Blue Gain value.

Module 2 (TK)



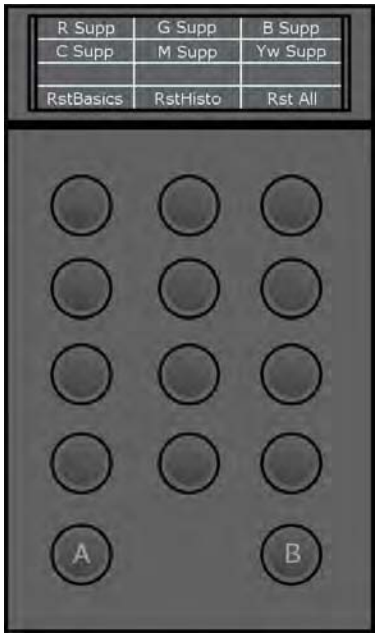
- **ShadowClr:** Move the trackball to adjust the colour of the shadows.
- **Mid Clr:** Move the trackball to adjust the colour of the midtones.
- **HighClr:** Move the trackball to adjust the colour of the highlights.
- **ShdOffset:** Turn the ring to adjust the shadows offset value.
- **Shd Gamma:** Turn the ring to adjust the shadows gamma value.
- **Shd Gain:** Turn the ring to adjust the shadows gain value.

Module 3 (MF)



- **Exit:** Exit the current menu.
- **Redo:** Reapply the last operation performed after an Undo operation.
- **Undo:** Undo the last operation performed.
- **Pan (Trackball + Alt):** Spin the ball along the X axis to pan the current shot left and right.
- **Scroll (Ring):** Turn the ring to scroll the current shot backward or forward.
- **Zoom (Ring + Alt):** Turn the ring to zoom the current shot in and out
- **Backward Play button:** Play the current shot backward. Results may vary according to the number of unrendered effects on the shot.
- **Backward Play + Stop buttons:** Move the positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the positioner one frame forward.
- **Forward Play button:** Play the current shot forward. Results may vary according to the number of unrendered effects on the shot.
- **Previous button:** Go to the start of the current segment.
- **Previous button (Alt):** Go to the previous keyframe.
- **Next button:** Go to the end of the current segment.
- **Next button (Alt):** Go to the next keyframe.

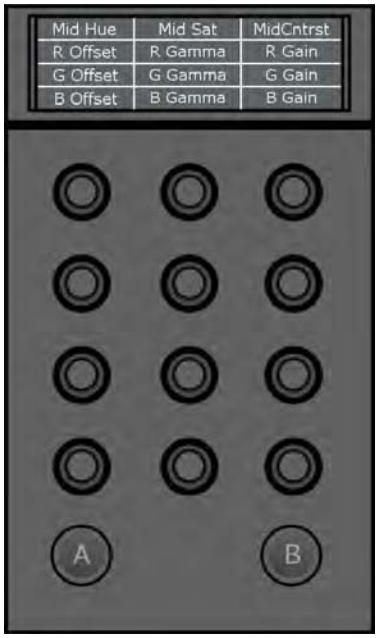
Module 4 (BT)



- **R Supp:** Adjust the Red suppression value.
- **G Supp:** Adjust the Green suppression value.
- **B Supp:** Adjust the Blue suppression value.
- **C Supp:** Adjust the Cyan suppression value.
- **M Supp:** Adjust the Magenta suppression value.
- **Yw Supp:** Adjust the Yellow suppression value.
- **RstBasics:** Reset the basic Colour Corrector parameter values.
- **RstHisto:** Reset the Histogram values.
- **Rst All:** Reset all parameter values.

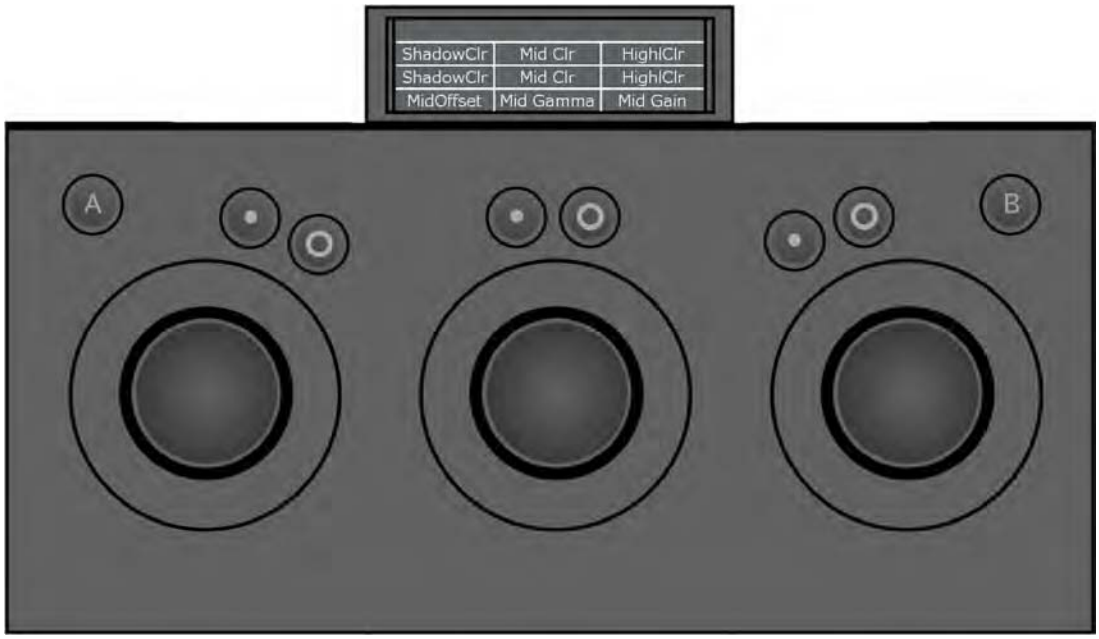
Colour Corrector Midtones Focus Layout

Module 1 (KB)



- **Mid Hue:** Adjust the hue value of the midtones.
- **Mid Sat:** Adjust the saturation value of the midtones
- **MidCntrst:** Adjust the contrast value of the midtones
- **R Offset:** Adjust the Red Offset value.
- **G Offset:** Adjust the Green Offset value.
- **B Offset:** Adjust the Blue Offset value.
- **R Gamma:** Adjust the Red Gamma value.
- **G Gamma:** Adjust the Green Gamma value.
- **B Gamma:** Adjust the Blue Gamma value.
- **R Gain:** Adjust the Red Gain value.
- **G Gain:** Adjust the Green Gain value.
- **B Gain:** Adjust the Blue Gain value.

Module 2 (TK)



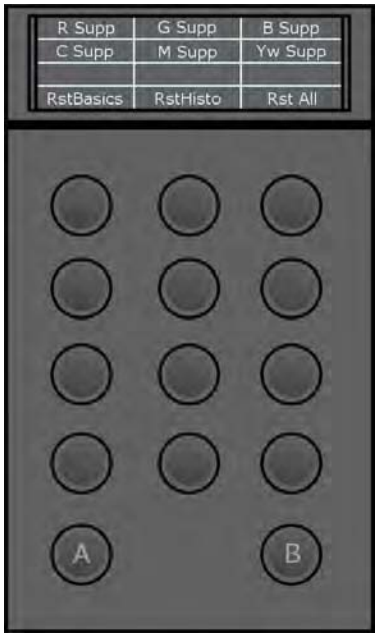
- **ShadowClr:** Move the trackball to adjust the colour of the shadows.
- **Mid Clr:** Move the trackball to adjust the colour of the midtones.
- **HighClr:** Move the trackball to adjust the colour of the highlights.
- **MidOffset:** Turn the ring to adjust the midtones offset value.
- **Mid Gamma:** Turn the ring to adjust the midtones gamma value.
- **Mid Gain:** Turn the ring to adjust the midtones gain value.

Module 3 (MF)



- **Exit:** Exit the current menu.
- **Redo:** Reapply the last operation performed after an Undo operation.
- **Undo:** Undo the last operation performed.
- **Pan (Trackball + Alt):** Spin the ball along the X axis to pan the current shot left and right.
- **Scroll (Ring):** Turn the ring to scroll the current shot backward or forward.
- **Zoom (Ring + Alt):** Turn the ring to zoom the current shot in and out
- **Backward Play button:** Play the current shot backward. Results may vary according to the number of unrendered effects on the shot.
- **Backward Play + Stop buttons:** Move the positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the positioner one frame forward.
- **Forward Play button:** Play the current shot forward. Results may vary according to the number of unrendered effects on the shot.
- **Previous button:** Go to the start of the current segment.
- **Previous button (Alt):** Go to the previous keyframe.
- **Next button:** Go to the end of the current segment.
- **Next button (Alt):** Go to the next keyframe.

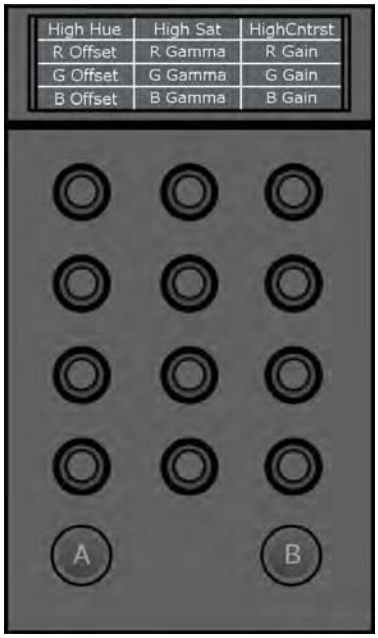
Module 4 (BT)



- **R Supp:** Adjust the Red suppression value.
- **G Supp:** Adjust the Green suppression value.
- **B Supp:** Adjust the Blue suppression value.
- **C Supp:** Adjust the Cyan suppression value.
- **M Supp:** Adjust the Magenta suppression value.
- **Yw Supp:** Adjust the Yellow suppression value.
- **RstBasics:** Reset the basic Colour Corrector parameter values.
- **RstHisto:** Reset the Histogram values.
- **Rst All:** Reset all parameter values.

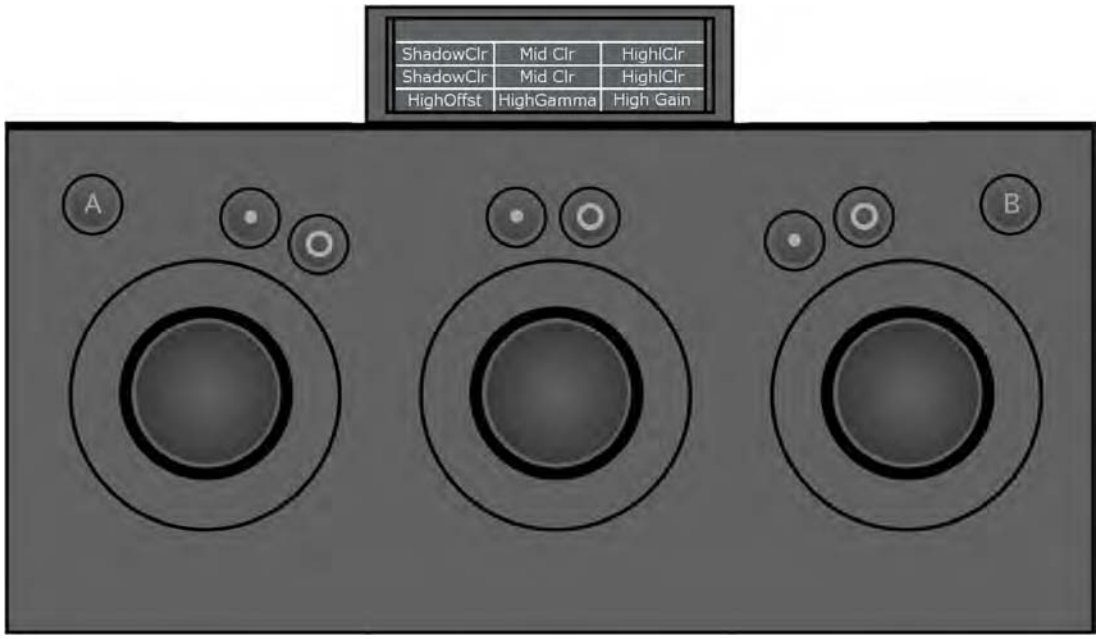
Colour Corrector Highlights Focus Layout

Module 1 (KB)



- **High Hue:** Adjust the hue value of the highlights.
- **High Sat:** Adjust the saturation value of the highlights.
- **HiCntrst:** Adjust the contrast value of the highlights.
- **R Offset:** Adjust the Red Offset value.
- **G Offset:** Adjust the Green Offset value.
- **B Offset:** Adjust the Blue Offset value.
- **R Gamma:** Adjust the Red Gamma value.
- **G Gamma:** Adjust the Green Gamma value.
- **B Gamma:** Adjust the Blue Gamma value.
- **R Gain:** Adjust the Red Gain value.
- **G Gain:** Adjust the Green Gain value.
- **B Gain:** Adjust the Blue Gain value.

Module 2 (TK)



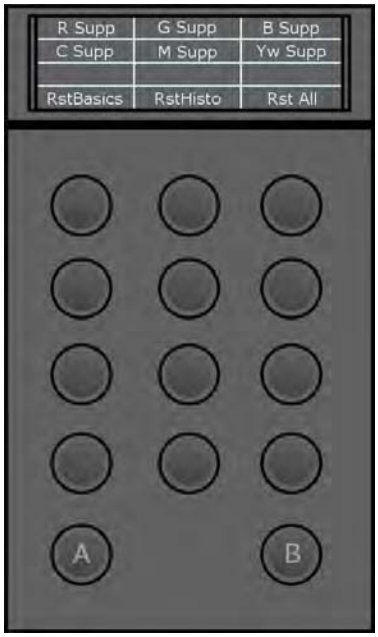
- **ShadowClr:** Move the trackball to adjust the colour of the shadows.
- **Mid Clr:** Move the trackball to adjust the colour of the midtones.
- **HighClr:** Move the trackball to adjust the colour of the highlights.
- **HighOffset:** Turn the ring to adjust the highlights offset value.
- **High Gamma:** Turn the ring to adjust the highlights gamma value.
- **High Gain:** Turn the ring to adjust the highlights gain value.

Module 3 (MF)



- **Exit:** Exit the current menu.
- **Redo:** Reapply the last operation performed after an Undo operation.
- **Undo:** Undo the last operation performed.
- **Pan (Trackball + Alt):** Spin the ball along the X axis to pan the current shot left and right.
- **Scroll (Ring):** Turn the ring to scroll the current shot backward or forward.
- **Zoom (Ring + Alt):** Turn the ring to zoom the current shot in and out
- **Backward Play button:** Play the current shot backward. Results may vary according to the number of unrendered effects on the shot.
- **Backward Play + Stop buttons:** Move the positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the positioner one frame forward.
- **Forward Play button:** Play the current shot forward. Results may vary according to the number of unrendered effects on the shot.
- **Previous button:** Go to the start of the current segment.
- **Previous button (Alt):** Go to the previous keyframe.
- **Next button:** Go to the end of the current segment.
- **Next button (Alt):** Go to the next keyframe.

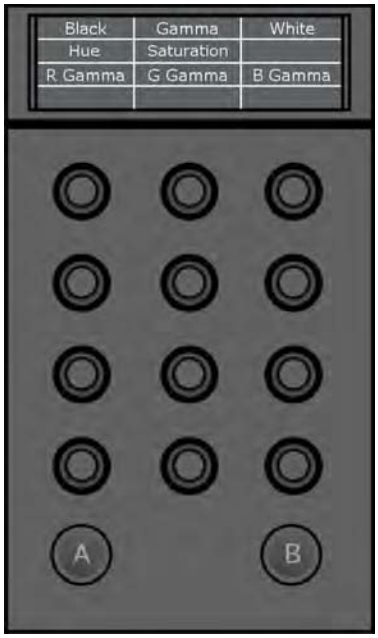
Module 4 (BT)



- **R Supp:** Adjust the Red suppression value.
- **G Supp:** Adjust the Green suppression value.
- **B Supp:** Adjust the Blue suppression value.
- **C Supp:** Adjust the Cyan suppression value.
- **M Supp:** Adjust the Magenta suppression value.
- **Yw Supp:** Adjust the Yellow suppression value.
- **RstBasics:** Reset the basic Colour Corrector parameter values.
- **RstHisto:** Reset the Histogram values.
- **Rst All:** Reset all parameter values.

Colour Warper QuickBar Focus Layout

Module 1 (KB)



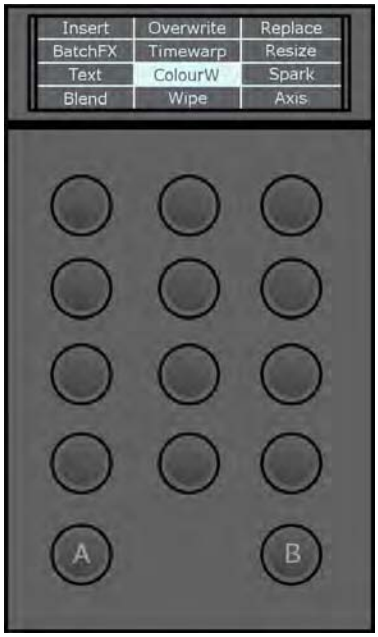
- **Black:** Adjust the black point.
- **Gamma:** Adjust the RGB gamma value.
- **White:** Adjust the white point.
- **Hue:** Adjust the RGB hue value.
- **Saturatio:** Adjust the RGB saturation value.
- **R Gamma:** Adjust the Red Gamma value.
- **G Gamma:** Adjust the Green Gamma value.
- **B Gamma:** Adjust the Blue Gamma value.

Module 3 (MF)



- **Timeline:** Enable Timeline Focus mode.
- **Pan Tmln (Trackball):** Spin the ball along the X axis to zoom the timeline.
- **Zoom Tmln (Trackball + Alt):** Spin the ball along the X axis to pan the the timeline left and right.
- **ScrolTmln (Ring):** Turn the ring to scroll the timeline backward or forward.
- **Backward Play button:** Go to the 1Up player mode and backward play the currently selected clip.
- **Backward Play + Stop buttons:** Move the timeline positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the timeline positioner one frame forward.
- **Forward Play button:** Go to the 1Up player mode and backward play the currently selected clip.
- **Previous button:** Move the timeline positioner to the previous transition.
- **Next button:** Move the timeline positioner to the next transition.

Module 4 (BT) - Page 1 of 2

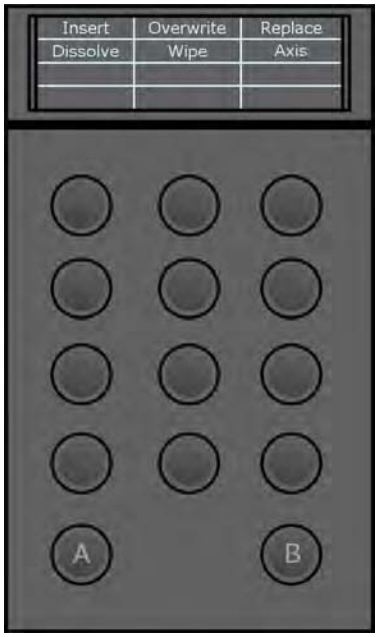


- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.
- **BatchFX:** Single press to apply a BatchFX to the currently selected timeline segment(s). You are taken into Batch.

  NOTE If the selected timeline segment already has a BatchFX applied to it, double pressing the button takes you to Batch for editing. Press and hold to mute the effect.

- **Timewarp:** Single press to apply a Timewarp Timeline FX to the currently selected timeline segment. Double press to enter the Timewarp editor. Press and hold to mute the effect.
- **Resize:** Single press to apply a Resize Timeline FX to the currently selected timeline segment. Double press to enter the Resize editor. Press and hold to mute the effect.
- **Text:** Single press to apply a Text Timeline FX to the currently selected timeline segment. Double press to enter the Text editor. Press and hold to mute the effect.
- **ColourC:** Single press to apply a Colour Corrector Timeline FX to the currently selected timeline segment. Double press to enter the Colour Corrector editor. Press and hold to mute the effect.
- **ColourW (Alt):** Single press (while holding Alt) to apply a Colour Warper Timeline FX to the currently selected timeline segment. Double press (while holding Alt) to enter the Colour Warper editor. Press and hold to mute the effect.
- **Spark:** Single press to apply an empty Spark Timeline FX to the currently selected timeline segment. The Quick Editor is displayed and you can select the spark to be applied. If a spark has already been applied to the segment, double press to enter the Sprak Editor. Press and hold to mute the effect.
- **Blend:** Single press to apply a Blend Timeline FX to the currently selected timeline segment. Double press to enter the Blend editor. Press and hold to mute the effect.
- **Wipe:** Single press to apply a Wipe Timeline FX to the currently selected timeline segment. Double press to enter the Wipe editor. Press and hold to mute the effect.
- **Axis:** Single press to apply a Axis Timeline FX to the currently selected timeline segment. Double press to enter the Axis editor. Press and hold to mute the effect.

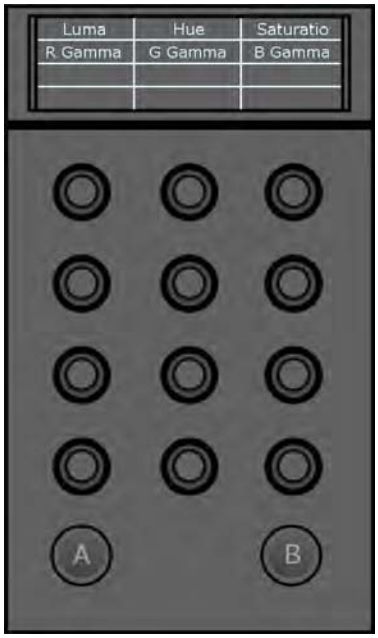
Module 4 (BT) - Page 2 of 2



- **Insert:** Perform an insert edit with the selected clip in the active timeline sequence.
- **Overwrite:** Perform an overwrite edit with the selected clip in the active timeline sequence.
- **Replace:** Perform a replace edit with the selected clip in the active timeline sequence.
- **Dissolve:** Single press to apply a Dissolve video transition to the currently selected timeline segment(s). Double press to enter the Dissolve editor.
- **Wipe:** Single press to apply a Wipe video transition to the currently selected timeline segment(s). Double press to enter the Wipe editor.
- **Axis:** Single press to apply an Axis video transition to the currently selected timeline segment(s). Double press to enter the Axis editor.

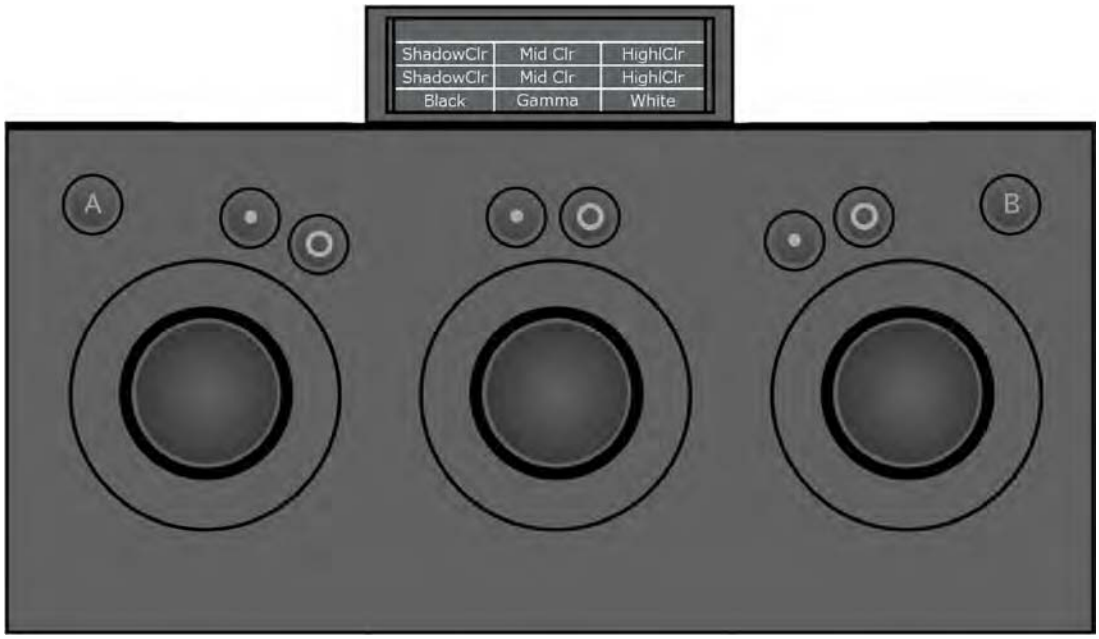
Colour Warper Master Focus Layout

Module 1 (KB)



- **Luma:** Adjust the RGB luma value.
- **Hue:** Adjust the RGB hue value.
- **Saturatio:** Adjust the RGB saturation value.
- **R Gamma:** Adjust the Red Gamma value.
- **G Gamma:** Adjust the Green Gamma value.
- **B Gamma:** Adjust the Blue Gamma value.

Module 2 (TK)



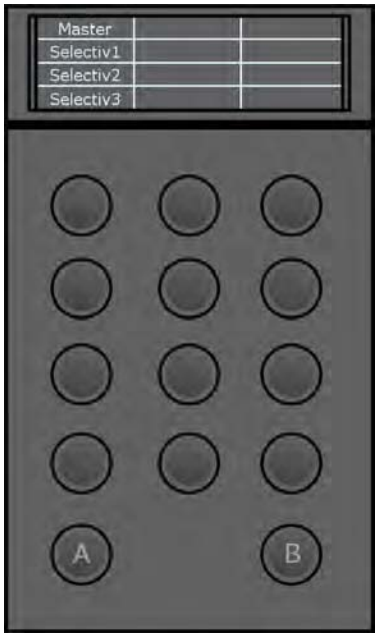
- **ShadowClr:** Move the trackball to adjust the colour of the shadows.
- **Mid Clr:** Move the trackball to adjust the colour of the midtones.
- **HighClr:** Move the trackball to adjust the colour of the highlights.
- **Black:** Adjust the black point.
- **Gamma:** Adjust the RGB gamma value.
- **White:** Adjust the white point.

Module 3 (MF)



- **Exit:** Exit the current menu.
- **Redo:** Reapply the last operation performed after an Undo operation.
- **Undo:** Undo the last operation performed.
- **Pan (Trackball + Alt):** Spin the ball along the X axis to pan the current shot left and right.
- **Scroll (Ring):** Turn the ring to scroll the current shot backward or forward.
- **Zoom (Ring + Alt):** Turn the ring to zoom the current shot in and out
- **Backward Play button:** Play the current shot backward. Results may vary according to the number of unrendered effects on the shot.
- **Backward Play + Stop buttons:** Move the positioner one frame backward.
- **Stop button:** Stop playback.
- **Forward Play + Stop buttons:** Move the positioner one frame forward.
- **Forward Play button:** Play the current shot forward. Results may vary according to the number of unrendered effects on the shot.
- **Previous button:** Go to the start of the current segment.
- **Previous button (Alt):** Go to the previous keyframe.
- **Next button:** Go to the end of the current segment.
- **Next button (Alt):** Go to the next keyframe.

Module 4 (BT)



- **Master:** Adjust the Red suppression value.
- **Selectiv1:** Press to target selective 1.
- **Selectiv2:** Press to target selective 2.
- **Selectiv3:** Press to target selective 3.

Troubleshooting

In the event that one or more of your panel modules are not recognized by Flame Premium, restarting the background service that manages the interactions between Flame Premium and the Tangent Element Control Surface can resolve the issue.

To restart the background service:

1. Exit Flame Premium.
2. Open a shell on your Linux machine.
3. Login as root.
4. Type: */etc/init.d/tangenthub restart*
5. Launch Flame Premium.

All of your module panels should now be recognized.

If the above procedure does not resolve the issue, you can manually connect the Control Surface to your software. To do this, you must define each module's unique ID number in the panel-list.xml file within the Flame Premium filesystem for the panel to be recognized.

To manually install the panel:

1. Plug and connect the four or five modules (depending on your setup) of your Tangent Element Control Panel to a power outlet and to your USB hub.

2 Once the modules are booted up and are displaying the Tangent logo, hit the A button on each module to display its unique ID number.



3 On your Linux machine, log in as root and use kEdit to edit the panel-list.xml file:
kedit /var/opt/Tangent/Hub/panel-list.xml

```
<?xml version="1.0" encoding="utf-8" standalone="yes"?>
<TangentWave fileType="PanelList" fileVersion="3.0">
    <Panel type="Element-Mf" number="1" serial="30"/>
    <Panel type="Element-Bt" number="1" serial="68"/>
    <Panel type="Element-Bt" number="2" serial="77"/>
    <Panel type="Element-Kb" number="1" serial="53"/>
    <Panel type="Element-Tk" number="1" serial="13"/>
    <Panel type="Wave"/>
    <Options>
        <DHCP enabled="true"/>
    </Options>
</TangentWave>
```

4 Add the corresponding ID numbers to the panel-list.xml file.

NOTE In a four module setup, the line <Panel type= "Element-BT" number="2" serial="00"/> should be left blank.

5 Save the file.

The Tangent Element Control Surface should be now recognized by Flame Premium.

Flame Premium Keyboard Shortcuts 30

Following is a selection of Flame keyboard shortcuts.

You can also find keyboard shortcuts by Searching the Keyboard Shortcut Catalogue (page 1666).

Navigation

Function	Flame Shortcut
MediaHub	Space - F1
Conform	Space - F2
Timeline	Space - F3
Batch	Space - F4
Tools	Space - F5
Tools - Composite	Space - 1
Tools - Look	Space - 2
Tools - Filter	Space - 3
Tools - Plugins	Space - 4
Tools - Clip	Space - 5
Tools - Utilities	Space - 6

Viewing Panel

Function	Flame Shortcut
Desktop Reels	~(tilde)
Thumbnail View	Ctrl - ~(tilde)
Standard Player	ESC

Function	Flame Shortcut
Standard Player Real-Time Deliverables	Alt - 1
Standard Player Audio Desk	Alt - 2
Standard Player Overlays	Alt - 3
Standard Player Vectorscope	Alt - 4
Standard Player Viewing Settings	Alt - 5
Player View	Ctrl - 1
Source/Sequence Player	Ctrl - 2
Triptych Player	Ctrl - 3
Trim View	Ctrl - 4
Full Screen Player	Ctrl - ESC
Player Source Clip Next	Shift - Right Arrow
Player Source Clip Previous	Shift - Left Arrow

Workspace

Function	Flame Shortcut
Tall	Shift - Tab
Details	Shift - ~
Hidden	Shift - ESC / Ctrl - Swipe

Desktop

Function	Flame Shortcut
Copy (Whole Clip)	Shift - Alt - drag
Copy (Current Frame)	Shift - Ctrl - Alt - drag
Copy In and Out	Shift - Ctrl - drag
Create Sub-Clip	Alt - C
Contextual Menu (Mouse right click tablet emulation)	Win (Smoke) - Click
Frames view (Cursor Over)	C

Function	Flame Shortcut
Storyboard view (Cursor Over)	Space - C
Delete (Cursor Over)	D
Rename (Cursor Over)	N
Lock (Cursor Over)	Space - L
Go To Frame (Cursor Over)	G
Go To Timecode (Cursor Over)	Alt - G
Cut After Selected Frame (Cursor Over)	Space - X
Cut At Each Splices (Cursor Over)	Alt - Space - X
Cut At (n) Frame (Cursor Over)	Ctrl - Space - X
Swap Segments (Cursor Over)	Space - S
Match Frame (Cursor Over)	M
Match Source (Cursor Over)	Alt - M
Match Content (Cursor Over)	Space - M
Match All Sources (Cursor Over)	Alt - Space - M
Render Segment (Cursor Over)	Space - R
Render Sequence (Cursor Over)	Alt - Space - R
Hard Commit Segment (Cursor Over)	Space - H
Hard Commit Sequence (Cursor Over)	Alt - Space - H

Timeline Navigation

Function	Flame Shortcut
Next Transition (Current Track)	Down Arrow
Next Transition (All Versions/Tracks)	Ctrl - Down Arrow
Previous Transition (Current Track)	Up Arrow
Previous Transition (All Versions/Tracks)	Ctrl - Up Arrow
Go to Clip Start	Ctrl - Left Arrow

Function	Flame Shortcut
Go to Clip End	Ctrl - Right Arrow
Go to In Mark	Alt - [
Go to Out Mark	Alt -]
Next Cue Mark	Ctrl - \
Prev Cue Mark	Alt - \
Zoom In	Ctrl - Space-Up Arrow
Zoom Out	Ctrl - Space - Down Arrow
Fit Timeline to Contents	Ctrl - Home
Fit Timeline to Selection	Shift - =

Editing

Function	Flame Shortcut
Cut (Current Version/Track)	X
Cut (All Versions/Tracks)	Ctrl - Shift - X
Cut Around Selection (Current Version/Track)	Alt - X
Cut Around Selection (All Versions/Tracks)	Alt - Shift - X
Remove Cut (Match Frame Edit)	Ctrl - Alt - X
Add Dissolve (All Versions/Tracks)	Shit - End
Add Dissolve (Current Version/Track)	End
Mark In	[
Mark Out	]
Mark In/Out (Current Shot)	'
Select Between In-Out (Current Track)	Shift - '
Select Between In-Out (Current Version)	Ctrl - '
Select Between In-Out (All Versions)	Ctrl - Shift - '
Clear In	Space - [

Function	Flame Shortcut
Clear Out	Space -]
Clear In-Out	Space - [-]
Add Cue Mark	\
Clear Cue Mark	Space - \
Insert	I
Overwrite	O
Replace	R
Ripple	Alt - R
Ripple Replace	Ctrl - Alt - R
Replace Media	Ctrl - Shift - R
Timeline Select Mode	Alt - A
Timeline Trim Mode	Alt - E
Trim to Positioner	Shift - P
Trim To In Mark	Shift - [
Trim To Out Mark	Shift -]
Trim 1 Frame Forward	. (period)
Trim 1 Frame Backward	, (comma)
Trim <n> Frames Forward	Shift - . (period)
Trim <n> Frames Backward	Shift - , (comma)
Slip Shot	Alt - S
Slide Keyframes	Shift - Alt - S
Slide Cuts	Shift - Alt - D
Slide	Alt - D
Snap	Alt - W
Invert Snap during manipulation	Shift

Function	Flame Shortcut
Invert Ripple during manipulation	Alt
Sub Clip	Alt -C
New Version	Shift - Q
New Video Track	Q
New Audio Track	Ctrl - Q

Effects View Shortcuts

Function	Flame Shortcut
Front View	F1
Back View	F2
Matte View	F3
Result View	F4
Batch or BFX Schematic View	Ctrl - ESC
Schematic View (All other tools with schematics)	ESC
1-Up View	Alt - 1
2-Up View	Alt - 2
Pan	Space
Zoom	Ctrl - Space
Previous Keyframe	Alt - Left
Next Keyframe	Alt - Right

Action Shortcuts

Function	Flame Shortcut
Select	M
Add	A
Delete Mode	D
Mimic Link	W

Function	Flame Shortcut
Schematic Autolink Mimic	Shift - W - drag
Add Action Media Input	Ctrl - N
Reset Media Layer	Ctrl - Alt - click
"Kissing" Nodes	Shift - drag
Reverse "Kiss"	Shift - Alt - drag
Toggle Navigator	Ctrl - Alt - N
Navigator Pan	Alt - drag

Animation Shortcuts

Function	Flame Shortcut
Context Menu (field)	Win (Smoke) - click
Add Channel to Selection	Shift - click
Reset Channel (keep current value)	Alt - click
Reset Channel (Default value)	Ctrl - Alt - click

Playback Controls

Function	Flame Shortcut
Play-Stop	Enter
Play Forward	L
Play Forward Increase Speed	L
Play Forward Decrease Speed	Shift - L
Stop Playback	K
Play Backward	J
Play Backward Increase Speed	J
Play Backward Decrease Speed	Shift - J

General Shortcuts

Function	Flame Shortcut
Keyboard Shortcut dialog box	Ctrl - Alt - F8

Function	Flame Shortcut
Preferences dialog box	Ctrl - Alt - F6
Display Tooltip	Ctrl - W
Exit	Alt - F12
Undo	Ctrl - Z
Redo	Ctrl - R
Help	Ctrl - F1

NOTE When you press a keyboard shortcut, a white keyboard appears in the lower-right corner of the screen. If a keyboard shortcut becomes stuck at any time, the white keyboard remains until you unstick the keyboard shortcut by pressing it. Click the white keyboard to display the name of the problematic keyboard shortcut in the message bar.

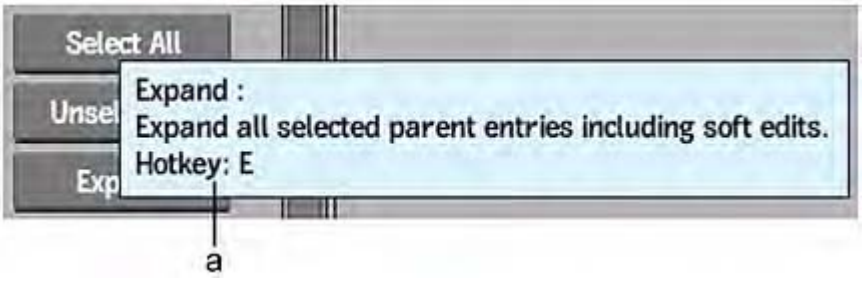
Determining a Button's Current Keyboard Shortcut

You can determine the keyboard shortcut assigned to a button from anywhere within Flame Premium.

To determine the keyboard shortcut for a button:

1. If auto display of tooltips is enabled in the Preferences, hover over the button for which you want to determine the assigned keyboard shortcut. If you have disabled the auto display of tooltips, press and hold `Ctrl+W` and hover over the button.

   If a keyboard shortcut exists for the button, it is displayed in the tooltip.



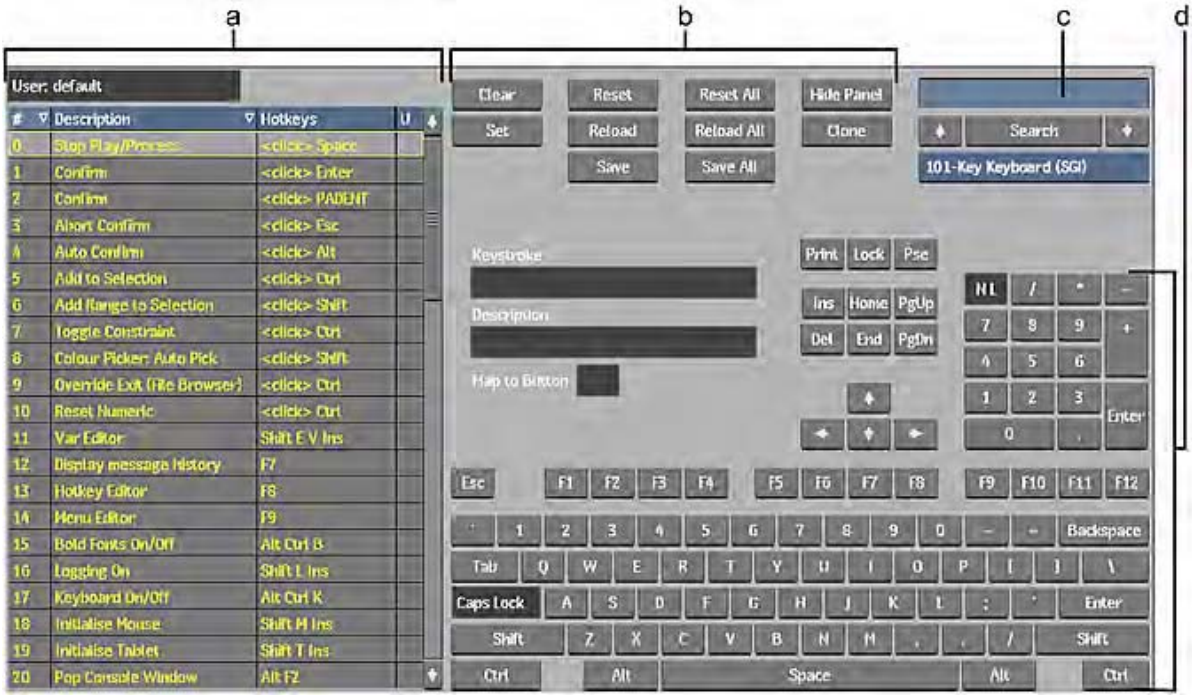
(a) Current keyboard shortcut message

Accessing the Keyboard Shortcut Editor

Use the Keyboard Shortcut Editor to view, modify, and create keyboard shortcuts.

To access the Keyboard Shortcut Editor:

1. From the Flame menu, select Keyboard Shortcuts.



(a) Keyboard Shortcut Catalogue (b) Keyboard Shortcut Manager (c) Search Field (d) Keystroke Editor

Keyboard Shortcut Catalogue

A catalogue of keyboard shortcuts is maintained for every user. If the user profile was created in the default home directory, the catalogue is found in */usr/discreet/user/effects/<user_name>/keyboard shortcut*. If the user profile was created elsewhere, the catalogue is found in *<user home directory>/keyboard shortcut*. The keyboard shortcuts in the keyboard shortcut catalogue are stored in the following formats:

- *default.<component>.butt.keyboard shortcut*
- *default.<component>.func.keyboard shortcut*
- *factory.<component>.butt.keyboard shortcut*
- *factory.<component>.func.keyboard shortcut*

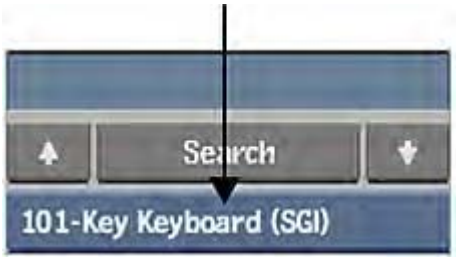
Where:	Means:
<component>	The module containing the keyboard shortcut.
default	The keystroke that is currently in use.
factory	The Autodesk factory default.
butt	A button keyboard shortcut.
func	A non-button function keyboard shortcut.

User-modified keyboard shortcuts are stored in a separate file: *default.<component>.butt.keyboard shortcut.user*. This file is loaded after the system keyboard shortcut file, *default.<component>.butt.keyboard shortcut*. Entries

in the user-modified keyboard shortcut file replace the system keyboard shortcut entries, with the exception of cloned keyboard shortcuts, which are duplicated.

Selecting Your Keyboard Type

In the Keyboard Shortcut Editor you can select the type of keyboard you are using to take advantage of extra keys on certain keyboards.



You have the choice of the following keyboards:

When you select your keyboard, the onscreen keyboard is changed to reflect your choice, and entries in the Keyboard Shortcut Catalogue are updated accordingly.

NOTE If a keyboard shortcut is set on a key not available for the keyboard selected, the Keyboard Shortcut Catalogue displays the entry in black.

Searching the Keyboard Shortcut Catalogue

You can search the Keyboard Shortcut catalogue to find keyboard shortcuts whose description match your search criteria.

To search the Keyboard Shortcuts catalogue:

1. In the Search field, enter the characters you want to search for.
2. Click Search.

   The catalogue is searched and the keyboard shortcuts whose description match your search criteria are highlighted.
3. If more than one keyboard shortcut is highlighted, use the previous and next buttons located next to the Search field to move through the list of search results.

Changing Keyboard Shortcuts

The Keyboard Shortcut catalogue shows all keyboard shortcuts for your product. Keyboard shortcuts appear in yellow text or in white text in the list:

- Yellow text: identifies shortcuts that are used everywhere in your product.
- White text: identifies keyboard shortcuts that are specific to the area you are in.

To edit a keyboard shortcut:

1. From the area where you want to use the new keyboard shortcut, open the Keyboard Shortcut Editor.
2. Select the keyboard shortcut in the Keyboard Shortcut catalogue.

The keystroke sequence and its description appear in the Keystroke Editor fields. If you select a map-to-button keyboard shortcut, the word “Yes” appears in the Map-to-Button field.

3 Click Clear in the Keyboard Shortcut Manager area to clear the existing keystroke sequence.

4 Enter the new keystroke sequence by clicking keys in the Keystroke Editor or by pressing keys on your computer's keyboard.

5 Click Set in the Keyboard Shortcut Manager area.

6 Click Save to save the changes to the current user's Keyboard Shortcut catalogue.

The edited keyboard shortcut is marked in the catalogue with a “Y” to show that it is user-modified.



NOTE If you enter a keystroke sequence that is already in use, an error message is displayed.

Cloning Keyboard Shortcuts

You can use Clone to map multiple keystroke sequences to a single button, field, or function using the regular system keyboard. This feature does not provide macro functionality.

To clone a button, field, or function:

1 Select a button, field, or function in the Keyboard Shortcut catalogue.

2 Click Clone in the Keyboard Shortcut Manager area to create a second entry for this button, field, or function.

A “Y” appears in the Keyboard Shortcut catalogue indicating that the cloned keyboard shortcut is user-modified.

3 Activate a control or enter a keystroke sequence on the system keyboard).

NOTE You must enter a keystroke sequence that is not in use. Otherwise, an error message appears.

4 Click Set in the Keyboard Shortcut Manager area of the Keyboard Shortcut Editor.

5 Click Save to save the changes to the current user catalogue.

Editing Local Keyboard Shortcuts

Almost every module has its own catalogue of keyboard shortcuts. The keyboard shortcuts listed in white are local keyboard shortcuts, and can be customized to suit your needs without affecting other modules. Modified keyboard shortcuts are saved as a user preference.

To edit a local keyboard shortcut:

1 In the module where you want to use the new keyboard shortcuts, access the Keyboard Shortcut Editor (`Alt+Ctrl+F8`).

2 Select the keyboard shortcut in the Keyboard Shortcut catalogue.

The keystroke sequence and its description appear in the Keystroke Editor fields. If you select a map-to-button keyboard shortcut, the word “Yes” appears in the Map-to-Button field.

3 Click Clear in the Keyboard Shortcut Manager area to clear the existing keystroke sequence.

4 Enter the new keystroke sequence by clicking keys in the Keystroke Editor or by pressing keys on your computer's keyboard.

5 Click Set in the Keyboard Shortcut Manager area.

6 Click Save to save the changes to the current user's Keyboard Shortcut catalogue.

The edited keyboard shortcut is marked in the catalogue with a “Y” to show that it is user-modified.



NOTE If you enter a keystroke sequence that is already in use, an error message is displayed.

Editing Global and Shared Keyboard Shortcuts

You can edit global and shared keyboard shortcuts wherever the Keyboard Shortcut Editor is available. When you access the Keyboard Shortcut Editor through the Preferences menu, click Global or Shared to view all current global or shared keyboard shortcuts, respectively. In this menu, the names of these keys appear in yellow.

In other modules, the global and shared keyboard shortcuts appear in yellow in the Keyboard Shortcut catalogue. When changing a global or shared keyboard shortcut, you are warned that the change will affect all modules, and are asked to confirm the action.

Mapping a Button to a Keyboard Shortcuts

You can create new map-to-button keyboard shortcut. New keyboard shortcuts are saved as a user preference.



To map a button to a keyboard shortcut:

1. With the Keyboard Shortcut Editor open, click the menu button for which you want to create the keyboard shortcut.

   NOTE If the button you want to click is hidden by the Keyboard Shortcut Editor, click the Hide Panel button.

   If the menu button you selected is a map-to-button keyboard shortcut, Yes appears in the Map to Button field, and the name of the menu button appears in the Description field.
2. Enter the new keystroke sequence by clicking keys in the Keystroke Editor, your computer's keyboard, or the keys on your pen tablet. The new keystroke appears in the Keystroke field.
3. If the menu button is a field, you can set a default value type and increment in the Value Type box and Increment field.
4. If the menu button is an option box, you can enable Cycle Pop-up to allow your new keyboard shortcut to cycle through the options each time it is pressed.
5. Click Set.
6. Click Save to save the changes to the current user's Keyboard Shortcut catalogue.

Keyboard Shortcut Editor Menu Settings

Keyboard Shortcut Manager Area



Clear button Clears the contents of the Keystroke field in the Keystroke Editor before you enter a new keystroke sequence.

Set button Sets the contents of the Keystroke field.

Reset button Resets the current local keyboard shortcuts to their default settings.

Reload button Reloads the current catalogue of keyboard shortcuts. This is useful if you made a change but have not yet saved it, and you wish to discard the change.

Save button Saves the current keyboard shortcuts to the Keyboard Shortcut Catalogue of the current user.

Reset All button Resets all keyboard shortcuts to their default settings.

Reload All button Reloads all keyboard shortcuts from the Keyboard Shortcut Catalogue of the current user.

Save All button Saves all keyboard shortcuts to the Keyboard Shortcut Catalogue of the current user.

Hide Panel button Hides the Keyboard Shortcut Editor panel when you want to select a menu button hidden by the panel. The panel reappears when you select a menu button. Alternatively, click in an open area to return to the Keyboard Shortcut Editor panel without selecting anything.

NOTE This button is only available when accessing the Keyboard Shortcut Editor from a module.

Clone button Assigns a regular keyboard keystroke sequence to a single button, field, or function.

Keystroke Editor Section



Keystroke field Displays the keystroke sequence. Click keys in the Keystroke Editor or the keyboard to enter a new sequence.

Description field Displays the name of the selected menu button. Non-editable.

Map to Button field Displays whether the selected keyboard shortcut is a map-to-button hot key (a keystroke sequence that is mapped to a button or a field on the current menu). Non-editable.

Value Type box Select a value type for the selected menu button.

Integer Increment field Displays the integer increment for the selected value type. Editable.

Float Increment field Displays the float increment for the selected value type. Editable.

Cycle Pop-up button Enable to allow the new keyboard shortcut to cycle through options for the selected box.

Search Section

Search field Displays the search criteria for the Hot Key Catalogue. Editable.

Search button Performs a search based on the characters in the Search field.

Previous button Selects the previous highlighted item in the search results.

Next button Selects the next highlighted item in the search results.

Miscellaneous Settings

User field Displays the current user that hot keys are saved for. Non-editable.

Keyboard Type box Select the type of keyboard being used to take advantage of extra keys on certain keyboards. See Selecting Your Keyboard Type (page 1666).

Close button Click to close the Keyboard Shortcut Editor, and return to your previous view.

Setting Flame Premium Preferences 31

Customize your working environment using preferences.

To open the Preferences dialog box:

1. Click Flame > Preferences.
2. Select the category of preferences you want to modify. The dialog box opens to the tab with the category of preferences you selected.

Audio Preferences

Player

Monitor Sync box Indicates which monitor will have audio sync when you play back a clip (the broadcast monitor or the high-resolution monitor).

Broadcast Delay field Displays the number of frames (-30.00 to 30.00) to delay the audio signal when syncing to the broadcast monitor. Editable.

Hi-Res Delay field Displays the number of frames (-30.00 to 30.00) to delay the audio signal when syncing to the graphics monitor. Editable.

Auto Fade field Displays the amount of time (0 to 10 ms) that a fade is added to the start and end of each audio segment. Use this setting to avoid audible clicks between audio sources. Editable.

Meters

Meter Range field Displays the overall scale for the fader meters. Editable.

Meter Green field Displays the green meter range. Editable.

Meter Yellow field Displays the yellow meter range. Editable.

Meter Step field Displays the step value for the meters. Enter a lower value to view a more detailed meter.

Meter Units box Select whether the meters display digital or analog units.

Tone

Audio Tone box Select the default audio tone.

Reference Level field Displays the reference level for custom audio tones. Editable.

Tracks

Consider Audio Head/Tail button Enable to include audio that extends before or after video when exporting movie files.

Supported Playback Tracks field Displays the number of playback tracks supported by your sound card. Non-editable.

Supported I/O Tracks field Displays the number of I/O tracks supported by your sound card. Non-editable.

Supported Audio Monitoring Field Displays the audio monitoring option supported by your sound card. Non-editable.

Outputs

Output Source field Displays all current outputs. Non-editable.

Output Sampling Rate field Displays the current output sampling rate. Non-editable.

Monitoring Level field Displays the default volume level. Does not effect output. Editable.

Backburner Preferences

Backburner Manager box The hostname of the Backburner Manager system that handles background jobs submitted by the workstation.

Backburner Group box Specifies a server group (a preset group of render nodes) used to process jobs submitted by the application. By default, Backburner Manager assigns a job to all available render nodes capable of processing it. If you have a dedicated group of render nodes for processing jobs, set the value to the name of the render node group. See the Backburner User Guide for information on creating groups.

Burn Job Type box Configure according to the GPU capabilities of the nodes in your background processing network.

- Software: none of the nodes in your background processing network is equipped with a GPU. The application will not send jobs that require a GPU to the background processing network, but only jobs that can be processed in software mode (using OSMesa) by the render nodes.
- GPU Burn: all the nodes in your background processing network are GPU-enabled. The application will send all jobs to the GPU-equipped nodes in the background processing network, even if some jobs do not specifically require a GPU node. The GPU-equipped render nodes will render jobs that require a GPU, as well as OSMesa jobs. If your background processing network also contains nodes without a GPU, and this setting is used, all jobs are sent only to GPU-equipped render nodes, and the nodes without a GPU are never used.

Burn Priority field Sets the priority for renderings jobs, from 1 (highest priority) to 100 (lowest).

Wire Priority field Sets the priority for transfers from and to this workstation, from 1 (highest priority) to 100 (lowest).

Store Priority field Sets the priority for caching imported media on this workstation, from 1 (highest priority) to 100 (lowest).

Burn Export field Sets the priority for exporting clips from this workstation, from 1 (highest priority) to 100 (lowest).

NOTE Regarding the different priorities, be careful when setting new values or you might end up with a system trying to transfer frames before they are rendered.

Broadcast Monitor Preferences

Broadcast Monitor Preferences

Broadcast Selection box Select what you want to preview on your broadcast monitor (can be changed on-the-fly).

Broadcast Monitor box Select the output device used by the broadcast monitor, or turn it off.

Always Send Grab Area button Enable to display Player or viewport output, including times when neither is in use. Active when the Screen Grab option is selected.

Scale Clip to Fit Monitor button Enable to resize the clip to fit the broadcast monitor. Active when the Show Selected Item option is selected.

Use Ratio button Enable to maintain the original proportions of the clip. Disable to use the monitor ratio. Active when the Show Selected Item option is selected.

Broadcast Multiview Preferences

Viewport Monitoring button Enable to send to the broadcast monitor the content of the viewport displaying the "Monitor" symbol. Disable to use the broadcast monitor as an additional viewport; then use the Broadcast Monitor toolbar to control the broadcast monitor as you would any other viewport.

Hide Broadcast Toolbar button Enable to hide the Broadcast Monitor toolbar. By default, the toolbar is displayed in the top-right corner of the graphics monitor. Spacebar+Click to move the Broadcast Monitor toolbar.

Image Data Type Preferences

Image Data Type box Select the type of image data you are displaying in the broadcast monitor. Your selection determines the type of transformation that is applied to the clip to modify the contrast. It is disabled if the Follow button is enabled.

Bypass button Enable to deactivate the image data type display settings in the broadcast monitor.

Broadcast LUT Preferences

Use 3D LUT in Monitor button Enable to apply a 3D LUT from the 3D LUT list to the broadcast monitor.

3D LUT List box Displays the list of 3D LUTs that you imported in the LUT menu.

Broadcast Overlay Preferences

Broadcast Overlay box Select the overlays displayed on the broadcast monitor. "Off" for no letterbox or reference overlay, "Letterbox" for aspect ratio and letterbox guides, "All" for letterbox and all reference overlay elements.

Broadcast Stereo

Broadcast Stereo box Displays the current stereoscopic display option.

General Preferences

Project Information

Project Framerate field Displays the framerate that corresponds with the current project's default resolution. Non-editable.

Incoming Sync

Incoming Sync field Displays the timing of the sync received by the workstation. Non-editable.

Timecode

Display Dual Timecodes button Enable to display both 29.97 and 23.97 fps timecode on the timeline and in the Player. This allows you to monitor which transitions will fall on jitter frames when 2:3 insertion is applied.

Dual TC: 23.97->29.97 fps NDF/DF box Select whether to view the 29.97 fps timecode in the Dual Timecode display as either drop frame or non-drop frame.

Drop Frame Reference Timecode field Displays the point at which the 29.97 fps material is in sync with the 23.97 fps material. Editable.

Drop Frame Bump Mode box Select a translation mode for invalid calculator entries.

Player

Source Time Mode box Select to display the timecode, keycode, or frame number of the source clip in the Player.

Auto-Toggle Player button Enable to open the Player automatically when you press the keyboard shortcuts for playback.

Display Sequence Viewer button Enable to display the currently selected sequence as a thumbnail with a red border in the Thumbnail view. When disabled, the selected sequence does not appear Thumbnail view.

Jog / Shuttle on Drag button Enable to shuttle a clip in the Player when jogging. The further from the center of the clip that the cursor is placed and dragged, the faster the shuttling.

Auto-Toggle Trim View button Enable to automatically display the Trim View when double-clicking a timeline cut, or when double-clicking a timeline segment with Slip or Slide mode selected.

Untitled Clip Name

Untitled Clip Name box Select an option to rename untitled clips to make it easier to distinguish multiple Untitled clips.

Rendered Clip Name

Rendered Clip Name box Select whether to add an acronym indicating the area of the software your clip was rendered from as a prefix or suffix to a rendered clip name, if a setup name does not already exist for the clip.

Stereo Naming

Left Eye field Displays the token used to identify left eye media in a stereo clip split or exported. Editable.

Right Eye field Displays the token used to identify right eye media in a stereo clip split or exported. Editable.

Animation

Default Interpolation box Select the default interpolation type for the channel editor.

Default Extrapolation box Select the default extrapolation type for the channel editor.

Audio Gain Animation

Default Audio Interpolation box Select the default audio interpolation type for the channel editor.

Autosave

Soft Save field Displays the time delay for a soft autosave. A small red icon appears indicating a two-second delay before a soft autosave occurs (allowing for a cancellation with any movement in the application). Editable.

Hard Save field Displays the time delay for a hard autosave. A hard autosave occurs at the specified time regardless of user intervention. Editable.

Tools Save field Displays the time delay for an autosave when you are using a tool that you accessed from the Tools tab. Editable.

Help Location

Help Location button Displays the location of the product help. By default, the product help is available online. You must have an internet connection to view it. You can download and install the help locally from the Autodesk website.

Browse Help button Opens a browser window. Select the folder that contains the help that you downloaded from the Autodesk website.

Reset Help button Resets the help location to the default.

Help Location field Displays the location of the help.

Setups

Scale Setups button Enable to scale setups when loading that were created in a resolution different from that of the current project.

Image Data Type

Image Data Type box Select a default image type setting.

Undo

Clear Buffer button Removes clips from the undo buffer and frees up disk space on the framestore

Undo Level field Displays how many levels of information are saved to the undo buffer. Editable.

Clip Select

Ignore Scan Format button Enable to disregard scan mode mismatches.

Background Tasks

Update field Displays a value, in seconds, for the wait time to update the Backburner Queue. Editable.

Default Rendering Options

Render box Select whether you want to include history and/or Burn when rendering.

Keep History Mode box Select whether Sources or Intermediates and Sources appear on the workspace when working with clip history.

Shared Libraries

Shared Libraries Refresh Rate field Displays the frequence at which the application refreshes the contents of Shared Libraries. Editable.

Shared Libraries Lock Timeout field Displays how long an inactive shared folder remains available for writing. Editable.

Media Library

Protect from Editing button Enable to protect all of the contents in the Media Library from editing. To work on media in a protected Media Library, you must bring it to a Batch or Desktop folder in the Media panel.

Input Devices Preferences

Pointer

Tablet Model field Displays the model of the currently installed tablet. Non-editable.

Pointer Reset button Returns all pointer preferences to their default values.

Threshold Test button Assesses the sensitivity of the pen interactively. Press the pen on the button and a sliding grey bar in the message bar displays the response to the amount of pressure applied.

Pressure Threshold field Displays the amount of pressure that you want to apply when using the pen. Use a higher value to decrease the sensitivity of the pen (more pressure required).

Use Mouse Cursor button When using a tablet, enable to use mouse cursors instead of tablet cursors.

Trackball Pressure button Enable to allow the trackballs to respond to changes in the pressure applied when using a stylus.

Tablet Margins

Proportional Margins button Enable to change the four margin controls proportionally.

Top Margin Control field Displays a percentage value to adjust the top boundary of the active area on the tablet. Editable.

Bottom Margin Control field Displays a percentage value to adjust the bottom boundary of the active area on the tablet. Editable.

Left Margin Control field Displays a percentage value to adjust the left boundary of the active area on the tablet. Editable.

Right Margin Control field Displays a percentage value to adjust the right boundary of the active area on the tablet. Editable.

LUT Preferences

1D LUT/3D LUT and Colour Transform list Displays the list of imported 1D LUTs or 3D LUTs and Colour Transforms in separate tabs. See Making 1D LUTs Available for Display (page 1556) and Making 3D LUTs and Colour Transforms Available for Display (page 1556).

LUT Type box Select whether you want to define a gamma function or import a ID LUT file in the highlighted row and assign the corresponding keyboard shortcut.

Import button Opens the Import LUT browser. Navigate to the 1D LUT file to import, and select it to load it into the list. This button is available when the LUT Type box is set to LUT File.

Value field Displays the gamma value. This field is available when the LUT Type box is set to Gamma.

Select button Apply the highlighted gamma or 1D LUT to all displayed images. See Applying 1D LUTs to the Monitor (page 1557).

Import button Opens the Import LUT browser. Navigate to the 3D LUT or Colour Transform file to import, and select it to load it to the 3D LUT and Colour Transform list.

Reset button Removes the highlighted item from the list.

Reset All button Removes all LUTs and Colour Transforms from the current list.

Shared Directory Displays the file path to the shared colour transforms directory. This is a useful location for storing custom transforms that can be used by other applications that use Autodesk Colour Management.

Browse Browse to select a different shared directory for color transforms. The path you select is used by all compatible applications on the same workstation.

Default Reset the shared directory to its installation default.

Broadcast Monitor Transform Specifies the colour transform that is used when the broadcastMonitor alias is referenced inside another transform. See Configuring the Colour Transform Aliases (page 1556).

Default Reset the Broadcast Monitor Transform to its installation default.

Browse Browse to select a different colour transform for the broadcastMonitor alias.

Graphics Monitor Transform Specifies the colour transform that is used when the graphicsMonitor alias is referenced inside another transform. See Configuring the Colour Transform Aliases (page 1556).

Browse Browse to select a different colour transform for the graphicsMonitor alias.

Default Look Transform Specifies the colour transform that is used when the defaultLook alias is referenced inside another transform. See Configuring the Colour Transform Aliases (page 1556).

Clear Sets the value of the defaultLook alias to the identity transform, which has no effect on colour values.

Browse Browse to select a different colour transform for the defaultLook alias.

Storage Preferences

Space Calculation

Space Calculation options Choose to display all available frames in the message bar, all available and purgeable frames in the message bar, or to purge frames before making a space calculation.

Volume Statistics button Lists the number of frames, proxies, and audio sources stored in the clip libraries, and main work area of each project.

Video Framestore Estimate button Displays the estimated amount of space left on the video framestore.

Framestore

Test Disks button Displays the number of frames read per second; the number of seconds required to read a single frame; and the amount of data, in megabytes, read per second.

Support Preferences

Upload Config Info Only button Click to upload only configuration information to Autodesk Customer Support. Use only if instructed by Autodesk Customer Support.

Upload Config Info and Logs button Click to upload pertinent configuration information and system logs to Autodesk Customer Support. Use only if instructed by Autodesk Customer Support.

Max Logged Days field Displays the number of logged days of information to be uploaded to Autodesk Customer Support. Editable.

Case ID field Enter the Case ID number given to you by Autodesk Customer Support so that it is included in the uploaded information.

Timeline Preferences

Editing

Default Timecode field Displays the default start timecode value for new items overwritten or inserted into a timeline sequence. Editable.

Merge Tracks Mode box Select how a timeline will be flattened when merging tracks.

Select:	To merge tracks by:
Simple Track Merge	Hard committing all soft transitions. Cut points between elements are preserved; clip handles are not.
Complex Track Merge	Hard committing soft transitions but creating separate elements with clip handles for each.
Committed Track Merge	Preserving editable soft transitions and clip handles.

Auto Timewarp button Enable to specify the behaviour of four-point replace and overwrite edits, when they involve a different number of frames between the in and out points on the source and sequence clips.

Framerate Converter button Enable to format the source clip to the correct destination framerate by applying a video timewarp to the source clip.

Snap Includes Marks button Enable to snap to timeline marks.

Snap With Positioner button Enable to snap segments to the positioner.

Display Phantom Marks button Enable to turn on phantom marks on the timeline and source clip. Phantom marks indicate the result of a 4-point edit regardless of whether you have marks set.

Containers

Uncontain options box Select whether new tracks and versions are added or overwritten when a container segment is uncontained.

Contain Timewarped Edits button Enable to preserve the edits in a fit-to-fill four-point edit. The incoming clip is contained and a timewarp is applied to the container instead of the clip.

Segment Display

Handles box Select an option for displaying heads and tails on segments in the timeline.

Navigation

Frame Positioner box Select whether to snap the timeline positioner frame-to-frame, or on a sub-frame basis (one-tenth of a frame).

Autoscroll field Displays the speed at which the timeline scrolls. Editable.

Scroll During Playback button Enable to scroll the timeline during playback, keeping the frame positioner visible.

Scroll Past First Frame button Enable to allow the positioner to move before frame one.

Transitions

Dissolve Duration field Drag left or right, or enter a value to set the default duration for dissolves.

Transition Alignment box Sets the default alignment for dissolves, wipes, axis, and custom transitions.

Curve Type box Select Bezier or linear as the default curve type for dissolves.

Default Wipe button Opens the file browser, where you select the default wipe type to be used when creating wipe transitions and timeline effects.

Default Wipe field Displays a SMPTE wipe number to be used as the default wipe for wipe transitions and timeline effects. Editable.

Rendering

Timeline Effects Render Mode box Select the render mode when a Timeline FX is added to a clip.

Timewarp Render Mode box Select the default rendering mode for timewarps created in the timeline.

Transitions Render Mode box Select a default rendering mode for transitions created in the timeline.

Cache Preview Effects button Enable to cache frames rendered in the timeline.

Auto Rendering button Enable to background render automatically when you enter the Axis, Wipe, Colour Corrector, or Sparks Editor.

Timeline Effects Rendering box Select whether to render an entire track first, then the next track (Track Based Render), or to render all tracks frame-by-frame (Frame Based Render).

Timeline FX / Batch FX Preferences

Batch FX

Interactive Max field Displays the number of seconds that the system uses to attempt a render when previewing the effect. Editable.

Add Axis On Matte Output button Enable to have an Axis timeline effect automatically added to the timeline when a matte is output from the output node to the timeline.

Always Load Nodes button Enable to automatically load all FX nodes into memory when a setup is loaded. Disable to load nodes when entering an FX level.

Clear Rendered Memory button Enable to automatically clear an FX node (and all nodes in setups nested within the clip) from memory when its output is rendered.

Batch FX Mode

Head Media option box Select how to interpret missing information before a clip.

Gap Media option box Select how to interpret missing information during a clip.

Tail Media option box Select how to interpret missing information after a clip.

Batch FX Automatic Nodes Copy

Automatic Nodes Copy box Select whether FX nodes are never copied into memory, always copied, copied when there is an Action node in the setup, or copied in a partially rendered setup.

Timeline Timewarp

Interpolation Option box Select an interpolation option for your interlaced timewarp.

Use Last Speed button Enable to apply the last timewarp speed to future timewarps.

Timewarp Speed Display box Select whether timewarp speed is displayed as a percentage (%) or in frames per second (fps).

Timewarp Sample Option box Select whether the speed of the timewarp is set in relation to the speed at the beginning, middle, or end of the timing curve for each frame or field. You can also change the Timewarp Sample option in the Timewarp Editor to override this selection.

Timeline Resize

Fit Method box Select a fit option for clips that must be resized when they are added to the timeline because their resolution is different from the project.

Resize Quality box Select a sampling algorithm for clips that must be resized when they are added to the timeline.

Keep Aspect button Enable to maintain the aspect ratio of clips that are resized when you add the to the timeline.

Timeline Text

Text Anti-aliasing box Select an anti-aliasing softness rate for text.

Default Font field Opens the font browser. Select a default font and click Load.

Timeline Blend

Timeline Blend box Select the type of blend that is applied by default when you add a blend timeline effect.

Timeline Colour Correct

Default Menu box Select whether the Colour Corrector or Colour Warper is the default menu displayed when you access the Colour Corrector through a timeline effect.

Viewport Settings

Default Viewport box Select whether to use a 1-Up or 2-Up view as the default Batch FX view. You can also change the viewport layout from within Batch FX.

Batch FX Cache Life Span

Cache Life Span field Displays the number of days that Batch FX clips are cached. Editable.

Batch Cache Life Span

Cache Life Span field Displays the number of days that Batch clips are cached. Editable.

Batch Read File Node

Absolute Frame Offset button Enable to allow all clips imported with a Read File node in Batch to automatically use the start frame of the clip.

Batch Snapshot

Also Save Batch File button Enable to save a Batch setup file when saving a Batch Snapshot.

User Interface Preferences

Colour

Background field Displays the background brightness value. Editable

Brightness field Displays the brightness of interface elements such as buttons and fields. Editable.

Contrast field Displays the contrast of interface elements such as buttons and fields. Editable.

On-Screen Widgets

On-screen Keyboard button Enable to have the on-screen keyboard appear when you enter text in a field.

Calculator Placement box Select where the calculator appears when you click in a numeric field.

Display

Clip Information box Select to display frames or timecodes in frame marking controls, the timebar, segments, and the fields below the timeline.

Bold Font button Enable bold fonts on buttons.

Drag Transparency field Displays the transparency level of clips while dragging. Editable.

Auto Key Button Look

Auto Key look box Select whether to apply a Classic (grey) or Coloured (yellow) look to the Auto Key button.

Thumbnail View

Snap to Grid button Enable to snap thumbnails to a grid. The grid becomes visible when this button is enabled.

Grid Size field Enter a value in pixels for the size of the boxes that make the grid.

Thumbnail Height field Enter a default value in pixels for the height of thumbnails displayed in the Thumbnail view. All new thumbnails are generated at this height. Existing ones are unchanged.

Tooltips

Auto Display button Enable to automatically display tooltips. If disabled, you can still display tooltips for selected buttons using the keyboard shortcut Ctrl+W.

Hover field Displays the amount of time you must position the mouse over the object before the tooltip displays (in seconds). Editable.

Duration field Displays the amount of time that the tooltip is visible (in seconds). Editable.

Reels

Friction field Displays a value that represents how much the reels scroll after you release your mouse/pen. Increase the value to limit the amount the reel scrolls after you release the mouse/pen.

Reels Orientation box Select a horizontal or vertical orientation for the reels.

Reverse button Enable to reverse the order that clips appear on the reels.

Clip Spacing field Enter a value to control the padding between clips on the reels.

Number of Default Reels field Enter a value for the default number of reels that appear in the Reels view.

Broadcast Display button Enable or disable Broadcast Monitor Display.

Gestural Workflow

Swipe Bars button Click to enable swipe bars that you can use to switch views in different areas of your product.

Layout Selection Overlay button Click to enable the menu overlay in the Timeline view. You can use the menu overlay to change what is displayed in the Timeline view.

Gateway Clip Description

32

The Gateway .clip is an XML structured file that describes at least the following information.

- Physical location of files, and additional metadata information, such as timecode and keycode
- Clip versions

NOTE This documents to use notation as close as possible to DTD definitions.

- xml-element?: zero or one instance of xml-element;
- xml-element+: at least one instance of xml-element;
- xml-element*: any number of instances of xml-element;
- #cdata: a placeholder for data described in the xml-element description, usually a string of any valid characters;

---

clip

The `<clip>` element defines the top-level data structure. The minimum elements required to create a valid `<clip>` are:

- `<tracks>`
- `<versions>`

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	clip	Required
version	Version of the xml element.	integer	3	Required

Attribute	Description	Data Type	Allowed Values	Attribute is...
uid	Optional unique identifier. When used, `uid` must be a unique ID within the .clip file.	ID	any	Implied

Children

handler?, name (page 1688)?, sourceName?, startTimeCode?, duration (page 1686)?, editRate?, dropMode (page 1685)?, userData (page 1694)?, comment (page 1684)?, tracks (page 1694), versions (page 1696)

comment

The `<comment>` contains an unparsed string. Use it to store comments.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	string	Implied
version	Version of the xml element.	unsigned integer	3	Implied

Children

#cdata

Found in

<clip> (page 1683), <feed> (page 1686), <track> (page 1693), <version> (page 1695)

creationDate

The `<creationDate>` contains an unparsed string. Use it to store the creation date of the version.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	string	Implied

Attribute	Description	Data Type	Allowed Values	Attribute is...
version	Version of the xml element.	unsigned integer	3	Implied

Children

#cdata

Found in...

<version> (page 1695)

denominator

The <denominator> must be an unsigned integer. It expresses, together with the <numerator> element, a frame rate.

The <denominator> element has no attributes.

Children

#cdata

Found in...

<rate> (page 1690), <sampleRate> (page 1691)

dropMode

The drop frame mode of the media.

The <dropMode> element has no attributes.

Allowed values for <dropMode>:

- DF: drop-frame media
- NDF: non drop-frame media

Children

#cdata

Found in...

<clip> (page 1683), <duration> (page 1686), <track> (page 1693)

duration

The <duration> element is the number of samples that make up the media. It is a is a 64-bit integer. A <duration> without a sibling <path> element is the defines the parent as a gap.

The <duration> element has no attributes.

Children
#cdata

Found in...
(page 1691)

duration

The duration and framerate of the media.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	time	Required
version	Version of the xml element.	unsigned integer	3	Implied

Children
rate (page 1690)?, nbTicks (page 1688), dropMode (page 1685)

Found in...
<clip> (page 1683), <track> (page 1693)

feed

The <feed> element allows you to manage and reference different qualities and resolutions for the same reference media.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	feed	Implied
version	Version of the xml element. Do not use, unless mixing and matching differing xml schema versions, which should not be done.	unsigned integer	3	Implied

Attribute	Description	Data Type	Allowed Values	Attribute is...
vuid	Identifies the version of the feed. In a <track> element, no two <feed> elements can have the same vuid. <feed> elements in different <track> elements can share the same vuid.	ID	any unique (within <feeds>)	Required
uid	Uniquely identifies the feed within the .clip file. Duplicate uid in a .clip are not allowed. We recommend that you use a GUID, but any unique identifier will do.	ID	any unique (within .clip)	Required

Children

handler?,storageFormat (page 1692)?,sampleRate (page 1691)?,startTimecode?,startOffset?,userData (page 1694)?,comment (page 1684)?,spans (page 1692)

Found in...

<feeds> (page 1687)

feeds

The <feeds> element contains the different <feed> elements. In a Gateway .clip, each <feed> can be conceptualized as a version of the media.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	feeds	Implied
version	Version of the xml element.	unsigned integer	3	Implied
currentVersion	Must match the vuid attribute of one of the children <feed>. currentVersion indicates the active <feed>, or current version the client application should use.	IDREF	vuid of a child <feed>	Required

Children

feed (page 1686)+

Found in...

<track> (page 1693)

name

The <name> element should be interpreted by the client application as the name of the parent element.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the xml element.	character data	string	Implied
version	Version of the xml element.	unsigned integer	3	Implied

Child Element

#cdata

Found in...

<clip> (page 1683), <track> (page 1693), <version> (page 1695)

Example

Flame displays <name> (with <clip> as immediate parent) in Batch as the clip name in the Batch setup. But <name> as child of <track> is used as the name of the channel.

nbTicks

The nbTicks element is the number of samples that make up the media. It is a is a 64-bit integer.

This number divided by the value of the <rate> element defines the duration of the media.

The nbTicks element has no attributes.

Children

#cdata

Found in...

<duration> (page 1686)

numerator

The <numerator> element must be an insigned integer. It expresses, when divided by the <denominator> element, a frame rate.

The <numerator> element has no attributes.

Children

#cdata

Found in...

rate (page 1690), sampleRate (page 1691)

path

The <path> element is the path to the media. The path can be relative or absolute. You can also use range brackets for file sequences.

```
/dir/clip.mp4
/dir/fileSequence/dpxSequence.[0-99].dpx
```

NOTE Regarding files sequence and padding in the <path> element: The Gateway reconstructs a sequence as indicated by brackets. dpxSequence.[1-99].dpx indicates to the Gateway there are 99 dpx files named dpxSequence.1.dpx to dpxSequence.99.dpx. It can also manage padding. dpxSequence.[001-099].dpx is read as a sequence of 99 files named dpxSequence.001.dpx to dpxSequence.099.dpx.

In a Smoke and Flame context, use the subFeedId attribute to pair up paths and create stereo spans.

```
<path subFeedId="Left">/dir/stereoClip.Left.[0-99].dpx</path>
<path subFeedId="Right">/dir/stereoClip.Right.[0-99].dpx</path>
```

The actual use and interpretation of the subFeedId attribute is left to the client application.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	path	Implied
version	Version of the xml element.	unsigned integer	3	Implied
subFeedId	Use this attribute to further identify the type of feed. In a Smoke or Flame context, you create stereo pairs by using the "Left" and "Right" values.	character data	any	Implied

Children

#cdata

Found in...

 (page 1691), <paths> (page 1690)

paths

The <paths> element contains all the <path> elements which contain the different paths to the media.

```
<span>
    <paths>
        <path>path_1_of_2</path>
        <path>path_2_of_2</path>
    </paths>
<span>
```

If there is only one child <path> element, <paths> does not have to be used.

```
<span>
    <path>singlePath</path>
</span>
```

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	paths	Implied
version	Version of the xml element.	unsigned integer	3	Implied

Children

path (page 1689)+

Found in...

 (page 1691)

rate

Frame rate of the media. If no frame rate is specified, it is up to the application to decide on the frame rate to use.

Use numerator/denominator elements to express as a fraction frame rates that use decimals; this provides more precision. The following expresses a frame rate of 23.976 fps.

```
<rate type="time">
 <numerator>24000</numerator>
 <denominator>1001</denominator>
</rate>
```

To express frame rates that use integers, use the short-hand notation without child-elements but only a single integer.

```
<rate type="time">25</rate>
```

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	rate	Implied

Attribute	Description	Data Type	Allowed Values	Attribute is...
version	Version of the xml element.	unsigned integer	3	Implied

Children

(numerator (page 1688), denominator (page 1685)) | #cdata

NOTE `#cdata` must be an unsigned integer.

sampleRate

The `<sampleRate>` element describes the frames per second rate of the media described by the `<feed>` element. If no frame rate is specified, it is up to the application to decide on the frame rate to use.

Use numerator/denominator elements to express as a fraction frame rates using decimals. Use an integer for other cases. The following expresses a frame rate of 23.976 fps.

```
<sampleRate type="time">
      <numerator>24000</numerator>
      <denominator>1001</denominator>
</sampleRate>
```

To express a frame rate using an unsigned integer, use the short-hand notation without child-elements but only a single integer.

```
<sampleRate type="time">25</sampleRate>
```

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	rate	Required
version	Version of the xml element.	unsigned integer	3	Implied

Children

(numerator (page 1688),denominator (page 1685)) | #cdata

span

The `<span>` element contains media that make up all or parts of the `<feed>`. A `<span>` element describes its duration and the path to the physical media files. A `<feed>` made up of multiple `<span>` elements means that multiple media files/sequences make up that one `<feed>`. The order in which each `<span>` element is listed is the order in which they are read to rebuild the sequence.

All media referenced in a `<span>` must have homogeneous properties (same codec, compression, format, length, rate...). To mix types of media in a .clip, use different `<track>` elements.

A with a <duration> but no <path> is the definition of a gap.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	span	Implied
version	Version of the xml element.	unsigned integer	3	Implied

Children

duration (page 1686),(paths (page 1690)|path (page 1689))?,userData (page 1694)?

Found in...

<spans> (page 1692)

spans

The <spans> element contains all the elements that make up the <feed>.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	spans	Implied
version	Version of the xml element.	unsigned integer	3	Implied

Children

span (page 1691)+

Found in...

<feed> (page 1686)

storageFormat

The <storageFormat> element explicitly describes the format of the media referenced. This element is populated by the Gateway and does not need to be populated at the creation of the .clip file.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	format	Required

Attribute	Description	Data Type	Allowed Values	Attribute is...
version	Version of the xml element.	unsigned integer	3	Implied

Children

type,channelsDepth,channelsEncoding,channelsEndianess,height,nbChannels,pixelLayout,pixelRatio,RowOrdering,width

Found in...

<feed> (page 1686)

track

The <track> elements contains all the information that make up a track in a Gateway clip.

Examples of tracks in a Gateway .clip context:

- A layer in a Photoshop file;
- A video, or one of multiple audio tracks in a QuickTime file;
- One of multiple render passes contained in a multi-channel OpenEXR file.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	track	Implied
version	Version of the xml element.	unsigned integer	3	Implied
uid	Unique identifier of the track element. This value must be unique within the .clip file.	ID	any	Required

Children

trackType (page 1694), extendedType?, handler?, sourceName?, startTimecode?, dropMode (page 1685)?, duration (page 1686)?, name (page 1688)?, editRate?, userData (page 1684)?, comment (page 1684)?, feeds (page 1687)

Found in...

<tracks> (page 1694)

tracks

The container for all the track elements making up the current .clip file.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	tracks	Implied
version	Version of the xml element.	unsigned integer	3	Implied

Children

track (page 1693)+

Found in...

<clip> (page 1683)

trackType

The type of media described in the track element. `<trackType>` must match the type of media later referenced by the feeds elements.

The `<trackType>` element has no attributes.

Allowed values for `<trackType>`:

- video
- audio

Children

#cdata

Found in...

<track> (page 1693)

userData

`<userData>` is dictionary object that stores client-defined elements, passed directly to the client application by the Gateway.

Here is an simple example of a `<userData>` structure used to store the name and version of the source application that created the .clip.

```
<userData type="dict">
 <appName type="string">Flame</appName>
```

```
  <appVersion type="int16">2012</appVersion>
</userData>
```

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	dict	Required
version	Version of the xml element.	unsigned integer	3	Implied

Children

#undefined

No children elements are defined for the `<userData>` element since its data structure is undefined; each child element created by the client must define its data type using the `type` attribute. The allowed data types are listed below.

Data Type	Defined Values
Integer (bound or unbound)	int, int8, int16, int32, int64
Unsigned integer (bound or unbound)	uint, uint8, uint16, uint32, uint64
Floating point	float, double
Boolean	bool
String	string

Found in...

<clip> (page 1683), <feed> (page 1686), (page 1691), <version> (page 1695)

version

The `<version>` element contains metadata for a clip version, and lists an existing version using its `uid` attribute.

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	version	Implied
version	Version of the xml element.	unsigned integer	3	Implied

Attribute	Description	Data Type	Allowed Values	Attribute is...
uid	Defines which <feed> of each <track> to use, based on matching vuid.	IDREF	Any vuid defined in a <feed>. Each uid must be unique across <version> elements.	Required

Children

name (page 1688)?,comment (page 1684)?,creationDate (page 1684)?,userData (page 1694)?

Found in...

<versions> (page 1696)

versions

The <versions> element lists two things: the available versions of a clip, and the version currently in use.

Different versions of a track are described by different <feed> elements each identified with a unique vuid attribute. In a Gateway clip XML, a specific version of a clip is defined as all the <feed> elements sharing the same vuid attribute accross different <track> elements.

The <version> element describes and lists the available versions, while the currentVersion attribute defines which <version> element is the current one. The client application that reads the .clip is not required to currentVersion; it only serves as a flag put there by the creator.

Consider the following, simplified example.

```
<tracks>
 <track uid="track1">
  <feeds>
   <feed vuid="v1" uid="t1f1"/>
   <feed vuid="v3" uid="t1f2"/> <!-- version 3 of the track, version 2 was never
created -->
  </feeds>
 </track>
 <track uid="track2">
  <feeds>
   <feed vuid="v1" uid="t2f1"/>
   <feed vuid="v2" uid="t2f2"/>
   <feed vuid="v3" uid="t2f3"/>
  </feeds>
 </track>
</tracks>
```

And you have the following <versions> structure.

```
<versions currentVersion="v2">
 <version uid="v1"/>
 <version uid="v2"/>
```

```
 <version uid="v3"/>
</versions>
```

You now have a clip that can display the following:

- Version 1: track1 using feed t1f1 and track2 using t2f1;
- Version 2: track2 using t2f2 only(there is no `vuid` in track1 matching "v2");
- Version 3: track1 using feed t1f3 and track2 using t2f3;

And in this example, the client application should display the version 2 of the clip because `currentVersion=` 2 .

Attribute	Description	Data Type	Allowed Values	Attribute is...
type	The data type of the element.	character data	versions	Implied
version	Version of the xml element.	unsigned integer	3	Implied
currentVersion	Defines the current version of the clip, as set by the client application that created the file. The client application is expected to view this version as the most up-to-date. If not defined, the Gateway considers the last `<version>` element defined in the file as the current version.	IDREF	any existing one `uid` from an enclosed `<version>` element.	Implied

Children

version (page 1695)+

Found in...

<clip> (page 1683)

Creating a Simple Gateway Clip

A simple Gateway clip contains one video track. It has no versioning and no setup information; it is the simplest case, and is a nice, practical, exercise.

Solution

Use the simplest form of .clip you can create: one video track, no audio. The following is the minimal structure required to to have a valid Gateway clip.

```
<?xml version="1.0" encoding="UTF-8"?>
<clip type="clip" version=3>
    <tracks>
        <track uid="client-defined unique identifier">
            <trackType>video</trackType>
            <feeds currentVersion="the <feed> vuid">
              <feed  vuid="client defined" uid="client-defined unique identifier">
```

```
                    <spans>
                        <span>
                            <duration>number of samples in the media
sequence</duration>
                            <path>path the media sequence</path>
                        </span>
                    </spans>
                </feed>
            </feeds>
        </track>
    </tracks>
    <versions currentVersion="one of the <version> uid">
        <version uid="the currentVersion of <feeds>"/>
    </versions
</clip>
```

NOTE Regarding files sequence and padding in the `<path>` element: The Gateway reconstructs a sequence as indicated by brackets. `dpxSequence.[1-99].dpx` indicates to the Gateway there are 99 dpx files named `dpxSequence.1.dpx` to `dpxSequence.99.dpx`. It can also manage padding. `dpxSequence.[001-099].dpx` is read as a sequence of 99 files named `dpxSequence.001.dpx` to `dpxSequence.099.dpx`.

Example and Discussion

Here is a commented .clip file. It does contain additional elements than the bare minimum, so as to play nicely with Smoke, Flame, or Lustre applications.

```
<?xml version="1.0" encoding="UTF-8"?>
<clip type="clip" version=3>
    <name type="string">NoVersionNoSetup</name>
    <tracks>
        <track uid="1b22da75-33e5-4dbe-80d0-7ed680a8b2b7">
            <trackType>video</trackType>
            <duration type="time">
                <rate type="rate">
                    <numerator>30000</numerator>
                    <denominator>1001</denominator>
                </rate>
                <nbTicks>26</nbTicks>
                <dropMode>NDF</dropMode>
            </duration>
            <name type="string">ClipNoVersionNoSetup</name>
            <feeds currentVersion="0">
                <feed vuid="0" uid="track1Feed1">
                    <spans>
                        <span>
                            <duration>26</duration>
                            <path>/var/tmp/UltimateFlick.[001-026].dpx</path>
                        </span>
                    </spans>
                </feed>
            </feeds>
        </track>
    </tracks>
    <versions currentVersion="0">
        <version uid="0">
```

```
            <creationDate>2010/12/14 11:54:06</creationDate>
        </version>
    </versions>
</clip>
```

NOTE If you compare this to the .clip structure described before, you will notice the use of user-defined elements such as `<userData>`. Both are optional. For example, Flame uses the following child of `<clip>`:

```
<userData type="dict">
    <appName type="string">Flame</appName>
    <appVersion type="string">2012</appVersion>
</userData>
```

This is used by Flame to determine the software and version used to create the .clip. But the Gateway ignores all `<userData>`: it is up to the client application to determine what to do with it.

Comments

```
<clip type="clip" version=3>
```

The `<clip>` element has two required attributes, with fixed values: `type="clip"` and `version=3`.

```
<name type="string">NoVersionNoSetup</name>
```

`<name>` is optional, but the client application uses it as the name of the clip. Note here that `type="string"` is mandatory so that the Gateway knows how to pass it to the client application.

```
<track uid="1b22da75-33e5-4dbe-80d0-7ed680a8b2b7">
```

Since this example has only one video track (or layer), there is only one `<track>` element.

The uid used here is a UUID, but it can be something as simple as "1", as long as it is unique accross `<track>` elements.

```
<trackType>video</trackType>
```

`video` is one of two allowed values in the `<trackType>` context, the other being `audio`. You must have a defined `<trackType>` for each `<track>`.

```
<duration type="time">
    <rate type="rate">
        <numerator>30000</numerator>
        <denominator>1001</denominator>
    </rate>
    <nbTicks>26</nbTicks>
    <dropMode>NDF</dropMode>
</duration>
```

`<duration>` defines the duration of the track, using the formula `<rate>`/`<nbTicks>`.

The `<rate>` element used here is the best definition of the 29.97 fps frame rate. This avoids rounding errors. Of course, if the frame rate is an integer such as 24 fps, use the simplified structure `<rate>24</rate>`.

And since 29.97 fps can be drop frame or non-drop frame, we have to define `<dropMode>`.

```
<name type="string">ClipNoVersionNoSetup</name>
```

Again, `<name>` is optional here, but Flame uses it as the name of the track.

```
<feeds currentVersion="0">
```

`currentVersion` refers to the `vuid` of the enclosed `<feed>`.

```
<feed vuid="0" uid="track1Feed1">
```

A feed forms a version of a track, identified with by its `vuid`.

As shown here (track1Feed1), a `<feed uid=>` can be any sort of value, as long as it is unique across all the `<feed>` elements.

```
<spans>
```

The `<spans>` element exists because you can have multiple spans. Such is the case with media cartridges that split clips into 2GB files.

```
<span>
    <duration>26</duration>
    <path>/var/tmp/UltimateFlick.[001-026].dpx</path>
</span>
```

Because we have 26 dpx files, and because we want to use them all, `<duration>` is set at 26. To use less of it, we can specify a lower value.

When linking to streaming media, `<duration>` becomes the number of samples to use.

```
<versions currentVersion="0">
    <version uid="0">
        <creationDate>2010/12/14 11:54:06</creationDate>
    </version>
    </versions>
</clip>
```

Every .clip has at least one version, defined using a `<version>` element. `currentVersion` refers to an existing `uid` from a `<version>`. The `uid` must be an existing `vuid`.

`<creationDate>` is not parsed by the Gateway, only passed to the client application.

Flame Premium Grading - Lustre 33

Lustre Workflow

It is a good idea to read this chapter thoroughly to understand how the workflow of a typical colour grading project corresponds to working in Lustre. The first section in this chapter describes the image processing order—the order in which modifications are applied to a shot. Each subsequent section summarizes a significant stage in a colour grading project and provides cross-references to sections containing procedural information.

This workflow is presented as a linear process. However, due to the flexibility and unpredictability of the creative process, it is unlikely that you will go through the stages in the order presented—often different stages are worked on simultaneously and some stages are revisited several times.

Image Processing Pipeline

In Lustre, image processing order is pre-determined—the order in which you perform image processing tasks does not change the order in which they are processed. You can perform these tasks on different stations simultaneously and get the same results. For example, if you apply an input LUT to a shot from a Lustre Station, it will be processed before the colour grading effects created on the Master Station are processed, regardless of the order in which you apply them.

Colour grading may occur at the beginning, the end, or the beginning and end of the post-production process. In Lustre, the order in which you create image processing effects parallels the different stages in the post-production process. Each task corresponds to a menu.

The image processing is as follows:

1. The application of a CDL from the CDL menu.
2. The application of input LUTs from the Input LUT menu.
3. Initial primary colour grading from the Grading menu (with the Input button enabled).
4. RGB and Hue curves colour grading from the Curves menu.
5. Secondary colour grading from the Secondaries menu.
6. Sparks® plugin effects from the Effects menu.
7. RGB curves colour grading from the Curves menu.

8 Final primary colour grading from the Primary menu (with the Output button enabled).

9 The application of output LUTs from the Render | Output menu.

NOTE Dust removal operations are processed separately. Shots where artefacts have been removed are not updated in other menus until you render.

Setting Up Users and Projects

At the beginning of every project, you specify the location of your footage, set up user profiles, the project, project parameters (such as logarithmic or linear colour space), and the scenes that contain the reel or scene footage that you are going to work on. During each session, you then select a user profile and project so that you can save metadata for your grades. See Project Management (page 1719).

Monitor Calibration

At the beginning of a project, and throughout, you should calibrate the monitors for all stations to ensure the proper display of your footage. Calibrate the monitors according to the colour space (logarithmic or linear) and footage you are using. See Monitor Calibration (page 1846).

Applying Print LUTs

For film print simulation, you can apply Print LUTs, which are updated regularly by your film lab. See Applying LUTs for Viewing (Print LUTs) (page 1853).

Editing

Once you store your media and resources into the appropriate directories, you can edit the footage to the timeline in preparation for colour grading. Procedures for editing footage may include:

- Browsing footage and bringing it to the timeline. See Loading Clips to the Library (page 1874).
- Assembling EDLs in the timeline. See Working with EDL, ALE, and Cutlist Files (page 1892).
- Editing shots in a multi-layer Timeline. See Editing in the Timeline (page 1936).
- Capturing footage from a VTR. See Capturing Material (page 2264).
- Performing basic gestural edits such as trimming, slipping, and sliding shots, as well as creating dissolves. See Editing (page 1912).

Rendering Proxies

For projects that use large files (such as 2K Cineon® files), you should render half-resolution (1K) proxies of all the shots in the timeline before you begin colour grading. Proxies are used to reduce processing time when interaction is slow, such as when you apply several secondaries to a shot. See Generating and Viewing Proxies (page 2227).

Removing Dust

You can remove dust and other artefacts from your shots at any time in the project workflow. However, it is recommended that you remove dust early in the project so that the colourist can work with clean shots and you have time to do more than one dust removal pass on the footage.

Dust removal metadata is saved separately from other colour grading data. This makes it possible to remove dust on Lustre Stations while developing the colour grade on the Master Station. The same applies if using more than one Lustre HD Station. You can then replace dusty shots with clean ones without affecting the colour grade. See Removing Dust (page 1998).

Repositioning Shots

Once your footage is organized in the timeline, you can reposition shots that need the viewable area adjusted. For example, reposition shots that come from a wide-screen film format and are going to video. See Repositioning Images (page 2016).

Applying Input LUTs

With your footage organized and prepared for creative work, you are ready to apply any necessary input LUTs before you begin colour grading. Apply an input LUT to a shot when you need to remap colour values in the entire shot. Procedures for applying input LUTs include:

- Applying an existing input LUT to create a particular look (such as a KODAK™ film stock look). See Applying an Existing Input LUT (page 2029).
- Creating an input LUT to convert logarithmic data to linear data when working with film footage in Linear mode. See Linear Mode: Creating Conversion LUTs (page 2030).

Colour Grading

Lustre is primarily a colour grading application. Project setup, editing, and basic image processing are all procedures that you perform in preparation for creating a convincing colour grade.

Colour grading procedures include:

- Performing primary colour grading to change the overall colour look of each shot used in a series of shots, scene, or entire project. See Primary Colour Grading (page 2058).
- Modifying RGB and Hue curves to colour grade specific ranges of colour without using keys. See Colour Grading Secondaries (page 2088).
- Performing secondary colour grading to specific ranges of colour or areas in an image through keys and geometries. See Colour Grading Secondaries (page 2088).
- Generating keys for secondary colour grading. See Creating a Secondary by Keying a Range of Colours (page 2116).
- Creating and tracking geometries for secondary colour grading. See About Geometries (page 2091)and Animating the Point Tracker (page 2144).

Using Sparks

While colour grading your shots, you can load Autodesk Sparks® plugins to create additional image processing effects such as a blur or a glow. Sparks are applied on a shot-by-shot basis. See Creating Lustre Sparks Effects (page 2156).

Animating Colour Effects

At various times, you may want to animate a colour grading parameter to enhance the colour look, ambience, or to make a static shot more dynamic. You can animate most parameters using the Animation controls. See Animation (page 2171).

Outputting the Result

At the end of a project, you output the final version of your grade. You may also want to output your grade in the middle of a project to preview the colour grade or show it to your client. Outputting procedures may include:

- Applying a Log to Lin conversion LUT. See Setting Colour Space Options (page 2226).
- Rendering a print of the final grade. See Rendering (page 2215).
- Outputting rendered files to a VTR or any other device. See Playing Out to a VTR (page 2280).

Interoperability Workflows in Lustre

Interoperability allows for Creative Finishing applications, on the same or on different workstations, to share clips and sequences (timelines).

To obtain a sequence, the Lustre artist simply browses the remote project's project folders, and drags and drops the Smoke/Flame sequence to the Lustre Storyboard. Grading proceeds as usual. Once done, Lustre renders its results back to the project's defined render destination (i.e Lustre Shared Library or any location on the Media List). Within the context of Source Grading, this process leaves all Timeline FX, transitions and metadata intact.

Conceptually, this is the simplest scenario. However, the interoperability features give you a high level of flexibility, whereas you can send your sequence to and back from any of the Creative Finishing applications at any point of the finishing process.

About Source Grading and Sequence Grading

There are two ways you can use Autodesk Lustre to grade media coming from Autodesk Smoke and Autodesk Flame, depending on the project you're working on and the requirements of you facility:

- You may want to work in parallel of the editorial process and grade the sources as the editorial decisions are being made (Source Grading).
- You may want to perform the colour grading at the end of your project, once the online editing, visual effects and titling are complete (Sequence Grading).

Source Grading Workflow

1. Assemble your sequence in Smoke/Flame.

2 Save your sequence to the Lustre shared Library.

3 Import your sequence into Lustre, with the Source Grading option enabled. Your multilayer sequence is imported into Lustre and the source media is displayed. You can grade your sources in context, taking into account the editorial decisions to date.

4 Grade your sequence sources.

5 Render the graded sequence in Lustre.

6 Open the graded sequence in Smoke/Flame for further enhancements.

Sequence Grading Workflow

1 Assemble your sequence in Smoke/Flame.

2 Render your sequence in Smoke/Flame and save it to the Lustre Shared Library.

3 Import your sequence into Lustre, with the Source Grading option disabled. Your multilayer sequence is imported into Lustre and all effects and unsupported transitions are committed.

4 Grade your sequence.

5 Render the graded sequence in Lustre.

6 Import the rendered sequence into Smoke/Flame or export the final media from Lustre.

Basic Interoperability Workflow

The setup of your interoperability project depends on whether you have Smoke/Flame and Lustre installed on separate workstations or whether you have them installed on the same workstation, as in Flame Premium.

To set up an interoperability project on standalone Smoke/Flame and Lustre workstations:

1 Create your projects in Smoke/Flame and Lustre.

2 Set your Scans and Renders paths in Lustre to correspond with your Smoke/Flame projects.

See:

Project Menu Settings (page 2327)

Setting Up the Lustre Wiretap Project (page 2326)

To set up an interoperability project with Smoke/Flame and Lustre on the same workstation, as in Flame Premium:

1 Create your project in Flame Premium as usual.

Smoke/Flame create a Lustre project and a shared library named Lustre in your project that Lustre renders back to by default.

NOTE Creating a Flame Premium project automatically creates a Lustre project. However, creating a Lustre project does not create a Flame Premium project.

Important Source Grading Considerations

To use the interoperability features between different Creative Finishing applications and workstations, you must be running the same version of Wiretap Server on each workstation. Depending on the Creative

Finishing product and the operating system it is running on, the naming convention of the Wiretap Server changes.

Current Release		
Linux: 2013 Extension 2	Mac OSX: 2013	Windows: 2013 Extension 2
Previous Release		
Linux: 2013 Extension 1	Mac OSX: 2012 Subscription Advantage Pack 2	Windows: 2013 Extension 1

File Path Translation

File path translation ensures that different workstations running different operating systems are able to convert file paths from other operating systems to a file path in their native file structure. This ensures compatibility between Linux, Mac OSX and Windows workstations in a collaborative workflow, as the different workstations send and receive commands to exchange media to and from shared storage.

Shared Storage in a Workgroup Environment

Before performing finishing and grading in a shared project from multiple workstations, using a shared storage (NAS/SAN), you must configure the following files:

- **Init.config:** located in: /usr/autodesk/lusterpremium_2013.1/
- **wiretap_path_translation_db.xml:** located in: /usr/discreet/wiretap/current/cfg/

Init.config

To make sure Lustre is able to read media files from its direct connection to a shared storage, you must edit the PathTranslationTable portion of the init.config file, as follows:

```
<PathTranslationTable>
<PathTranslation src="172.16.129.101@wtg:/Volumes/QASAN01"
dst="localhost@wtg:/QASAN01"/>
```

This example translates a Mac Wiretap Gateway mountpoint to its corresponding Linux Lustre mountpoint, which allows Lustre to read the media from its own storage mount point, rather than from the remote workstation.

wiretap_path_translation_db.xml

It is possible to render media files on a shared storage and write the sequence and clips metadata in the Smoke / Flame Premium Media Library or Shared Folder (Wiretap Render). If the shared volume does not have the same mount point on the two workstations, you must add the corresponding mount points to the wiretap_path_translation_db.xml file, as follows:

Add the corresponding mount points:

```
<map src_os="Linux" src_path="/QASAN01" dst_os="MacOSX" dst_path="/Volumes/QASAN01"
 />
<map src_os="MacOSX" src_path="/Volumes/QASAN01" dst_os="Linux" dst_path="/QASAN01"
 />
```

Save the file and restart Stone+Wire

Performing a Wiretap render writes the files and the generated clips and sequence(s) that will reference the translated path.

NOTE

- Lustre is able to read & write to any folder in a Workspace that is not Read-Only.
- Flame Premium Batch Snapshots cannot be accessed in Lustre.

BatchFX

You must render BatchFX segments and BatchFX on gaps prior to importing the sequence into Lustre.

- Rendered effects are seen as source media by Lustre and can be graded along with the rest of the sources.
- Unrendered BatchFX are displayed as black frames in Lustre.

NOTE Rendered BatchFX segments in Lustre do not display the BatchFX icon. And the BatchFX icon is not displayed on the BatchFX timeline segments in Smoke/Flame after being rendered in Lustre.

Transitions

All transitions are preserved by Lustre.

- Only dissolves (with or without animation) are displayed.
- Axis and Wipe transitions are displayed as dissolves. The original Wipe/Axis transitions are preserved in Lustre and are recreated in Smoke/Flame upon reimport. A special icon is displayed in the timeline in Lustre whenever Axis and Wipe transitions occur.



Axis Transition in Smoke/Flame (Image courtesy of EVN PRODUCTIONS)



Axis Transition displayed as a dissolve in Lustre (Image courtesy of EVN Productions)

Custom Transitions

Custom transitions are rendered media and appear as a clip in Lustre. This clip is maintained in Smoke/Flame when you reload the sequence after it is rendered in Lustre. It is recommended to create your custom transitions after the source media has been graded and replaced in Smoke/Flame.



Custom Transition in Smoke/Flame (Image courtesy of EVN PRODUCTIONS)



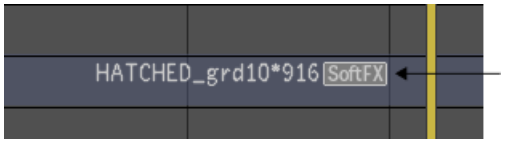
Custom transition in Lustre



Custom transition in Smoke/Flame after render from Lustre

Timeline FX

All Timeline FX are preserved in Lustre. Lustre retains the Timeline FX metadata and displays the source clip rather than the result clip of the Timeline FX. Timeline segments with Timeline FX are displayed in the Lustre sequence with a Timeline FX icon. The Timeline FX are reapplied in Smoke/Flame when you reload the sequence.



Adjustment layers are supported like any other Timeline FX. However, they are not displayed when scrubbing or playing back in Lustre and cannot be graded. The original adjustment layer is restored in Smoke/Flame when you reload the sequence after it has been rendered in Lustre.

- If there is media under the adjustment layer, that media is displayed in the viewport.
- If there is no media under the adjustment layer, a red "X" is displayed in the viewport.

NOTE In the case of a soft timewarp that is not long enough to cover the duration of the segment (for example, a slow-motion effect), Lustre preserves the duration of the segment but displays a red X on missing frames. This is normal because you are grading the sources, which are not being affected by the timewarp animation curve.





Containers and Matte Containers

Containers and matte containers created in Smoke/Flame are preserved in Lustre.

TIP

- To add a matte to the first frame of a source clip in a matte container/multi-channel clip in Lustre, press Ctrl and drag and drop the matte to the container.
- To insert the matte at the first frame of the In point of the timeline segment, press Ctrl+Alt and drag and drop the matte to the container.

See:

About Containers (page 1942)

About Matte Containers (page 1944)

Adding a Matte to a Matte Container (page 1945)

Selecting a Matte to Grade Secondary Layers (page 1946)

Multi-channel Clips

Multi-channel clips created in the Lustre sequence are recreated as matte containers in Smoke/Flame.

See:

About Multi-channel Clips (page 1947)

Saving a Sequence for Grading in Lustre

To save a sequence for grading in Lustre (Source Grading Workflow):

1. Render your BatchFX.
2. Save your sequence to the Lustre shared library.

To save a sequence for grading in Lustre (Sequence Grading Workflow):

1. Render your entire sequence.
2. Save your sequence to the Lustre shared library.

Importing the Sequence from Smoke/Flame

To import a sequence from Smoke/Flame:

1. Open the Editing/Browse menu in Lustre.
2. Determine whether you want to work using the Source Grading or the Sequence Grading workflow. If you are working in the Source Grading workflow, enable Source Grading. If you are working in the Sequence Grading workflow, disable Source Grading.



3. Navigate to the folder that contains the sequence you want to import.
4. Drag your sequence directly from the file browser to the storyboard area.
5. Your sequence and its sources are added to the library. You are now ready to grade.

NOTE

- If you import an unrendered sequence without enabling Source Grading, Lustre looks for the processed Timeline FX segments in the sequence rather than the source clips and displays black frames on unrendered effects. This also applies to relink media. If you relink media to an unrendered sequence without enabling Source Grading, Lustre looks for the processed Timeline FX segments in the sequence rather than the source clips and displays a red X on unrendered effects.
- The behavior in Lustre has not changed when importing with Timelines Only disabled, even if Source Grading is enabled. The one sequence version of the sequence sources will be imported as before.

Displaying and Editing the Sequence in Lustre

It is not recommended to make editorial changes in Lustre to clips with Timeline FX or transitions. Editing clips with Timeline FX or transitions may cause unexpected results when you open the sequence in Smoke/Flame.

To display the Smoke/Flame sequence for editing in Lustre, it is recommended to use the Colour/Timeline menu. Some editorial functions are locked in this menu to prevent any accidental editorial changes.

All editorial changes you make in Lustre are supported in Smoke/Flame.



See:

Timeline Menu (page 1932) for a complete list of supported editing operations in Lustre.

Rendering and Exporting in Lustre

Rendering Back to Smoke/Flame

There are two ways to render your sequence back to Smoke/Flame from Lustre:

- Rendering to the shared Lustre Library
- Rendering to a shared storage location

Rendering to the Lustre Shared Library

When rendering to a Wiretap location from Lustre, you are rendering to a Smoke/Flame standard file system. To render to a Smoke/Flame standard file system, you must set your Renders paths to refer to a Wiretap server path (A destination folder on the Smoke/Flame standard file system. The Lustre shared library is the recommended destination).

NOTE You can create Shared Libraries and folders in the Media Library of the Smoke/Flame projects directly from Lustre and use it as a rendering destination from Lustre.

For example:

oshawa@wt:/stonefs7/Hatched_Project/Lustre

The resulting rendered media is located on your standard file system and is managed by Smoke/Flame.

NOTE By default, Flame Premium configures the Lustre project to render to a Smoke/Flame library.

To render to the Lustre shared folder:

1. Access the Project Settings, in the Setup/Settings menu.
2. Make sure your Renders Home paths are set correctly. For example: oshawa@wt:/stonefs7/Hatched_Project/Lustre.
3. Access the Render/Local menu in Lustre.
4. Enable the Src Grade render mode.



5. (Optional) Enable the Heads & Tails option.
6. Click Start.

Rendering to a Shared Storage Location

If you want Lustre to render back to shared storage, such as a SAN or a NAS, your Renders paths must refer to a storage path outside your standard file system.

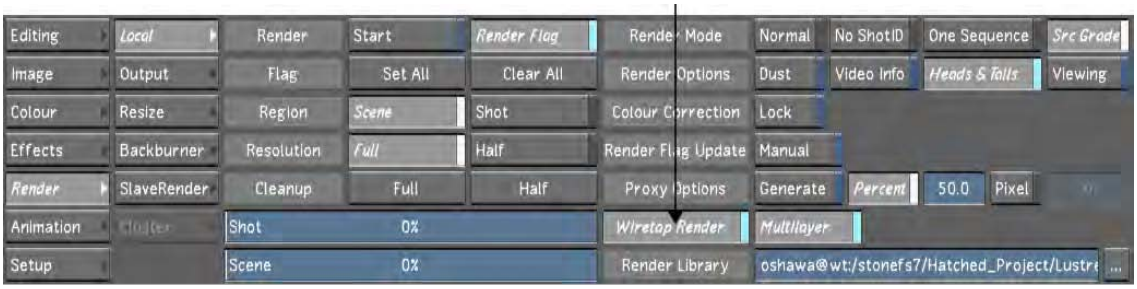
For example:

/mnt/SAN01/Hatched_Project

The resulting media is rendered to a location on your SAN/NAS while the metadata is sent to the Wiretap server. The resulting sequence is seen as soft-imported in Smoke/Flame . This media is not managed by Smoke/Flame.

To render to a shared storage location:

1. Access the Project Settings, in the Setup/Settings menu.
2. Make sure your Renders Home paths are set correctly. For example: /mnt/SAN01/Hatched_Project.
3. Access the Render/Local menu in Lustre.
4. Enable the Src Grade render mode.
5. (Optional) Enable the Heads & Tails option.
6. Define the Wiretap location to which you want Lustre to save the metadata in the Render Library field.
7. Enable the Wiretap Render option.



8 Click Start.

NOTE

- You can define the Wiretap location for the metadata permanently for your project in the Project Settings Network Rendering tab, in the Render Library field.
- Re-rendering the same sequence from Lustre replaces the media on the shared storage. If you make changes to your grades and re-render the sequence, all Smoke/Flame sequences using the same media are updated with the new rendered sources.
- If you are using workstations running different operating systems within your interoperability workflow (Smoke for Mac OSX, Lustre on Microsoft Windows, Linux) and want to render to a shared storage location, you must configure your Wiretap Path Translation file, as each operating system (Mac, Windows, Linux) has a different path to the same location.

See *Installation Guides*

Outputting the Final Version from Lustre

There are two ways to output your final version from Lustre:

- Rendering the final version from Lustre as one sequence in an uncompressed format to a location outside the Standard fs.
- Exporting the final version from Lustre to a compressed format to a location outside of the Standard fs.

Rendering the Final Version from Lustre

Depending on the state of your sequence at the point where you decide to perform your grading in Lustre, or the nature of your project, you may want to render the final version of your project from Lustre.

To do this, you must set your Lustre Renders paths to point to a location outside of the Smoke/Flame Standard fs.

1 Access the Project Settings, in the Setup/Settings menu.
2 Make sure your Renders Home paths are set correctly. For example: mnt/StorageMedia/Renders.
3 Access the Render/Local menu in Lustre.
4 Enable the One Sequence render mode.
5 (Optional) Access the Render/Output menu.
6 (Optional) Disable Same format as scans and select an output format.
7 In the Render/Local menu, Press Start.

Lustre renders your timeline as one sequence at the specified destination.

Exporting the Final Version from Lustre

Depending on the state of your sequence at the point where you decide to perform your grading in Lustre, or the nature of your project, you may want to export the final version of your project from Lustre to a compressed format. This is done through the Render/Export menu.

NOTE Your sequence must be rendered prior to exporting.

1. Render your project in One Sequence mode.
2. In the Render/Export menu, create a new Export version. See Export Versions.
3. Set the Media Origin field to One Sequence Renders.
4. Set the Region field to Scene to export all shots.
5. If the sequence has audio media and that you want to be included in the export, enable the Video & Audio option.
6. Set the Video File Format and the Compression type for your audio and video or select a profile, if available.
7. Define the File Name using tokens or an actual alphanumeric filename.
8. Define the Export Destination.
9. Save your Export Version.
10. Click Export.



Advanced Interoperability Workflows

RED Media Interoperability Workflow

When importing RED footage in Flame Premium, you can transcode the media prior to import. The transcode settings determine the resolution, bit depth, colour settings, etc. of the media created from your R3D files. If you choose to soft-import your media (i.e. linking to the media from its original location), you are able to access the media at any of the resolutions supported by its debayer in Lustre, after importing the timeline. This can be helpful for grading, where a high level of detail can sometimes be critical.

To illustrate this workflow, we are importing an FCP xml file and soft-importing its associated RED sources for conforming in Smoke. We then import the Flame Premium sequence in Lustre, where we can modify the media's transcode settings to perform our grading tasks on the full resolution footage. Once done, we render the sequence back to Smoke for further enhancement.

We are assuming that you have already created your Flame Premium and Lustre projects and that the Lustre library has already been generated.

Importing RED footage to Flame Premium

1. Click the Conform tab.
2. From the contextual menu, select Load new FCP XML / AAF.
3. Navigate to the location of the RED footage (with xml) to soft-import.

You can apply resize settings and apply a LUT on import from the Transcode menu. See

- Transcode Resize Options (page 1886)
- LUT Options Settings (page 1883)

NOTE If you are accessing media that was imported to a Smoke/Flame system with the Cache Source Media option enabled, that media is no longer considered Gateway media. It is seen by Lustre as managed media and hence the LUT and Resize on import options are not available. To work around this, you can perform a Flush Cache Media operation in Smoke / Flame, prior to importing in Lustre.

If you want to change the transcode settings of your media, you can do so from the File I/O menu under the Media Browser prior to importing. If you don't make any changes, the media is imported with the camera settings.



4 Select the FPC XML and click Import.

Smoke finds the sources and reassembles the sequence.

Your sequence is now available in Flame Premium for editing. Once you are satisfied with the sequence and are ready for grading in Lustre, save it to a library and exit Smoke.

See Saving a Sequence for Grading in Lustre (page 1710).

Accessing the Flame Premium sequence for grading in Lustre

Because we are working with soft-imported RED footage, once the Flame Premium sequence is imported into Lustre, you can modify the transcode settings set in Flame Premium and access the footage at up to full resolution for grading.

To access the Flame Premium sequence from Lustre

1 Access the Editing/Browse menu.
2 Navigate to your Flame Premium project.
3 Enable Source Grading.
4 Drag & drop the Flame Premium sequence onto the storyboard.

The sequence and its associated sources are automatically added to the Library.

Modifiying the Flame Premium transcode settings

Access the Image/Transcode menu. You can modify any of the transcode settings that you like; one of which being the debayering settings. Changing the debayering settings will enable you to work at any of the resolutions supported by the debayer: a higher resolution for greater detail or a lower resolution for more responsiveness. The resolution of the media is immediately updated in the viewport and you are ready to grade your sources.



Footage taken from "Other People's Children", courtesy of Golden Films.

Once you are happy with your grades, you have a few options:

Option 1

Render the sequence back to Flame Premium for further enhancement. Lustre creates new (graded) sources and a new sequence (containing all the metadata from the original sequence). The format of the footage is no longer RED media. Instead the sources are in uncompressed RGB, on the Flame Premium Standard fs.

See Rendering and Exporting in Lustre (page 1711).

Option 2

Render the final version from Lustre as one sequence in an uncompressed format to a location outside the Standard fs.

See Rendering and Exporting in Lustre (page 1711).

For more details on rendering media in Lustre, see About Rendering in Lustre (page 2215).

Option 3

Export the final version from Lustre to a compressed format to a location outside of the Standard fs.

NOTE The sequence must be rendered prior to exporting.

See Rendering and Exporting in Lustre (page 1711).

For more details on exporting media from Lustre, see Background Media Export (page 2240).

ProRes Interoperability Workflow

It is possible to import and export ProRes footage to and from Autodesk Smoke, Flame and Lustre applications. To accomplish this, you must have access to a Mac computer running Autodesk Backburner (which is automatically installed if you are running Smoke on the Mac computer).

For information on installing Backburner, see the *Installation Guides*.

This workflow demonstrates how to import ProRes footage from a Final Cut Pro .xml into Smoke and send the conformed sequence to Lustre for grading in Source Grading mode. We then export the graded ProRes sources from Lustre to relink them with the original Smoke sequence, preserving all metadata and soft effects.

Importing the ProRes footage and sequence into Smoke/Flame

To import a ProRes sequence into Smoke/Flame

1. Click the Conform tab.
2. From the contextual menu, select Load new FCP XML / AAF.
3. Navigate to the Mac computer with the ProRes media (with xml) on your local network.
4. Select the FPC XML.
5. Enable Consolidate on Import (recommended).
6. Set your Max Handles (recommended).



7. Click Import.

The sequence is now open in your Flame Premium workspace.

Once you are ready to grade the source files, save your sequence to the Lustre shared library.

Then, launch Lustre, navigate to the project where you saved your sequence, enable Source Grading and drag and drop the sequence to the storyboard. Your sources and the sequence are automatically imported to your Library. You are ready to grade your sources.

Once you are satisfied with your grades, you have a few options:

Option 1

Render the sequence back to Smoke/Flame for further enhancement. Lustre creates new (graded) sources and a new sequence (containing all the metadata from the original sequence). The format of the footage is no longer ProRes media. Instead the sources are in uncompressed RGB, on the Smoke/Flame Standard fs.

See Rendering and Exporting in Lustre (page 1711).

Option 2

Render the final version from Lustre as one sequence in an uncompressed format to a location outside the Standard fs.

See Rendering and Exporting in Lustre (page 1711).

For more details on rendering media in Lustre, see About Rendering in Lustre (page 2215).

Option 3

Export the final version from Lustre to a compressed format to a location outside of the Standard fs.

NOTE The sequence must be rendered prior to exporting.

See Rendering and Exporting in Lustre (page 1711).

For more details on exporting media from Lustre, see Background Media Export (page 2240).

Option 4 (Linux Only)

Export the graded sources from Lustre to the ProRes format and relink the graded sources back to the original Smoke/Flame sequence, for further enhancement.

NOTE The sequence must be rendered prior to exporting.

Exporting the Graded Sources from Lustre to the ProRes Format (Linux Only)

After having rendered the sequence in Lustre, you can set up a Source Grade export.

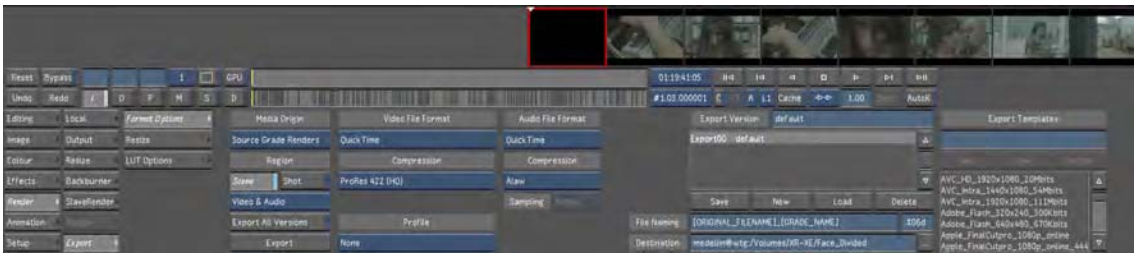
1. Access the Render / Export menu.
2. Set your export destination. This must be a Mac computer running Backburner. The Backburner manager.host file must be edited to have the Flame Premium or Lustre Linux system as the Backburner manager.

   To edit the manager.host file

   1. From a shell, login as root.
   2. Navigate to /usr/discreet/backburner/cfg/manager.host.
   3. Open the file for editing with a text editor.
   4. By default, the backburner manager is set to localhost. Change this to the name or IP address of your Flame Premium or Lustre Linux workstation.
   5. Save the file and exit.

      NOTE If the backburner manager is set to localhost in the manager.host file on your mac, this workflow will not work. You will get error messages when trying to export.

3. In the Render / Export menu, select Source Grade Renders, under Media Origin.
4. Select Video & Audio, under the Region buttons.
5. Select QuickTime under Video File Format.
6. Select one of the ProRes codecs under Compression.
7. Press Export.



Footage taken from "Face Divided", courtesy of Red Truck Productions.

Lustre begins exporting the media.

NOTE If the sequence is not rendered, Lustre automatically starts a render before exporting.

Updating the Sequence with the Graded Sources

Once the media is exported:

1. Launch Smoke/Flame.
2. Click the MediaHub tab.
3. Navigate to the folder containing exported ProRes sources.
4. Drag the folder to the Media Library in the Media panel.
5. Still in the MediaHub, navigate to the folder containing your original sequence and drag it to the Media Library in the Media panel.
6. Click the Timeline tab.
7. Select the original sequence you just copied to the Media Library in the Media panel.

   This displays the sequence with all of its sources unlinked.
8. Click the Conform tab.

   The list of unlinked media is displayed on top of the sequence.
9. Select the folder containg your exported ProRes sources.
10. From the contextual menu, select **Conform ➤ Set As Search Location**.

    Your sources are found. A green check mark appears next to the matched sources.
11. Click Link Matched Sources.

    Your original sequence is assembled with the new graded ProRes sources, with all its original metadata intact.

You can now continue refining your graded sequence. This workflow can be repeated as many times as you wish. When you are happy with your sequence, you can render or export it to the format of your choice.

Project Management

Project Management is definition of project and user settings to customize the colour grading environment.

In Lustre, a project uses the following hierarchy:

- Project: Generally speaking, a Lustre project corresponds to a large work effort such as a film. A project contains all the information regarding the location of:
  - Project data files
  - Original footage
  - Rendered files
- Scene: A scene contains the Libraries, the Scene-based Grade Bin and degrained material within the project.
- Library: The library contains the source clips, cuts, grades, audio and related metadata.
- Cut: A group of shots that get rendered.
- Grade: The operations performed on a cut.

In addition, a project also contains information about the users accessing the project, and their preferences. Lustre saves the preferences of each user of a project at the end of every work session. The saved settings are restored when the user logs back on.

To prepare a project for colour grading:

1. (Optional) Before running Lustre for the first time, configure the *init.config* file. See Configuring System Settings (page 1720).
2. Create and set up the project. See Project Configuration (page 1730).
3. Create and set up the users for the project. See User Configuration (page 1755).
4. Create scenes. See Working With Scenes (page 1769).
5. Organize the footage in cuts. See Managing Cuts (page 1773).
6. Create grades. See Creating a Grade Version for a Cut (page 1777)

Incinerator-Specific Project Management Configuration

Lustre with Incinerator® in-line processing uses the same project management concepts. The only difference is that in order to use Incinerator on the Lustre workstation, the Scans and Render Home paths must point to the Gigabit Ethernet (GigE) Browsed® IP address of the Lustre Media Server. In the Lustre Project menu, you can set the Project Home path to use the nfs mount of the media server. For further information on project management configuration in Incinerator, refer to the *Autodesk Incinerator Installation and User Guide*.

Configuring System Settings

When Lustre is installed, it is pre configured with the default system-level settings. These system-level settings are found in the *init.config* file.

You can define these settings manually if you like or you may want to view the *init.config* file for troubleshooting purposes.

To configure/view the system settings:

1. Open the *init.config* file in a text editor. The location of the file depends on the workstation's operating system.

Operating System:	Location:
Windows	*C:\Program Files\Autodesk\<Lustre_version>*
Linux	*/usr/autodesk/<Lustre_version>*

2. Edit the *init.config* file as required to define:
   - BrowseD.
   - Wiretap® and Wiretap Gateway server
   - Calibration settings
   - Autodesk Control Panel
   - Slave Renderer
   - Backburner ™
   - Internal keywords
   - DPX keycode emulsion list
3. Save your changes and exit the text editor.

See Lustre Software, Project, and User Configuration Files (page 1721)

Lustre Software, Project, and User Configuration Files

Configuration File Overview

The configuration files define all the necessary settings your software needs to recognize various hardware and software settings on the Master Station and on the dedicated render station.

The following is a list of the Lustre configuration files.

Configuration file	Lustre Standalone Location	Flame Premium Grading Location	Description
init.config	*/usr/autodesk/lustre <version>/*	*/usr/autodesk/lustre <version>/*	This file stores system settings as well as Wiretap servers, Slave Renderer settings, and film stock information.
login.config	*/usr/autodesk/lustre <version>/*	*/usr/autodesk/lustre <version>/*	This file records the last user and last project used so at the next session, the user and project login defaults will reflect these values.
project.config	*/usr/autodesk/projects/<project>/project.config*	*/usr/discreet/projects/<project>/project.config*	This file stores project-level information including project settings, calibration, rendering, engineering, and Backburner and Wiretap settings specific to the project.
user.config	*/usr/autodesk/users/<user>/user.config*	*/usr/discreet/users/grading/<user>/user.config*	This file stores user settings specific to a particular user, such as Autosave, printer light, GUI background/gain/gamma.
<user>Context.config	*/usr/autodesk/projects/<project>/<user>Context.config*	*/usr/discreet/projects/<project>/<user>Context.config*	This file stores several settings relevant to the context of a particular user working on a particular project. The purpose of storing context-specific information is that there are some parameter settings that, by their nature, are more likely to be needed in the next session, but that are not configurable in the Project Management pages.

System Settings

You can configure the system settings manually in the *init.config* file. If these system settings are not configured, Lustre uses the default system settings. The following table describes the relevant system settings that can be configured.

NOTE All keyword values are case-sensitive.

Parameter	Data type	Default	Function
<Locations>			
<MainProjectHome>	String		Location of the Project configuration settings folder.
<MainUserHome>	String		Location of the User configuration settings folder.
<GlobalGradeBinHome>	String		Defines the location of the global Grade bin.
<LutHome>	String	[LUSTRE_HOME]/lut	Defines the location of the LUTs. **NOTE** If you a re configuring a Slave Renderer system, and [LUSTRE_HOME] is located on a mounted network drive, make sure you use the full UNC path of the network folder instead of the mapped drive letter.
<PresetsHome>	String	[LUSTRE_HOME]/presets	Defines the location of the presets. **NOTE** If you a re configuring a Slave Renderer system, and [LUSTRE_HOME] is located on a mounted network drive, make sure you use the full UNC path of the network folder instead of the mapped drive letter.
<PluginsHome>	String	[LUSTRE_HOME]/plugins	Defines the location of the plug-ins. **NOTE** If you a re configuring a Slave Renderer system, and [LUSTRE_HOME] is located on a mounted network drive, make sure you use the full UNC path of the network folder instead of the mapped drive letter.

Parameter	Data type	Default	Function
<TemplatesHome>	String	[LUSTRE_HOME]/templates	Defines the location of the templates. **NOTE** If you a re configuring a Slave Renderer system, and [LUSTRE_HOME] is located on a mounted network drive, make sure you use the full UNC path of the network folder instead of the mapped drive letter.
<Help>			
<HelpWebBased state>	String	On	Defines the location of the Lustre Help that is opened by the Help button and shortcut. Lustre Help is web-based by default.
<HelpLocalPath>	String		Defines the default location for Lustre Help when it is installed locally.
<Browser>			
<HideFolder> (within <MediaBrowseHideFoldersGroup>)	String	degrain_cache,original, matte,marry_grade, and 128x96	Hides the degrain cache, original, matte, marry_grade, and 128x96 folders from the file browser. **NOTE** You can hide any additional folder by adding the folder name to the list.
<Wiretap>			
<ServerAuto>	State	On	Specifies whether Lustre automatically scans for Wiretap servers.
<WiretapServer> (within <WiretapServer Group>)	String	0.0.0.0	The IP address or DNS host name for a specific Wiretap server. If one or more <WiretapServer> keywords are set to valid Wiretap server addresses, Lustre lists the content of these servers in the browser before the results of the auto scan. If <ServerAuto> is set to OFF, only the specified Wiretap servers are listed in the browser.
<WiretapGatewayServer> (within <WiretapGatewayServer Group>)	String	0.0.0.0	The IP address or DNS host name for a specific Wiretap Gateway server.

Parameter	Data type	Default	Function
<PathTranslation> (within <PathTranslationTable>)	String	0.0.0.0	Allows you to map hostnames to IP addresses, as well as to define path translation rules.
<NTSCImageAspectRatio>	Float	1.3333333333333	Applies the correct aspect ration to NTSC resolution footage rendered to the Wiretap server, but not imported from the Wiretap server.
<PALImageAsepectRatio>	Float	1.3333333333333	Applies the correct aspect ratio to PAL resolution footage rendered to the Wiretap server, but not imported from the Wiretap server.
<MonitoringAndCalibration>			
<DisplayType>	Enumerated (abstract data)	LCD	Specifies the monitor display type (e.g., LCD or CRT).
<Calibration_Steps>	Integer	10	Specifies the number of monitor calibration steps performed.
<ControlSurface>			
<AutodeskPanels>	State	On	When ON, this keyword enables the Autodesk Control Surface (ACS). When OFF, Lustre enables the Tangent CP100 control surface (if applicable). You must also set the path for the control surface rules file in the Panel Setup File field.
<PanelIDs>		function="0" grading="0" navigation="0"	The ID numbers for the Function, Navigation, and Grading panels of the ACS. You must manually configure these keywords. If your control surface is a Tangent CP100, Lustre does not use these keywords.
<PanelIPTags>		function="101" grading="100" navigation="102"	
<Browsed>			

Parameter	Data type	Default	Function
<Port>	Integer	1055	All computers on the *BrowseD* network must use the same port to communicate. **NOTE** For Linux over IP, set the port value to 1044.
<Username>	String	root	Administrative user on the *BrowseD* server.
<Password>	String	xxx	Password for the administrative user. **NOTE** To encrypt the password, set the attribute `toEncrypt` to *yes*. The next time the application starts, the password string is encrypted in the configuration file.
<UseInfi>	State	Off	Switch to ON if the networking protocol to use with *BrowseD* is the InfiniBand.
<ReadCacheBuffer Num>	Integer	0	
<ReadCacheThread Num>	Integer	0	
<BrowsedServer> (within <BrowsedServer Group>)	String	0.0.0.0	Defines the IP address or DNS host name for a specific *BrowseD* server.
<SlaveRenderer>			
<HostName>	String		The IP address or DNS host name for a specific Slave Renderer machine.
<Timeout>	Integer	5000	Sets the timeout duration (in milliseconds) of the automatic Slave Renderer detection.
<Backburner>			
<Hostname>	String	localhost	The IP address or DNS host name for a specific Backburner Manager machine.
<Transcode>			
<Transcode>	String	localhost	The IP address or DNS host name for a specific WT Gateway that will be used to create and access the transcoded media.

Parameter	Data type	Default	Function
<MatchCustomGroup>			
<MatchCustom>	Name	"AliasName" type="s"	Specifies the XML metadata field to be used by the Custom match option selected from the Browse menu. "s" represents string.
<MatchCustom>	Name	"DPXTimeCode" type="tc"	"tc" represents timecode.
<MatchCustom>	Name	"DPXkeycode" type="kk"	"kk" represents keycode.
<MatchCustom>	Name	"DPXTapeNAME" type="s"	"s" represents string.
<MatchCustom>	Name	"EDLReelName" type="s"	"s" represents string.
<MatchCustom>	Name	"DL_EDLClip_ name" type="s"	"s" represents string.
<MatchCustom>	Name	"DL_edlFrameID" type="i"	"i" represents integer.
<MatchCustom>	Name	"DLEDL_startTC" type="tc"	"tc" represents timecode.
<AVIO>			
<PlayoutHighSpeed>	State	Off	When OFF, it refreshes the Player as you are performing a playout. When ON, the Player is no longer refreshed and therefore the performance of the playout improves.
<WTTFirstSDLeadIn Correction>	Integer	1	Sets the delay for the SD first lead-in.
<WTTSubsequentSD LeadInCorrection>	Integer	2	Delays all the lead-ins following the SD first shot.
<WTTSDLeadIn Increment>	Integer	0	Corrects the delay after the SD third shot.
<WTTFirstHDLeadIn Correction>	Integer	0	Sets the delay for the HD first lead-in.
<WTTSubsequentHD LeadInCorrection>	Integer	0	Delays all the lead-ins following the HD first shot.

Parameter	Data type	Default	Function
<WTTHDLeadIn Increment>	Integer	0	Corrects the delay after the HD third shot.
<VtrTCTD>	Integer	10	The TimeCode Transition Delay specifies the number of milliseconds the application waits before asking for timecode from the VTR. The default value of 10 ms works for most decks. For HDCAM-SR decks use the following values: ■ 14, for 1080/59i/60 or 720/50/59/60 ■ 21 for other timings
<Miscellaneous>			
<DPXHeaderOffsetCheck>	State	On	When this keyword is enabled, Lustre reads the DPX / Cineon® file header of each frame in a sequence. This is useful when using DPX/Cineon files that have a different media offset within the same sequence of files. When disabled, Lustre reads only the first frame header of a sequence. **WARNING** Turning this keyword on can alter playback performance.
<DisablePanScan Frame>	State	On	
<NVidiaSDISync>	State	Off	
<NVidiaPlayoutDelay>	Integer	0	
<BlockSize>	Integer	2048	
<AudioResyncTime>	Float	-1	
<LoadingSetup>	Integer	0	
<EnableKeykode Sending>	Enumerated (abstract data)	Off	
<LoginGUIGain>	Float	1.0	
<LoginGUIGamma>	Float	1.0	
<ClusterPlayDelay>	Integer	0	Available for Incinerator® only.

Parameter	Data type	Default	Function
<ClusterSmooth Playback>	State	Off	Available for Incinerator only.
<ChangeCutOffset>	State	On	Allows the change cut or match grade feature to preserve the keyframe animations based on therecord timecode whenever a shothas been replaced or moved withina timeline. **NOTE** Be sure to enable only the Record button within the Match Option when performing a change cut or match grade.
<AnimCopy_UsingTrimOffset_Off>	State	On	"On" enables the first animation keyframe to load on the first frame of the shot. "Off" enables the first keyframe to be loaded according to the source frame number (and thus load with the offset).
<PreallocEnable>	State	Off	Allows Burn rendering from a Lustre Windows workstation to use CXFS SAN preallocation.
<FirstFrameNumber>	Integer	On	
<StartShotFrameNumber>	Integer	On	
<DefaultDelivrablePanScanFilter>	Enumerated (abstract data)	Fast	Possible values are: ■ **Fast**(Lanczos2 filter) ■ **Quality** (Lanczos2 filter) ■ **Custom** (BSpline filter)
<RenderNaming> See Render Naming Settings (page 1747)			
<NormalMode>	String		Normal render mode file path and token based naming convention.
<NoShotIdMode>	String		No Shot Id render mode file path and token based naming convention.
<OneSeqMode>	String		One Sequence render mode file path and token based naming convention.
<SrcGradeMode>	String		Source Grade render mode file path and token based naming convention.

Parameter	Data type	Default	Function
<StereoNormalMode>	String		Stereoscopic Normal render mode file path and token based naming convention.
<StereoNoShotIdMode>	String		Stereoscopic No Shot Id render mode file path and token based naming convention.
<StereoOneSeqMode>	String		Stereoscopic One Sequence render mode file path and token based naming convention.
<StereoSrcGradeMode>	String		Stereoscopic Source Grade render mode file path and token based naming convention.
<NormalPadding>	String		Defines the padding of the file name (the number of digits used for file numbering) for the Normal render mode.
<NoShotIdPadding>	String		Defines the padding of the file name (the number of digits used for file numbering) for the No Shot Id render mode.
<OneSeqPadding>	String		Defines the padding of the file name (the number of digits used for file numbering) for the One Sequence render mode.
<SrcGradePadding>	String		Defines the padding of the file name (the number of digits used for file numbering) for the Source Grade render mode.
<StereoNormalPadding>	String		Defines the padding of the file name (the number of digits used for file numbering) for the Streroscopic Normal render mode.
<StereoNoShotIdPadding>	String		Defines the padding of the file name (the number of digits used for file numbering) for the Stereoscopic No Shot Id render mode.
<StereoOneSeqPadding>	String		Defines the padding of the file name (the number of digits used for file numbering) for the Stereoscopic One Sequence render mode.

Parameter	Data type	Default	Function
<StereoSrcGradePadding>	String		Defines the padding of the file name (the number of digits used for file numbering) for the Stereoscopic Source Grade render mode.
<Debug>			
<DumpDLEDLPath>	String		Path to place the contents of a DLEDL when it is loaded in the timeline or Shot bin.
<DumpWiretapCreate Clip>	State	Off	When ON, the attributes of a Wiretap clip are placed into a log file each time a clip is created on a Wiretap server. The file *WiretapClipDump.log* is located in the Lustre home folder. Use this keyword for debugging only.
<FilmTypeGroup>			
<FilmType>	Name		Various film types for AGFA, Kodak™, Eastman, and Fuji emulsions. It defines the relationship between the DPX film emulsion code and the DPX film code number.

Project Configuration

Projects are a means of organizing work performed in Lustre, as well as the directories where the original footage and rendered files are stored. A project typically corresponds to an entire creative piece, such as a film or commercial.

All the work you do in Lustre is non-destructive. Operations performed on the footage are saved as metadata, separate from the original footage. A render permanently applies operations to a copy of the original footage.

When creating a project, you specify a main project directory—known as *Project Home*—in which all metadata files for the project are stored. You can store footage and render files in sub-directories of the Project Home or in another location. If you store footage and renders outside the Project Home, you must specify the location when you define the project.

NOTE Before creating a project, you must have a good idea of how you will structure your data directories. See Recommended Directory Structure for Projects (page 1734).

You can configure the following default settings for your project:

- Project
- Calibration
- Rendering
- Render Naming

NOTE Render Naming Settings are exclusive to Lustre 2012 Extension 1.

- Engineering
- Network Rendering

Select:	To configure:
Project	The name and location of the project. See Project Settings (page 1738).
Calibration	Default system calibration settings and define the default LUTs for a project. See Calibration Settings (page 1741).
Rendering	Default rendering settings for a project. See Rendering Settings (page 1743).
Render Naming	Default render naming settings for a project. See Render Naming Settings (page 1747).
Engineering	Default video and graphics settings for a project. See Engineering Settings (page 1749).
Network Rendering	Default Backburner/Burn®, Slave Renderer, and Wiretap directory paths and settings for a project. See Network Rendering Settings (page 1754).

About Project Defaults and Session Settings

When configuring your project settings in Lustre, you can configure some parameters as project defaults and others as user defaults. The parameters that are configurable as project defaults become the default settings each time a project is loaded, regardless of the logged in user. However, if you make changes in the Lustre application that affect the default settings and then save these changes to a grade file, the default settings will be overridden the next time you load the project.

While logged in, the user can override some of the project default settings, such as Render Mode in the Local menu. However, these overrides only last for the duration of the session. Restarting the application resets the settings to the values set at project creation.

Creating a Project

Creating a Lustre project is the first step to prepare Lustre for a new colour grading project. You can use templates to accelerate project setup. See Working with Templates (page 1793).

NOTE You can create multiple projects with the same Project Home. This allows you to work on a project with different settings while sharing the same scenes, cuts, and grades.

To create a new project:

1. Do one of the following:
   - From the Lustre splash screen, click New in the Project group.
   - From the Main menu, click Setup, then Settings, and then click New in the Project group.



NOTE If you have a template named *default*, then its settings will be loaded automatically.

2 Configure the project. Use Reset to reset all settings, from every menu, to their defaults. See Project Configuration (page 1730).

3 To save the settings of the new project, click Save Project.

4 Click Exit Project.

To create a project based on an existing project:

1 Do one of the following:

- From the Lustre splash screen, select the baseline project, and then click Edit in the Project group.
- From the Main menu, click Setup, and then Settings. Select the baseline project, and then click Edit in the Project group.



2 In the Project Name field, enter a new name for the new project, or you will overwrite the original project's settings with the new settings.

3 Configure the project. Use Reset to reset all settings, from every menu, to their defaults. See Project Configuration (page 1730).

4 To save the settings for the project, click Save Project.

5 Click Exit Project.

To create a new project using a template:

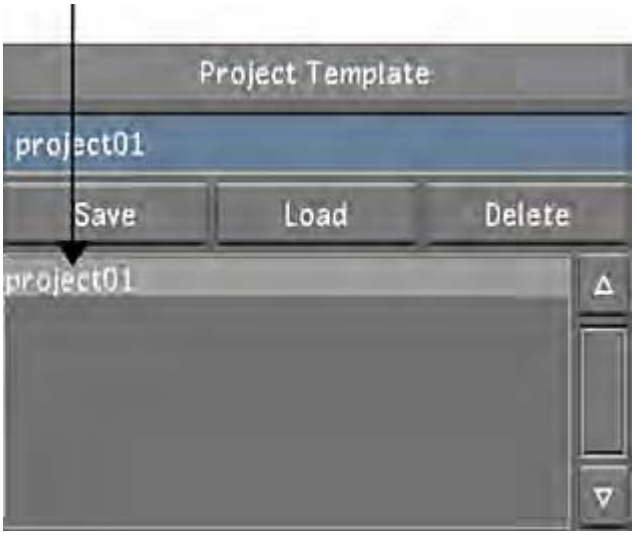
1 Do one of the following:

- From the Lustre splash screen, click New in the Project group.
- From the Main menu, click Setup, then Settings, and then click New in the Project group.

NOTE If you have a template named "default", then its settings will be loaded when you click New.



2 From the Project Template list, select a template. See Creating a Project Template (page 1793).



3 Click Load.
The template's settings are now loaded.

4 Configure the project. Use Reset to reset all settings, from every menu, to their defaults. See Project Configuration (page 1730).

5 To save the settings for the project, click Save Project.

6 Click Exit Project.

Editing an Existing Project

While working on a project, you can edit some of its default settings.

To edit the settings of a project:

1 Do one of the following:
- From the Main menu, click Setup, and then Settings. Select the project to edit, and then click Edit in the Project group.
- From the Lustre splash screen, select the project to edit, and then click Edit in the Project group.



2 Configure the project. Use Reset to reset all settings, from every menu, to their defaults. See Project Configuration (page 1730).

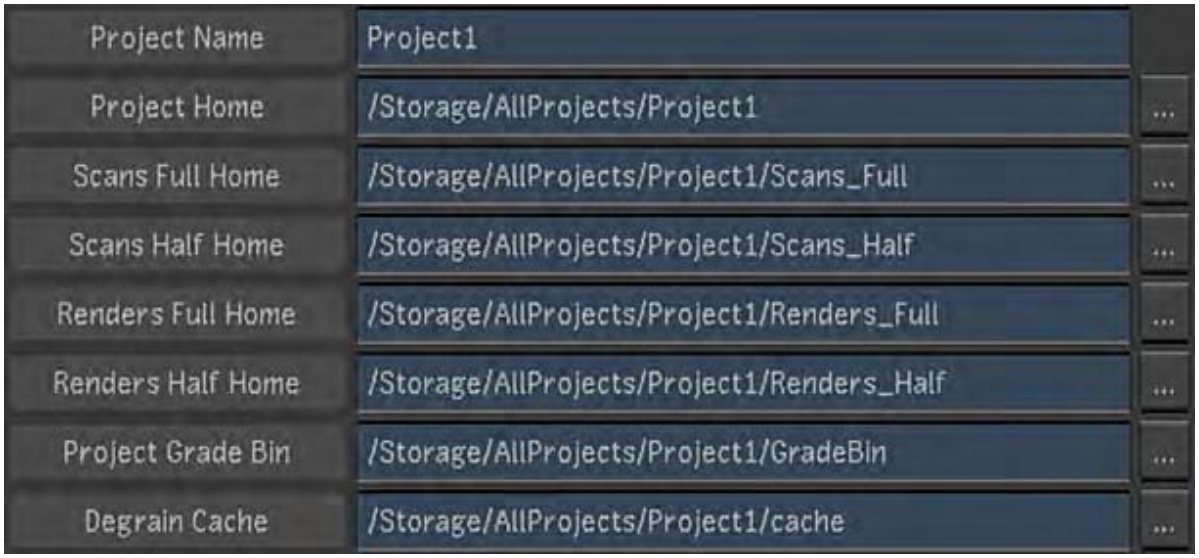
3 To save the settings for the project, click Save Project.

4 Click Exit Project.

Recommended Directory Structure for Projects

The following files can be stored per scene in the Project Home directory, or they can be stored in separate directories, as decided within the project settings. See Project Settings (page 1738).

- Original footage, both full and half resolution
- Render files, both full and half resolution
- Audio
- Grade files
- Grade Bin
- Cached degrained frames
- Library: this new folder contains all the metadata created in the scene.



Example of file directory structure configuration in the Project menu

When directories are not located in Project Home, they can be located on another drive, mount point, or even on a remote SAN or file server. Select the location that best suits your hardware configuration. For example, if you work on one project at a time, you may have enough space to store all footage on the Master

Station. If you are working on multiple projects, you may not have the required space on the Master Station; in this case, you could store your full scans and render files on a large remote server, and store the half-resolution scans locally.

NOTE If configuring Lustre on an Autodesk Incinerator® high-speed network, see Incinerator-Specific Project Management Configuration (page 1720).

Configuring File Locations in Project Home

When project data is stored in Project Home, the following structure is recommended during product configuration. In this example, the Project Home is defined in Windows as the *H:* drive, whereas in the Linux version it is usually defined as the */mnt/md0/* mount point.

Directory for:	Windows Folder:	Linux Directory:
Project Home	*H:\<project name>*	*/mnt/md0/<project_name>*
Scans	*H:\<project name>\<scene name>\<scans folder>*	*/mnt/md0/<project_name>/<scene_name>/ <scans_directory>*
Renders	*H:\<project name>\<scene name>\<renders folder>*	*/mnt/md0/<project_name>/<scene_name>/ <renders_directory>*
Grade bin	*H:\project name\<scene name>\<grade bin folder>*	*/mnt/md0/<project_name>/<scene_name>/ <grade_bin_directory>*
Degrain cache	*H:\project name\<scene name>\<degrain cache folder>*	*/mnt/md0/<project_name>/<scene_name>/<degrain_cache_directory>*

Configuring Project Sub-directories

Once you configure file locations for your project files, you must create the sub-directories for your scans manually. Lustre generates many sub-directories automatically for specific types of file.

When you create a scene in your project, Lustre generates directories for these scenes in the project directory. Alternatively, you can manually create the scene's directory and Lustre will recognize it as an existing scene directory if you later define scenes with the exact names you gave to the directories. See Working With Scenes (page 1769).

Lustre stores all project metadata in sub-directories of the project directory. The metadata is saved per scene, each scene having its sub-directory. The sub-directory is created automatically and is named *Library*.

The render directories related to the scenes are created automatically by Lustre. The directory structure differs depending on the Render Place option specified when rendering. See Specifying the Destination for Local Render Files (page 2220).

In the following table, the Project Home is defined in Windows as the *H:* drive, whereas in the Linux version it is usually defined as the */mnt/md0/* mount point.

Directory for:	Windows Folder:	Linux Directory:
Scene	*H:\<project name>\ <scene name>*	*/mnt/md0/<project_name>/<scene_name>*
Library	*H:\<project name>\ <scene name>\Library*	*/mnt/md0/<project_name>/<scene_name>/ Library*
Scans sub-directories associated with a shot	*H:\<project name\ <scene name>\<scans folder>\<shot name>*	*/mnt/md0/.../<scans_direct-ory>/<shot_name>*
Scans at half- and full-resolution footage, which should both be at this level, in different directories. The directory name must include the scan resolution. Example: *2048x1556*. If you generate proxies in Lustre, the proxies directory is created automatically.	*H:\<project name\ <scene name>\<scans folder>\<shot name>\<resolution>*	*/mnt/md0/.../<scans_direct-ory>/<shot_name>/<resolution>*
Degrain cache	*H:\project name\ <scene name>\<degrain cache folder>\<resolution>\degrain_cache*	*/mnt/md0/<pro-ject_name>/<scene_name>/<de-grain_cache_directory>/<resolution>/de-grain_cache*
Degrain cache (when saved with Scans)	*H:\<project name\ <scene name>\<scans folder>\<shot name>\<resolution>\degrain_cache*	*/mnt/md0/.../<scans_direct-ory>/<shot_name>/<resolution>/de-grain_cache*
Renders	*H:\<project name>\ <scene name>\<render mode>*	*/mnt/md0/<pro-ject_name>/<scene_name>/<render mode>*

Configuring File Locations Separately from Project Home

To have the scanned footage and renders stored on a separate storage location, the Scans Full Home and the Renders Full Home must be explicitly defined as a separate location. On Windows, an example of this separate location could be a shared folder coming from a remote file server, and mapped to the local drive *G:* under the *films\myfilm* folder. On Linux, a corresponding example would be a directory exported from a remote file server, and mounted on the */mnt/fileserver/* NFS mount point under the *films/myfilm* directory.

Directory:	Windows Folder:	Linux Directory:
Project Name	*H:\<project name>*	*/mnt/md0/<project_name>*
Scene	*H:\<project name>\ <scene name>*	*/mnt/md0/<pro-ject_name>/<scene_name>*

Directory:	Windows Folder:	Linux Directory:
Library	*H:\<project name>\ <scene name>\Library*	*/mnt/md0/<project_name>/<scene_name>/ Library*
Where you create sub-directories to store original footage	*G:\films\myfilm\ <scans folder>*	*/mnt/fileserver/ films/myfilm/<scans_directory>*
Where Lustre creates sub-directories for render files	*G:\films\myfilm\grd*	*/mnt/fileserver/ films/myfilm/grd*

NOTE If you decide to store half-resolution scans in a different location from the originals, the project directory structure must be identical in both locations and must include the root directory of the project. For example, if the Project Home is defined as *V:\Data\Lustre_project\My_movie*, the Scans Full Home could be defined as *W:\mnt\San\Lustre_project\My_movie* and the Scans Half Home could be *X:\Lustre_project\My_movie*.

The following table shows the sub-directory structure that CEV_ProdName_CEV creates for renders. The structure differs depending on the Render Place option specified when rendering. See Specifying the Destination for Local Render Files (page 2220). For the sake of simplicity, in this table, *G:\...* represents *G:\films\myfilm*, and */mnt/fileserver/.../* represents */mnt/filesever/films/myfilm* as shown in the first table in this section.

Directory:	Windows Folder:	Linux Directory:
Where sub-directories of the renders are located.	*G:\...\<render mode>*	*/mnt/fileserver/.../<render mode>*

Browsing for Paths

Instead of typing the path, you can browse for paths using the path browser.

To browse for paths:

1. Display the path browser by clicking '...'.



2. Using the path browser, navigate the directory structure to reach the desired directory.
   The selected path is automatically displayed in the path field.
3. To exit the browser, click Enter.

Using Environment Variables

You can use your operating system's environment variables to define all or a portion of a path. In Lustre, if you choose to use environment variables, enclose the variable name inside diamond brackets. For example, the Project Home field might be:

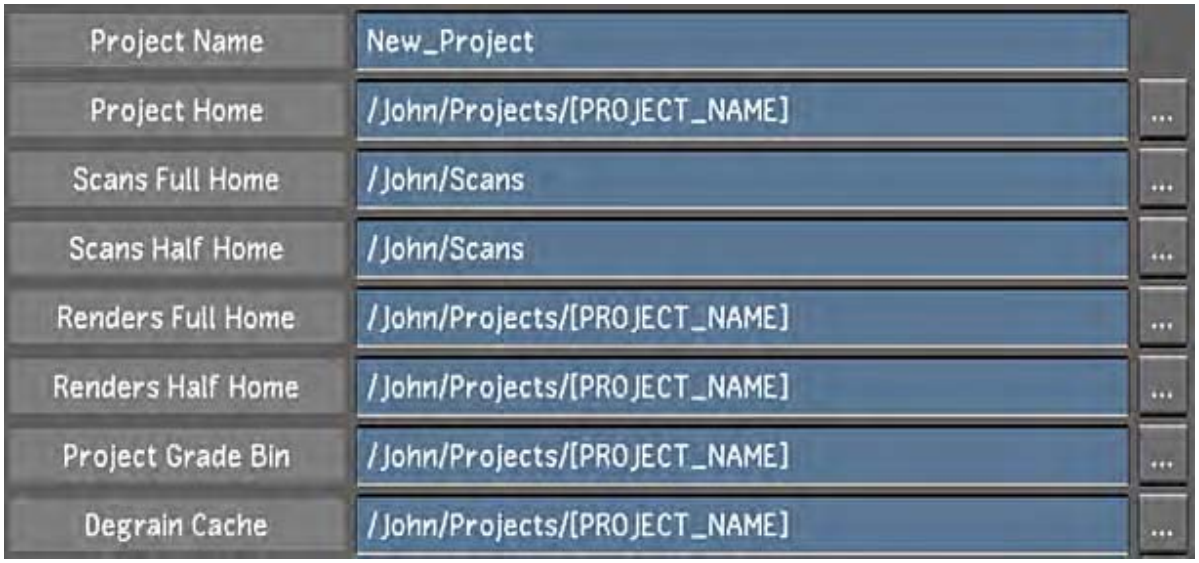
<HOME>\Projects\Zeus

Using the [PROJECT_NAME] and [USER_NAME] Variables

You can also use [PROJECT_NAME] and [USER_NAME] in your paths. You do not need to define these variables as they are predefined. The value of [PROJECT_NAME] is always the value you assign to Project Name in the Project settings page of Project Management. Likewise, the value of [USER_NAME] is always the value you assign to User Name in the Display & Interface settings page of User Management.

NOTE Both the [PROJECT_NAME] and [USER_NAME] variables require opening and closing square brackets.

The following graphic shows the use of both types of variables.



Project Configuration Settings

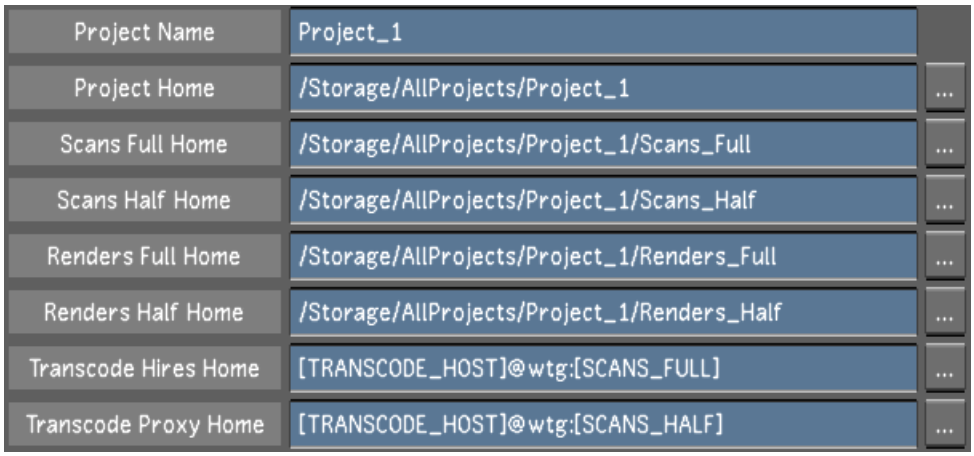
Project Settings

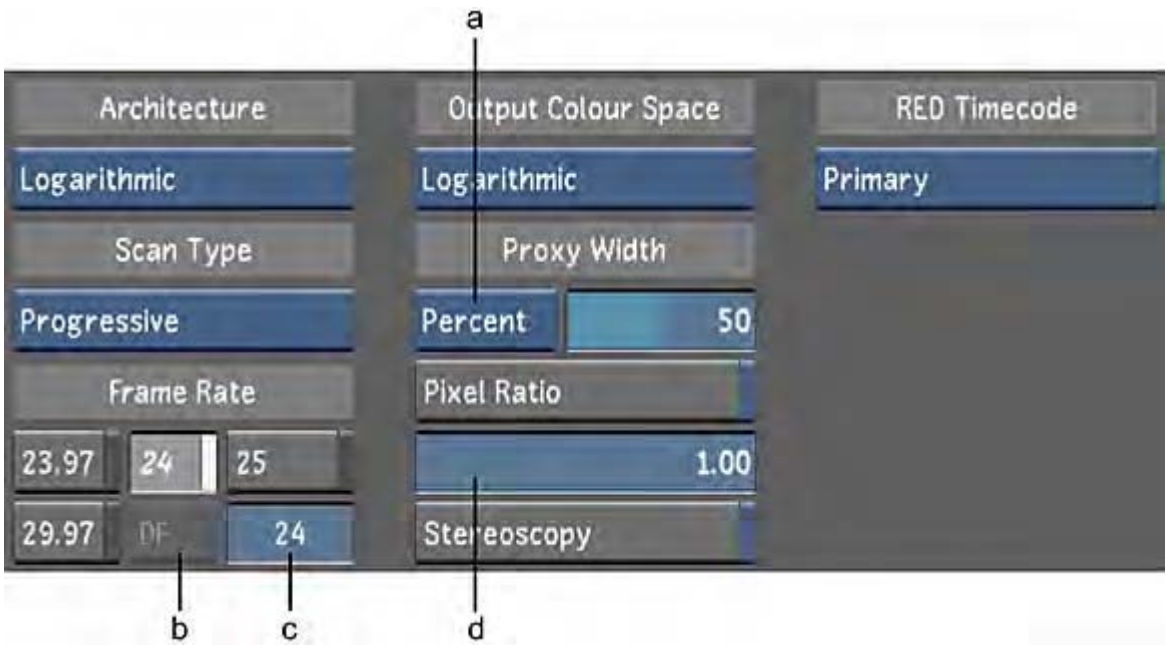
Use the Project settings to define:

- The Project Name
- The Project Home directory
- The Scans Home directory
- The Renders Home directory
- The Transcode Home directory
- The architecture

- The scan type
- The frame rate
- The proxy size
- The pixel ratio
- The Stereoscopy state
- The RED Timecode

For the project's directory structure, follow the guidelines described in Recommended Directory Structure for Projects (page 1734). When entering a directory path, you can use the path browser. See Browsing for Paths (page 1737).





(a) Percent/Pixel option box (b) Drop Frame button (c) Frame Rate field (d) Pixel Ratio slider

Project Name field Enter the name of the project.

Project Home field Enter, or browse to, the location of the main project directory.

Scans Full Home field Enter, or browse to, the location of the scanned footage directory.

Scans Half Home field Enter, or browse to, the location of the proxy footage directory.

Renders Full Home field Enter, or browse to, the location of the render directory for renders of (full resolution) footage.

Renders Half Home field Enter, or browse to, the location of the render directory for renders of proxy footage.

Architecture option box Toggle to set the default grading architecture and colour space while working in Lustre to logarithmic or linear.

Scan Type option box Toggle to set the default scan type.

Frame Rate buttons Enable the default frame rate for the project's timeline and editing tools.

Drop Frame button Enable when your shot contains drop frames. For the majority, this button is greyed-out since most of the default frame rates do not support drop frames. This button becomes available when you select a frame rate that supports drop frames (e.g., you have selected 29.97 or you have entered a personal frame rate within the frame rate field).

Frame Rate field Displays the selected default frame rate or you can enter your own frame rate within the field.

Output Colour Space option box Toggle to set the default colour space architecture when rendering to logarithmic or linear.

Percent/Pixel option box Toggle to set whether Lustre will determine the proxy width as a percentage of the original footage, or as a fixed number of pixels.

Proxy Width slider Use to define the default proxy width. Whether it is a percentage or a number of pixels is defined by the Percent/Pixel option box.

Pixel Ratio button Enable to configure the default aspect ratio.

Pixel Ratio slider Sets the default aspect ratio. Available if Pixel Ratio is enabled.

Stereoscopy button Enable to work on a stereoscopic project.

RED Timecode option box Select which RED media timecode to transcode. Note that when conforming an EDL, overiding the timecode data of the clips with a different timecode track will prevent relinking in the Autodesk Visual Effects and Finishing applications. For more information about RED media and transcoding, see Transcode Options (page 1878).

Select:	To:
Primary	Select either the Time of Day or Edgecode timecode. Only select this option if you know which timecode has been set as the primary.
Time of Day	Establish a timecode field that is set by the internal clock from the RED camera.
Edgecode	Establish a timecode field that is set by the camera operator.

Project Template field Enter the name of a project template you wish to create in this field before clicking Save.

Save button Click to save the project settings of the current project to the template named in the Project Template field. Use to create a new template or to update an existing template.

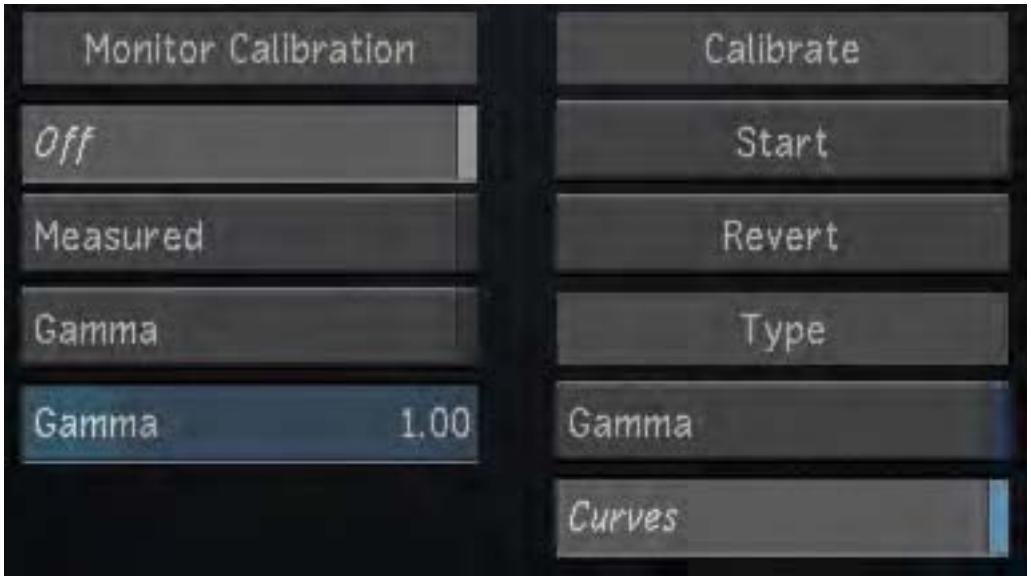
Load button Click to load a project template after you select a template in the Template Name list.

Delete button Click to delete a project template after you select a template in the Template Name list.

Template Name list Select an existing project template from this list before loading or deleting the selected template. See Working with Templates (page 1793).

Calibration Settings

Use the Calibration settings to configure default system calibration settings and to define the default LUTs for a project.



Off button Enable to disable all monitor calibration settings.

Measured button Enable to apply configured monitor calibration settings.

Gamma button Enable to adjust the monitor's gamma level manually (without the use of an external colourimeter).

Gamma slider Use to increase or decrease the default gamma.

Start button Click to begin the calibration. See Monitor Calibration (page 1846).

Revert button Click to revert back to the previous automatic calibration.

Gamma button Enable to set the calibration type to 'Gamma'.

Curves button Enable to set the calibration type to 'Curves'.



Print LUT buttons Enable one of the LUT 1, 2, or 3 buttons, then select a displayed print LUT to perform a quick comparison among three LUTs. For example, to designate a LUT as Print LUT 1, enable LUT 1 and select one of the displayed print LUTs. Repeat this for LUT 2 and 3.

LUT Type menu Select the type of Print LUT: **1D Lut**, **3D Lut**, or **Colour Transform**. See Applying LUTs for Viewing (Print LUTs) (page 1853).

Print LUT list Select a print LUT from each list after you enable one of the Print LUT buttons. You can import a LUT into the list. See Importing a LUT File (page 2027).



Custom button Enable to select a custom input LUT (an input LUT other than the Log-to-Lin LUT). See Input LUTs (page 2026).

NOTE If you enable Custom, make sure you also select a custom LUT in the Input LUT list.

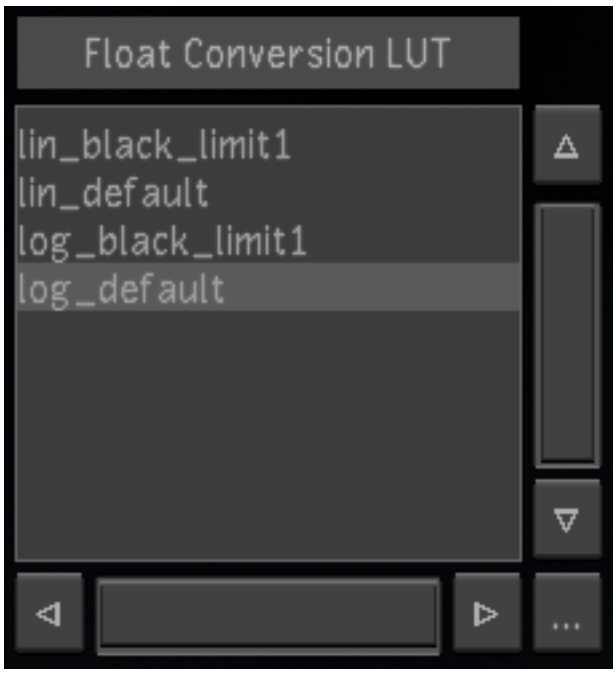
Log>Lin button Enable to configure Lustre to perform a logarithmic-to-linear conversion on shots as they are dragged into the timeline or when assembling an EDL. See Input LUTs (page 2026).

Input LUT list Select a custom input LUT from this list to configure Lustre to use this LUT when:

- Dragging a shot from the Library or Browser to the timeline, or
- Assembling an EDL

You can import a LUT into the list. See Importing a LUT File (page 2027).

NOTE To select a custom input LUT, enable the Custom button.



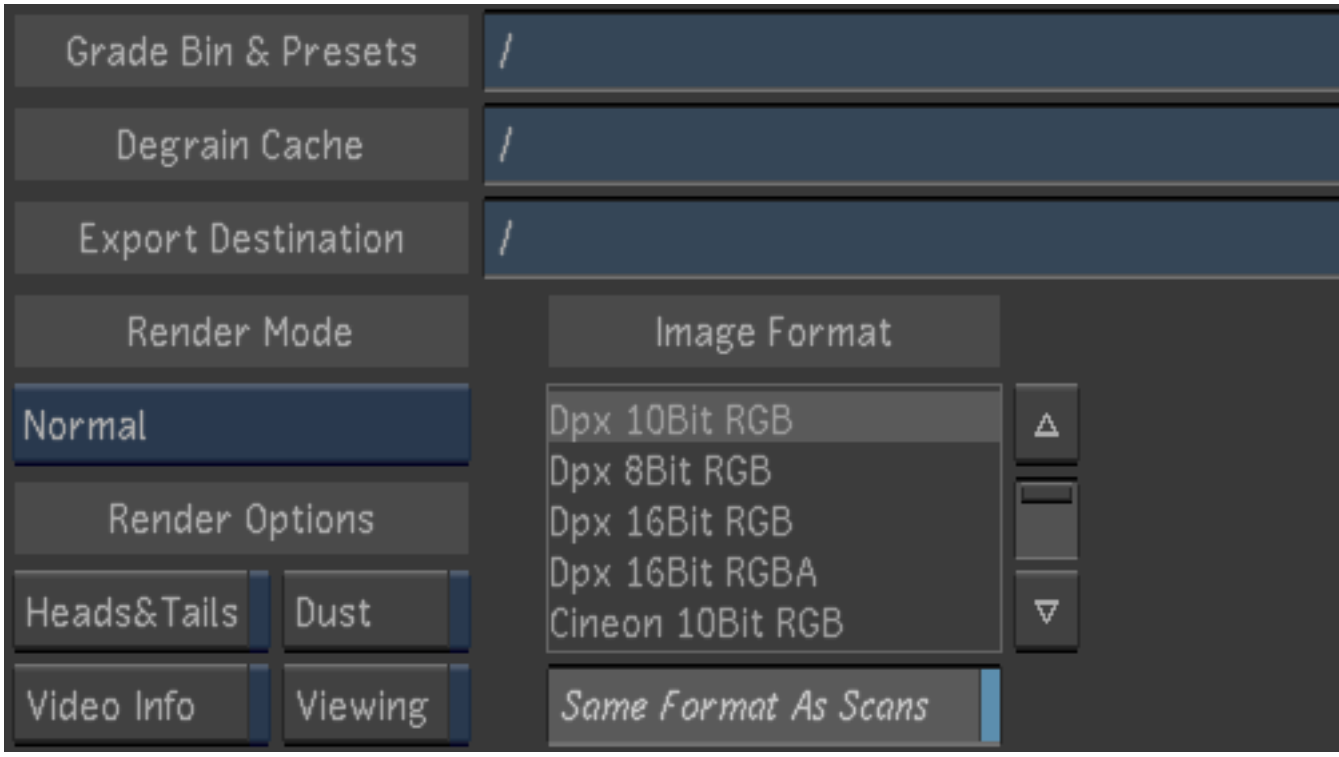
Float Conversion LUT list Select a LUT to convert the OpenEXR 16-bit float data to 16-bit integer Lustre internal format. You can import a LUT into the LUT list. See Importing a LUT File (page 2027).

Rendering Settings

Use the Rendering settings to:

- The Grade Bin & Presets directory.
- The Degrain Cache directory.
- The Export Destination directory.
- The Render Mode.
- The Render Options.
- The Image (Output) format.
- The Marry grade settings.
- The Dust settings.
- The Degrain settings.

- The Proxy Generation filters.
- The Reposition filter.
- The Output Lut.



Project Grade Bin field Enter, or browse to, the location of the grade files for the current project.

Degrain Cache field Enter, or browse to, the location of the user-defined cache location for degrained frames. When there is no access to the Scan directories, and the Degrain File Location option box in the Rendering menu is set to Saved with Scan, cached frames are saved in this location. See Rendering Settings (page 1743).

NOTE Verify that the cache location supports the bandwidth you require for the project.

Export Destination field Enter, or browse to, the location to export to.

Render Mode Set the default render option.

Select:	To have:
Normal	Each shot of a cut rendered to a separate directory, with a Unique ID, under the render path specified in the project.
No Shot ID	Each shot in the cut rendered to a separate directory under the render path specified in the project.
One Sequence	Shots rendered into one directory as a single sequence of renamed images.
Src Grade	Individually-graded shots rendered to their own UID (unique ID) folder in the Renders Home location. This option is mostly used in the Interoperability workflow.

Heads & Tails Enable to set the project's default to render shots with head and tail frames.

Dust Enable to set the project's default to render shots with dust metadata.

Video Info Enable to set the project's default to burn in the timecode into the render.

Viewing Select to render viewing LUT; however, it renders 8-bit BMP files. Use this option when you want to render the image displayed in the monitor.

Image Format list Select the default format of the media to be rendered from this list.

Same Format As Scans Enable to render media in the same format as the source media.

NOTE If you apply a colour transform that converts between integer and floating-point data or vice versa in the Transcode menu, you should disable Same Format As Scans and specify the output format manually.



Marry Grade Format option box Toggle to specify the default marry grade format. For more information about the marry grade formats, see Saving and Loading Marry Grades (page 1784).

Select:	To:
Based on Wedge	Save marry grade files based on the wedge number, defined by the first and last frame in the shot's frame range.
Based on UID	Save marry grade files based on their unique identifiers.

Marry Grade File Location option box Toggle to specify the location where marry grade files are saved. Marry grades are saved in resolution-specific directories.

Select:	To save marry grade files to a resolution-specific directory:
Saved with Scan Full	Under the Scans Full Home directory.
Saved with Scan Half	Under the Scans Half Home directory.

Dust File Location option box Toggle to specify the location where Lustre saves dust files. Dust files are saved in resolution-specific directories.

Select:	To save dust files to a resolution-specific directory:
Saved with Scan Full	Under the Scans Full Home directory.
Saved with Scan Half	Under the Scans Half Home directory.

Degrain File Location option box Toggle to specify the location for caching degrained frames.

Select:	To save cached frames:
Saved with Scan	In the Scans directories.
Saved in Degrain Cache	In the user-defined Degrain Cache directory.

Clear Cache Click to delete all files in the degrain file cache. This button is disabled when the Degrain File Location option box is set to Saved with Scan.

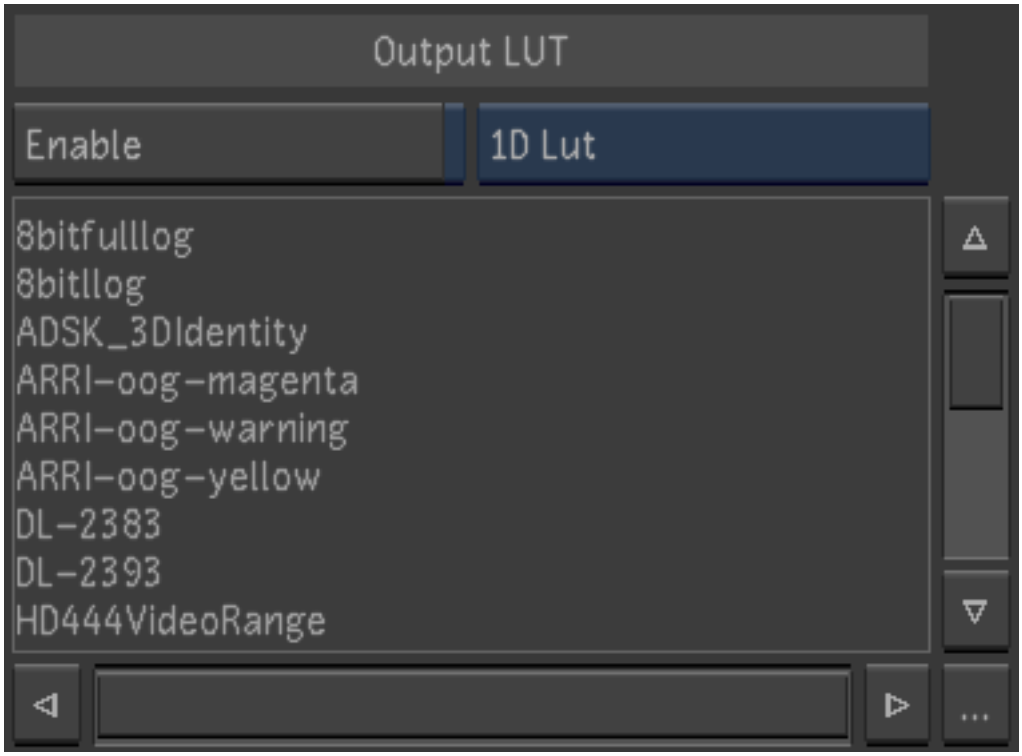
Proxy Filter Select the default filter to use when creating a proxy.

Select:	To use as default:
Normal	An average filter if the resolution of the proxy is half, and to use a Sync filter if the resolution of the proxy is full.
CatmullRom	The CatmullRom proxy generation filter when creating proxies.
Lanczos	The Lanczos proxy generation filter when creating proxies.
Sinc	The Sinc proxy generation filter when creating proxies.
Mitchell	The Mitchell proxy generation filter when creating proxies. Use the sliders below to define the filter settings.

Reposition Filter Select the reposition filter to use by default when creating proxies.

Select:	To use as default:
Normal	An Average filter if the resolution of the proxy is half, and to use a Sync filter if the resolution of the proxy is full.

Select:	To use as default:
CatmullRom	The CatmullRom reposition filter.
Lanczos3lobe	The Lanczos3lobe reposition filter.
Mitchell	The Mitchell reposition filter. Use the sliders below to define the filter settings.



Enable button Apply the output LUT or colour transform when rendering.

Output LUT Type option Choose 1D LUT, 3D LUT, or Colour Transform.

Output LUT list Select a default output LUT or color transform to use when rendering from this list, or click the Browse (...) button at the lower right to select a file.

Render Naming Settings

Use the Render Naming Settings to configure the token based folder structure and naming conventions of rendered media. These settings are included in any project templates you create.

The project is provided with default path structures for each render mode based in the RenderNaming section of the init.config file. You can customize the folder structure and naming conventions of your renders by modifying the fields in the new Render Naming menu in the Project settings.

If you modify a render naming path for a new grade, and you load a previous grade that has a different rendering path for the same render mode, Lustre will be able to link the grade to its previously saved render path. The render naming paths are saved in the file called <grade name>.renderinfo.



Standard Render Button Select to set standard (monoscopic) rendering.

Stereoscopic Render Button Select to set stereoscopic rendering.

Normal Field The location and naming convention for Normal renders.

No ShotID Field The location and naming convention for No ShotID renders (individually rendered shots).

One Seq Field The location and naming convention for One Sequence renders.

Src Grade Field The location and naming convention for Source Grading renders.

Padding Field Defines the padding of the file name (the number of digits used for file numbering) for each render mode. For example: <NormalPadding string=".%06d" /> will generate a six digit file like this: filename.123456.dpx.

NOTE It is possible to create a file with only a frame index (no filename), by removing the "." in the Padding Field. This is the default padding value of the One Sequence render mode.

The following tokens can be used for rendering:

- [PROJECT_NAME]: Name of the current project.
- [PROJECT_HOME]: Home folder of the project.
- [SCANS_HOME]: Location of the scans home according to the current Full or Half resolution view at the time of render.
- [SCANS_FULL_HOME]: Location of the full resolution scans.
- [SCANS_HALF_HOME]: Location of the proxy resolution scans.
- [RENDERS_HOME]: Location of the renders home according to the current Full or Half resolution view at the time of render.
- [RENDERS_FULL_HOME]: Location of the full resolution renders.
- [RENDERS_HALF_HOME]: Location of the proxy resolution renders.
- [SCENE]: Current scene name.
- [USER]: Current user.
- [RESOLUTION]: Output resolution, i.e. 1920x1080.
- [REEL_NAME]: Reel/tape name of the source clip.
- [STEREO_LAYER]: Left or Right.
- [GRADE_NAME]: Grade name (i.e grd01).
- [GRADE_NOTE]: Text data entered by user and displayed between the grade name and cut --> grd01 FinalGrade [cut01], "FinalGrade" is the grade note.
- [CUT]: Name of the cut.
- [UID]: Shot-based Unique ID.
- [ORIGINAL_FILENAME]: The original file name of the source clip (minus the extension), like "12345_clip.0000001.dpx", "12345_clip" is the file name.
- [ORIGINAL_PATH]: The original path and file name of the source clip (including the extension). Use this token to replace the source footage when rendering.

NOTE

When using the [ORIGINAL_PATH] token, the padding field is not available as the original file padding is used.

When rendering to a Wiretap destination, ie. <wiretap_server>@wt:/stonefs…, the clip names are not affected by the Render Naming convention. However, the naming of the Media files will be affected if the Wiretap Render option is enabled (i.e. media files in this case are written to a shared location as unmanaged media).

Avoid using spaces in the names you choose for your folders. Use underscores instead.

You must include the brackets when adding or modifying a token.

Grade-based Render Location

Each grade saves the current render naming path associated with its render mode every time you start a new render, submit a render job to Burn or the Slave render. This means if you modify a render naming path for a new grade, and you load a previous grade that has a different rendering path for the same render mode, Lustre will be able to link the grade to its previously saved render path. The render naming paths are saved in a file called <grade name>.render<render mode>, according to the selected render mode. For example:

- grd01.renderNormal
- grd01.renderNoShotId
- grd01.renderOneSequence
- grd01.renderSourceGrade

Modifiying the Render Naming

If you type an invalid token and press enter (or select another path field), Lustre will highlight the invalid token to warn you about the error.

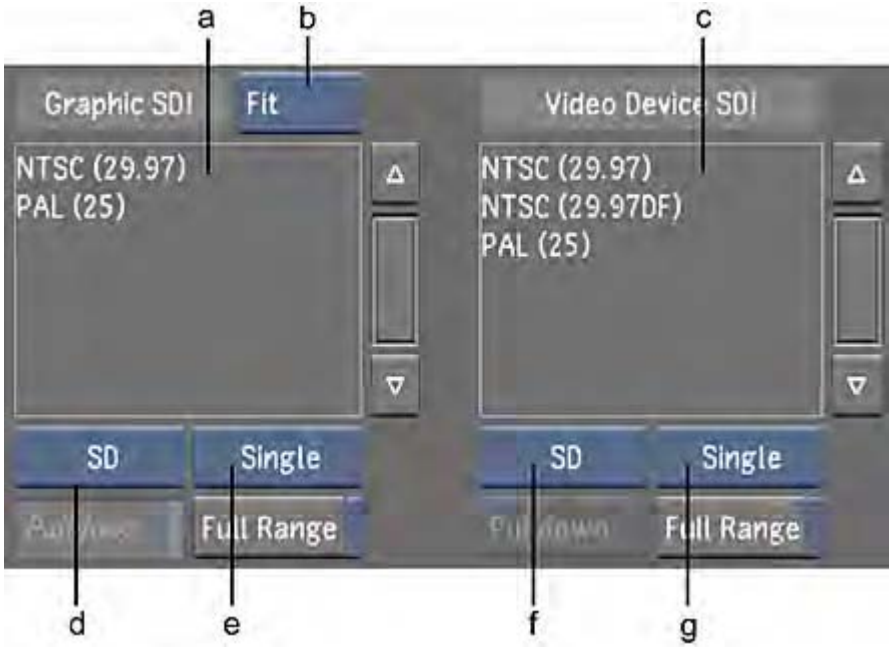


Also, after modifying a render location field, if you want to revert to the default location defined in the init.config, press the CTRL key and click on the field you want to reset.

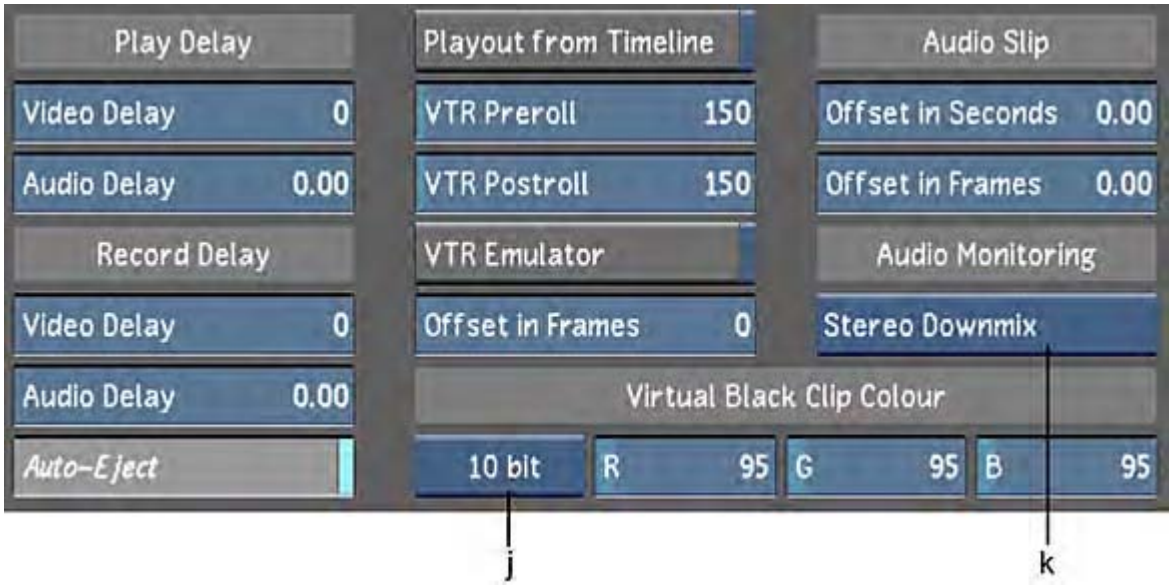
Engineering Settings

Use the Engineering settings to configure:

- Video device SDI settings
- Graphic SDI settings
- Sync mode settings
- Video capture and playout settings
- Audio capture and playback settings
- Virtual black clip colour


a
b
c
Graphic SDI
Fit
Video Device SDI
NTSC (29.97)
PAL (25)
NTSC (29.97)
NTSC (29.97DF)
PAL (25)
SD
Single
SD
Single
Full Range
Full Range
d
e
f
g


Sync Mode
InternalSync
ExternalSync
TrilevelSync
Video Capture Format
Dpx 10Bit RGB
Tiff 8Bit RGB
Audio Capture Format
Horizontal Delay 0
Vertical Delay 0
Wave
48kHz/24bit
h
i



(a) Graphic SDI Raster list (b) Graphic SDI Display option box (c) Video Device SDI Raster list (d) Graphic SDI Format option box (e) Graphic SDI Link Type option box (f) Video Device SDI Format option box (g) Video Device SDI Link Type option box (h) Audio File Type option box (i) Audio Sampling Rate/Bit Depth option box (j) Bit Depth option box (k) Audio Monitoring option box

Graphic SDI Display option box Allows you to select how you would like to display your video on the Graphic SDI output. Cycle through the list to select one of the options.

Select:	To:
Fit	Enlarge the image as much as possible while preserving the aspect ratio at the SDI out (graphics card).
Fit W	Fit the image to the width of the monitor at the SDI out.
Fit H	Fit the image to the height of the monitor at the SDI out.
Stretch	Fit the image to the dimensions of the monitor by adjusting the aspect ratio at the SDI out (graphics card).
Center	Center the image without zooming at the SDI out (graphics card).

Graphic SDI Raster list Select one of the graphics rasters to set the default format for the graphic card's SDI output.

Graphic SDI Format option box Toggle to display, in the Rasters list, rasters using the selected resolution format.

Graphic SDI Link Type option box Toggle to display, in the Rasters list, rasters using the selected link.

Graphic SDI Pulldown button Enable to configure Lustre to add a pulldown frame when outputting through the Graphic SDI. See About 3:2 Pulldown (page 2297).

Graphic SDI Full Range button Enable to configure Lustre to use the full range of YUV and RGB when outputting to the Video device SDI. When this button is disabled, Lustre uses the Normal mapping matrix for the configured video raster.

Video Device SDI Raster list Select one of the video rasters to set the default video resolution for the video device's SDI.

NOTE You must choose a video device SDI raster with the same frame rate as the graphic SDI output raster. Otherwise, the video raster will not be enabled in the application.

Video Device SDI Format option box Toggle to display, in the Rasters list, rasters using the selected footage format.

Video Device SDI Link Type option box Toggle to display, in the Rasters list, rasters using the selected link.

Video Device SDI Pulldown button Enable to configure Lustre to add a pulldown frame when outputting through the video device SDI. See About 3:2 Pulldown (page 2297).

Video Device SDI Full Range button Enable to configure Lustre to use the full range of YUV and RGB when outputting to the Video device SDI. When this button is disabled, Lustre uses the Normal mapping matrix for the configured video raster.

Sync Mode list Select one of the sync mode options from this list.

Select:	To:
InternalSync	Set the sync mode to a free running internal sync (SD and HD).
ExternalSync	Genlock to analog sync connected to the sync input (SD only).
TrilevelSync	Genlock to trilevel sync connected to the sync input (HD only).

Horizontal Delay slider Use to define the number of pixels by which the horizontal sync output is delayed in relation to the incoming sync. The step interval is half a pixel for SDTV devices and two pixels for HDTV devices. The maximum delay is one line length.

Vertical Delay slider Use to define the number of pixels by which the vertical sync output is delayed in relation to the incoming sync. The step interval is +/- 1 field for SDTV devices and +/- 8 line pixels for HDTV devices. The maximum delay is one line length.

Video Capture Format list Select a video capture format to define Lustre's video capture format to a specific file type, colour space, and bit depth.

Select:	To configure Lustre to capture only:
Dpx 10Bit RGB	DPX files with an RGB colour space at a bit depth of 10 bits.
Tiff 8Bit RGB	TIFF files with an RGB colour space at a bit depth of 8 bits.

Audio File Type option box Choose whether to save audio captured from a VTR/ATR, or imported from Wiretap as a Wave (.WAV) or AIFF (.AIF) file.

Audio Sampling Rate/Bit Depth option box Select to capture the audio tracks at a sampling rate of 48 kilohertz (kHz) and a bit depth of 16-bit, or 48 kHz and a bit depth of 24-bit.

Single File Capture button When enabled, all the selected audio tracks are saved in a single file. When disabled, each audio track is saved as an individual file. This option is not available when you are importing audio from Wiretap.

Video Delay (Play) slider Define the default number of video frames to add before the start of the shot when capturing from the VTR.

Audio Delay (Play) slider Define the default number of audio frames to add before the start of the shot when capturing from the VTR.

Video Delay (Record) slider Define the default number of video frames to add before the start of the shot when recording to the VTR.

Audio Delay (Record) slider Define the default number of audio frames to add before the start of the shot when recording to the VTR.

Auto-Eject button Enable to set the project's default to automatically eject the tape during EDL capture.

Playout from Timeline button Enable to set the project's default to play out from the Output view. If this button is disabled, the playout to tape function plays out rendered material only.

VTR Preroll slider Define the time (in seconds) allowed for the VTR to run up to speed and stabilize to perform an edit.

VTR Postroll slider Define the time (in seconds) allowed for the VTR to roll after an edit.

VTR Emulator button Enable to set the project to emulate a VTR.

VTR Emulator Offset in Frames slider Define the default number of frames you wish to offset the Start Timecode from the timeline.

Offset in Seconds slider Define the default number of seconds you wish to offset the audio. See Offsetting the Audio File (page 2312).

Offset in Frames slider Define the default number of frames you wish to offset the audio. See Offsetting the Audio File (page 2312).

Audio Monitoring option box Toggle to select a monitoring option to make sure your playback settings correspond to your audio configuration. See Audio Monitoring Options (page 2311).

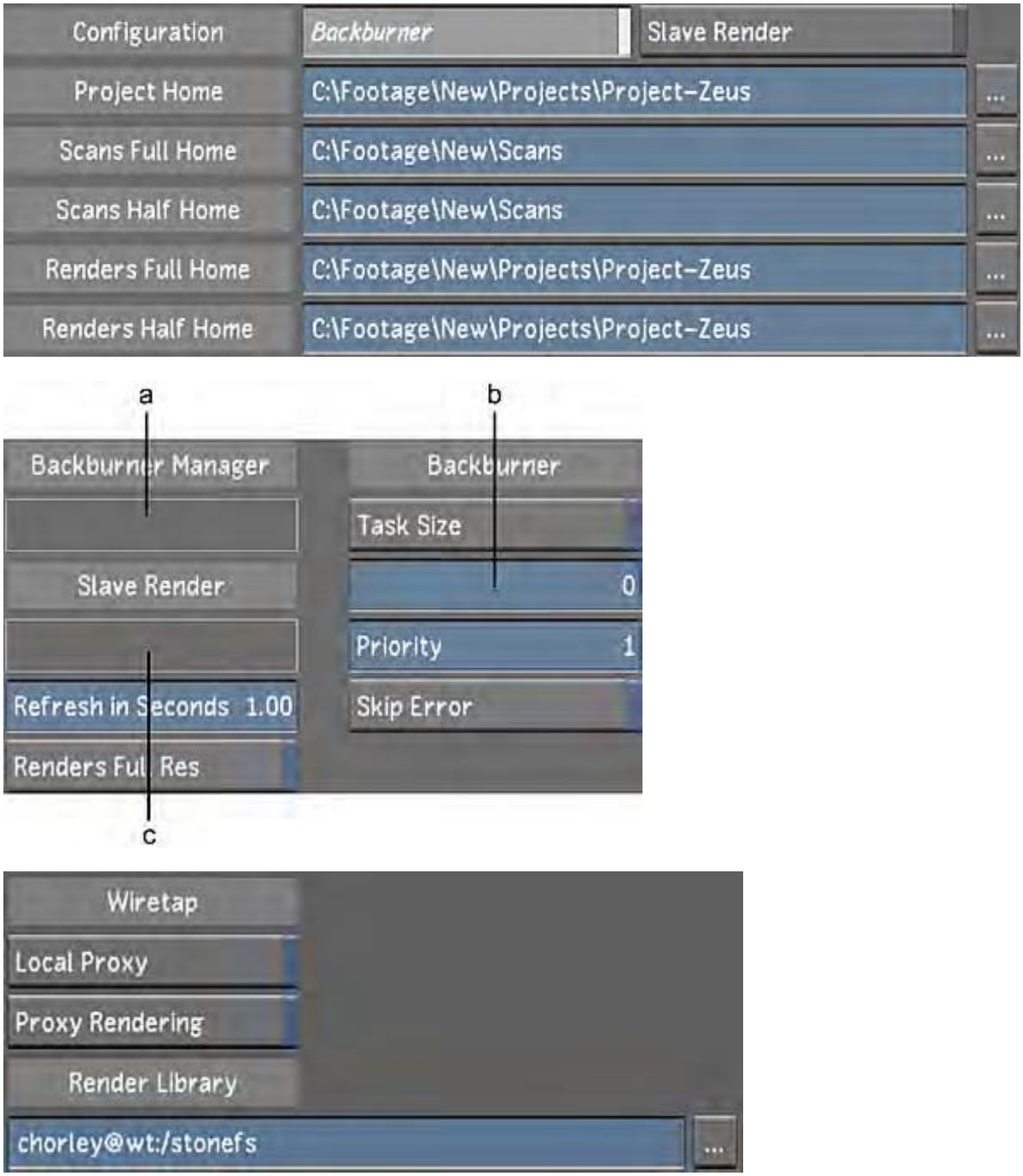
Select:	Audio Device:	When:
Stereo Downmix	DVS or AJA	There are only two audio tracks that are supported on the audio monitoring device. All the odd numbered tracks (e.g., 1, 3, 5, etc.) go through port 1 and all the even numbered tracks go through port 2.
4-Track Downmix	DVS or AJA	There are only four audio tracks that are supported on the audio monitoring device.
8 Tracks	DVS only	There are up to eight tracks and each track is supported on the audio monitoring device.
8-Track Downmix	AJA only	There are only eight tracks that are supported on the audio monitoring device.
16 Tracks	AJA only	There are up to 16 tracks and each track is supported on the audio monitoring device.

Bit Depth option box Toggle to select the default bit depth of virtual colour clips in your project. Possible values are 8, 10 (default), and 16 bit.

RGB sliders Define the default colour of virtual colour clips in this project.

Network Rendering Settings

Use the Network Rendering settings to configure default Backburner/Burn, Slave Renderer, and Wiretap directory paths and settings for a project.



(a) Backburner Manager host field (b) Task Size slider (c) Slave Render host field

Backburner button Enable to display Backburner render path.

Slave Render button Enable to display Slave Render render path. See Network Rendering Settings (page 1754).

Project Home field The location of the main project directory.

Scans Full Home field The location of the high-resolution (full) scans directory.

Scans Half Home field The location of the proxy (half) scans directory.

Renders Full Home field The location of the high-resolution renders directory.

Renders Half Home field The location of the half-resolution renders directory.

Backburner Manager Host field The IP address or host name of the Backburner Manager node. This address is read from the *init.config* file.

Slave Render Host field The IP address or host name of the Slave Renderer host. This address is read from the *init.config* file.

Refresh in Seconds slider Use to define the default number of seconds allowed to elapse before the Slave Renderer progress indicator is updated.

Renders Full Res button Enable to set the default to render high resolution media when working in Proxy mode and using the Slave Renderer.

Task Size button Enable to configure the size of the task (in number of frames) sent to individual Backburner render nodes.

Task Size slider Use to define the number of frames in each task sent to the render nodes.

NOTE You must first enable the Task Size button before configuring the size of the task with the slider.

Priority slider Use to define the default priority of a Backburner task.

Skip Error button Enable to configure Lustre to skip errors when performing remote renders with Backburner.

Local Proxy button Enable to generate and view, in the Half Resolution Player, the proxies Lustre creates for Wiretap media. Disable to see the proxies located on the Wiretap server.

Proxy Rendering button Enable to automatically generate proxies when you render a hires timeline to the Wiretap server according to the project's Proxy Options. This makes it efficient to load clips in a Visual Effects and Finishing application as you no longer have to generate proxies first.

Render Library field Set the Wiretap location on a Visual Effects and Finishing system where Lustre creates soft-import links for the rendered shots. See Browsing for Paths (page 1737).

WARNING For publish metadata to be sent to the Render Library path, the Wiretap Render button must be enabled in the Render > Local menu before you render your project.

User Configuration

In Lustre, using user configuration, you can:

- Create a new user profile.
- Create a new user from an existing user profile.
- Create a new user from an existing user template.
- Edit the settings of an existing user profile.

Your options are:

Select:	To configure:
Display & Interface	The default display and interface settings for this user. See Display & Interface Settings (page 1759).
System & Menu	The system, match grade, assemble, and playout settings for this user. See System & Menu Settings (page 1762).
Tools	Keyframe interpolation, animation, dust, and colour isolation settings for this user. See Tools Settings (page 1765).

About User Defaults and Session Settings

When configuring your project settings in Lustre, you can configure some parameters as project defaults and others as user defaults. The parameters that are configurable as user defaults become the default settings each time that user logs in and loads a project. However, if you make changes in the Lustre application that affect the default settings and then save these changes to a grade file, the default settings will be overridden the next time you load the project.

While logged in, the user can override some of the user default settings. However, these overrides only last for the duration of the session. Restarting the application resets the settings to the values set at user creation.

Setting Context Parameters

When a user logs into a particular project in Lustre for the first time, Lustre creates the *<user>context.config* file, where *<user>* represents the user name associated with a user profile. The context file is found in the *C:\project\<project name>* folder in Windows and the *usr/autodesk/project/<project_name>/* directory in Linux. The context parameters include some attributes not defined in the Project or the User Management pages. They are saved and reloaded each time the context (same user and same project) applies. Contextual settings include:

- Last scene
- Last grade
- Viewing options
- Timeline settings
- Crop value
- Audio file
- State of the surface control
- Stereo sync state

Context parameters are stored in the *<user>Context.config* file. For information, see the *Autodesk Lustre Software Installation Guide*.

NOTE The *Context.config* file can be deleted by holding `Shift` and clicking the User Delete button.

NOTE It is recommended that you verify your user-level and project-level parameters in the User and Project configuration menus, prior to starting a project. See User Configuration (page 1755)and Project Configuration (page 1730).

Creating a User Profile

The user settings menu allows you to customize your settings so they are used whenever you enter the Lustre application. The settings are broken down into three groups: Display & Interface, System & Menu, and Tools.

Use templates to accelerate user creation. See Working with Templates (page 1793).

To create a new user:

1. Do one of the following:
   - From the Lustre splash screen, click New in the User group.
   - From the Main menu, click Setup, then Settings, and then click New in the User group.

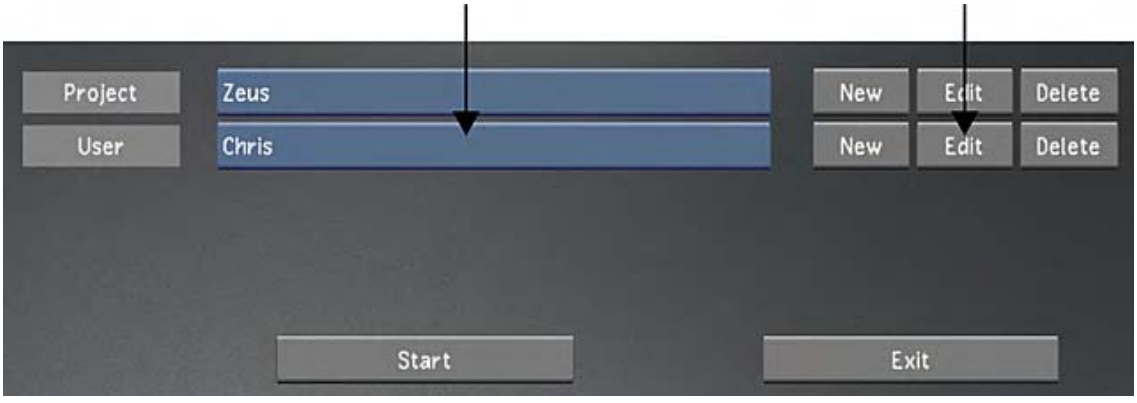


NOTE If you have a template named *default*, then its settings will be loaded automatically.

2. Configure the new user. Use Reset to reset all settings, from every menu, to their defaults. See User Configuration (page 1755).
3. To save the settings for the user, click Save User.
4. Click Exit User.

To create a new user from an existing user:

1. Do one of the following:
   - From the Lustre splash screen, select the baseline user, and then click Edit in the User group.
   - From the Main menu, click Setup, and then Settings. Select the baseline user, and then click Edit in the User group.



2 In the Project Name field, enter a name for the new project, or you will overwrite the original project's settings with the new settings.

3 Configure the new user. Use Reset to reset all settings, from every menu, to their defaults. See User Configuration (page 1755).

4 To save the settings for the user, click Save User.

5 Click Exit User.

To create a new user using a template:

1 Do one of the following:

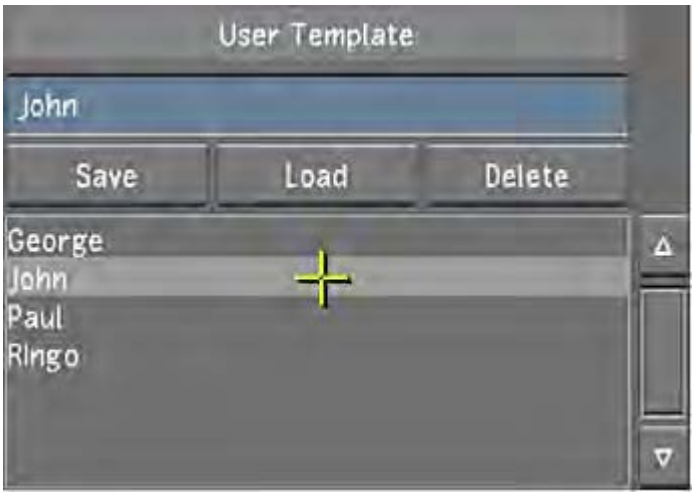
- From the Lustre splash screen, click New in the User group. If there is a template named *default*, it will load automatically.

  NOTE If you have a template named *default*, then its settings will be loaded automatically.

- From the Main menu, click Setup, then Settings, and then click New in the User group.



2 From the User Template list, select the template to use. See Creating a User Template (page 1797).



3 Click Load.

The template settings are now loaded.

4 Configure the new user. Use Reset to reset all settings, from every menu, to their defaults. See User Configuration (page 1755).

5 To save the settings for the user, click Save User.

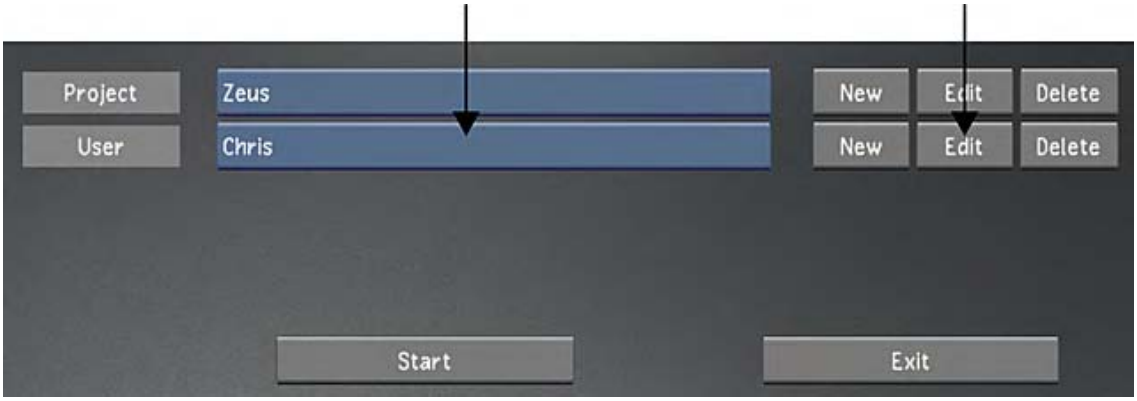
6 Click Exit User.

Editing an Existing User

The editing feature allows you to change the settings of an existing user profile.

To edit user settings:

1 Do one of the following:
- From the Lustre splash screen, select the user, and then click Edit in the User group.
- From the Main menu, click Setup, and then Settings. Select the user, and then click Edit in the User group.



2 Edit the user settings. Use Reset to reset all settings, from every menu, to their defaults. See User Configuration (page 1755).

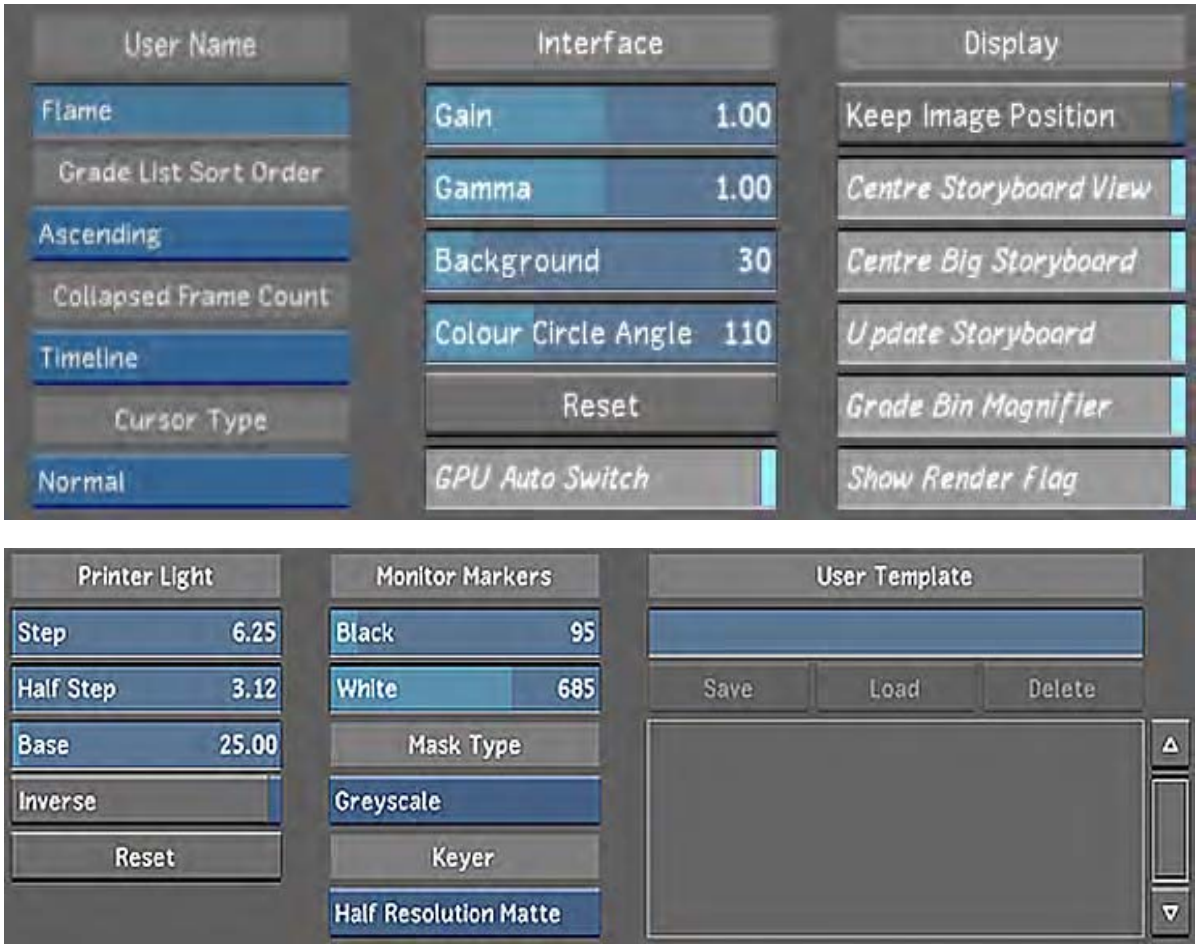
3 To save the edits, click Save User.

4 Click Exit User.

User Configuration Settings

Display & Interface Settings

Use the Display & Interface Settings page to configure the default display and interface settings for this user.



User Name field Enter your user name in this field. This name appears in the list of users after the user profile is saved.

Grade List Sort Order Define the sort order of the Grade List. Options are:

- Ascending: From grade 001 and up (this is the default option).
- Descending: From the last grade to grade 001.

Collapsed Frame Count Define the numbering of the counter used when working in Collapse mode. Options are:

- Timeline: Displays the counter numbering (record timecode / frame count / feet+frame) relative to the current uncollapsed timeline. This is the default option when creating a new user.
- Collapsed: Displays the continuous counter numbering (record timecode / frame count / feet+frame), based on the record timecode of the first selected shot. Frame counters start at frame 1. This is the default option used in previous versions.

Cursor Type option box Use to select cursor type.

Select:	Cursor:
Normal	Displays the default yellow crosshair cursor.
Micro	Displays a yellow dot as cursor.

Select:	Cursor:
Transparent	Displays a transparent cursor.

Gain slider Use to adjust the white point value in the user interface.

Gamma slider Use to increase or decrease the gamma setting in the user interface.

Background slider Use to increase or decrease the background grey settings in the user interface.

Colour Circle Angle slider Use to modify the colour circle angle of the grading colour wheels.

Reset button (Interface group) Use to revert the gain, gamma, background, and colour circle angle back to their default settings.

GPU Auto Switch Automatically disable the GPU processing when using features not supported by the GPU.

Keep Image Position button Enable to maintain the image position in the display when you access the Player. When disabled, the image centres itself in the Player when you hide the user interface.

Centre Storyboard View button Enable to centre the selected shot in the Storyboard.

Centre Big Storyboard button Enable to centre the selected shot when you are viewing the Storyboard in the large Storyboard view.

Update Storyboard button Enable to automatically update the Storyboard thumbnails whenever you make changes to your shot.

Grade Bin Magnifier button Enable to magnify the thumbnail in the Grade bin when the mouse cursor is over a storage container.

Show Render Flag button Enable to display the render flag in the upper-left corner of the screen.

Step slider Use to set the value for one lab light.

Half Step slider Use to change the value for half a lab light.

Base slider Sets the default value for your printer light settings (i.e. red, green, blue (RGB) values).

Inverse button Enable this button in the Printer Light group to invert the printer light density mode.

Reset button (Printer Light group) Click to revert the step, half step, base, and inverse settings to their default value.

Black slider Defines the values of the lines set in the histogram and waveform monitor for black.

White slider Defines the values of the lines set in the histrogram and waveform monitor for white.

Mask Type option box Defines the default display mode for the F11 hotkey and the Matte (M) button in the View mode panel.

Select:	To display mattes:
Greyscale	In Greyscale mode. This Mask Type is referred to as Matte view.
Overlay	In Overlay mode. The matte is displayed in its original colour and the unselected region of the matte is overlayed with a uniform colour. This Mask Type is referred to as Secondary view.

Keyer Matte Resolution option box Select the default Keyer matte resolution. Your options are:

- Full Resolution Matte
- Half Resolution Matte

User Template field Enter the name for the template you are creating. See Working with Templates (page 1793).

Save button Click this button to save the user settings to the template named in the User Template field.

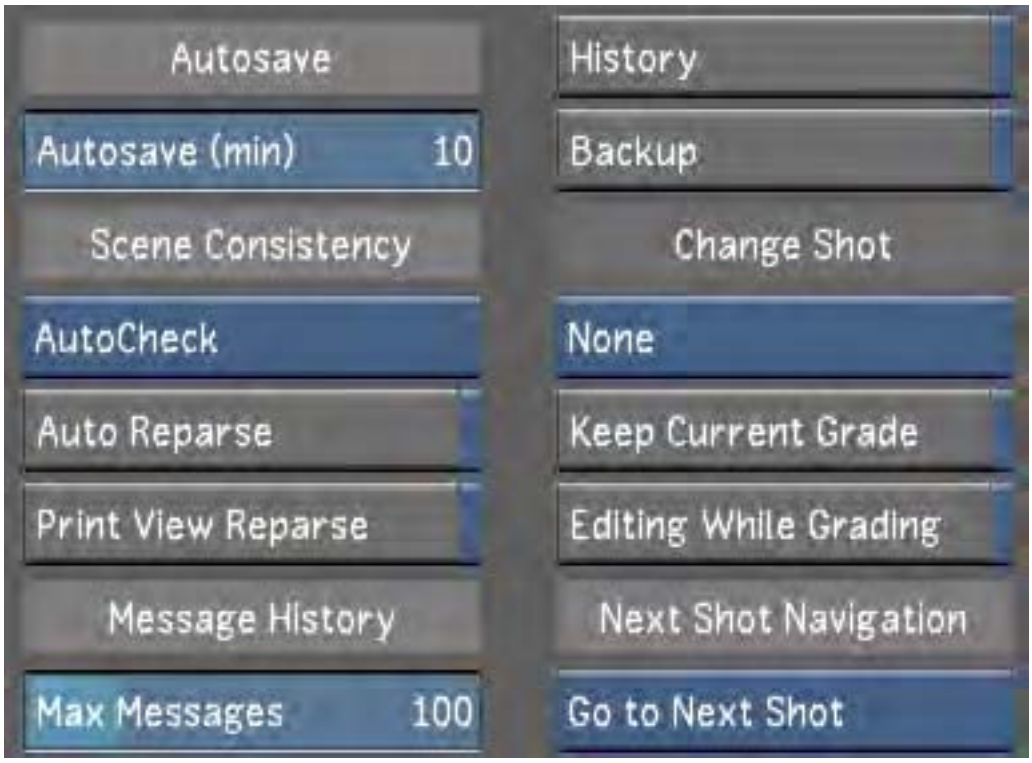
Load button Loads the user template selected in the User Template list.

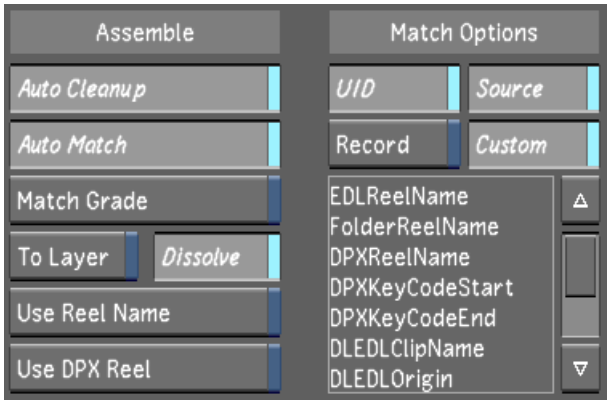
Delete button Deletes the user template you have selected in the User Template list.

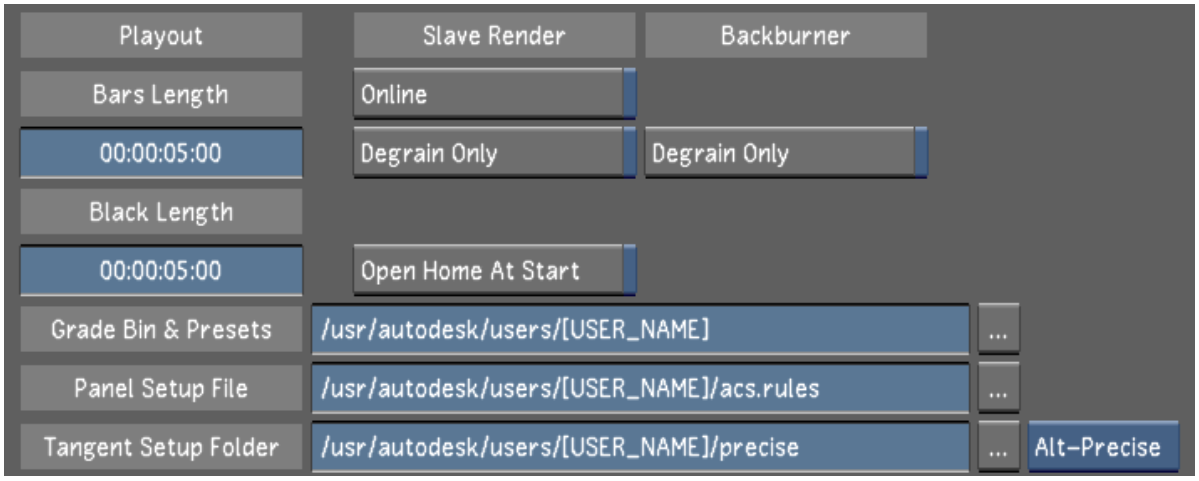
User Template list Displays a list of all the user templates which have been created for Lustre.

System & Menu Settings

Use the System & Menu Settings page to configure the backup options, change shot settings, assemble options, match options, playout settings and background render user preference as well as the panel setup file and the user grade bin and presets directory for this user.







Autosave slider Use to define, in minutes, the frequency with which Lustre automatically saves a backup of the grade and cut you are working on.

Scene Consistency option box Toggle to verify inconsistencies in the file metadata when loading.

Select:	To:
AutoCheck	Check the cut files when loading to look for inconsistencies in the metadata. Results are printed out to the console window.
AutoRepair	Perform the same task as above but additionally, to repair all inconsistencies.
None	Disable the Scene Consistency function in Lustre.

Auto Reparse button Enable to force Lustre to reparse the project structure and source material at start-up.

Print View Reparse button Enable to have Lustre reparse the project structure and source material in Print View.

Message History slider Use to define the number of system messages which Lustre will save in its message history.

NOTE Message History is exclusive to Lustre 2012 Extension 1.

History button Enable to create a history file for each grade whenever you save the grade. Any subsequent grade that you save will also save the cut and grade files to a unique *<grdxx_bak>* backup directory, which

is automatically created in the current scene's library sub-directory in the Project Home location. The backup directory name has the same name as the grade.

Backup button Enable to save a copy of the current grade metadata on the system disk. If you save over your current grade, or save a new grade, a backup copy of all the grade metadata from your current grade is saved in a folder named *backup* that is automatically created in the default location where Lustre is installed (*C:\Program Files\Autodesk\Lustre<version>* for Windows or *usr/autodesk/lustre_<version>* for Linux).

NOTE When History is enabled along with Backup, all the cut and grade files from the history backup will also be saved in the system *backup* folder.

Change Shot option box Toggle to define the change operation settings.

Select:	To:
Do	Force a Do operation after moving to another shot.
DoAndClearUndo	Force a Do operation and delete the undo buffer after moving to another shot.
None	Do nothing when moving to another shot.
ClearUndo	Delete the undo buffer after moving to another shot.

Keep Current Grade button Enable to configure Lustre not to remove a current grade from memory when loading a scene.

Editing While Grading button Enable to make editing hotkeys available when you are in the colour grading menus.

Auto Cleanup button Enable to automatically perform cleanup during an EDL assembly. When Lustre performs an EDL cleanup, it arranges EDL events according to record timecode and deletes problematic events. For example, if an EDL has two events with conflicting timecodes, the event with the lower number is deleted during cleanup.

Auto Match button Enable to automatically match EDL event timecodes to available shot timecodes in the Library during assembly.

Match Grade button Enable to apply the current grade settings to a new assembled timeline without having to create a new cut file and use the Change Cut option.

To Layer button Enable to assemble a cut to a new layer above the existing layer(s) in the currently loaded cut.

Dissolve button Enable to include all dissolves in the EDL. If this button is disabled, dissolves are not part of a timeline after the EDL assembly.

Use DPX Reel button Enable to match the EDL reel name to the reel name in the DPX header during an EDL assembly.

UID button Enable to configure matching to be based on the shot's unique ID during EDL assembly.

Source button Enable to configure the matching to be based on the source data, such as a shot's source timecode, during an EDL assembly.

Record button Enable to configure the matching to be based on the record data, such as an EDL's record timecode, during an EDL assembly.

Custom button Enable to use one of the cut file's metadata as matching criteria during EDL assembly.

XML Tag Options list Select one of the cut file's metadata from this list to use as a matching criteria during an EDL assembly. You must enable the Custom button before performing this operation.

Bars Length slider Use the slider to define the default minimum colour bars length when recording in Record mode.

Black Length slider Use the slider to define the default black length time when recording in Record mode.

Slave Render Online button Enable to automatically enable the Slave Renderer when starting Lustre.

Slave Render Degrain Only button Enable to set the default state of the Degrain Only button in the Render/Slave Render menu. When this option is enabled, the shot will be rendered with only the Degrain data applied.

Backburner Degrain Only button Enable to set the default state of the Degrain Only button in the Render/Backburner menu. When this option is enabled, the shot will be rendered with only the Degrain data applied.

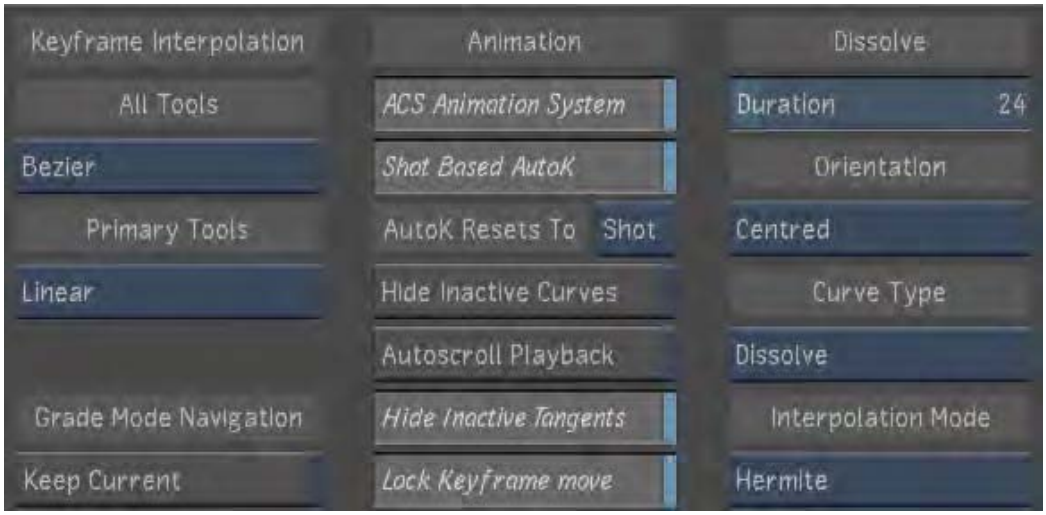
Open Home At Start button Enable to set the file browser to display the contents of your project's Scans Full directory at start-up.

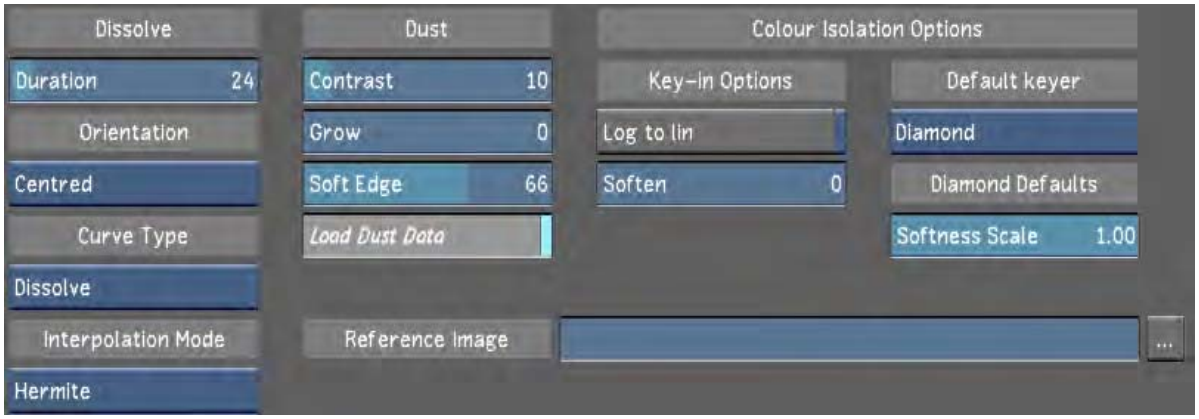
Panel Setup File field Enter, or browse to, the location of the control panel rules file. The default path for this file is the User folder of the application. For information about defining paths, see Browsing for Paths (page 1737).

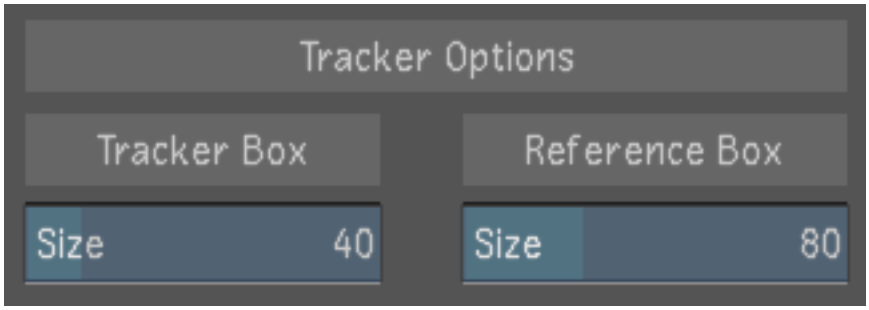
User Grade Bin & Presets Home field Enter, or browse to, the location of the grade files and presets for the current user. The default path for these files is the User folder of the application. For information about defining paths, see Browsing for Paths (page 1737).

Tools Settings

Use the Tools Settings page to configure keyframe, animation, dissolve, dust, colour isolation and reference image settings for this user.







All Tools option box

Toggle to select the default keyframe interpolation mode for all tools, except the input and output primary grading.

NOTE The settings in the All Tools sub-group do not affect tools in the Colour > Grading menu.

Select:	
Linear	Select to use a linear default interpolation for all tools. The linear interpolation joins keyframes with a straight line. This affects all parameters (other than the input and output primary grading) in the Colour menu.
Bezier	Select to make the default interpolation for all tools to be Bezier. The bezier interpolation produces a smooth curve with a smooth transition between keyframes and each keyframe on the curve has tangent handles. This affects all parameters (other than the input and output primary grading) that are in the Colour menu.
Constant	Select to use a constant default interpolation for all tools. Constant interpolation produces a square curve. This affects all parameters (other than the input and output primary grading) in the Colour menu.
Hermite	Enable to make the default interpolation for all tools Hermite. Hermite interpolation produces a smooth curve with a smooth transition between keyframes. This affects all parameters (other than the input and output primary grading) in the Colour menu.

Primary Tools option box

Toggle to select the default keyframe interpolation mode for all primary grading tools.

Select:	
Linear	Select to use a linear default interpolation for all primary grading tools. The linear interpolation joins keyframes with a straight line. This affects all parameters within the Colour menu for input and output primary grading.
Bezier	Select to make the default interpolation for primary grading tools to be Bezier. The bezier interpolation produces a smooth curve with a smooth transition between keyframes and each keyframe on the curve has tangent handles. This affects all parameters within the Colour menu for input and output primary grading.
Constant	Select to use a constant default interpolation for all tools. Constant interpolation produces a square curve. This affects all parameters within the Colour menu for input and output primary grading.
Hermite	Select to make the default interpolation for primary grading tools Hermite. The hermite interpolation produces a smooth curve with a smooth transition between keyframes. This affects all parameters within the Colour menu for input and output primary grading.

Grade Mode Navigation You can enable the Keep Current button to preserve the current grading stage of shots (Primary, Secondary, etc.) as you navigate. This means that when you navigate back to a shot, the last grading stage is automatically recalled.

ACS Animation System button Enable to have Lustre mimic the behavior of the ACS panel when animating from the user interface. Using the ACS Panel, Lustre automatically updates the first keyframe in your timeline whenever a keyed parameter is modified before the first keyframe or the last keyframe in your timeline whenever a keyed parameter is modified after the last keyframe.

Shot Based Auto Key button Enable to activate the Auto-key button on a shot basis rather than having AutoKey active for all the shots.

Hide Inactive Curves button Enable to hide animation curves that are not in use. Hiding inactive curves can make it easier for you to manage your keyframes and modify your animations.

Autoscroll Playback button Enable to configure Lustre's default behaviour to scroll automatically when the positioner plays off the edge of the Channel Editor or Timeline.

Hide Inactive Tangents button Enable to hide tangent handles for keyframes that are not selected.

Lock Keyframe move button Enable to allow only vertical movement when dragging a keyframe.

NOTE You can temporarily override locked movement by holding down the Shift key and dragging the keyframe horizontally. If you want to move the keyframe without any restrictions, disable the Lock Keyframe Movement button.

Duration Slider Use to set the default duration, in frames, of dissolves.

Orientation option box Toggle to select Centered, From Cut or Up To Cut as the default orientation of dissolves.

Curve Type option box Toggle to select Dissolve or Blend as the default curve type for dissolves.

Interpolation Mode option box Toggle to select Constant, Linear, Hermite or Bezier as the default interpolation mode for dissolves.

Contrast slider Use to set the amount of contrast used to find dust during automatic analysis. Low contrast values cause the algorithm to accept low colour value differences between corresponding pixels on the current, previous, and next frames. Therefore, low values result in more complete removal of the artefact. However, if the contrast is set lower than actually needed, adjacent pixels are modified more than actually required to remove the dust.

Grow slider Use to control the number of pixels around the pixels detected by Lustre.

Soft Edge slider Use to soften the stroke of the pixels specified by the Grow slider in order to replicate a natural paint stroke.

Load Dust Data button Enable to load the dust removal metadata after you have already loaded the grade.

Reference Image field Enter, or browse to, the path of the reference image. Then, you can press `Shift+K` to load the image into the single file system for reference and then view the image by pressing `L`. For information about defining paths, see Browsing for Paths (page 1737).

Log to Lin button Enable to apply colour transformation prior to keyer input processing.

Soften slider Use to reduce image grain and noise and to keep edges intact prior to keyer input processing.

Default keyer option box Toggle to select the Diamond Keyer or the HLS Keyer as the default keyer in the secondary menu.

NOTE The default keyer setting only applies to ungraded shots and unused secondaries. A loaded grade file has priority over this setting. This means that the keyer that was used to grade a shot or modify a secondary is the keyer that will be displayed.

Softness Scale slider Use to adjust the scale of the Softness Diamond on the hue cube for the Diamond Keyer.

Tracker Options Define the default size of Tracker and Reference boxes.

Tracker Box slider Use to define the default size of Tracker boxes.

Reference Box slider Use to define the default size Reference boxes.

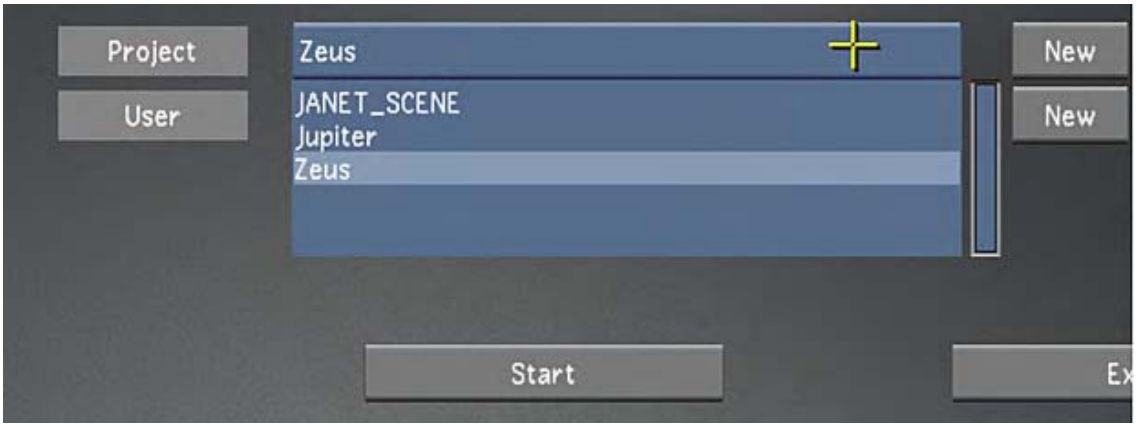
Loading Users and Projects

After creating a project and defining users, you can start using it. You do so by loading a project and selecting a user profile. This can be done from the Lustre splash screen, or from an already started session.

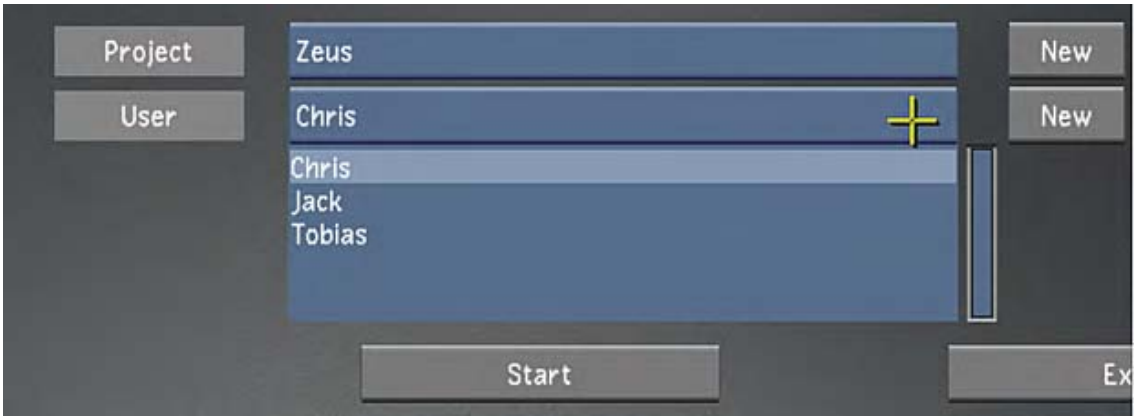
NOTE When you are already in a Lustre session, you can change both the user and the project independently.

To load a project from the Lustre splash screen:

1 Select a project in the Project list.



2 Select a user in the User list.



3 Click Start.

To load a project or a user during a Lustre session:

1 From the Main menu, click Setup, and then Settings.

2 (Optional) To load a project, select a project in the Project list and click Load in the Project group.



3 (Optional) To load a user, select a user in the User list and click Load in the User group.



4 Leave the Settings menu to start working on the loaded project using the loaded user profile.

Working With Scenes

A scene is used to organize your libraries within a given project. A scene called "default" is automatically created with every new project. You can create as many scenes as needed per project.

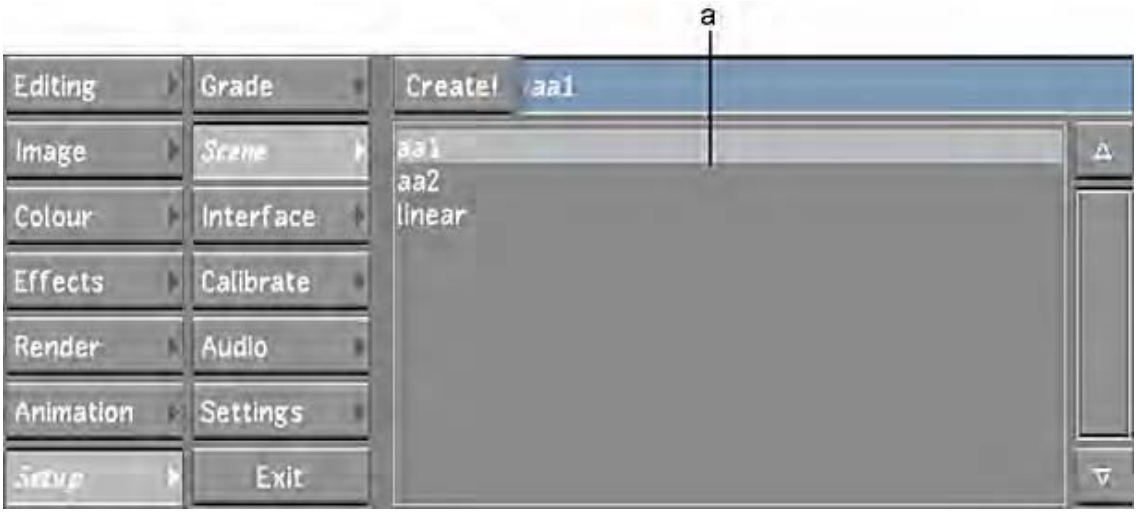
Each Scene folder contains the following folders:

- Audio
- Grade Bin
- Library: this contains all the metadata such as cuts and grades files.

For example, the grade metadata is stored in a file named *grdxx.grade*, where *xx* is the number of the grade file, and the cut metadata is in a file named *cutname.cut*, where *cutname* is the name of the cut.

To create a scene:

1 In the Main menu, click Setup, and then click Scene.
The Scene menu appears.



(a) Scene list

2 Enter a name for the new scene in the Create Scene field. If there is already a name in the Create Scene field, delete it and then enter the new name. This does not affect the existing scene.

NOTE You can create a scene with the name of an existing sub-directory of the Project Home; in this case, Lustre recognizes the directory as a scene and places the required scene metadata directory inside it.

3 Click Create.
The new scene is added to the Scene list. It is highlighted in grey, indicating it is the currently selected scene.

Selecting Scenes

When you select a scene, you gain access to the metadata associated with the scene such as cuts, grade files, the scene-based presets and grade bin, EDLs and audio. Once you select a scene, you then select the grade you want to work on from a list of those associated with the scene. The grade and its associated cut will then be loaded.

You can select a scene from either the Scene or Grade menu.

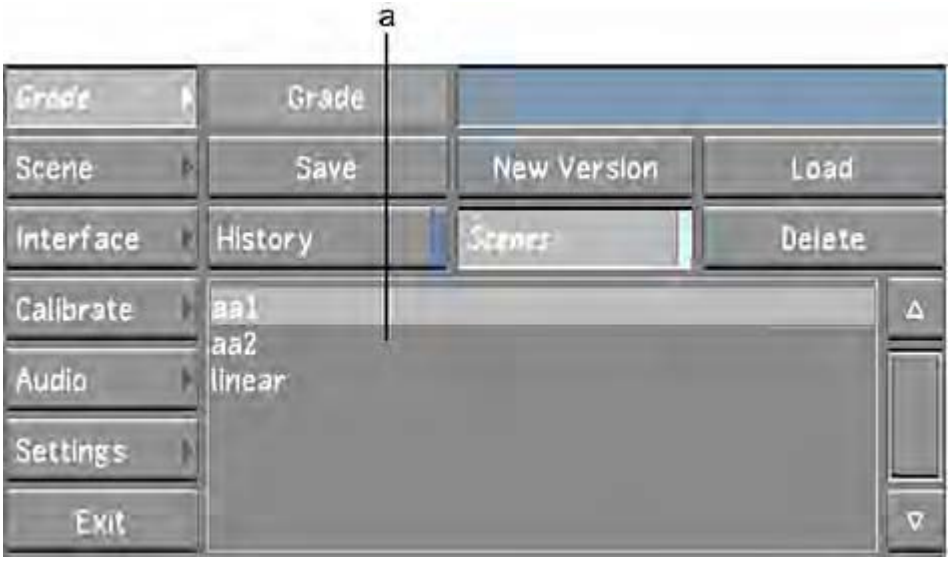
To select a scene from the Scene menu:

1 In the Main menu, click Setup, and then click Scene.
The Scene menu appears.

2 In the Scene list, click a scene to select it.
It is highlighted in grey, indicating it is the selected scene.

To select a scene in the Grade menu:

1 In the Main menu, click Setup, and then click Grade.
The Grade menu appears.



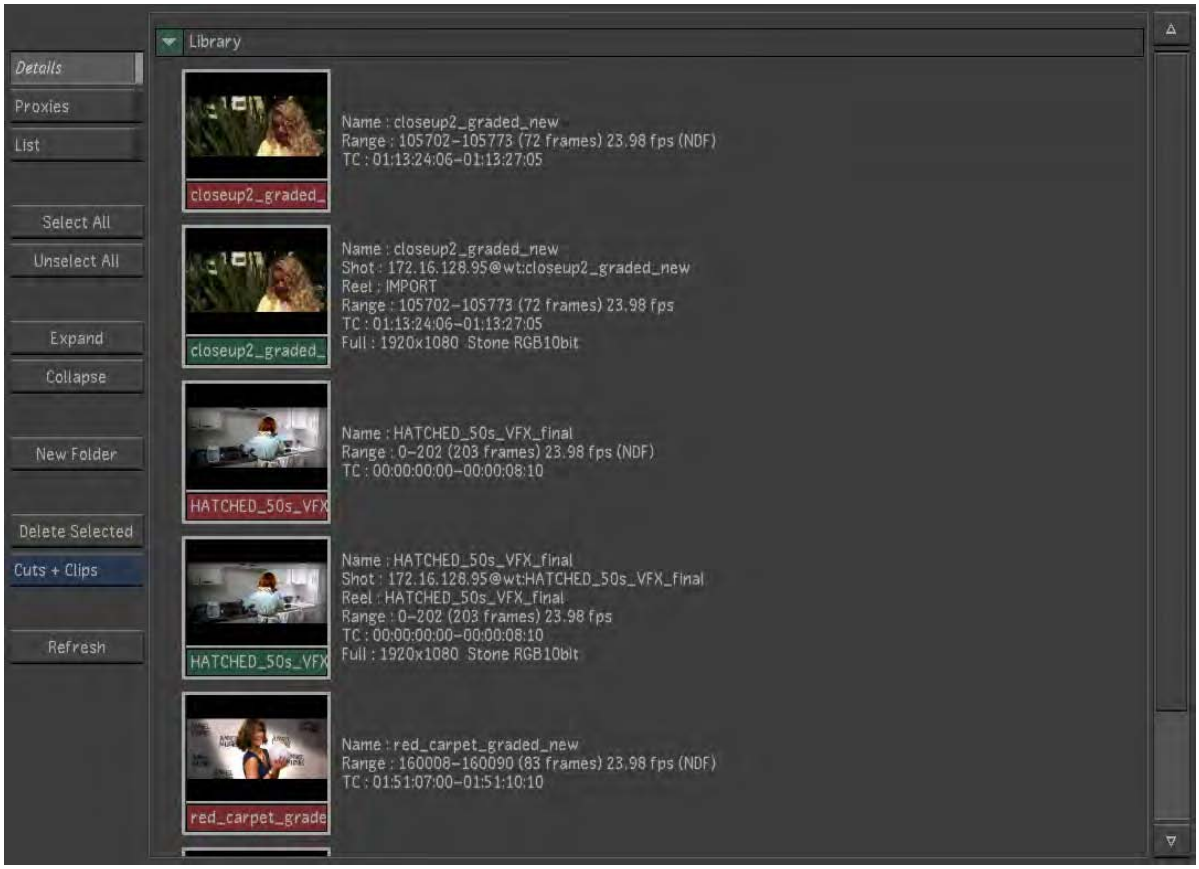
(a) Scene list

2 Enable Scenes.
A list of scenes created in the current project appears.

3 In the Scene list, click the scene to select it.
It is highlighted in grey, indicating it is the selected scene.

Working with Cuts and Grades

When you create a project, a default scene is automatically created. Within each scene is a Library which contains your source clips (.clip) and timelines or cuts (.cut). A cut is automatically created when importing a Wiretap timeline or assembling an EDL. With source clips imported from the Library, you must create the cut manually.



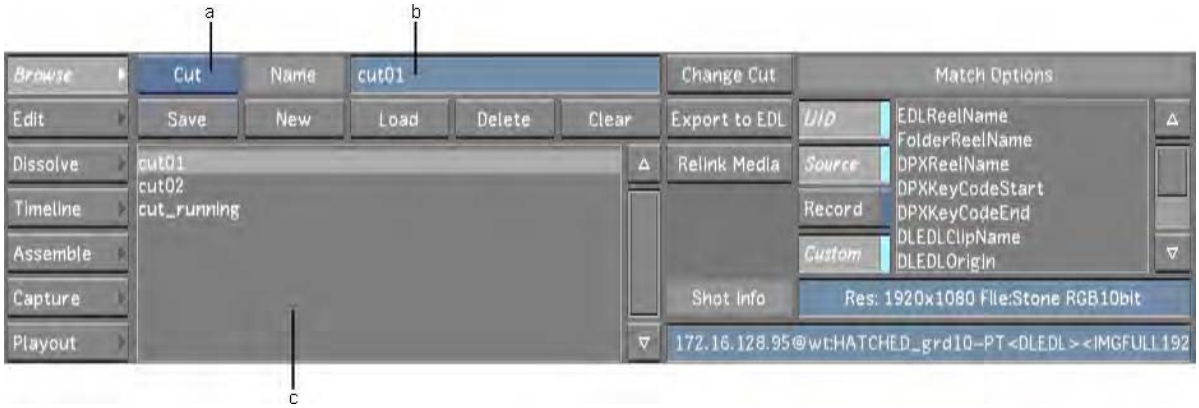
Footage courtesy of EVN PRODUCTIONS

Source clips have green labels; Timelines (or cuts) have red labels.

NOTE Grades are not displayed in the Library because they are managed separately.

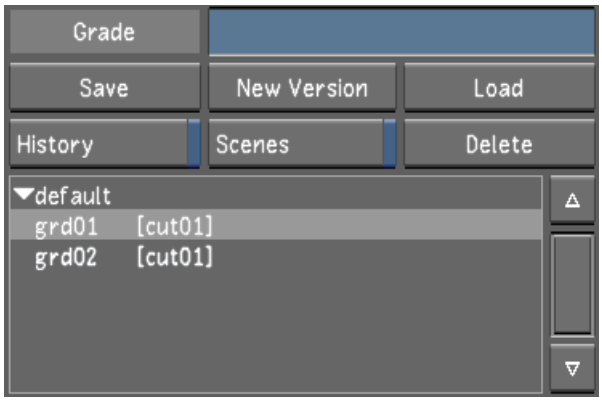
The Browse menu gives you access to all the tools you need to create and manage cuts and change lists. A cut contains edited shots in a timeline, as well as any modifications to the shots such as transcoding, transition effects or soft effect metadata from Wiretap.

A change list applies to a specific cut and contains information about how the contents of a cut has changed. When you apply a change list to its parent cut, the cut is modified to match the change list. Change lists are described in Using Change Lists (page 1775)



As soon as you have created or imported a cut, you need to create an associated grade to contain all the grading information.

A grade is always linked to the cut, but the metadata contained in the grade file is independent of the cut . You can, for example, edit the cut, reorder the shots or add cross-fades, and the grade will not be affected. The grade data for a particular shot will still be linked to it after editing.



When you save the grade, any modifications made to each shot in the associated cut are saved to a grade metadata file. This file is named *grdxx.grade*, where *xx* is the number of the grade file. In addition, any changes made in the Dust menu are saved to a dust metadata file and changes made to the cut (such as trims or dissolves) are saved to the cut metadata file when the grade is saved.

Usually, you save the grade for an entire cut. In some circumstances, it can also be useful to save the grade of an individual shot. See Saving and Loading Marry Grades (page 1784)

Cardinal rules:

- You can have multiple versions of cuts, but each cut should have an associated grade.
- You can create multiple grade versions that reference the same cut.
- A cut can exist without a grade, but **a grade cannot exist without a cut**, therefore we do not recommend ever deleting a cut still referenced by a grade. This results in the grade still being displayed in the list, without you being able to use it.
- You cannot move a cut to a different folder in the Library once a grade is associated with it, as this will break the link between the cut and the grade.

Managing Cuts

Use the tools in the Browse menu to manage your cuts. You can load, save, change, and delete cuts from this menu.



The New button is used for creating new cuts from scratch, or creating a new timeline based from an existing cut. When you create a new cut or cut version, you always need to provide a new name for the cut in the Cut name field.

The Save button is used to overwrite an existing cut.

Creating a New Cut

When you create a new scene, you have an empty timeline. Unless you are assembling from an EDL or importing a Wiretap timeline, you must manually create a new cut.

The Save button is for saving editorial changes to a currently loaded cut.

If you want to create a new cut, you will need to click New.

To create a new cut:

1. Click Editing, and then click Browse to display the Browse menu.
2. Drag source clips from the Browser or Library and drop into the Storyboard or Timeline.
3. Enter the name for the new cut in the Cut Name field.
4. Click New.

   A new cut is created. The name given to the cut appears in the Cut Name list.

To create a new cut version or to save a cut with a new name:

1. Click Editing, and then click Browse to display the Browse menu.
2. Display the Cut Name list by selecting Cut from the Cut/Change List.
3. Select an existing cut from the Cut Name list and click Load.
4. Enter the name for the new cut in the Cut Name field.
5. Click New.

A new cut is created from the current timeline. The new name given to the cut appears in the Cut Name list.

NOTE It is important to create an associated grade version for the cut, see Creating a Grade Version for a Cut (page 1777).

Saving a Cut

The most recent changes to a cut are not saved automatically. You explicitly save your cut in the Browse menu, although you can set up a regular autosave interval.

To save a cut:

1. Click Editing, and then click Browse to display the Browse menu.
2. Click Save.

The cut is saved.

Loading a Cut

You can load an existing cut to resume editorial work on a previously saved timeline.

To load a Cut:

1. Click Editing, and then click Browse to display the Browse menu.
2. Display the Cut Name list by selecting Cut from the Cut/Change List option box.

3 Select the cut in the Cut Name list, and then click Load.

The selected cut is loaded into the timeline.

NOTE You can also load a cut by dragging a Timeline from the Library and dropping into the Storyboard or the timeline.

Clearing a Cut

You can clear the current cut when you no longer need it.

To clear a cut:

1 Click Editing, and then click Browse to display the Browse menu.
2 Click Clear.
3 Click Confirm.

The cut is cleared from the storyboard and the timeline.

Deleting a Cut

You can delete a cut when you no longer need it.

IMPORTANT Do not a delete a cut if it has an associated grade file. The grade is dependent upon its cut.

To delete a cut:

1 Click Editing, and then click Browse to display the Browse menu.
2 Display the Cut Name list by selecting Cut from the Cut/Change List option box.
3 In the Cut Name list, select the cut you want to delete.
4 Click Delete.
5 Click Confirm.
The cut is deleted.

NOTE You can also delete a cut from the Library by selecting the cut thumbnail and clicking 'Delete'.

Using Change Lists

A change list tracks editorial changes to the shots and layers that make up a cut. You use a change list when you want to save changes to a cut without overwriting the cut file. Using this process you can review changes in a cut prior to saving the cut file itself.

The change list tracks the following types of changes:

- Addition/Removal of shots to the cut
- Addition/Removal of layers to the cut
- Rearrangement of shots
- Replacement of shots

A change list always maintains a relationship with the cut from which it is created (the parent cut). You can apply a change list only to its parent cut.

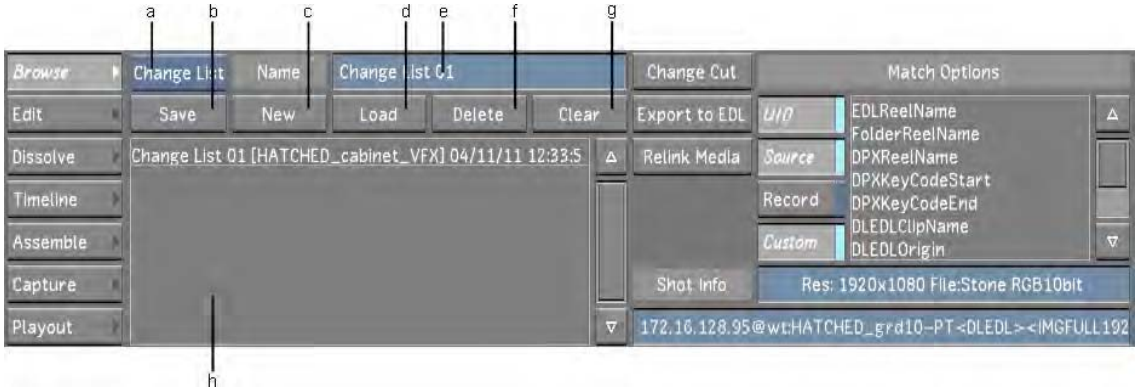
Creating a Change List

After making modifications to the shots in a cut, you can create a change list to save the changes.

To create a change list:

1. After modifying the shots in your cut, click Editing, and then click Browse to display the Browse menu.
2. Select Change List from the Cut/Change List option box.

   The Change List controls appear.



(a) Cut/Change List option box (b) Save button (c) New button (d) Load button (e) Change List Name field (f) Delete button (g) Clear button (h) Change List Name list

3. Type a name for the change list in the Change List Name field.
4. Click New.

   A change list is created. You can see the name of the parent cut in parentheses next to the change list name.

Loading a Change List

You apply the changes stored in a change list to the cut from which the change list was created. This cut is the parent cut.

After you have loaded a change list to its cut, you can continue making changes and save them as a second change list. This second change list contains the changes in the first change list, so should you need to reload all the changes to the parent cut, you do not need to reload the first change list.

To load a change list:

1. Load the parent cut to which you want to apply the changes.
2. Select Change List from the Cut/Change List option box.
3. Select the change list in the Change List Name list.
4. Click Load.

   The changes stored in the change list are applied to the parent cut.

Deleting a Change List

You can delete a change list when you no longer need it.

To delete a change list:

1. Click Editing, and then click Browse to display the Browse menu.
2. Display the Change List Name list by selecting Change List from the Cut/Change List option box.
3. In the Change List Name list, select the change list you want to delete.
4. Click Delete.
   The change list is deleted.

Managing Grades

Though grades are managed separately from their associated cuts, a grade is dependent upon a cut. You cannot create a grade file without an associated cut.

NOTE Using the Change Cut feature, you can apply a grade to a different cut that contains some or all of the same shots. The grade will be applied to the common shots in the new cut.

See Changing a Cut (page 1779).

Creating a Grade Version for a Cut

Save the grade to store the grading data on a specific cut. Grades are named using sequential numbers, but you can optionally add a text name to the number.



NOTE You cannot create a grade if a cut is not loaded since a grade needs to always be linked to a cut.

To create a grade for a cut:

1. Create or load a cut for which you want to save a grade. See Managing Cuts (page 1773).
2. Work on the grade as needed.
3. In the Main menu, click Setup, and then click Grade.
   The Grade menu appears.
4. In the Grade menu, do one of the following:
   - Click New Version to save the new grade version as a binary file.
   - Hold down `Ctrl` and click New Version to save the grade version as an XML file.
5. (Optional) Type a name for the grade in the Grade field and press Enter.

The grade appears in the Grade list under the current scene. It is given a sequential number. If you gave it a name, that name appears in addition to the number. The name of the associated cut appears in brackets to the right of the grade name.

Saving a Grade

When you save a grade by clicking the Save button under the Grade name field in the Setup/Grade menu, the changes are saved by overwriting the currently loaded grade.

IMPORTANT If you have a cut loaded without an associated grade, clicking Save will create grade called "default". If you want to create an actual grade version number, ie. grd01, you need to click New Version.

To Save Changes to an Existing Grade:

1. In the Main menu, click Setup, and then click Grade. The Grade menu appears.
2. Make sure the grade you want to save is loaded.
3. Do one of the following:
   - Click Save to save the grade as a binary file.
   - Hold down Ctrl and click Save to save the grade as an XML file.

The current settings are saved to the existing grade version.

Loading Grades for a Cut

When you start a work session in Lustre, switch scenes in a project, or want to revert to an older version of a grade, you can select the required grade and load it. This loads both the grade and its associated cut.

WARNING You must save the currently loaded grade before switching to another grade, or you will lose any changes you have made to it.

To load a grade:

1. In the Main menu, click Setup, and then click Grade.
   The Grade menu appears.
2. Click the grade you want to load.
   The grade is highlighted in the list.
3. Do one of the following:
   - Click Load to load a grade file in binary format.
   - Hold down the `Ctrl` key and click Load to load a grade file in XML format (the XML suffix follows the grade name).

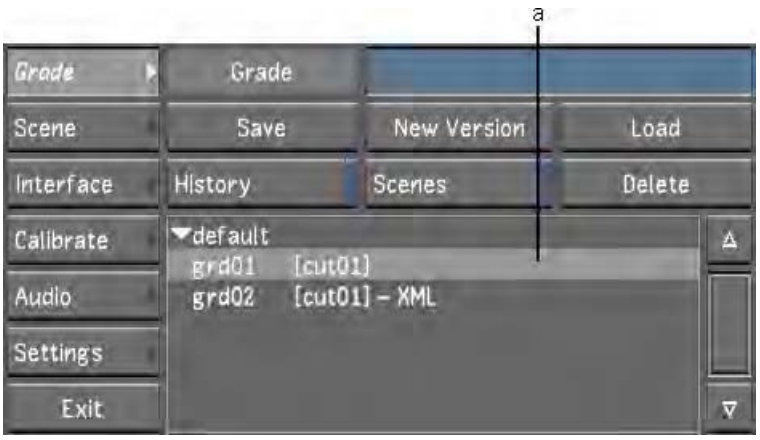
   The grade and associated cut are loaded.

Deleting Grades

You can delete grades that you no longer need.

To delete a grade:

1. In the Main menu, click Setup, and then click Grade.
   The Grade menu appears.



(a) Grade list

2 Display and select the grade you want to delete in the Grade list.
It is highlighted in light grey. You do not have to load it.

3 Click Confirm.

Changing a Cut

The Change Cut tool allows you to apply the grade settings from the current cut to the new cut you are loading.

For example, assume you create a grade with a cut file. You then receive a new EDL and create a new cut from it. This new cut is similar to the first one because it uses many of the same shots, so you would like to use the grade settings that are already done to prevent manually reloading individual grade settings. You load the grade with the original cut, and then change to the new assembled cut. The grade settings that were in the original cut are transferred to the shots that correspond to the Match Options (described below).

The grade is applied to the new cut based on the Match Options (i.e., UID, Source, Record, and Custom). You can base the match on one or multiple options.



NOTE Though the Custom option is enabled by default, nothing is initially selected from the Custom list.

Select:	To:
UID	Match the unique ID of the shots in the original cut to the new cut.
Source	Match the tape/reel name, source ID, and source timecode of the original cut to the new cut.

Select:	To:
Record	Match the record timecode of the original cut to the new cut.
Custom	Match the original cut to the new cut based on the keyword that is selected in the custom list.

If the Custom option is enabled, there is a list of keywords to which you can associate the custom option.

Select:	To:
EDLReelName	Match the reel name of the EDL.
FolderReelName	Match the folder reel name.
DPXReelName	Match the DPX reel name.
DPXKeyCodeStart	Match the DPX header keycode start.
DPXKeyCodeEnd	Match the DPX header keycode end.
DLEDLClipName	Match the clip name of the shot coming from the Wiretap server.
DLEDLOrigin	Match the image import path and file name of the timeline coming from the Wiretap server.
DLEDLSourceId	Match the media source's unique ID of the timeline coming from the Wiretap server.
DLEDLSegmentId	Match the segment's unique ID of the timeline coming from the Wiretap server.
DLEDLStartTc	Match the start source timecode of the timeline coming from the Wiretap server.
EDLComment	Match the unique comment that is applied to the original cut from another application.

The more match options that are enabled, the easier it is to complete an accurate change cut. For example, if you enable UID, Source, and Custom, the same shot can be in the timeline numerous times and the grade will be matched according to the shot's unique ID, tape/reel name, source ID, source timecode, and the custom option you have selected.

Lustre can also apply grading to a new cut on a shot-to-shot (and layer-to-layer) basis, regardless of any matching criteria. This is similar to a direct transfer of grading, depending upon the location of the shots within the timeline. To apply this type of change cut, disable all of the Match Option buttons.

For example, if the original cut is a single layer with two shots, the grade is applied to the new cut's first two shots (if there are more than two shots), even though there is no matching criteria between the shots in the original and new cut.

On a multi-layered cut, if there are two layers of shots in the original cut, the grade is applied to the first two layers of the new cut, on a shot-to-shot basis, even if the new cut has more than two layers. Note that within a multi-layered timeline, the grading is applied to the bottom layer first.

NOTE An alternative to the procedure below is the Match Grade feature. It allows you to apply current grade settings to a new assembled timeline without having to first create a cut file and then apply the Change Cut option. See Match Grade (page 1901).

To apply a grade with Change Cut:

1. Make sure a graded cut is already loaded into the timeline.
2. Click Editing, and then click Browse to display the Browse menu.
3. Display the Cut Name list by selecting Cut from the Cut/Change List option box.
4. Select the cut you want to change the grading to in the Cut Name list.
5. Enable the match options you want applied to the change cut.
6. Click Change Cut.

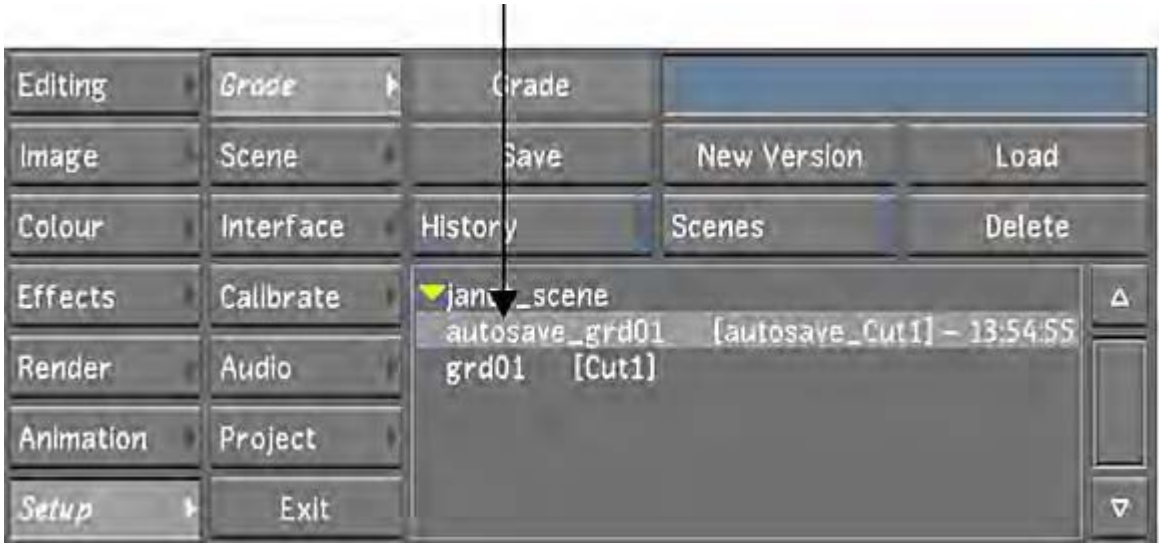
   The new cut is loaded with the grade settings from the old cut.

   When working with a multi-layer timeline, and Solo mode is disabled, the layers are flattened and only the grade settings from the shots that are visible in the Storyboard are applied to the new cut. When working with a multi-layer timeline and Solo mode is enabled, only the grade settings from the active layer are applied to the new cut. If the new cut has multiple layers, then the grade setting is applied to each layer.
7. Click Setup, and then click Grade to display the Grade menu.
8. Click New Version to save the grade with the newly associated cut.

Autosave Grades and Cuts

The Autosave feature ensures that the current grade and cut from the scene you are working on are backed up at specified time intervals. Only one grade and one cut per scene can be backed up. To enable the Autosave feature, enter an interval in the Autosave field of the System & Menu tab. See System & Menu Settings (page 1762).

When you are using Autosave, a snapshot of the current grade and cut appears in the Grade list. The word '*autosave_*' is added to the beginning of the grade and cut name and a time stamp of the autosave is added to the end.



NOTE The autosave information in the Grade list is not updated according to the Autosave time interval, therefore, you need to press Ctrl+R to refresh the information.

A snapshot of the grade, cut, and all their associated files is captured in the Library folder.

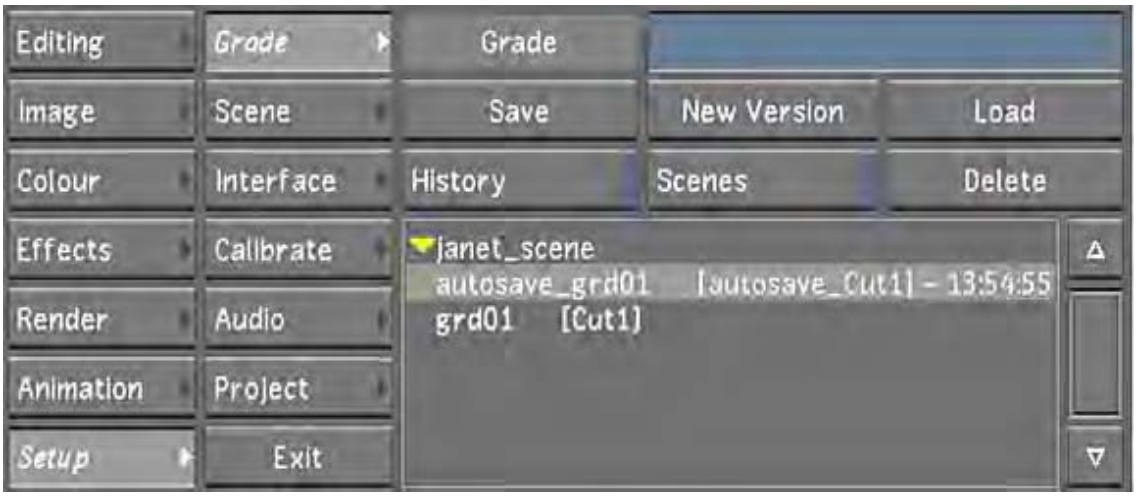
There are certain scenarios where an Autosave is performed before its scheduled time interval. These scenarios include the following:

- After you generate proxies
- When you create a new grade (after performing a change cut or restoring an autosaved grade or cut)

Autosave is suspended if you perform a change cut. See Applying a Grade to a Cut (page 1779). If a change cut is performed, you need to load a grade, save a grade, or create a new grade for the Autosave to reactivate.

To restore an autosaved grade and cut:

1. In the Main menu, click Setup, and then Grade.



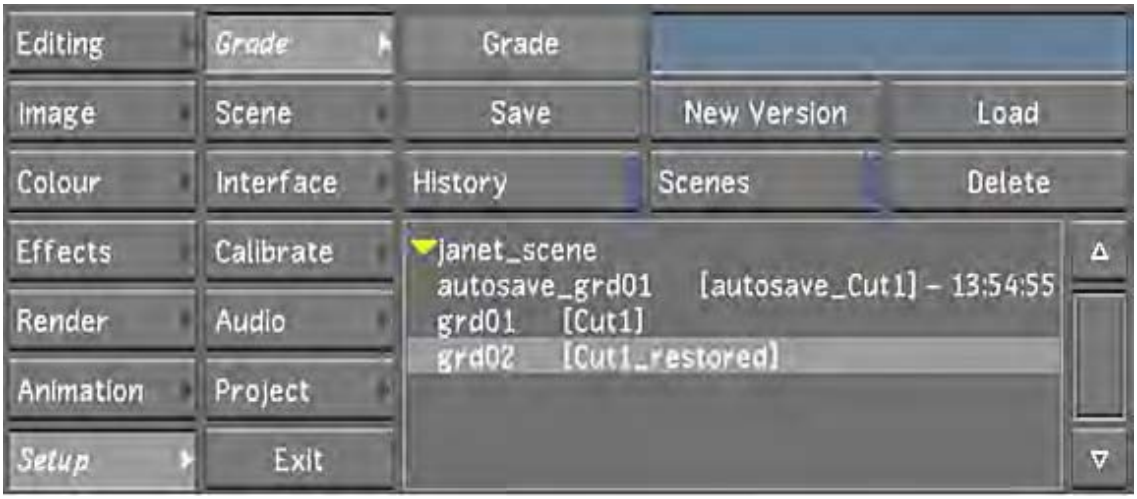
2. Press Ctrl+R to make sure the most current autosave information is displayed in the Grade list.
3. Select the autosaved grade and cut.
4. Click Load.

   NOTE The autosaved version of the cut file appears in the cut name list (e.g., *autosave_Cut1*).

5. Click New Version.

   NOTE You cannot click Save because you cannot overwrite a loaded Autosave cut and grade.

   A new grade is saved and associated with the restored cut.



For the cut file, the existing cut file name is used and *_restored* is added to the end of the name to differentiate the original cut from the one that is restored by using Autosave.



NOTE If you are working with Autosave and multiple grades, you need to use the Grade History list for a list of all the backup grades.

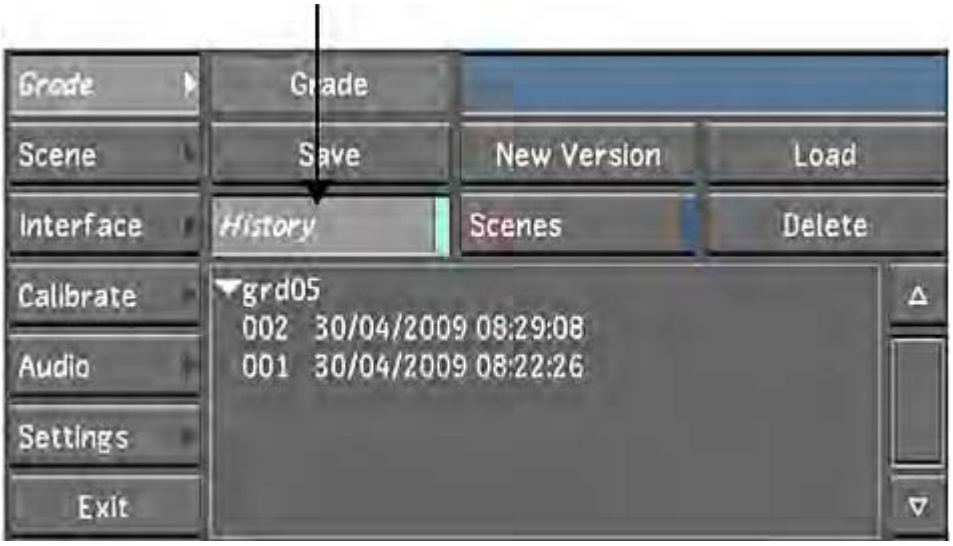
Loading Backup Copies of a Grade

As you save a grade, you can have Lustre save a copy of it each time you click Save. These copies are then available in case you need to revert to a previous version. The date and time of the save is provided to help you identify the version you want to load.

NOTE To save backup copies of grades, you need to first enable the History button when configuring the user. See System & Menu Settings (page 1762).

To revert to a backup copy of a grade:

1. Make sure the grade for which you want a backup copy is loaded. See Loading Grades for a Cut (page 1778).
2. In the Grade menu, click History.



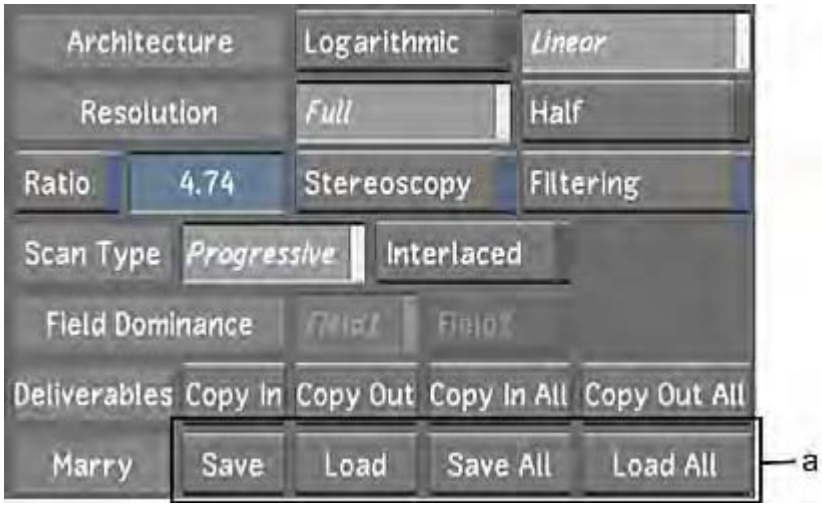
The Grade History list appears. This is a list of backup copies of the currently loaded grade.

3 Click the backup version you want to load.

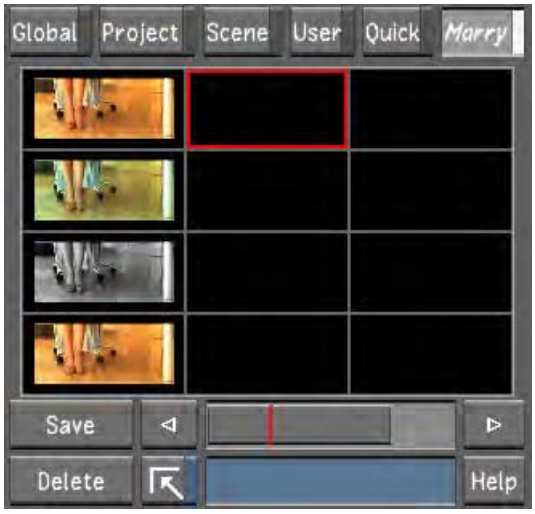
The backup grade is loaded.

Saving and Loading Marry Grades

When you save a grade using the Marry controls, the colour grading metadata is saved separately from the regular grade file. Marry grade files are saved in the same location as the original shots (except for shots imported from Wiretap or the Wiretap Gateway). Marry grade files can also be saved and loaded with the Marry Grade bin thumbnails and Grade view. See Using the Marry Grade Bin (page 1821).



(a) Marry controls



Marry Grade bin

You can save an individual grade for the current shot or grades for all the shots in the timeline. Similarly, you can load the grade back to the current shot or to all shots in the timeline. One situation where Marry grade files can be useful is if you start your project working on a rough cut of a scene, and later are given the final EDL for the scene. You can save the grades created for particular shots on the rough cut and apply them to shots on the final cut.

Marry grade files can be created with any type of content and are saved along with the original scans files. Marry grades created from Wiretap or Wiretap Gateway media are stored under a local *wt* folder under the Scans Full Home or Scans Half Home.

Marry grade files are saved as binary files by default. You can also save the grading metadata as an XML file.

NOTE You can load Marry grade files from previous versions of Lustre that are in text format. They will be automatically converted into binary files.

The folders in which Marry grade files are saved are named based on the unique identifier (UID) or wedge number of the shot. When you use the Scene Detect function to splice a larger shot into individual shots, each scene-detected shot is also assigned a unique identifier (UID). A shot is also assigned a wedge number, which is a range based on the first and last frame in the shot. A wedge number is also associated to shots that are spliced manually.

NOTE You can find the UID for each shot in the cut file located in your scene's Library folder.

When configuring the Marry Grade panel, select Based on Wedge or Based on UID in the Marry Grade File Location option box. See Rendering Settings (page 1743).

To save the grade of one or all shots in a cut:

1. Click Setup, then click Grade to display the Grade menu.
2. If you are saving the grade for one shot, navigate to that shot. See Navigating through Shots (page 1975).
3. Using the Marry controls, do one of the following:
   - Click Save to save the grade for the current shot as a binary file.
   - Click Save All or hold Shift and click Save under the Marry Grade bin to save individual grades for each shot in the cut.
   - Hold down the `Ctrl` key and click either Save or Save All to save the grade(s) in XML format.

To load the grade of one or all shots in a cut:

NOTE To load grades using the Marry controls, the shots in the two cuts must point to the same original footage. If you want to apply the grade of one shot to a different shot, use the Grade bin. See Using Grade Bins (page 1821).

1. Load any cut containing one or more shots for which individual grades have been saved.
2. If you are loading the grade of one shot, navigate to the shot.
3. Using the Marry controls, do one of the following:
   - Click Load or double-click the appropriate thumbnail in the Marry Grade bin to load the grade of the current shot.
   - Click Load All or hold Shift and double-click the appropriate thumbnail in the Marry Grade bin to load the grades for all shots in the cut.

   The grades are loaded to the shots. If the cut contains shots for which no individual grades were saved, those shots are unchanged.

Setting Logarithmic or Linear Mode

Use the Colour Space Architecture options to select between the two basic colour grading modes — Logarithmic and Linear — for use on the project. These modes configure the system and set certain user interface options specific to the mode.

Generally speaking, a project should consist entirely of either Logarithmic or Linear files. For example, film-based projects are usually graded in Logarithmic mode, while video-based projects are usually graded in Linear mode. You should set the architecture to the mode you are planning to work in. If some source files are different from the mode you want to work in, you can convert them before using them. For example, if you are working in Linear mode, and you have a few Logarithmic files, you can convert them to Linear after bringing them into Lustre. To convert a shot from Logarithmic to Linear and vice versa, use the Input LUT menu. See About LUTs (page 2026).

The colour space architecture can be set differently per grade, which may be useful in some circumstances, but typically you should set the architecture for the entire project before beginning. Once you start working on your shots, you should not change the architecture as this affects the images and may create unwanted results.

NOTE The colour space setting for rendering also affects images. See Setting Colour Space Options (page 2226).

Colour Grading Toolset Differences

The colour grading toolset varies depending on if you are in Linear or Logarithmic mode:

- When the colour space is set to Linear, the controls in the Grading menu are customized to reflect the workflow typically used on a video-based project. When set to Logarithmic, the controls reflect the workflow of a film-based project. See Primary Colour Grading (page 2058).
- Printer lights only appear in Logarithmic mode. See Adjusting Printer Lights for Primary Grading (page 2061).
- In the Input LUT menu, the controls for creating a Logarithmic to Linear conversion LUT are only available in Linear mode. See Linear Mode: Creating Conversion LUTs (page 2030).

To set or modify the colour space architecture for the project:

1. In the Grade menu, enable Logarithmic or Linear.

See Project Configuration Settings (page 1738)

Saving Timeline RGB Offsets

You can save the grade for the entire timeline in a file called *primary.txt*. This file is similar to the *.dsc* file, except that the information is timeline-based as opposed to folder-based and cannot be loaded back into Lustre. The benefit of saving timeline RGB offsets is that you can exchange primary grading data with other systems.

The following is an example of what the *primary.txt* file looks like:

GRAO 2

46928774000050 <HOME>/2k/dpx/2048x1556/008_%05d.dpx -64.457 80.586 -16.129

46928774000052 <HOME>/2k/dpx/2048x1556/009_%05d.dpx*-52.922 84.429 -31.507

To save the grade for the timeline:

1. Press `Shift+G`.

   The *primary.txt* file is generated in the Lustre export directory, e.g. *C:\Program Files\Autodesk\Lustre<version>\export* on Windows, and */usr/autodesk/lustre<version>/export* on Linux.

The `Shift+G` hotkey has another function. In addition to saving the primary grading data for the timeline, the hotkey automatically runs batch scripts. This could be used, for example, by a facility to convert the grading data to a LUT or third-party Color Decision List (CDL).

To automatically run a batch script:

1. Create a batch script called *primexport.bat*.
2. Place *primexport.bat* in the *C:\Program Files\Autodesk\Lustre<version>\export* directory on Windows, and */usr/autodesk/lustre<version>/export* on Linux.
3. Press `Shift+G`.

   Lustre saves the primary offset RGB values for the timeline in the *primary.txt* file, and automatically runs the batch script.

Setting the Resolution

You can work on the full-resolution version of shots or proxies. Using proxies speeds up interaction and playback of shots containing many effects.

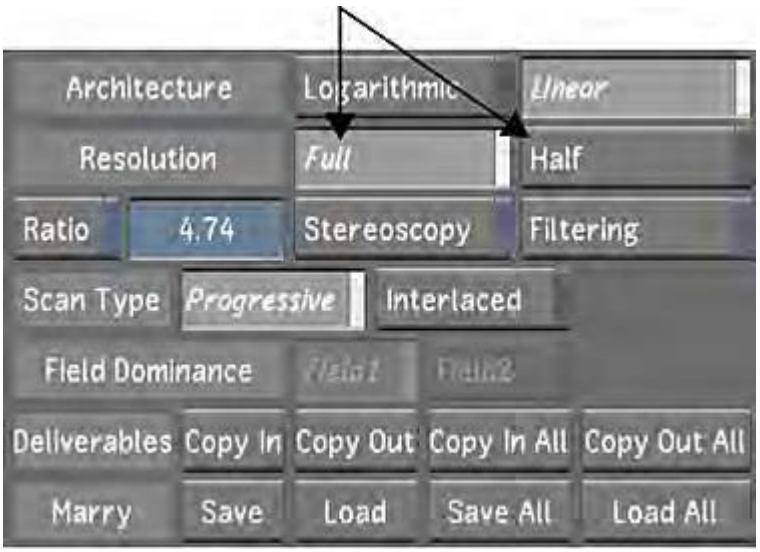
You can switch between full and half resolution at any time. Generating and displaying proxies does not affect your original footage in any way. The resolution is independent of the grade metadata and is therefore completely non-destructive to your work. Metadata is applied to footage only when you render it. You specify the resolution you want for the render files at the time you render.

To display proxies, you must first generate them within Lustre or create them outside Lustre and place them in the project directory structure. If you opt to work on proxies, it is recommended that you generate them as soon as the cut is created so that they will be immediately available for use. See Generating and Viewing Proxies (page 2227) and Recommended Directory Structure for Projects (page 1734).

To set the resolution:

1. In the Grade menu, enable Full or Half resolution.

   TIP You can also use the F9 hotkey or the 1 / 1/2 button in the View mode panel.

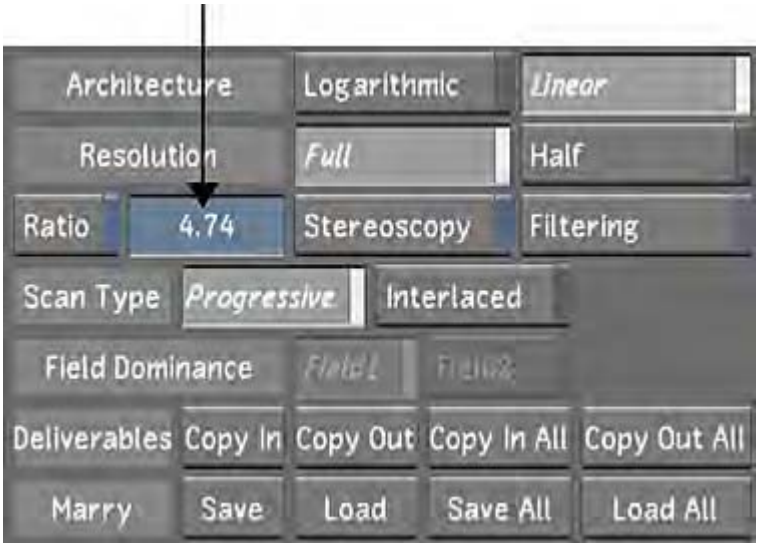


Setting the Aspect Ratio

You can set the display aspect ratio for shots in the project. When you change the aspect ratio, the shots are stretched accordingly.

To set the project aspect ratio:

1. In the Grade menu, enable Ratio.
2. Enter the aspect ratio in the Ratio field.



The shots are stretched to fit in the specified aspect ratio.

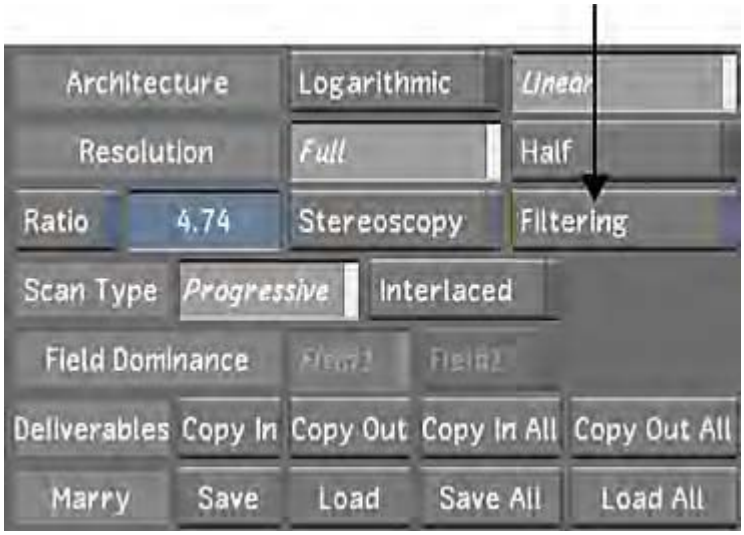
Setting the Filtering

The Filtering option allows you to soften your shot so when you zoom in, the pixels are not as sharp or obvious.

If the Filtering option is enabled during playout with GPU acceleration, the filtered image will be played out to tape.

NOTE The Filtering option should be disabled when you want to isolate the colour of a pixel so it can be used with the HLS and Diamond Keyers.

To enable the filtering option, access the Grade menu and enable the Filtering button.

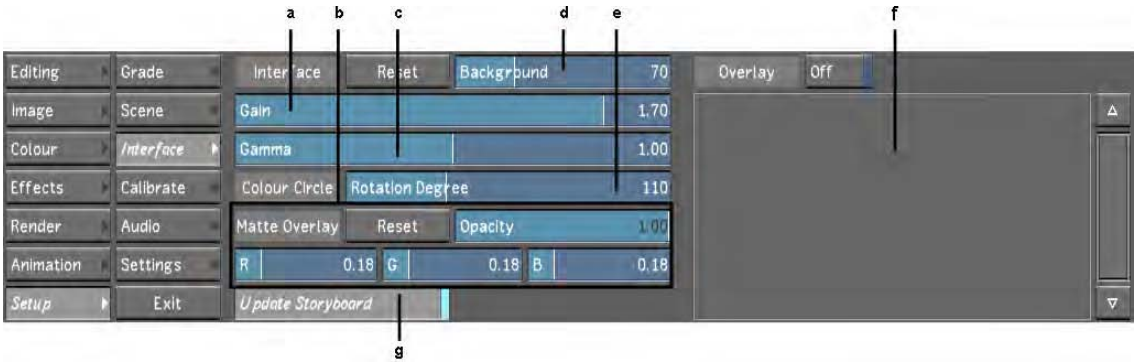


Setting Up the User Interface

Use the controls in the Interface menu to set the appearance of the user interface to your liking. For example, you can set the user interface to suit the room lighting conditions as well as control the display of a matte overlay by adjusting the opacity and RGB parameters of the unselected region of the matte. From the Interface menu, you can also add an overlay, such as a logo, to your cut.

To access the Interface menu:

1 In the Main menu, click Setup, and then click Interface.
The Interface menu appears.



(a) Gain slider (b) Matte Overlay (c) Gamma slider (d) Background slider (e) Colour Circle slider (f) Overlays field (g) Update Storyboard button

The Interface menu is made up of the following elements.

Background field Sets the shade of grey of the area surrounding the player window.

Gamma field Sets the brightness of the midtones of the menu area. The gamma primarily affects the grey areas of the menu, such as button backgrounds.

Gain field Sets the overall brightness of the menu area. The gain primarily affects lighter areas of the menu, such as text.

Colour Circle field Sets the position of colours in the colour wheels found in the Grading menu. The default rotation degree is 110.

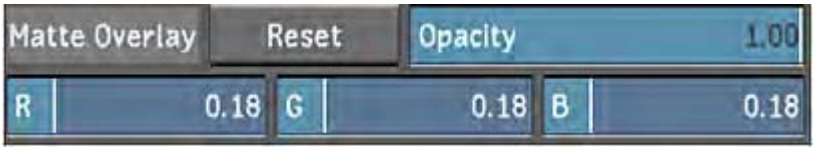
Reset button Resets all parameters in this menu to the default values.

Overlay list Includes a list of all files that can be applied as an overlay. See Applying Overlays (page 1791).

Update Storyboard button Enables and disables the option to automatically update the storyboard thumbnails as you make changes to the shots within your timeline. See Updating the Storyboard Thumbnails (page 1980).

Matte Overlay Settings

When you perform secondary colour grading, whether with a geometry, a key, or an external matte, it is important to see what part of the image is being affected by the matte. The Matte Overlay feature displays the matte in its original colour, while overlaying the unselected region of the matte with a uniform colour, allowing you to see the precise area of your image that will be colour corrected.



Opacity Sets the opacity of the unselected region of the matte between fully transparent (0) and fully opaque (1).

RGB Set the intensity of the red, blue, and green colours for the unselected region of the matte.

Reset button Resets the Matte Overlay parameters to default values.

You can modify the Matte Overlay settings at any time when you are performing secondary colour grading without affecting the image output. See Adjusting the Colour and Opacity of a Matte Overlay (page 2106).

NOTE Matte Overlay settings are saved per user; therefore, any changes are saved to the current user profile.

Applying Overlays

Use this feature to apply an overlay to your cut. For example, you can add a logo to every frame. An overlay consists of a single uncompressed RGBA, TIFF, or SGI® file containing alpha channel information. The white areas of the file are viewable in the cut.

To make files available for use as overlays:

1. Exit Lustre.
2. Navigate to the Lustre directory by doing one of the following:
   - If using the Windows version of Lustre, in Windows Explorer, go to *C:\Program Files\Autodesk\Lustre<version>*, where *C* is the drive where Lustre is installed.
   - If using the Linux version of Lustre, type:
     cd /usr/autodesk/lustre<version>
3. Create a directory called *overlay*. To do this in Linux, type:
   mkdir overlay
4. Place any files you want to use as overlays in this directory. To do this in Linux, type:
   cp / `<path>/<filename>` **/usr/autodesk/lustre<version>/overlay**
5. Restart Lustre.

To apply an overlay to a cut:

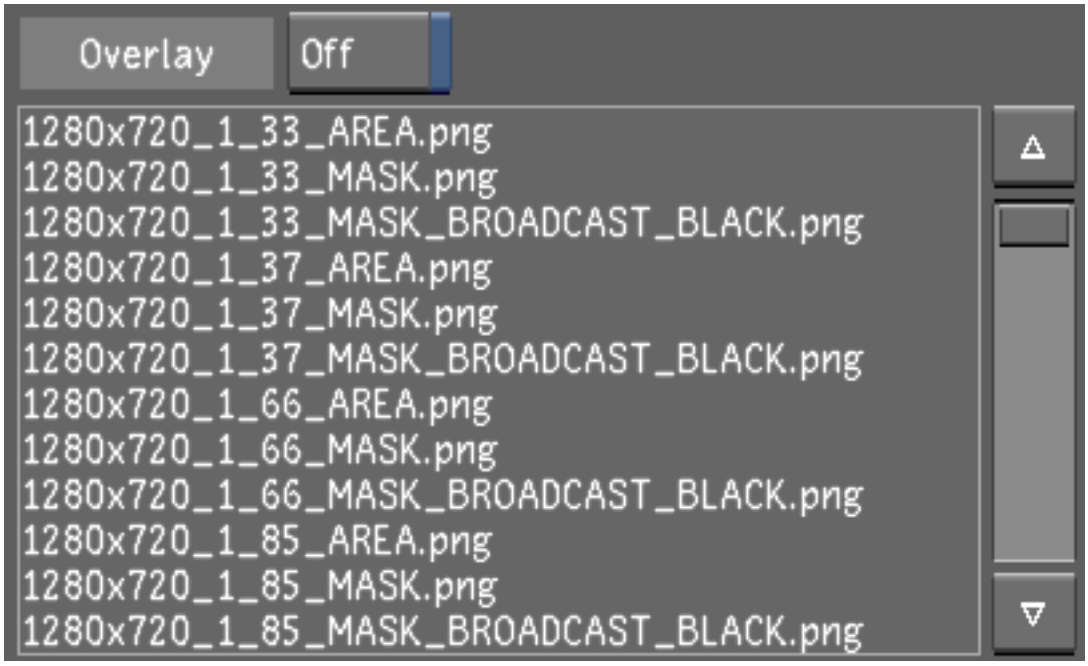
1. Load the cut to which you want to apply an overlay. See Managing Cuts (page 1773).
2. In the Main menu, click Setup, and then click Interface.
   The files in the *overlay* directory appear in the Overlay list.
3. Click the overlay file that you want to apply.
   It is highlighted.
4. Enable On/Off.



The overlay is applied to the cut.

Overlay Presets

In the Setup / Interface menu, you can select an overlay from a list of overlay presets to apply to your media. You can use these overlays for play out to VTR.



The following overlay presets are available:

- **Area:** White outline showing the framing of the selected resolution and aspect ratio.
- **Safe:** White outline showing the full image area, the safe action and the safe title.
- **Mask:** Black mask showing the framing of the selected resolution and aspect ratio.
- **Mask Broadcast:** Use this mask when you want to playout the full video signal of the grade to VTR, using the Full Range playout option.

Overlays are available for the following resolutions:

- NTSC (720x486)
- PAL (720x576)
- 1280x720
- 1920x1080
- 2048x1556

NOTE Overlays are not saved with the grade file. They are a user interface attribute that is saved with the session. Overlays cannot be rendered to files.

See VTR Playout and Full Range Video Signal (page 1792).

VTR Playout and Full Range Video Signal

You can play out grades from Lustre to VTR, using the full video range (0 to 255 in 8-bit and 0 to 1024 in 10-bit). To do so, select the Full Range playout option. In this mode, the black and white levels are remapped to the full video range. Make sure the following settings are in place prior to output to VTR:

- In the Project Management / Engineering tab, set the Virtual Black Clip Color to RGB 16-16-16 (for 8-bit) or 64-64-64 (for 10-bit). This creates a legal black video signal for black media.
- If your timeline contains one or multiple gaps on video layer 1, make sure to replace these segments with virtual black. Alternatively, if your timeline contains so many gaps as to render the substitution

impractical, insert a video layer underneath the first video layer and insert a virtual black clip spanning the length of the timeline.

- To use letterbox on your playout, use one of the Broadcast Black overlays provided instead of using Image / Reposition or Render / Resize. These Broadcast Black overlays have a black video level that, when remapped, is set to legal black video level.

Working with Templates

Use templates to define re-usable presets for future projects or users.

Creating a Project Template

When you are choosing your project settings, you can create a template for your future projects.

From the Project Template group, within the Project settings page, you can complete the following:

- Create a new project template.
- Create a new project template from an existing project.
- Create a new project template from an existing project template.

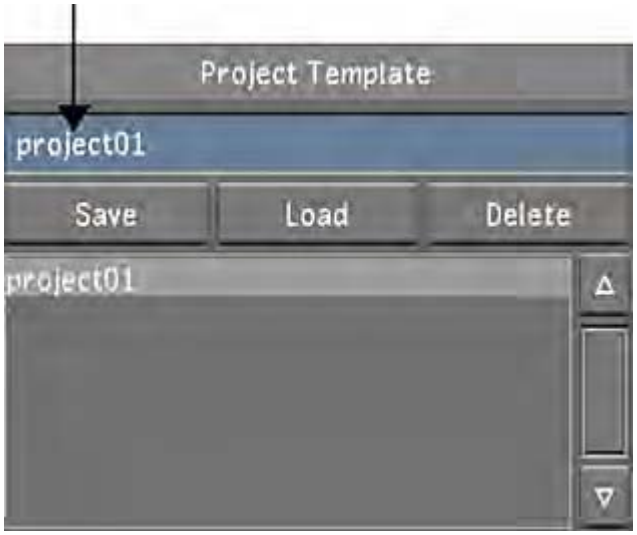
To create a new project template:

1. Do one of the following:
   - From the Main menu, click Setup, then Settings, and then click New in the Project group.
   - From the Lustre splash screen, click New in the Project group.



NOTE If you have a template named *default*, then its settings will be loaded automatically.

2. Configure the settings for the template. Use Reset to reset all settings, from every menu, to their defaults.
3. When you are done editing the settings, click Project, and enter a project name in the Project Template field. See Project Configuration (page 1730).



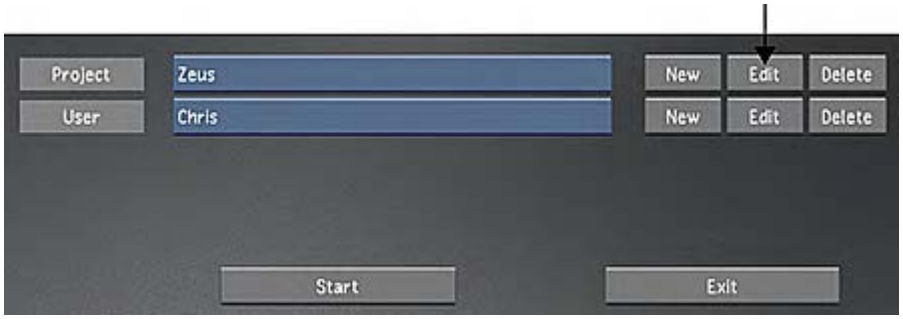
4 In the Project Template group, click Save.

5 To exit, click Exit Project.

To create a new project template from an existing project:

1 Do one of the following:

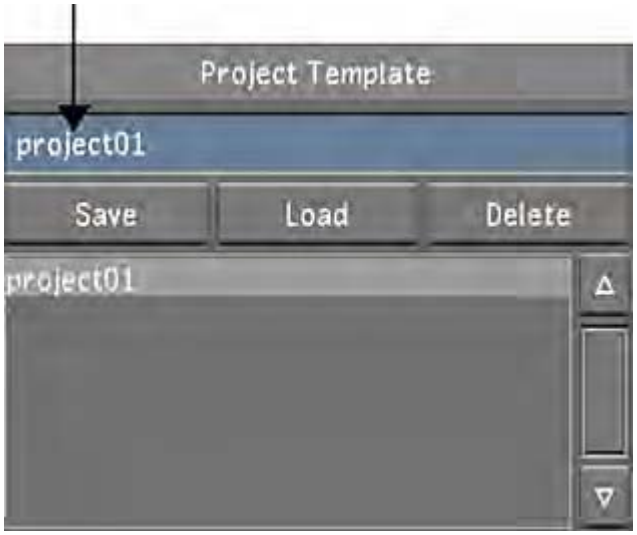
- From the Main menu, click Setup, and then Settings. In the Project group, select the baseline project, and then click Edit.
- From the Lustre splash screen, select the baseline project, and then click Edit in the Project group.



2 Configure the project. Use Reset to reset all settings, from every menu, to their defaults. See Project Configuration (page 1730).

WARNING Rename the new project, or you will overwrite the original project's settings with the new settings.

3 Click Project and enter a name in the Project Template field.



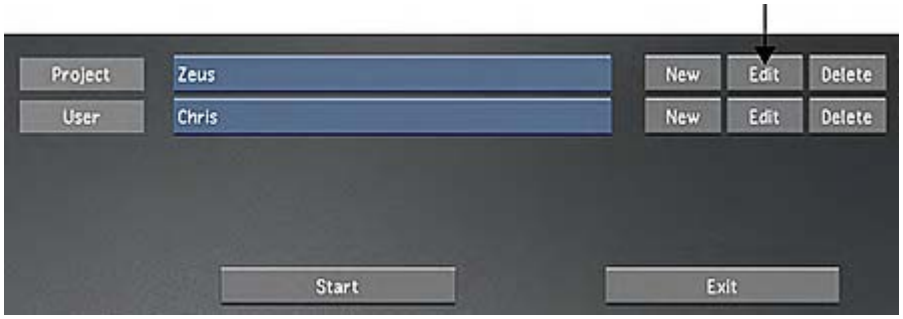
4 To save the template, click Save in the Project Template group.

5 Click Exit Project.

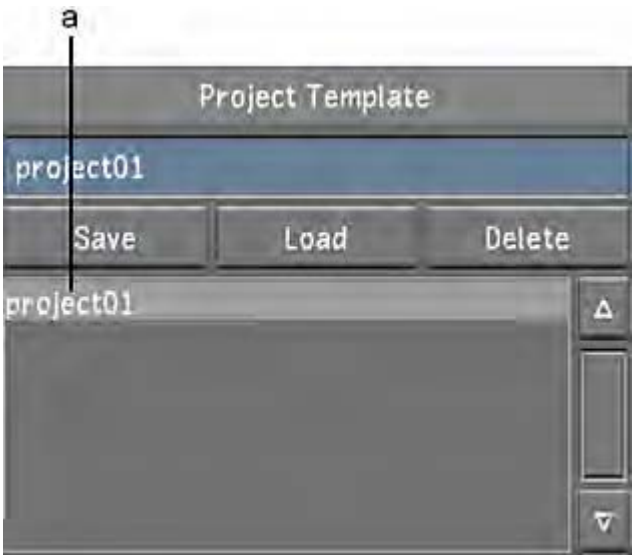
To create a new project template from an existing project template:

1 Do one of the following:

- From the Main menu, click Setup, and then Settings. Select the project to edit, and then click Edit in the Project group.
- From the Lustre splash screen, select the project to edit, and then click Edit in the Project group.



2 From the Project Template list, select a template.



(a) Project Template list

3 Click Load.

4 In the Project Template field, enter a new template name, and then click Save.

WARNING Rename the new template, or you will overwrite the original template with the new settings.

5 Configure the template. Use Reset to reset all settings, from every menu, to their defaults. See Project Configuration (page 1730).

6 To save the template, click Save in the Project Template group.

7 Click Exit Project.

Editing a Project Template

As you create project after project, there may come a time when you need to change a few of the settings and you do not want to create an entire new template. You can accomplish this by using the edit feature.

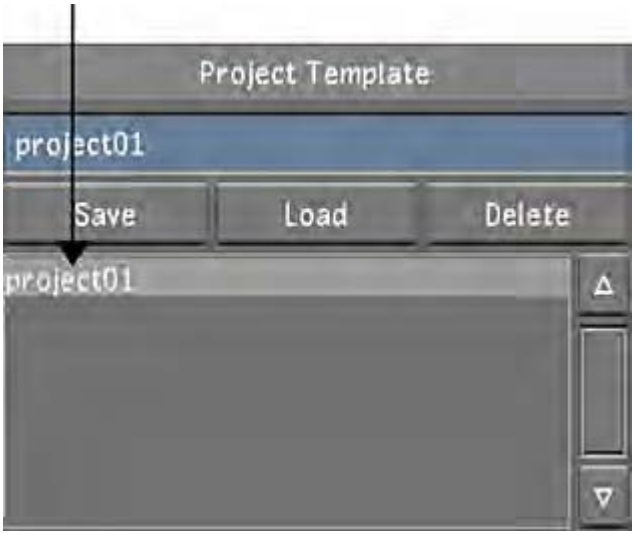
To edit a project template:

1 Do one of the following:

- From the Main menu, click Setup, then Settings, and then click New in the Project group.
- From the Lustre splash screen, click New in the Project group.



2 From the Project Template list, select the template to edit.



3 Click Load.

4 Configure the template. Use Reset to reset all settings, from every menu, to their defaults. See Project Configuration (page 1730).

5 To save the template, click Save in the Project Template group.

6 Click Exit Project.

Creating a User Template

When creating a user, you can save the setup as a template for other users.

From the User Template section in the Display & Interface settings, you can do the following:

- Create a new user template.
- Create a new user template from an existing user profile.
- Create a new user template from an existing user template.

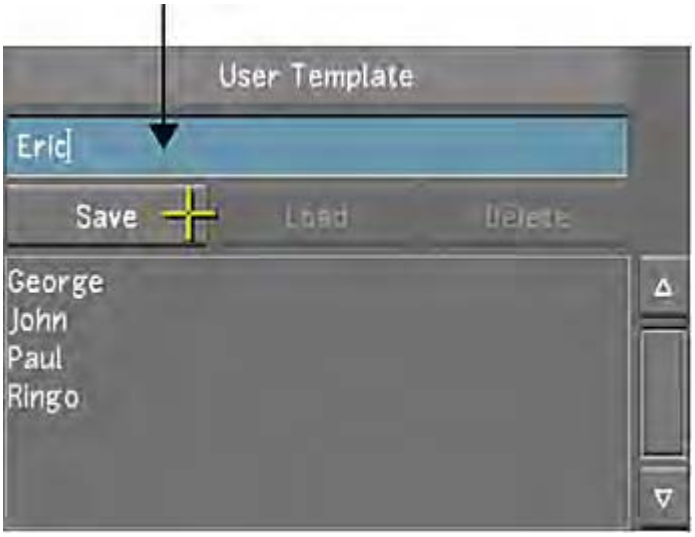
To create a new user template:

1 Do one of the following:
   - From the Main menu, click Setup, then Settings, and then click New in the User group.
   - From the Lustre splash screen, click New in the User group.



2 Configure the new user. Use Reset to reset all settings, from every menu, to their defaults. See User Configuration (page 1755).

3 Click Display & Interface, and enter a name in the User Template field.



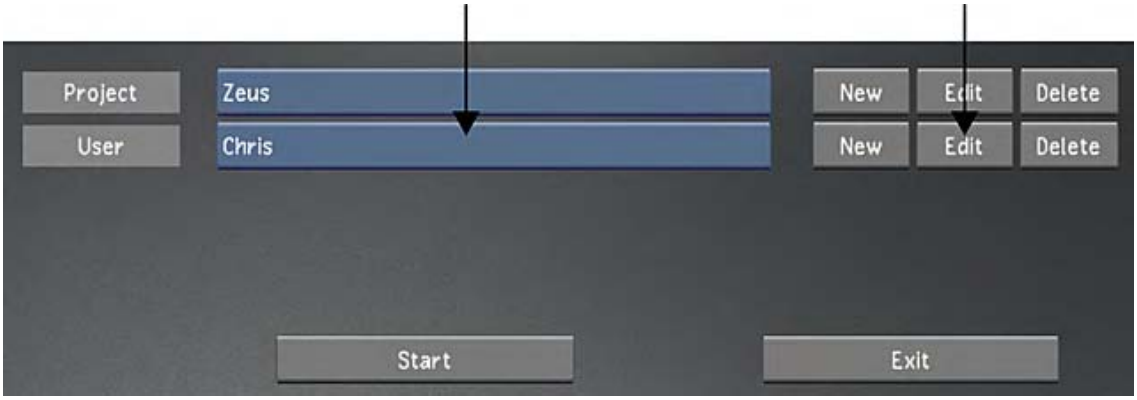
4 To save the template, click Save in the User Template group.

5 Click Exit User.

To create a user template from an existing user profile:

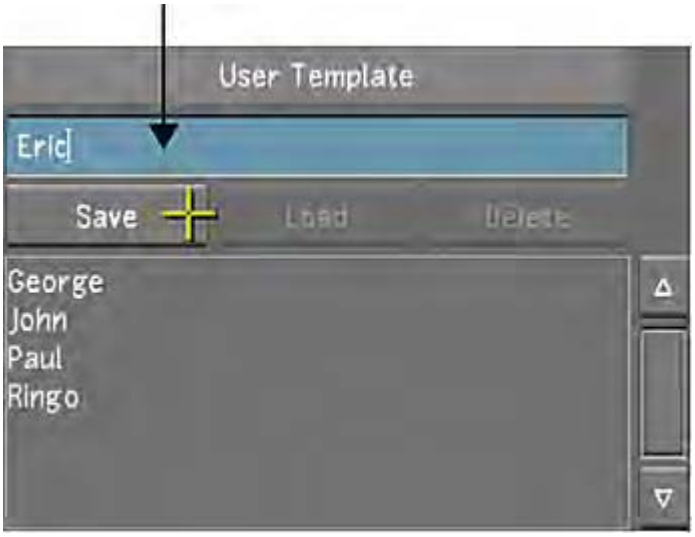
1 Do one of the following:

- From the Main menu, click Setup, and then Settings. Select the baseline user, and then click Edit in the User group.
- From the Lustre splash screen, select the baseline user, and then click Edit in the User group.



2 Configure the new template. Use Reset to reset all settings, from every menu, to their defaults. See User Configuration (page 1755).

3 Click Display & Interface and enter a new name in the User Template field.



4 To save the template, click Save in the User Template group.

5 Click Exit User.

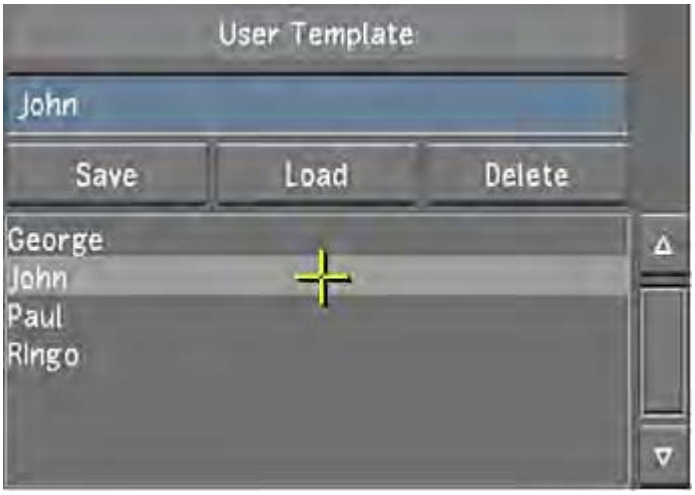
To create a new user template from an existing user template:

1 Do one of the following:

- From the Main menu, click Setup, then Settings, and then click New in the User group.
- From the Lustre splash screen, click New in the User group.



2 From the User Template list, select the template to use as baseline.



3 Click Load.

4 In the User Template field, enter a new template name, and then click Save.

WARNING Rename the new template, or you will overwrite the original template with the new settings.

5 Configure the template. Use Reset to reset all settings, from every menu, to their defaults. See User Configuration (page 1755).

6 From the User Template list, select the new template.

7 To save the template, click Save in the User Template group.

8 Click Exit User.

Editing a User Template

The template editing feature is available for you to make minor changes to existing templates as they are needed.

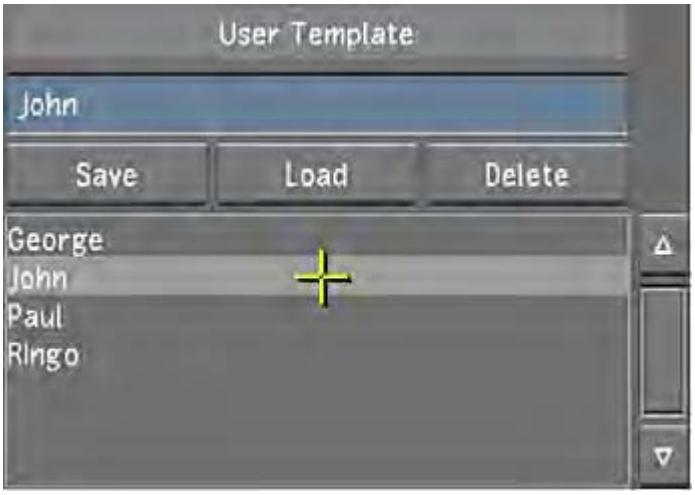
To edit a user template:

1 Do one of the following:

- From the Main menu, click Setup, then Settings, and then click New in the User group.
- From the Lustre splash screen, click New in the User group.



2 In the User Template group, select the user template to edit.



3 Click Load.

4 Configure the template. Use Reset to reset all settings, from every menu, to their defaults. See User Configuration (page 1755).

5 To save the template, click Save in the User Template group.

6 Click Exit User.

Basics

This chapter introduces basic functionality for Lustre. It provides an overview of the user interface and control surfaces, and explains global features such as undo/redo and the Grade bin.

Starting and Exiting Lustre

To start Lustre, use any of the standard methods for opening an application in Windows or Linux.

To start Lustre:

1 Start the application by double-clicking the desktop icon, or by doing one of the following:
- If using the Windows version of Lustre, select Start | Programs | Autodesk | lustre from the Windows task bar.

- If using the Linux version of Lustre, it is recommended to log into the Lustre environment. If you open a Linux shell, it will automatically place you in the Lustre directory:

  /usr/Autodesk/<lustre version>/

  Next, start Lustre. Type:

  ./lustre to start the Lustre application.

Lustre starts with all the settings you used when you last exited. For example, it loads the project, cut, and grade you were working with and places the positioner on the frame you were on.

NOTE If Lustre cannot find a license, the Flex LM® wizard appears, requesting the location of the license file.

To exit Lustre:

1. In the Main menu, click Setup.
2. Click Exit and confirm.

TIP You can also exit the application by pressing SHIFT + Insert.

Understanding the Lustre User Interface

The Lustre user interface is divided into four main sections.



Footage courtesy of EVN PRODUCTIONS (a) Player display controls (b) Player (c) Storyboard (d) Menu

Most sections display different kinds of information, depending on user preferences and the action being performed.

The Player

This section of the user interface is generally used for viewing the currently selected shot in the Storyboard. However, you can also use it to:

- Display the Library and file browser in order to select footage to add to the Storyboard. See Browsing for Footage (page 1856).
- View and modify EDL details. See Working with EDL, ALE, and Cutlist Files (page 1892).
- Display Colour menus. See Displaying Colour Menus in the Player (page 2042).
- Monitor capture progress. See Capturing Material (page 2264).

The Storyboard

Use the Storyboard to select shots, arrange shots, navigate through shots, and isolate certain types of shots. Using hotkeys, you can also copy grades to shots via the Storyboard. See Navigating through Shots (page 1975) and Storyboard Viewing Options (page 1979).

Player Display Controls

Below the Storyboard, there are controls for:

- Resetting the current menu. See Resetting Parameters (page 1804).
- Bypassing the current menu. See Bypassing Menu Parameters (page 1805).
- Undoing and redoing actions. See Undoing and Redoing Actions (page 1805).
- Setting play and view options for shots. See Playing, Viewing, and Sorting Shots (page 1968).
- Adjusting printer light settings. These display only when a Colour menu is active. See Adjusting Printer Lights for Primary Grading (page 2061).
- Setting keyframes automatically. See Animating Reposition Values (page 2024) and Animating with Keyframes (page 2176).

The Menu Section

The menu section in Lustre is organized from left to right. The first column of buttons are the main menus. Clicking a main menu displays its sub-menu buttons in the next column. Clicking a sub-menu displays its functions in the central part of the menu. Menu functionality is addressed by other chapters in this guide.

On the right side of the menu are features that, for the most part, are globally applicable. These features include:

- The Notes window. See Assigning Notes to Shots (page 1807).
- The Flags system. See Flagging Shots (page 1808).
- Grouping functionality. See Creating Shot Groups (page 1810).
- The Selector. See Copying Parameters with the Selector (page 1816).
- The standard Lustre calculator. See Assigning Values With the Standard Lustre Calculator (page 1819).
- The Grade bin. See Using Grade Bins (page 1821).

Working with Control Surfaces

Lustre supports the Tangent Element control surface, the Tangent CP100 control surface and the Autodesk control surface (ACS). All three control surfaces improve interactivity when colour grading film and video footage, and allow you to accomplish many of the tasks you do in the Lustre user interface.

See About the Tangent Element Control Surface (page 2377)

See The Tangent CP100 Control Surface (page 2484)

See The Autodesk Control Surface (page 2420)

Resetting Parameters

You can reset parameters on a shot-by-shot basis. You can either reset a parameter, reset all parameters in a menu, or reset a shot to its original state.

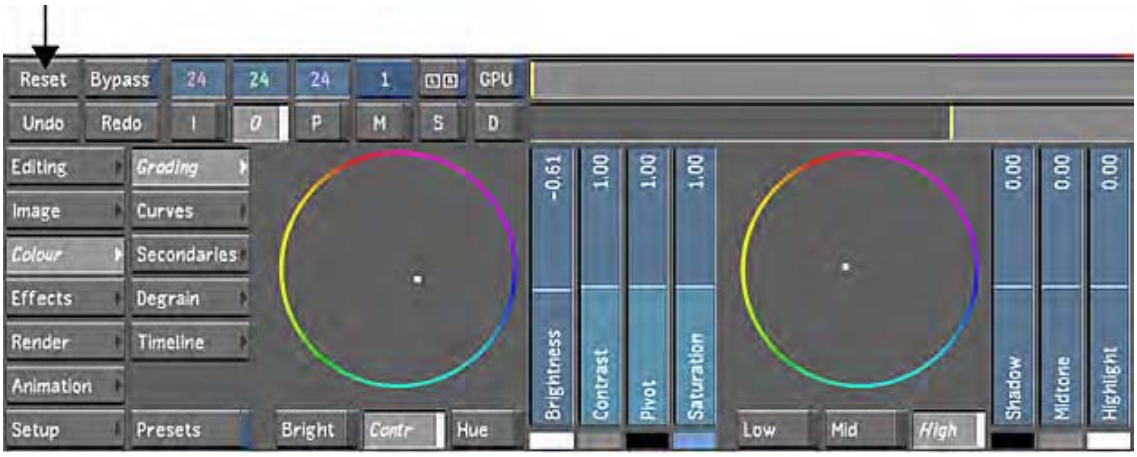
To reset a parameter:

1. After making a change to a parameter, hold down the `Ctrl` key and then click the parameter (field, graph, or option).

   The parameter is reset.

To reset all parameters for a menu:

1. Select the shot you want to reset.
2. Access the menu you want to reset.
3. Click Reset, and then confirm.



The menu is reset.

To reset a single shot to its original state:

1. Select the shot that you want to reset.
2. Press the `R` key twice.

   TIP You can also double-click an unused storage container in the Grade bin to remove any grade or repositioning from the selected shot.

   The shot is reset to its original state.

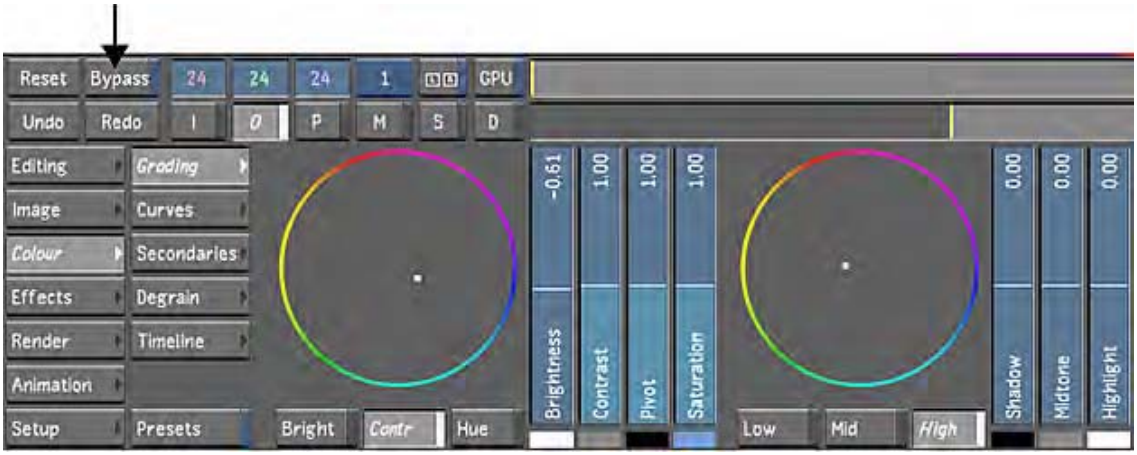
Bypassing Menu Parameters

You can bypass the parameters of a single menu or all menus to view a shot as if the menu or menus had not been modified. Do this when you want to compare the original shot to the modified shot. Bypassing differs from resetting in that you can toggle Bypass on and off, whereas you have to undo a reset operation to return to the previous state. By toggling Bypass off and on, you can easily compare the original shot to the modified one. You can bypass the following menus either singly or simultaneously:

- Grading
- Curves
- Secondaries
- Effects

To bypass menu settings:

1. Load the selected modified shot into the Player.
2. To bypass a single menu's parameters, click Bypass.



3. To bypass all menu parameters, do one of the following:
   - `Shift`-click Bypass.
   - Press `Shift+B`.

   Changes made to the current menu are temporarily turned off.
4. Enable Bypass again to turn the menu back on.

In a stereoscopic project, you can use the Bypass feature on the Left Eye and Right Eye layers. See Bypassing Menu Parameters of an Eye (page 2215).

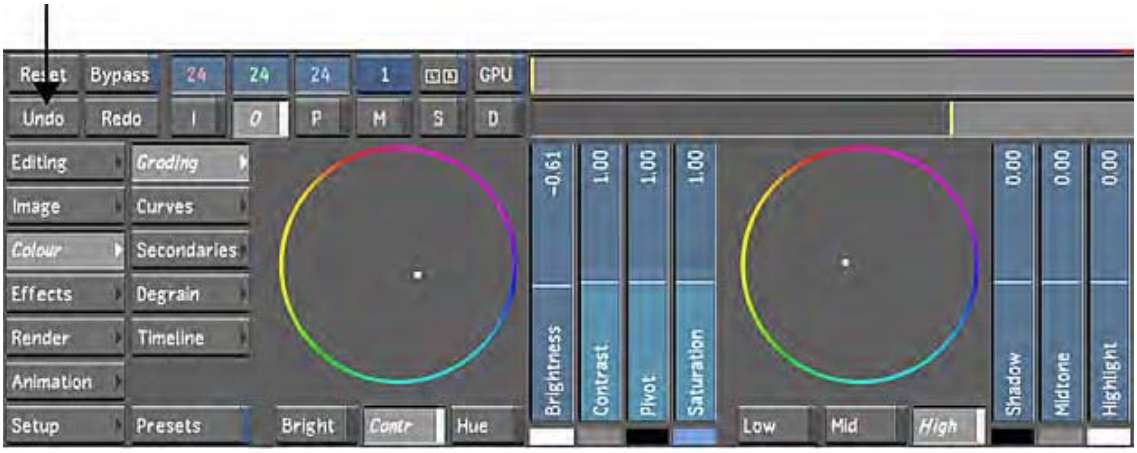
Undoing and Redoing Actions

Each shot in Lustre has its own undo and redo history.

You can undo most operations in Lustre. You can use Redo immediately after undoing an operation.

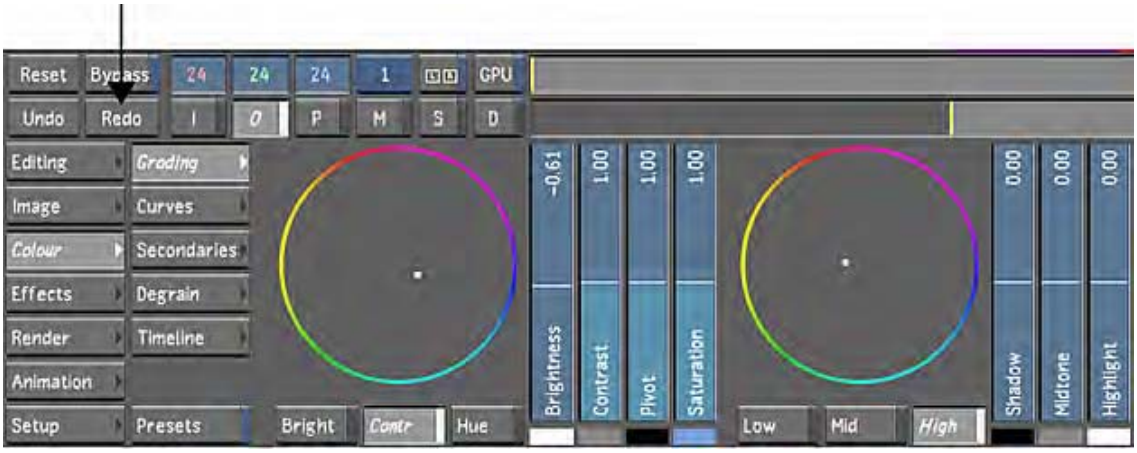
To undo an operation:

1. After performing an operation that you want to undo, click Undo.



To redo an operation:

1 After undoing an operation, click Redo.

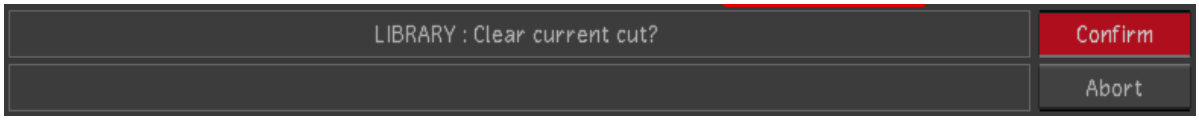


Confirming and Cancelling Actions

There are several operations in Lustre that must be confirmed before they begin. For example, when rendering, capturing, playing out, or deleting material, you must first confirm the action.

When you launch an action that must be confirmed, the Message Bar will show a message asking you to confirm or cancel the action and a red Confirm button along with a grey Abort button will appear to the right of the Message Bar. Click the appropriate button to confirm or abort the action.

NOTE The Message Bar is exclusive to Lustre 2012 Extension 1. In prior versions of Lustre, there is simply a Confirm button.



TIP You can also click anywhere in the grey area of the menu to cancel the action.

Assigning Notes to Shots

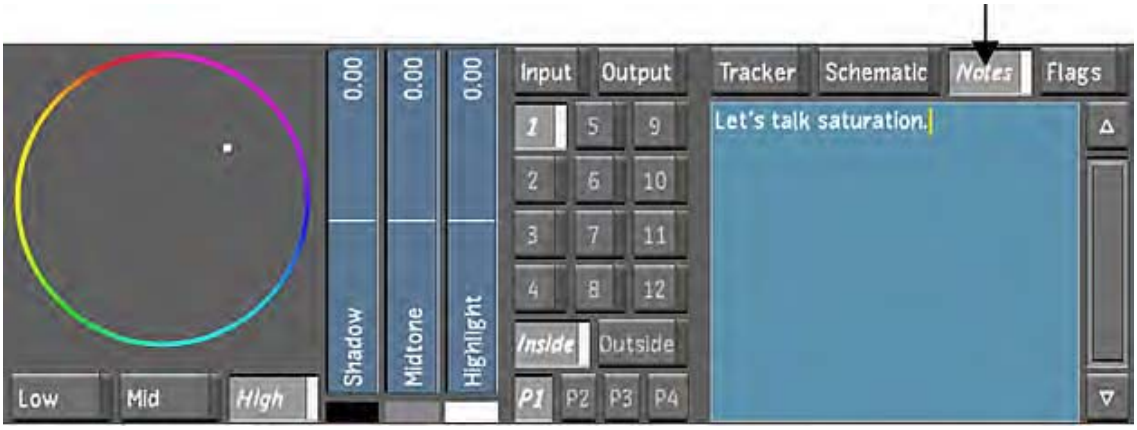
The Notes window is available from almost every menu and allows you to enter unlimited wrapping text pertaining to the current shot. Notes are retained as you jump from one shot to another, and saved when you save the grade.

Notes are saved to an .XML file in the scene's *Library* directory. The file is identifiable by a *.desc.xml* extension. You can open the DESC file in a text editor to view and make changes that will be reflected in Lustre the next time you load the grade. This could be useful if, for example, you wanted to review the notes for each shot between grading sessions. You would copy the file to a laptop, and then copy it back to the scene's *Library* directory if any changes were made. See XML in Lustre (page 2362).

When editing the DESC file, only the Notes field should be changed. Editing any other field could produce unexpected results.

To use the Notes window:

1 Enable the Notes button.



This displays the Notes window below the Notes button.

You can use any of the following standard text editor functions when drafting notes.

Press:	To:
`cursor arrows`	Move up, down, left or right.
`Backspace, Delete`	Delete text.
`Home`	Jump to the start of the current line.
`Ctrl+Home`	Jump to the start of the text.
`End`	Jump to the end of the current line.
`Ctrl+End`	Jump to the end of the text.
`Page Up`	Jump one text screen up.
`Page Down`	Jump one text screen down.

Press:	To:
`Shift+cursor arrow` or `left mouse button-drag`	Select text.
`Ctrl+A`	Select all text.
`Enter`	Add an empty line.
`Esc` or click outside the text editor	Disable the text editor.

NOTE Lustre hot keys are not functional while the Notes text editor is enabled.

2 Save the scene's notes by saving the grade. See Creating a Grade Version for a Cut (page 1777).

Flagging Shots

The flag system is accessible from almost any menu and allows you to enable and disable flags, create and delete flags, and copy flag states to one or more shots.

NOTE When in the timeline or the colourist's timeline, the flag system is always displayed to the right of the timeline canvas.

System flags are available by default for all shots. There are five types.

Render Flags the shots to be rendered.

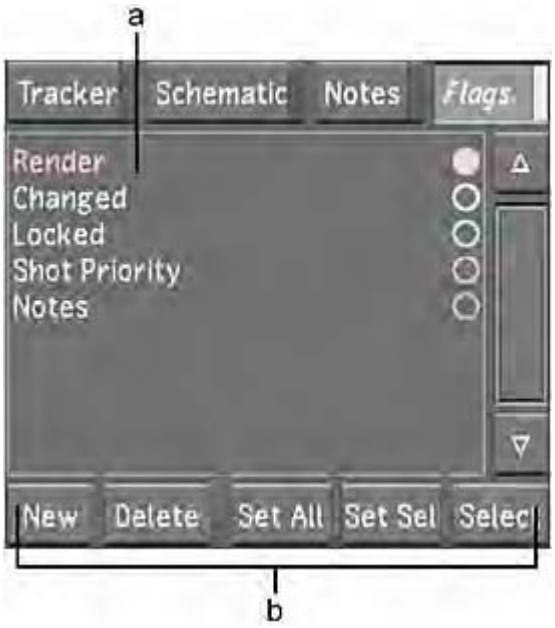
Changed Indicates whether or not the current shot has been modified since the last save.

Locked Locks the current shot to prevent modifications. The shot's locked status is also indicated by the word Locked that spans the three printer light fields.



Shot Priority Assigns priority to the current shot in the timeline. See Shot Priority (page 1960).

Notes Denotes whether notes exist against the current shot. Unlike the other system flags, the Notes flag cannot be turned on or off. Instead, its state is automatically updated depending on the contents of the Notes window. The Notes flag is set only when content is added to the Notes window. See Assigning Notes to Shots (page 1807).



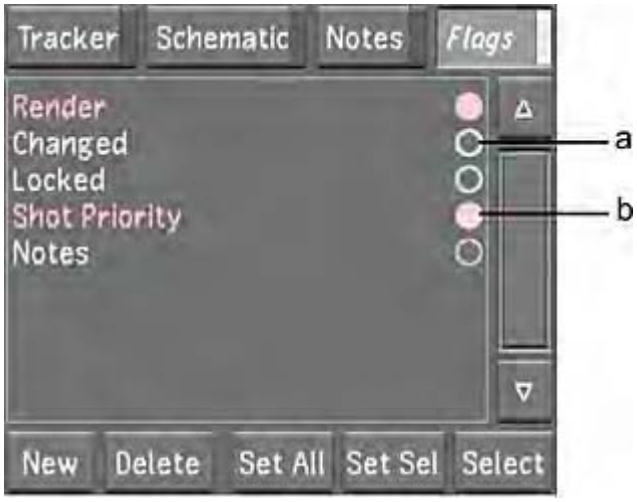
(a) Flags list (b) Flag controls

To access the Flags window:

1 Enable the Flags button.
This displays the flags for the current shot.

To enable or disable a flag:

1 Right-click the flag row.
When the flag is enabled, the flag name is in colour and its circle is filled.



(a) Disabled flag (b) Enabled flag

TIP To select a flag without enabling or disabling it, left-click the flag row.

Custom Flags

Apart from the five system flags, you can also create your own custom flags. When creating a custom flag, you must adhere to the following:

- You cannot rename a system flag, nor can you assign the name of a system flag to a custom flag.
- Spaces are not supported in the naming convention. It is recommended that underscores be used instead of spaces when you are working on either Linux or Windows.

Custom flags are saved to the *custom.flags* file in the Lustre version folder. Consequently, the custom flags are applied globally on all projects. After you create a custom flag, newly created grades save the *custom.flags* data to the *grdXX.desc.xml* file in the project's *Library* folder.

To create a custom flag:

1. Click New under the Flags window.
   A new flag is added to the bottom of the list.
2. To rename the custom flag, middle-click it and enter a new name.

To delete a custom flag:

1. Select the flag and click Delete under the Flags window.

   NOTE System flags cannot be deleted. When deleting custom flags, only one can be selected at a time.

To copy a custom flag's state to all shots:

1. Select the Custom flag (the selected flag will be highlighted).
2. Click Set All in the Flag controls.

To copy a custom flag's state to one or more shots:

1. Select the Custom flag (the selected flag will be highlighted).
2. Select the shots in the Storyboard or Multi-Layer Timeline by doing one of the following:
   - Right-click the shots you wish to flag in the Storyboard.
     In the Storyboard, the borders of the selected shots will turn aqua in colour.
   - Select the shots you wish to flag in the Multi-Layer Timeline. See Selecting Elements (page 1937).
     In the Multi-Layer Timeline, the selected shots will turn yellow in colour.
   - Create a shot group, then select the group. See Creating Shot Groups (page 1810).
3. Click Set Sel.

Creating Shot Groups

You can group shots together. This is useful, for example, when you have a series of exterior shots and a series of interior shots in the Storyboard. You can define a group for the exterior shots and another group for the interior shots. When you want to adjust parameters for the exterior shots, you select that group. Then, when you want to work on the interior shots, you select the Interior Shots group. Each time you select a different group, the selected shots in the Storyboard change.

In addition to being able to group shots, you can also define what parameters are relevant for the group. In the Exterior Shots group example, you may want to group Primary colour and Effects, but not Secondary

colour. This allows you to perform primary colour grading on all shots in the group, but secondary colour grading is applied only to the current shot.

If, at any point, you need to apply changes to a parameter defined as part of a group, but you want the changes to apply to the current shot only, use Solo. This function temporarily isolates the shot for modification—your changes are not applied to other shots in the group.

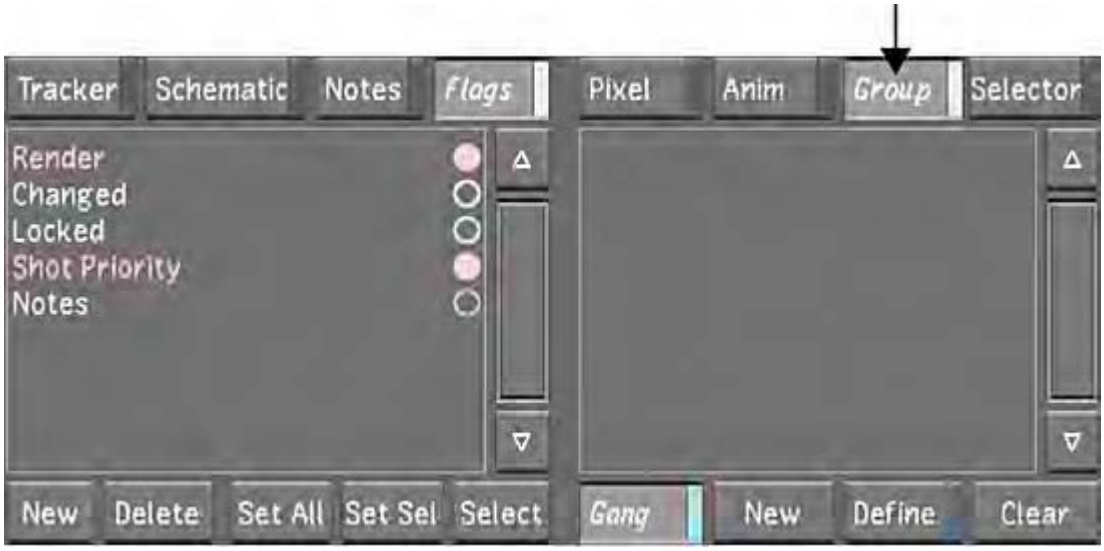
Grouping Shots

Use the Group tool to select shots to group and the parameters to include with the grouping. Any change made to one shot in the group propagates to the other shots in the group, provided that the changes are to parameters that have been included in the group.

Changes made to grouped shots are relative, not absolute. For example, if a non-grouped shot's printer light values are 25, 25, 25 and a new setup is loaded in which the printer light values are 37, 28, 10, then the shot's printer light values will be changed to 37, 28, 10. On the other hand, if the shot were part of a group, the new printer light values would be added, resulting in printer light values of 62, 53, 35. When in group mode, values are added to all the grouped shots, even those that have already been colour corrected.

To create a shot group:

1 Click Group.



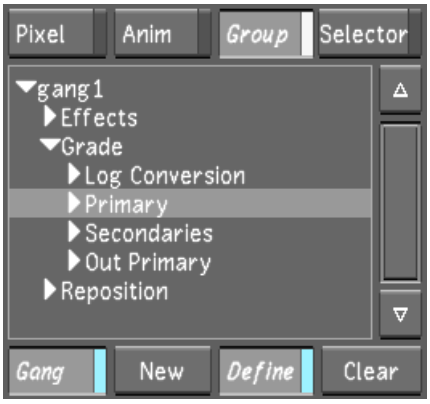
NOTE You can access the Group button from any menu except the Animation menu.

2 Click New.

A new group called gang1 is created.

3 Enable Define.

This allows you to add shots to the shot group and select the parameters that are applied to the grouping.

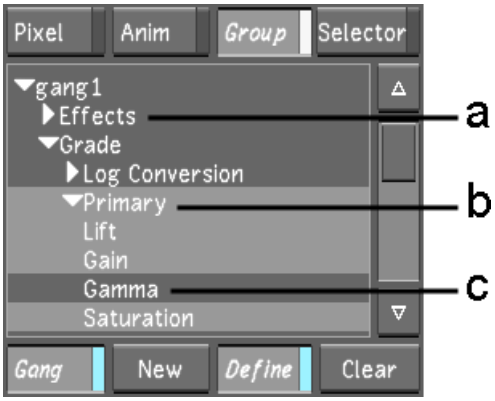


4 Right-click shots in the Storyboard to add them to the group.

Grouped shots are highlighted with a yellow border.

5 Select the channels in the list for the parameters that you want to apply to the group:

- To select all channels in a directory, select the directory.
- To select a subset of the channels in a directory, click the arrow to the left of the directory to expand the directory, and then select the channels.
- To deselect a directory or a channel, click it.



(a) Deselected directory (b) Selected expanded directory (c) Selected channel

6 When you have finished setting up the group, disable Define.

7 You can name the group by middle-clicking the group and entering a new name.

NOTE Spaces are not supported in naming conventions. It is recommended that underscores be used instead of spaces when working on either Linux or Windows.

8 You can delete a group by selecting it and pressing `Delete`, while the cursor is placed over the group menu.

9 You can activate split view with grouped shots by right-clicking the group.

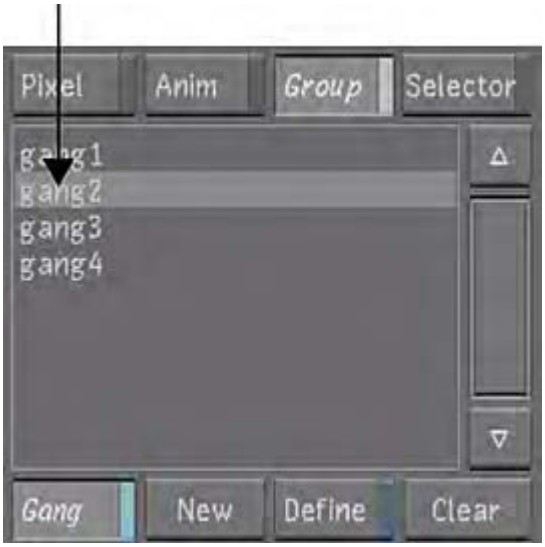
Applying Changes to Grouped Shots

After you group shots, changes made to one shot are applied to all other shots in the group.

To apply changes to all shots in a group:

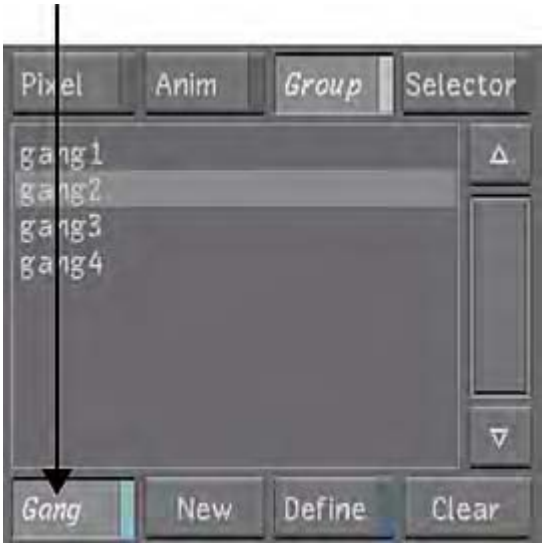
1 Click Group.

2 Select the group to which you want to make changes.



The grouped shots are selected in the Storyboard.

3 Enable the Solo/Gang button.



The Solo button turns to a Gang button and the grouped shots are outlined in yellow.

4 Select one of the grouped shots and make modifications to it.

NOTE You can also apply a grade to one of the shots.

The changes are applied to all the other grouped shots.

Soloing Shots

Use Solo to temporarily disable grouping for a shot.

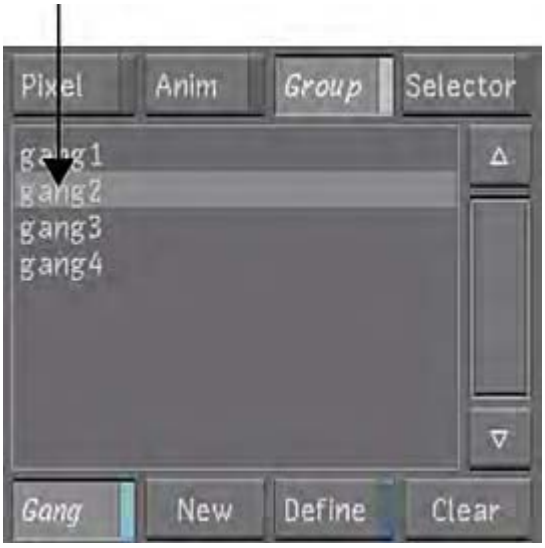
When Solo mode is enabled, you can only modify the current shot. You can copy grades (drag and drop from the Selector or from the Grade bin) only to the current shot. It is possible to copy a grade from any shot to the current shot.

In Solo mode, the outline around the ganged shots is now a muted orange.

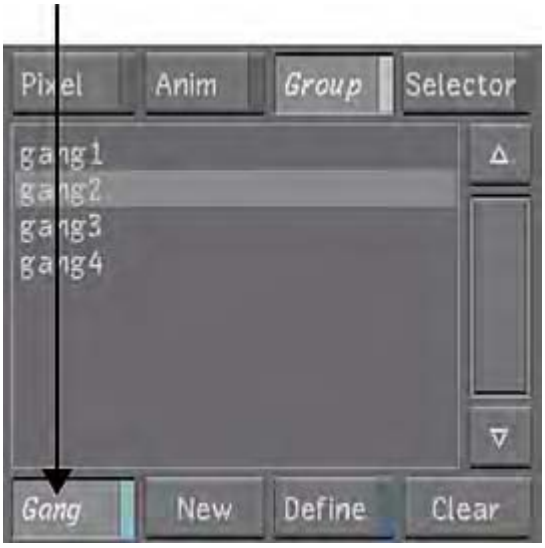
- Group (Gang mode): all shots from the gang can be modified.
- Group (Solo mode): only the current shot can be modified.

To solo shots in a group:

1. Click Group.
2. Select the group with the shot you want to solo.



3. In the Storyboard, select the shot to which you want to make changes.
4. Disable the Solo/Gang button.



   The Gang button turns to a Solo button.
5. Make the changes to the shot.
   The changes affect the soloed shot only.
6. Enable the Solo/Gang button to turn off solo.
   Any changes made to the shot are now applied to all grouped shots.

Soloing Shots

Use Solo to temporarily disable grouping for a shot.

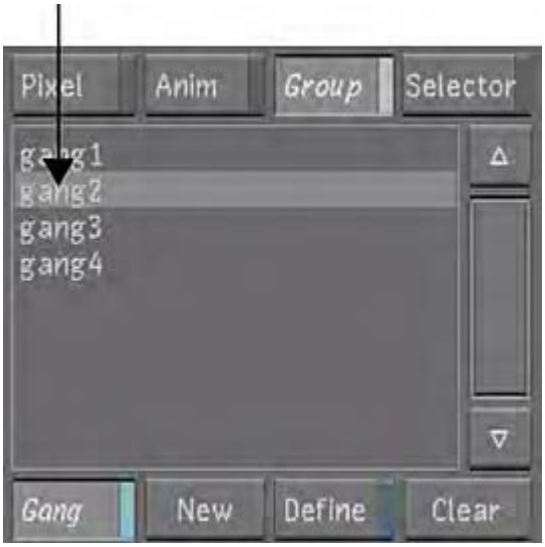
When Solo mode is enabled, you can only modify the current shot. You can copy grades (drag and drop from the Selector or from the Grade bin) only to the current shot. It is possible to copy a grade from any shot to the current shot.

In Solo mode, the outline around the ganged shots is now a muted orange.

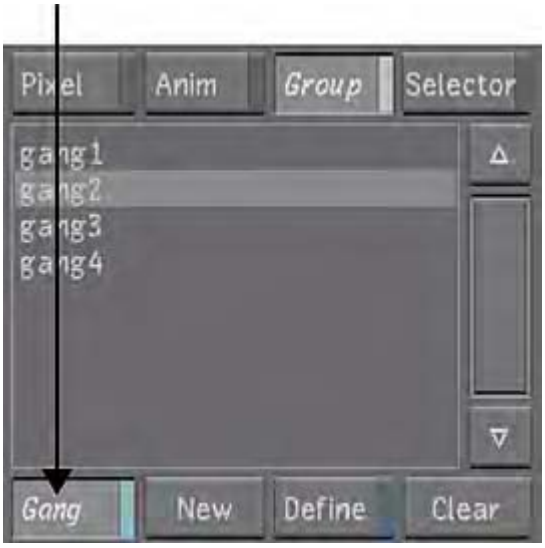
- Group (Gang mode): all shots from the gang can be modified.
- Group (Solo mode): only the current shot can be modified.

To solo shots in a group:

1. Click Group.
2. Select the group with the shot you want to solo.



3. In the Storyboard, select the shot to which you want to make changes.
4. Disable the Solo/Gang button.



The Gang button turns to a Solo button.

5 Make the changes to the shot.

The changes affect the soloed shot only.

6 Enable the Solo/Gang button to turn off solo.

Any changes made to the shot are now applied to all grouped shots.

Copying Parameters with the Selector

Use the Selector to copy channel parameters from the current shot to other selected shots in the cut. This feature is useful when you have a series of shots that need to have a similar grade applied to them, for example, shots that are too dark. You can adjust the brightness for the current shot, select all the shots, and then copy the grade setting to all other selected shots. For more information about using the Selector to copy grading information in the Multi-Layer Timeline, see Copying Grading Information in the Colourist Timeline (page 2055).

You can also use this feature to copy keyer information. For example, if you blurred a shot and adjusted its hue with the HLS Keyer, you can choose to copy only the hue change to the other selected shots; or, if you modified the tolerance and softness of a shot with the Diamond Keyer, you can choose to copy only the tolerance change to another selected shot.

You can copy the following parameters between shots using the Selector:

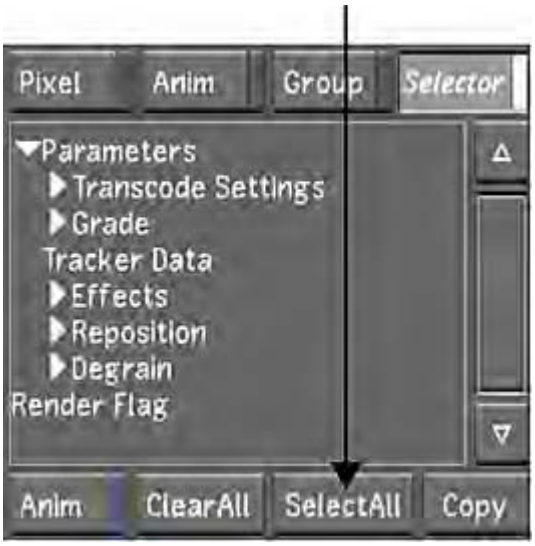
- Colour grading
- Input LUT (1D LUTs and 3D LUTs) selections
- CDL values
- Transcode Settings
- HDRx Settings (ARRIRAW, RED)
- Repositioning values
- Tracker data
- Effects
- Degrain settings
- Render flag settings
- Look Settings (ARRIRAW)

To copy channel parameters from the current shot to selected shots:

1 Click Selector.



2 Select your target shots by completing one of the following:
- To select your shots one at a time in the Storyboard, right-click specific shots in the Storyboard.
- To select your shots one at a time in the Timeline, left-click specific shots in the Timeline.
- To select all the shots in the Storyboard or the Timeline, click SelectAll in the Selector.



The target shots are highlighted in aqua.

3 Click the shot from which you want to copy parameters.

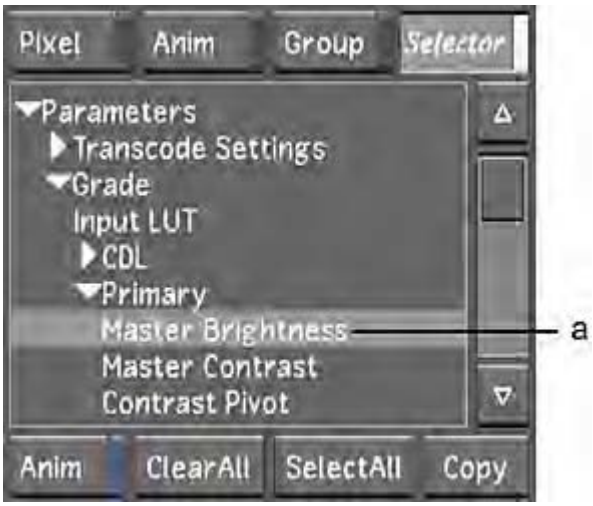
4 Specify what parameters to copy by selecting their channels in the Selector channel list:
- To select all the channels in a directory, select the directory.



(a) Collapsed directory (b) Expanded directory

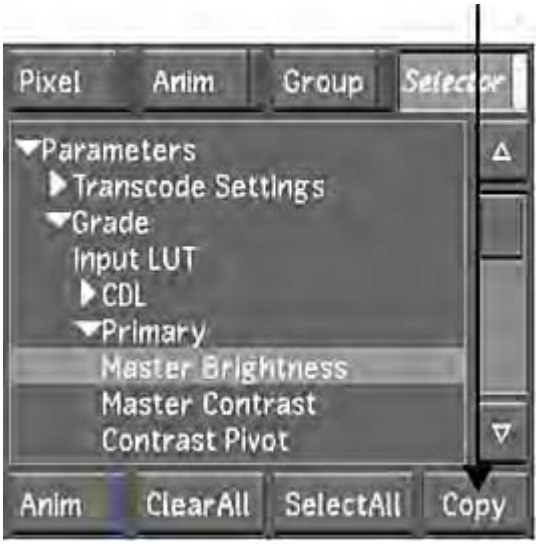
Selected channels are highlighted.

- To select specific channels in a directory, expand the directory and select any of its channels.



(a) Selected channel

5 Click Copy.



The parameters for the selected channel are copied to the selected shots.

Copying and Pasting Grading Parameters Between Grading Stages

You can copy or move grading parameters between the Input, Output and any of the Secondary stages with the following key strokes:

- To copy the parameters from one of the grading tabs (Colour/Grading, Colour/Curves or Colour/Secondaries) of the current grading stage (Input, Secondary or Output), to another stage, hold `CTRL`and click the destination stage (Input, Secondary or Output).
- To copy the parameters of all the grading tabs (Colour/Grading, Colour/Curves and Colour/Secondaries) of the current grading stage (Input, Secondary or Output), to another stage, hold `SHIFT` and click the destination stage (Input, Secondary or Output).
- To move the parameters of all the grading tabs (Colour/Grading, Colour/Curves and Colour/Secondaries) of the current grading stage (Input, Secondary or Output), hold `SHIFT + CTRL` and select the destination stage (Input, Secondary or Output). The source stage is reset to the initial values.
- To copy the parameters as defined by the Selector of the currently selected stage (Colour/Grading, Colour/Curves or Colour/Secondaries), hold `ALT` and select the destination stage (Input, Secondary or Output).

See:

Primary Colour Grading (page 2058)

Secondary Colour Grading (page 2083)

Assigning Values With the Standard Lustre Calculator

Use the standard Lustre calculator to assign values to sliders and certain fields. You can display the calculator by right-clicking any of the following user interface elements.

Sliders Adjust the value of the currently selected slider.

Zoom field Adjusts the zoom value of the currently selected shot.

Printer Lights Adjust the value of the colour wheel via the printer lights. The calculator displays only after you have already left-clicked in a printer light to display colour wheel values instead of the printer lights. Then, when you right-click a colour wheel value, the calculator appears.

NOTE When you right-click in the timeline Information field, the timecode calculator is displayed. The timecode calculator is used primarily for navigating the timeline. See Navigating With the Timecode Calculator (page 1977).



(a) Active slider (b) Calculator

NOTE The calculator's default value is always zero.

The calculator features a numeric keypad and buttons for performing standard operations such as addition, subtraction, multiplication, and division. To enter numbers, you can click the calculator buttons, or press the keyboard's number keys or numeric keypad.

It is also possible to assign a negative value to a slider with the calculator.

To assign a negative value to a slider with the calculator:

1. Right-click a slider to display the calculator.
2. Type '-' on the keyboard.
3. Use the calculator to enter the value.

   WARNING You must enter the complete number (e.g., 0.654 and not .654).

4. Press `Enter`.

The following hotkeys can be used with the calculator.

Press:	To:
`Backspace`	Erase the last digit in the calculator's numeric field.
`Delete`	Reset the calculator's numeric field to zero.
`Page Down`	Change the sign of the value in the calculator's numeric field.
`Up cursor arrow`	Add the current calculator value to the slider or field value.
`Down cursor arrow`	Subtract the current calculator value from the slider or field value.
`Enter` or click the calculator's active numeric field	Calculate the value, apply it, and close the calculator.

Press:	To:
Enter or click the calculator's = (equals) button	Calculate the value and display the result without applying it.
Ctrl+Enter or right-click the calculator's = (equals) button	Calculate the value, display the result, and apply it.
Esc or click outside the calculator	Close the calculator and leave the slider or field value unchanged.

Using Grade Bins

Store shot settings to a Grade bin when you need grades for future reference. Intermediary grades are useful when you want to experiment with different looks. Grades are saved on a shot-by-shot basis and contain all shot and animation settings.

The location of grade files determine how they can be shared between users scenes, and projects.

Grade bin type:	Contains:
Global	Grades saved for use between all users and projects on the current workstation.
Project	Grades saved for the current project only.
Scene	Grades saved for the current scene only.
User	Grades saved for the current user only.
Quick	Grades saved in a user-defined folder. See Defining the Quick Folder (page 1835).
Marry	Grades saved as Marry grade files. See Using the Marry Grade Bin (page 1821).

TIP Place the cursor anywhere outside the file browser or expanded Grade bin, and press Shift+– to cycle through the display of different types of Grade bins.

You can also use Grade bin thumbnails as reference images. See Viewing Reference Images (page 1970).

Using the Marry Grade Bin

Unlike the other Grade bins, which are used to apply grading to any shot(s) in a given timeline, a Marry Grade bin is shot-specific. Each shot in the cut has its own dedicated Grade bin for its Marry grade files. For example, if a cut is composed of three shots, three Marry Grade bins exist, although the only visible Marry Grade bin is the one for the current shot. This allows you to save multiple Marry grades to the same shot.

Marry Grade file information is also displayed in the Grade view. Marry grade files are saved with their source media files (Marry grade files for media imported from Wiretap and Wiretap Gateway are saved with local links to the shots). Enable the Marry button under the file browser to navigate to the file folder in which Marry grades are located. See Using the Expanded Grade Bin (page 1828).

You can also use the Marry grade controls in the Setup > Grade menu to save Marry grade files into the Marry Grade bin. See Saving and Loading Marry Grades (page 1784).

Saving Grades

You can save grades to the Grade bin. When you save grades, all the settings are saved with a high-resolution PNG image and a thumbnail for identification. By saving your shot settings to the appropriate Grade bin, you can save your shot settings globally, to a predefined folder, or limit them to a user, scene, or project. You can also save them with your shots as Marry grade files.

If you save grades to a global, project, scene or user Grade bin, each selected grade is saved in a separate storage container, starting from the selected container and proceeding column by column and left to right.

If you save grades to a Marry Grade bin, each selected grade is saved in a separate storage container in the current container position (highlighted in red) in the Grade bin for the shot.

To save a grade to a Grade bin:

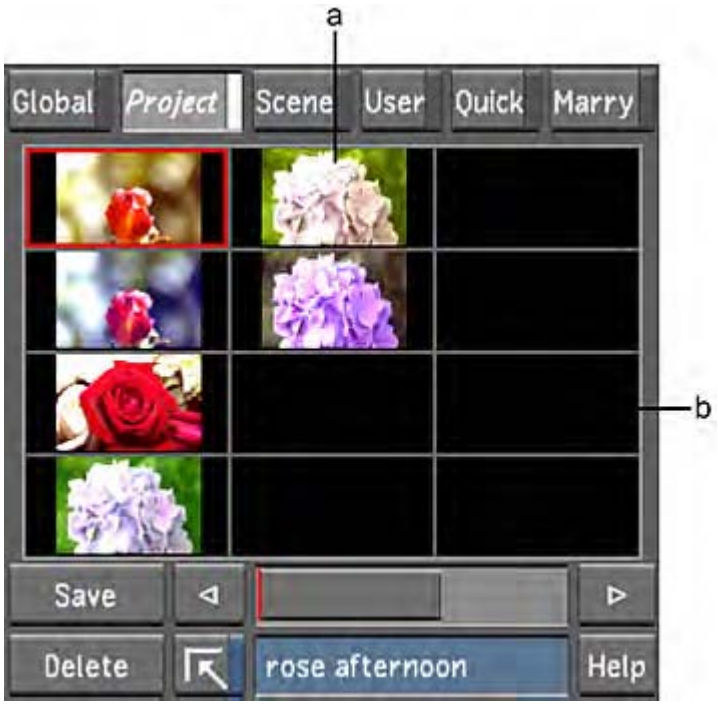
1 Select the Grade bin you want to use (for example, the Grade bin for the current project).



2 (Optional) Scroll to the area of the Grade bin you want to use. To scroll through a Grade bin, middle-click and drag right or left, or use the horizontal slider located immediately below the Grade bin. The red line in the slider indicates the current position.

3 Do one of the following:

- To create a grade, select an unused storage container.
- To update a grade with modified settings, select the storage container of the grade you want to update.



(a) Used storage container (b) Unused storage container

NOTE When you add a grade to a container in the last column of the Grade bin, three new columns are automatically added to the right. You can add an unlimited number of grades.

4 Click Save.

The settings from the current shot in the timeline are saved to the Grade Bin and a thumbnail of the current frame appears in the Grade bin.

TIP You can use the thumbnail as a reference image while you work on another shot. See Viewing Reference Images (page 1970).

5 (Optional) Enter a name or comment in the Note field and press `Enter`.



To save grading information from several shots to the Grade bin:

1. Do one of the following:
   - To choose grades for specific shots, use the Storyboard or the Multi-Layer Timeline to select the shots whose grading information you wish to save to the Grade bin. For more information about selecting elements in the Multi-Layer Timeline, refer to Selecting Elements (page 1937).
   - To choose all shots in the timeline, leave all shots unselected.
2. In the Grade bin, select the storage container where you wish to save your grade.
3. Hold down the `Shift` key while pressing the Save button.

   WARNING Storage containers with existing grading information will be overwritten. If you are saving to storage containers that already have grading information, Lustre prompts you to confirm you want to overwrite these grades.

4. If you are saving to storage containers that already have grading information, confirm or cancel the action when Lustre prompts you to confirm the action.

   The selected grades are saved to the Grade bin.

Applying Grades to Shots

You can apply grades to one or multiple shots in the Storyboard or timeline. Apply intermediary grades to your shots when you want to experiment with different looks. You can control which saved settings you want to apply—all the settings from the intermediary grade, only those selected in the Selector, or only those in the current menu.

To load intermediary grades from a Grade bin:

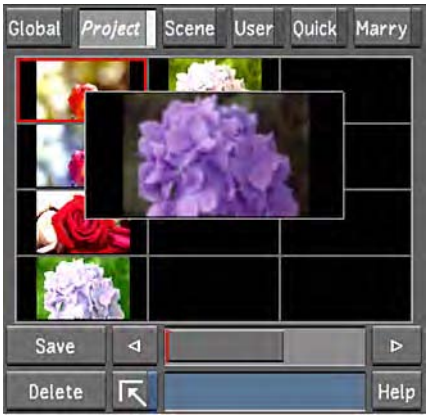
1. In the Storyboard, select the shots to which you want to apply an intermediary grade.

TIP You can select multiple shots by right-clicking them in the Storyboard. You can also create groups of shots. See Selecting Shots in the Storyboard (page 1980) and Creating Shot Groups (page 1810).

2 Select the Grade bin you want to use.

NOTE If you select the Marry Grade bin, each shot that you select will be loaded with the corresponding Marry grades for that specific shot. See Using the Marry Grade Bin (page 1821).





TIP To display an enlarged image of the thumbnail, place your mouse cursor over a thumbnail without clicking.



3 Do one of the following:

Double-click a shot's thumbnail

- To load all the settings in the grade to the current shot only.

  TIP Double-click an unused storage container in the Grade bin to reset selected shots to their default settings.

Press Shift and double-click a shot's thumbnail

- To load all the settings in the grade to selected shots.

Press Ctrl and double-click a shot's thumbnail

- To load the grade settings to the current shot only from the current menu only. For example, if the Grading menu is displayed, the saved settings from the Grading menu are applied.

Press Ctrl+Shift and double-click a shot's thumbnail

- To load the grade settings to selected shots from the current menu only. For example, if the Grading menu is displayed, the saved settings from the Grading menu are applied.

Press Alt and double-click a shot's thumbnail

- To load the grade settings that are selected in the Selector to the current shot only. See Copying Parameters with the Selector (page 1816).

Press Alt+Shift and double-click a shot's thumbnail

- To load the grade settings that are chosen in the Selector to a selection of shots. See Copying Parameters with the Selector (page 1816).

The grade is applied. In the Player, the image is updated to reflect the current settings.

TIP To revert to the previous settings, click Undo.

Within the Storyboard view, you can drag and drop the grading from one shot to another. You can reapply the grading in one or more different shots without having to drop it in the Grade bin first.

To drag and drop the grade from one shot to another:

1 Do one of the following:
- Hold down `Ctrl+Shift` and drag and drop to copy grades to multiple destinations.
- Hold down `Ctrl+Alt` and drag and drop to use the Selector to copy grades to a single destination.
- Hold down `Ctrl+Shift+Alt` and drag and drop to use the Selector to copy grades to multiple destinations.

Moving Grades

You can reorganize the Grade bin by moving grades from one storage container to another. You can also drag a Grade bin to the file browser or expanded Grade bin in the Grade view. See Moving Grade Files Between Grade Bins (page 1835).

Deleting Grades

You can delete grades one at a time, or delete all grades from the Grade bin simultaneously.

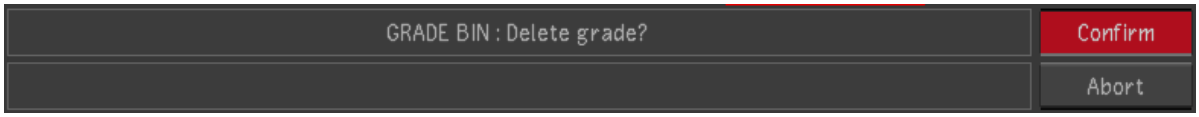
To delete grades from a Grade bin one at a time:

1 Select the Grade bin from which you want to delete grades. The name of the selected thumbnail appears in the Note field.



2 The Message Bar will show a message asking you to confirm or cancel the action and a red Confirm button along with a grey Abort button will appear to the right of the Message Bar. Click the appropriate button to confirm or abort the action.

NOTE The Message Bar is exclusive to Lustre 2012 Extension 1. In prior versions of Lustre, there is simply a Confirm button.



To delete all grades from the Grade bin simultaneously:

1 Select the Grade bin from which you want to delete grades.



2 Hold down `Shift` while clicking Delete.

The Message Bar will show a message asking you to confirm or cancel the action and a red Confirm button along with a grey Abort button will appear to the right of the Message Bar. Click the appropriate button to confirm or abort the action.

NOTE The Message Bar is exclusive to Lustre 2012 Extension 1. In prior versions of Lustre, there is simply a Confirm button.

Using the Expanded Grade Bin

Use the expanded Grade bin to locate and display grade files. You can manage intermediary grade files at the global, project, scene, or user level, and Marry grade files at the shot-based level. You can also define a custom folder and bookmark it for quick access.

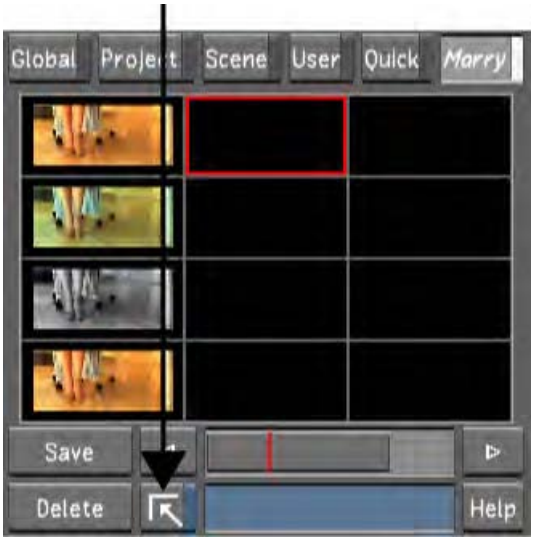
In addition to move, copy, and delete operations, you can create new folders, enable a player to view grades and shots, and work in different views to view grade files.

Accessing the File Browser

Use the file browser to locate grade data and load it into the Grade bin.

To access the file browser:

1 In the Grade bin, click the Expand button.



Footage courtesy of EVN PRODUCTIONS

The Grade view appears.



Image courtesy of EVN PRODUCTIONS (a) File browser (b) Expanded Grade bin (c) Player

When you are in the Grade view, the work area is divided into multiple areas.

File browser Use to locate grade data and display it in the expanded Grade bin.



Click:	To:
Global	Display the location of the global Grade bin folder and its contents.
Project	Display the location of the project Grade bin folder and its contents.
Scene	Display the location of the scene Grade bin folder and its contents.
User	Display the location of the user Grade bin folder and its contents.

Click:	To:
Quick	Display the location of the folder selected with the Set Quick Folder option.
Marry	Display the location of a Marry Grade bin folder and its contents. See Using the Marry Grade Bin (page 1821).
Set Quick Folder	Set the currently selected folder in the File Browser as the Quick Folder.
New Folder	Create a new folder under the currently selected folder. Enter a new name. Rename the folder at any time by right-clicking it and entering a new name (the global, scene, project, and user directories, cannot be renamed).
Delete Folder	Delete the currently selected folder (for which you have deletion permissions enabled). The Message Bar will show a message asking you to confirm or cancel the action and a red Confirm button along with a grey Abort button will appear to the right of the Message Bar. Click the appropriate button to confirm or abort the action. **NOTE** The Message Bar is exclusive to Lustre 2012 Extension 1. In prior versions of Lustre, there is simply a Confirm button.

TIP Place the cursor over the file browser or expanded Grade bin, and press `Shift+-` to cycle through the display of different expanded Grade bins.

Expanded Grade bin Use to display and organize the grade files.



Footage courtesy of EVN PRODUCTIONS

Click:	**To:**
Details	Display grade files in Thumbnail and Text view.
Proxies	Display grade files in Thumbnail view.
List	Display grade files in Text view.
Player	Display the contents of the expanded Grade bin and the Player. Disable button to display the expanded Grade bin only. Press Q to alternate the display between a large Storyboard view and the Player.

Click:	To:
Select All	Select all grade files in the expanded Grade bin.
Unselect All	Deselect the selected grade files in the expanded Grade bin.
Refresh	Rescan the file systems and update the file browser and expanded Grade bin with up-to-date information.
Delete Selected	Delete selected grade files.
Delete All	Delete all grade files from the expanded Grade bin. The Message Bar will show a message asking you to confirm or cancel the action and a red Confirm button along with a grey Abort button will appear to the right of the Message Bar. Click the appropriate button to confirm or abort the action. **NOTE** The Message Bar is exclusive to Lustre 2012 Extension 1. In prior versions of Lustre, there is simply a Confirm button.
Player	Show/hide a player in the Grade view.

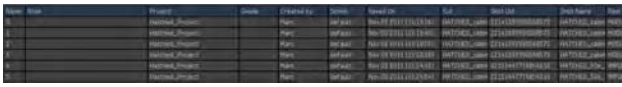
Footage courtesy of EVN PRODUCTIONS



Details

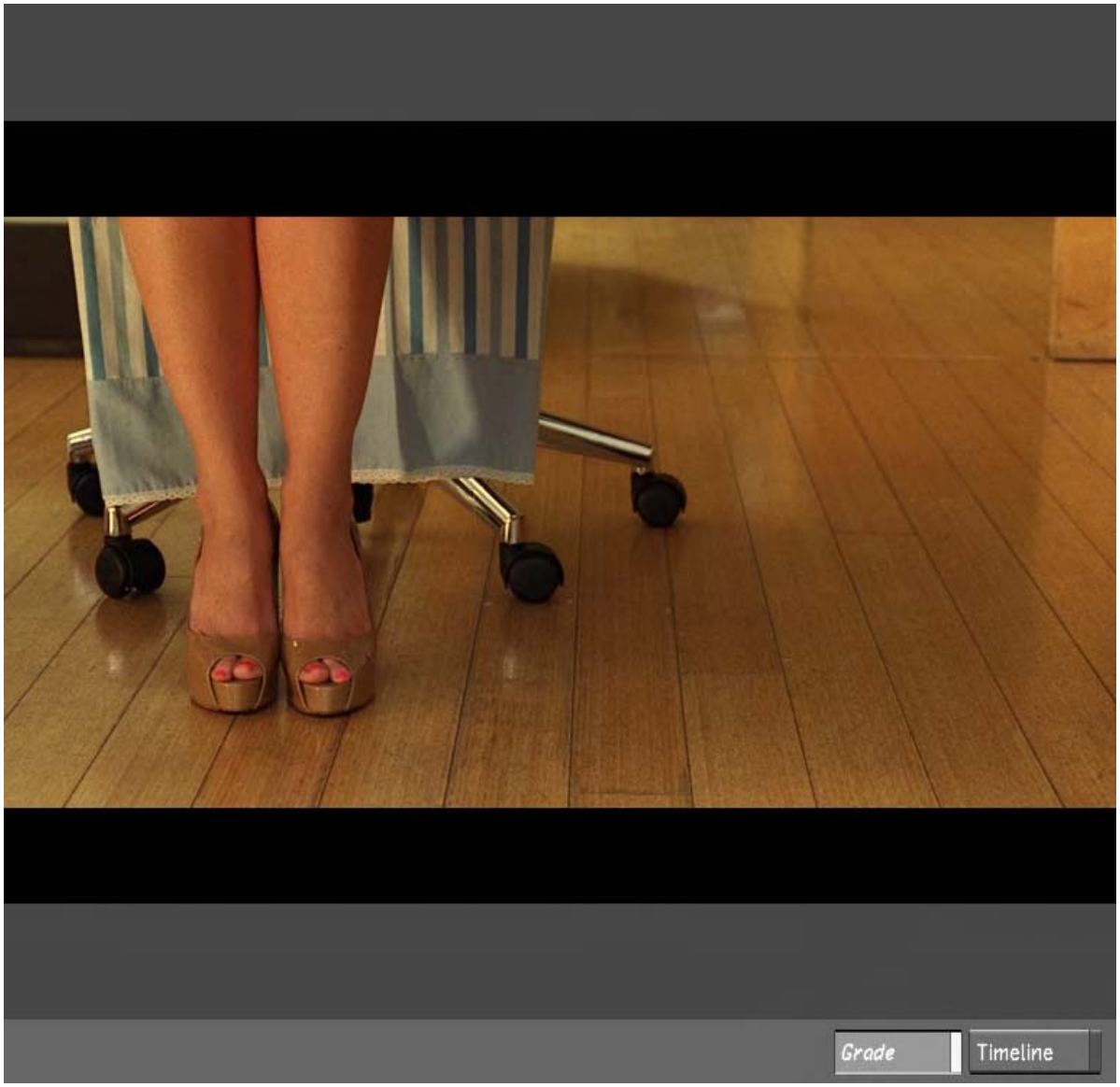


Proxies



List

Player Use to display selected shots and grade files.



Footage courtesy of EVN PRODUCTIONS

Click:	To:
Grade	Display the current grade selected in the expanded Grade bin.
Timeline	Display the current shot selected in the Timeline or Storyboard.

NOTE When in Timeline mode, the Player displays the shot from the timeline with its current grade applied to it, which is not necessarily the grade selected in the Expanded Grade Bin.

Defining the Quick Folder

You can use the file browser to select any folder and define it as a location in which you want to store grade files. Once you have stored the location of this folder, you can quickly display and select it in the file browser, and use the Quick buttons to display its contents in the Grade bin and expanded Grade bin.

To set the Quick folder:

1. Navigate to the folder you want to set as your Quick folder.
2. Click the name of the folder to select it.
3. Click Set Quick Folder.

Loading Grade Files from the Expanded Grade Bin

Once you locate your Grade bin folder in the file browser and display its contents in the expanded Grade bin, you can move grade files into the Grade bin storage containers. Grade files in the expanded Grade bin are pointers to the actual files on the file system. You can view information about the grade files in expanded Grade bin.

Moving Grade Files Between Grade Bins

In addition to moving grade files from the expanded Grade bin to a storage container, you can use drag-and-drop operations to move and copy between the expanded Grade bin, Grade bin storage containers, and file browser.

NOTE To copy a grade file from one location to another, hold down `Ctrl` as you drag files from the source Grade bin to its destination.

Drag from:	Drop in:
Expanded Grade bin	■ A folder in the file browser ■ A Grade bin storage container
Grade bin storage container	■ Another Grade bin storage container ■ An expanded Grade bin ■ A folder in the file browser
File browser	Illegal operation: Using a drag-and-drop operation, the file browser can only have a grade file moved or copied to it.

Applying Grade Files to Shots in a Cut

You can select grade files in the expanded Grade bin and apply them to a shot in the Storyboard or Timeline.

To apply grade files from the expanded Grade bin to a shot:

1. Do one of the following:
   - In the Proxies or Details view, click the grade file.
   - In the List view, click the grade name.

   The cursor picks up the grade file. If you click more than one grade file, only the first grade file will be applied.
2. Move the cursor to the Storyboard or Timeline.

   If the shot is moved to the Storyboard, an orange highlight appears around the selected shot.



3. Click to drop the grade file on the shot.

TIP Double-click an unused storage container in the Grade bin to reset selected shots to their default settings.

Deleting Grades From the Expanded Grade Bin

You can delete all or selected grade files from the expanded Grade bin.

NOTE Deleting grade files from the expanded Grade bin does not delete grade data from the Storyboard.

To delete selected grade files from the expanded Grade bin:

1. Select the grade files you want to delete.
2. Click Delete Selected, and then confirm the action.

   All the selected grade files are deleted.

To delete all grade files from the expanded Grade bin:

1. Click Delete All.
2. The Message Bar will show a message asking you to confirm or cancel the action and a red Confirm button and a grey Abort button will appear to the right of the Message Bar. Click the appropriate button to commence or cancel the action.

   All the grade files are deleted.

Grading Presets

From the Grade Bin's Global folder, you can access various grading presets that provide different looks you can use as a starting point for your grades.

The grading presets are organized into seven categories:

- **Color Filters:** 15 color filters, such as Color Temperature Blue, Color Temperature Orang, Cool, Warm, Sepia, etc.
- **Diffusion Filters:** 21 diffusion filters, such as Center Focus, Diffusion, Blur, Fog, etc.
- **Film:** 24 presets emultaion film lab processing and film stock, such as xProcess, Bleach Bypass, 2 Strips, 3 Strips, Very Old Film, etc.
- **Grad Filters:** 8 gradient filters, such as Blur, Green, Red, Amber, etc.
- **Image:** 28 image manipulation presets, such as Flip, Flip, Flip/Flop, Sharpen, Negative, Overexposed, Noise, etc.
- **Looks:** 38 look types, such as Day For Night, Film Noir, Expressionism, 8mm, Atomic Blast, etc.
- **Vignette:** 16 circle and rectangle vignettes with various highlight and blur values.

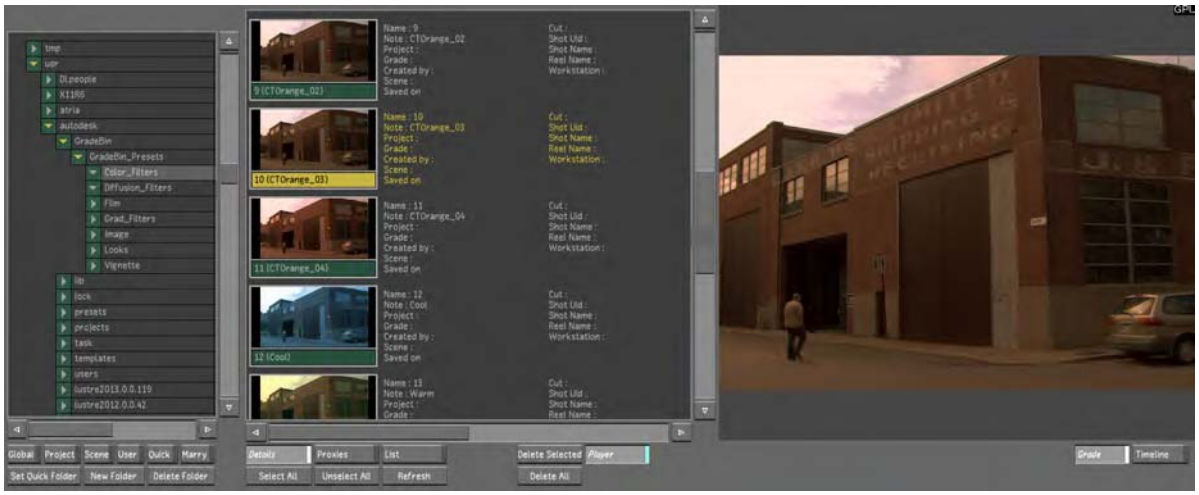
For the complete list of grading presets, see Grading Presets List (page 1838).

Accessing the Grading Presets

The grading presets are located in the Global Grade Bin folder, in a sub-folder named GradeBin_Presets.

To access the grading presets:

1. Navigate to the Golbal Grade Bin folder with the Expanded Grade Bin and expand the GradeBin_Presets folder to see the different preset categories, listed above.
2. Select a preset folder to see a thumbnail representation of the preset in the Expanded Grade Bin.
3. Double-click on the name of a preset thumbnail to see the preset in the Expanded Grade Bin Player.



TIP You can also preview the selected preset on the currently selected shot in the storyboard by selecting Timeline rather than Grade in the Expanded Grade Bin Player.

Using the Grading Presets

To use the grading presets, first copy them to either the Global, Project, Scene or User Grade Bin.

To copy grading presets to the Grade Bin:

1. Hold the Ctrl key on the keyboard.
2. Select one or more grading presets from the Expanded Grade Bin.

3 Drag & drop the grading presets to the Grade Bin of your choice (Global, Project, Scene, User).

The selected grading presets are now displayed in the Grade Bin.

To apply a grading preset to a shot:

1 Select the shot you want to apply the grading preset to in the Storyboard.
2 Double-click on the grading preset in the Grade Bin.

The grading preset is applied to the selected shot.

To edit a preset:

1 Apply the grading preset to the selected shot.
2 Modify the parameters.
3 Press the Save button at the bottom left of the Grade Bin, if you want to save the modified preset.

NOTE You can also load the entire contents of a Grade Bin Presets sub folder (Color_Filters, Diffusion_Filters, Film, etc.) by assigning it as a Grade Bin Quick Folder. However, this creates a symbolic link from the Grade Bin to the location of the grading presets. It does not copy them. This means that if you modify and save a modified grading preset, you are overwriting the original.

See Accessing the File Browser (page 1829).

See Applying Grades to Shots (page 1824).

Grading Presets List

The following tables list the complete grading presets available in Lustre. Some presets include multiple variations.

Color Filter Presets:	Description:
Clear	Resets the grading parameters.
CTBlue (01 to 04)	Color temperature presets that attempt to fix white balance issues (tungsten to daylight).
CTOrange (01 to 04)	Color temperature presets that attempt to fix white balance issues (daylight to tungsten).
Cool	Blue/green overall cast.
Warm	Yellow/orange overall cast.
Sepia	Soft brownish cast.
Blue	Soft blue cast.
Solar_Flare	Strong orange cast.
Neo	Greenish overall cast.

Color Filter Presets:	Description:
Purple	Soft purple cast.

Diffusion Filter Presets:	Description:
Clear	Resets the grading parameters.
Center_Focus (01 to 04)	Variations of a Circle shape with blur on the outside of the mask.
Diffusion_Filter_Blur	Soft blurry highlights.
Black_Diffusion	Soft blurry shadows.
Black_Diffusion_Mist	Heavy blurry shadows.
Diffusion_Glamour	Sublte diffusion preset.
Fog (01 to 04)	Variations of overall fog.
Blur (01 to 04)	Variations of blur.
High_Diffusion (01 to 04)	Variations of blurry highlights.
Heavy_Diffusion	Heavy highlights diffusion.

Film Presets:	Description:
Clear	Resets the grading parameters.
xProcess (01 to 04)	Four variation of cross processing photographic effect.
Bleach_Bypass	Bleach bypass color processing looks.
2_Strips	Emulation of the 2 strips color process.
3_Strips	Emulation of the 3 strips color process.
Kchrome	Film stock looks.
Red_Scale_100	Emulation of photographic processes involving exposing the back side of color negative.
Echrome	Film stock looks.
Infrared	Black and white film stock look.

Film Presets:	Description:
Night_Vision_Blue	Overall blue cast.
Very_Old_Film	Light black and white look with heavy grain.
BW_Aged	Old film with a yellow, green and blue cast.
BW_HighCon	High contrast black and white looks.

Gradient Filters Presets:	Description:
Clear	Resets the grading parameters.
Grad_Blue	Gradient shape with a dipping blue cast at the top of the image.
Grad_Blue_Green	Gradient shape with a dipping blue cast at the top of the image and a green cast at the bottom.
Grad_Amber_Green	Gradient shape with a dipping amber cast at the top of the image and a green cast at the bottom.
Grad_Amber_Purple	Gradient shape with a dipping amber cast at the top of the image and a purple cast at the bottom.
Grad_Red	Gradient shape with a dipping red cast at the top of the image.
Grad_Green	Gradient shape with a dipping green cast at the top of the image.
Grad_Sunset	Gradient shape with a dipping orange cast at the top of the image.
Grad_Sunrise	Gradient shape with a dipping purple cast at the top of the image.

Image Presets:	Description:
Clear	Resets the grading parameters.
Flip	Image is vertically inverted.
Flop	Image is horizontally inverted.
Flip_Flop	Image is both vertically and horizontally inverted.
Sharpen (01 to 04)	Variations of image sharpening.
Anamorphic	Converts anamorphic images to square pixels.

Image Presets:	Description:
Color_Negative	Color film negative look.
BW_Negative	Black and white film negative look.
Negative_Masked	Exposed color film negative look.
Vertical_Blur	Vertical blurring.
Horizontal_Blur	Horizontal blurring.
Photocopy	Variations of photocopy looks (color and black and white).
Overexposed	Variations of overexposed looks.
Noise	Variations of black and white looks with noise.
Moon	Moon in the sky created with shapes.
Stars	Stars in the sky created with shapes.
Moon_and_Starts	Moon and stars in the sky created with shapes.
Moon_ Starts_and_Clouds	Moon, stars and clouds in the sky created with shapes.

Looks Presets:	Description:
Clear	Resets the grading parameters.
Day_For_Night	Converts a day scene to a night scene.
Film_Noir	1950s drama look.
Expressionism	1960s drama look.
Matrix	Postmodern drama look.
Sixties_slides	Reversal slides look.
Berlin	1970s drama look.
Black_and_White	Black and white realistic look.
BW_Newspaper	Black and white look with grain.
8mm	Projected film stock look.

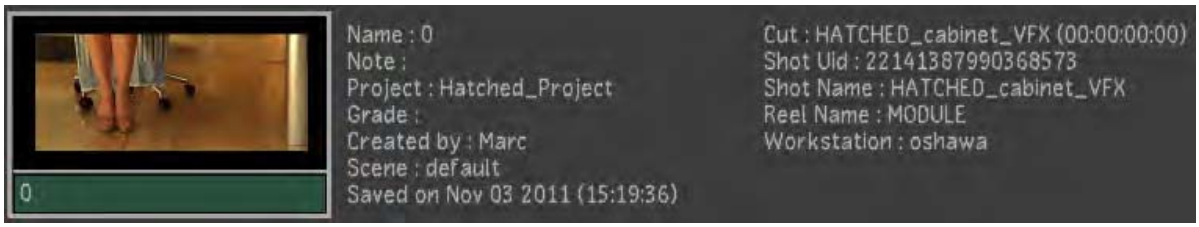
Looks Presets:	Description:
Drama	Variations of dramatic cold and warm looks.
Dusk	Bluish drama look.
Vivid	Variations of a realistic look.
Murky_Old_Negative	Dramatic low contrast brown cast.
Atomic_Blast	Science fiction dramatic look.
Faded	Variations of a faded look.
Glowy	Pale bluish look.
Glowy_Warm	Pale yellowish look.
Low_Contrast	Low contract look.
Purple_Smear	Purple cast with blurry vertical lines.
Ultraviolet_Flare	Purple cast with blur.
Cool_Darker	Dramatic dark look.
Bright_Yellow_Glow	Hard orange cast.
Pinkish	Pink cast.
Gold	Heavy gold look.
Streaks	Overexposed color flares look.
Warm_Sky	Orange cast sky.
Cool_Thriller	Bluish dramatic look.
Warm_And_Soft	Warm blurry look.
End_Of_Day	Warm and dark overall look.
Reddish	Dark and blurry red look.

Looks Presets:	Description:
Early_Morning	High contrast red and blurry look.

Vignette Presets:	Description:
Clear	Resets the grading parameters.
Vignette_Circle	Variations of circle vignettes with blur and highlights / shadows look
Vignette_Rectangle	Variations of rectangle vignettes with blur and highlights / shadows look.

Viewing Media File Information

When you view grade files in the expanded Grade bin, you can display information about the media files.



Details view

Name	Note	Project	Grade	Created by
0	rose afternoon	tamu_2010	default	Tamu
1	rose daytime	tamu_2010	default	Tamu
2	rose morning	tamu_2010	default	Tamu
3	butterfly black morning	tamu_2010	default	Tamu
4	stripe butterfly dusk	tamu_2010	default	Tamu
5	stripe butterfly day	tamu_2010	default	Tamu

Footage courtesy of EVN PRODUCTIONSList view

NOTE Text information is not displayed in Thumbnail view.

Name Displays the name of the grade file.

Note Displays any notes saved with the grade. Right-click to edit the note.

Project Indicates the project in which the grade was created.

Grade Indicates the original grade in which the grade file was created.

Created by Indicates the user that created the grade.

Scene Indicates the original scene in which the grade was created.

Saved on Displays the date on which the grade was saved.

Cut Indicates the original cut in which the grade was created.

Shot UId Displays the shot's unique ID.

Shot Name Displays the name of the shot saved with the grade.

Reel Name Displays the name of the source reel on which the shot is saved.

Workstation Displays the workstation on which the grade was created.

To sort List view information:

1 Do one of the following:
   - Click a column header to sort the list in ascending order by selected column. Re-click the column to sort the list in descending order.
   - Press `Ctrl` and click any column header to reset the sort order by name.

Loading Grading Presets

You can select which grading information you load from a grade file in the expanded Grade bin or the Grade bin using preset lists.

See Saving and Loading Presets Using the Presets Lists (page 2051).

Using Audio

You can hear audio during timeline playback if you import or capture a WAV or AIFF file into the system. The imported/captured file is associated with a cut and begins playback at the same time as the timeline. You can also hear audio during timeline playback if your system is using an external device which detects timecode. When you have selected an audio file or audio tracks to playback with your timeline, all the information (i.e., playback status, selected audio file or audio track, offset, LTC Chase option, and Sync option) are saved in the *Context* file.

NOTE Only one audio file can be associated with the timeline.

For more information on importing/capturing, playing back, and playing out audio, see Audio (page 2306).

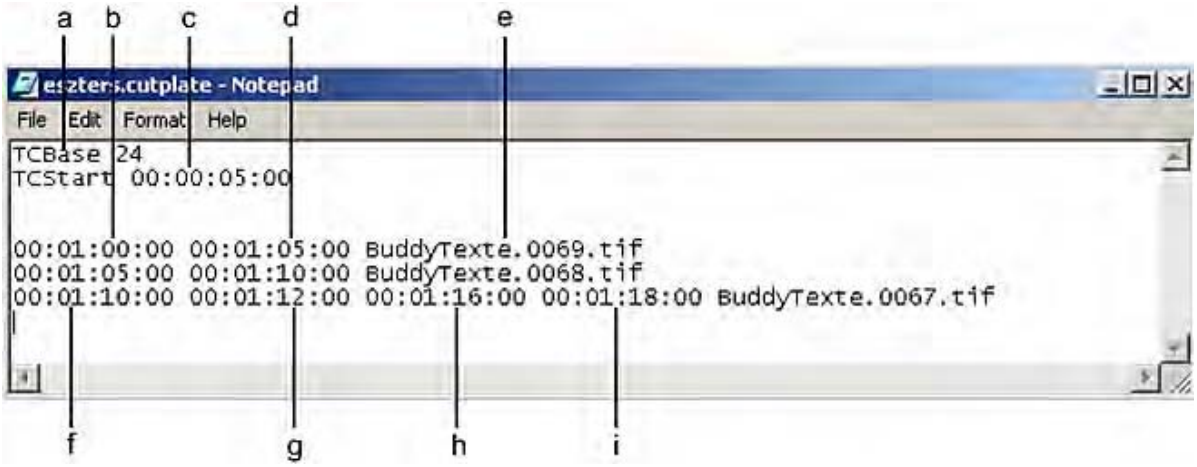
Using Text

When you want to add credits to a cut loaded to the timeline, you can automatically key text files containing an alpha channel over shots in the timeline. To do so, use a text plate EDL that describes how the text files are arranged in the timeline.

To use a text plate with a cut, the text plate file must use a *.cutplate* extension, as well as have the same filename as the *.cut* file you are using. For example, a text plate for *movie.cut* would be named *movie.cutplate*. Store the cut plate for the text files in the *Library* directory—where the loaded cut file is stored. Store the text files in the same scene's *Library\plate* directory.

NOTE If using the Linux version of Lustre, the above filename will use forward slashes.

The following is an example of a text plate EDL.



(a) TCBase (frame rate) (b) Record in timecode (c) TCStart (timecode offset) (d) Record out timecode (e) Text file name (f) Record in timecode and start of fade-in (g) Fade-in end timecode (h) Fade-out start timecode (i) Record out timecode and end of fade-out

The text plate EDL contains the following timecode information.

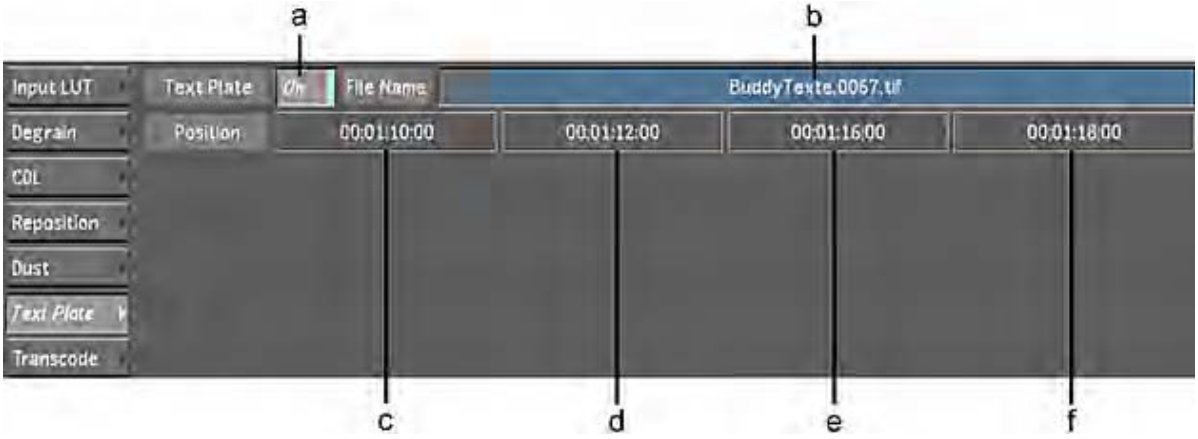
TCBase Set the frame rate. You can set the frame rate to any rate, such as 24, 25, or 30 fps.

TCStart Offset the timecode to slip the entire timeline. For example, 00:00:05:00 would slip the text plate 5 seconds earlier in the timeline.

MaxColor Set the colour intensity from 0-1023, where 0 is black and 1023 is white. If this value does not appear in the EDL, the value defaults to 800.

Timecode List the record timecode for each referenced text file. Set the record in timecode and record out timecode, as well as durations for fade-ins and fade-outs.

Use the Text Plate menu to turn text plates on or off.



(a) On/Off button (b) Text file name (c) Record in timecode and start of fade-in (d) Fade-in end timecode (e) Fade-out start timecode (f) Record out timecode and end of fade-out

To turn a text plate on:

1. Load the cut file with the text plate to the timeline. See Managing Cuts (page 1773).
2. In the Image menu, click Text Plate.
   The Text Plate menu appears.
3. In the timeline, go to the timecode where the text plate appears.
4. Set the On/Off button to On.

The Text plate is keyed over the shot in the Player. When the text plate is on, you can render the cut with the text plate.

To turn a text plate off:

1. In the Image menu, click Text Plate.
   The Text Plate menu appears.
2. In the timeline, go to the timecode where the text plate appears.
3. Set the On/Off button to Off.
   The text plate is turned off and no longer appears in the Player.

   NOTE The On/Off button applies to all text plates in the timeline.

Monitor Calibration

All monitors display colours differently. The colours that you see on your digital monitor change slightly over time because the back light degrades. This causes the maximum brightness of the monitor to decrease slightly. To ensure a perfect colour match between film and monitor, you need to calibrate the monitor. By regularly calibrating the monitor, you ensure that the image on the screen is consistent from day to day by creating a uniform environment—the basis for the accurate display of image colours.

WARNING When you set monitor calibration settings in the Setup > Calibrate menu, you override the monitor calibration settings already configured through User Management and Project Management. This override only survives for the session. After exiting Lustre and restarting the application, the monitor calibration settings will revert to the default settings configured in User and Project Management.

After you calibrate your monitor in Lustre, the image that you view is an accurate replica of that which is sent to the output buffer—with the monitor calibration Lookup Table (LUT) applied on top. If you output to a log file, you will need a log-to-display LUT. If you output to a linear file, you will need a linear-to-display LUT. The calibration design ensures that any modifications you make to the image are as accurately displayed as technically possible—thus reducing the chance of downstream errors.

You can calibrate the monitor automatically or manually. You can also select and compare up to three Print LUTs (either 1D or 3D) to calibrate your system display with the output from the printer. This nullifies the need for subsequent printer light adjustments.

NOTE You can have any number of Print LUTs on your system; however, you can only select three for quick comparisons.

However, with a 3D LUT, you amalgamate the different stages of calibration into the LUT including the calibration of the monitor and the application of the Print LUT measurements.

Logarithmic and Linear Colour Spaces

You colour grade all the shots in your project in either logarithmic or linear colour space. The colour grading toolsets are dependant on the colour space you select. In Log mode, the toolset is designed for digitized film images originating from a datacine conversion process. In Linear mode, the toolset is designed for images that contain linear data.

If you want to use a logarithmic image in Linear mode, you can linearize it by creating a Log to Lin conversion LUT with the LUT Editor. See Linear Mode: Creating Conversion LUTs (page 2030).

NOTE In Log mode, no conversion is necessary on input since you are working on digitized film images in logarithmic colour space.

Monitor Calibration Workflow

Because different facilities may be set up differently, there are many monitor calibration scenarios, particularly if you are working with film images. The following workflow scenarios are guidelines for setting up the display environment that you need.

The recommended steps involved in setting up the proper display environment are as follows.

Step:	Refer to:
1. Adjust the black level and picture settings of the graphics monitor.	Adjusting Black Level and Picture Settings (page 1847).
2. Select the colour space you intend to use for the project—logarithmic or linear.	Selecting the Project Colour Space (page 1849).
3. If you are using a colourimeter, automatically calibrate the monitor.	Calibrating the Monitor Automatically (page 1850).
4. If you do not have a colourimeter, manually calibrate the monitor.	Calibrating the Monitor Manually (page 1853).
5. If your video card is set to Dual-Head mode, set LUTS for each monitor.	Calibrating for Dual-Head Mode (page 1853).
6. Select and compare up to three Print LUTs obtained from your lab to determine which best calibrates the display with the printer.	Applying LUTs for Viewing (Print LUTs) (page 1853).

NOTE Before you set up the proper display environment, make sure the graphics board is set to the supported resolution. See the *Autodesk Lustre Software Installation Guide* for your platform.

Adjusting Black Level and Picture Settings

All monitors come with controls to adjust brightness and contrast. Brightness affects the black level setting of the monitor; contrast controls the picture setting.

You should adjust these settings when you first unpack and connect the monitor. When you adjust the monitor, make sure lighting conditions in the room are what they usually are when you are working. Adjust brightness to set the black level first, and then adjust contrast to control the picture setting.

You should not have to change these settings provided you always use your system under consistent lighting conditions. These settings are perception-based, so do not worry about getting perfect results. In the end, you are trying to ensure that black looks black and white looks white.

Brightness (black level) offsets the red, green, and blue signals. Adjust the black level to display black as black without losing the distinction between black and very dark greys. Note the following:

- When the black level is too high, true black appears slightly grey.

- When the black level is too low, dark greys appear black, so you lose subtle details in shadows.

Contrast (picture setting) applies gain to the red, green, and blue signals. Adjust the picture setting to display white properly on the monitor. Note the following:

- If the picture setting is too low, white appears off-white on your display.
- If the picture setting is too high, very light grey is the brightest colour the monitor displays, so you lose subtle detail in highlights.

For more information about adjusting the black level and picture settings of your monitor, read Charles A. Poynton's monitor adjustment guide at:

http://www.poynton.com/notes/brightness_and_contrast/index.html

To adjust the black level setting:

1. Load a black frame (RGB 0, 0, 0) to the Storyboard. See Creating a Cut with Shots in the Library (page 1876).

   The black frame appears in the Player. Use this image to set your black level.

   TIP You can also create a black frame (RGB 0, 0, 0) in the Curves menu by dragging the RGB curves down.

2. In the Main menu, click Setup, and then click Interface to display the Interface menu.
3. Set Background to the default value of 30.



   A value of 30 creates a subtle contrast between the background colour and the black frame.
4. On the monitor, set Contrast to the minimum value.
5. On the monitor, set Brightness to the minimum value.
6. Slowly increase Brightness on the monitor to increase the black level. When the black image appears lighter than the surrounding background, stop increasing brightness and then decrease it a little.

   You should have a good black level value.

To adjust the picture setting:

1. Adjust the black level as described in the previous procedure.
2. Load a white frame (RGB 1023, 1023, 1023 in Log mode) to the Storyboard. See Creating a Cut with Shots in the Library (page 1876).

   The white frame appears in the Player. Use this image to set your picture settings.

   TIP You can also create a white frame (RGB 1023, 1023, 1023) in the Curves menu by dragging the RGB curves up.

3. In the Main menu, click Setup, and then click Interface to display the Interface menu.
4. Set Background to the default value of 30.



A value of 30 creates a good contrast between the background colour and the white frame.

5 On the monitor, set Contrast to the maximum value.

6 Slowly decrease Contrast on the monitor to adjust the picture setting. When you begin to see that the white frame turns light grey, stop decreasing contrast and then increase it a little.

TIP If anything, err on the side of dark. Sensitivity to monitor flicker increases with the Contrast value. For the most part, the end result of monitor adjustment processes should be good blacks and whites on a screen you are comfortable looking at.

Selecting the Project Colour Space

Select the colour space that you intend to use for the project. The colour space should correspond to the footage you are using. If you are using film images, select logarithmic colour space. If you are using images with linear data (or you prefer the toolset), select linear.

To select the project colour space:

1 In the Main menu, click Setup, and then click Grade.

The Grade menu appears.



(a) Logarithmic colour space button (b) Linear colour space button

2 Select a colour space:

- Click Log to select the logarithmic colour space.
- Click Linear to select the linear colour space.

Calibrating a Monitor

In Lustre, you can calibrate your monitor automatically or manually. Automatic monitor calibration is much more accurate than manual monitor calibration methods because manual methods hinge on perception, and your perception of the monitor can vary depending on the surrounding environment.

Access the Calibration tools in the Calibrate menu.

To access the Calibrate menu:

1 In the Main menu, click Setup, and then click Calibrate.
The Calibrate menu appears.



(a) Print LUT buttons (b) Calibration tools (c) Print LUT list

Calibrating the Monitor Automatically

Use a colourimeter (an external monitor measurement device) to scan your monitor's output and automatically correct the display.

You should automatically calibrate the monitor at the beginning of each project. If the project is long, you may want to recalibrate the monitor from time to time. Every 200 working hours is a good guideline.

To work with film, using a colourimeter is strongly recommended to get the best results.

Gamma Calibration

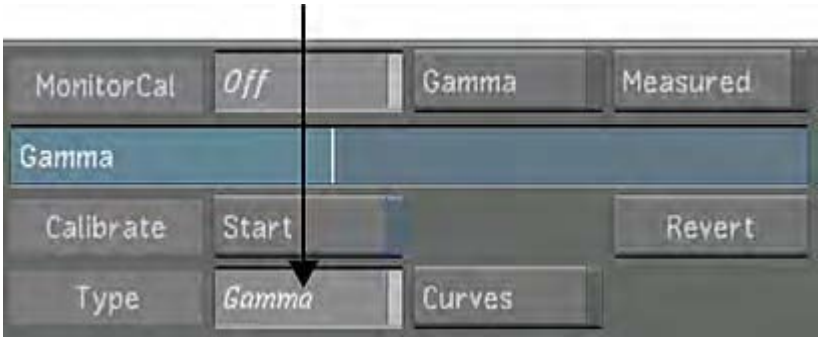
You can calculate a gamma correction LUT using one measure. This creates a simple gamma curve.

To automatically calibrate the monitor based on a gamma measure:

1 In the Calibrate menu, click Off to disable existing monitor calibration settings.



2 Set the calibration type to Gamma.



3 Attach the colourimeter to the centre of the screen.

4 Enable Start.

A series of grey patches appears on the screen. After a few moments, a monitor calibration LUT is generated based on the readings and saved to the *monitor.calib* file.

5 Click Measured to apply the monitor calibration settings.



Curves Calibration

You can calculate the gamma correction LUT using a series of colours, which appears on the screen.

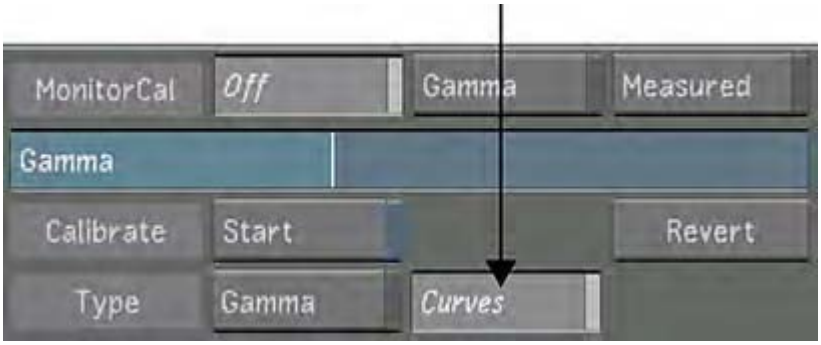
NOTE You can set the colour steps in the *init.config* file. See the *Autodesk Lustre Software Installation Guide*.

To automatically calibrate the monitor based on a curves measure:

1 In the Calibrate menu, click Off to disable existing monitor calibration settings.



2 Set the calibration type to Curves.



3 Attach the colourimeter to the centre of the screen.

4 Enable Start.

A series of colour patches appears on the screen. After a few moments, a monitor calibration LUT is generated based on the readings and saved to the *monitor.calib* file.

5 Click Measured to apply the monitor calibration settings.



Reverting to a Previous Calibration Setting

If you are not satisfied with the results from the automatic calibration, you can revert to the previous automatic calibration.

To revert to a previous calibration setting:

1 In the Calibrate menu, click Revert.



The LUT that was previously saved to the *monitor.calib* file is used to calibrate the monitor. If you click Revert again, you will apply the settings from the latest LUT.

Calibrating the Monitor Manually

If you do not have access to a colourimeter, you can adjust the monitor gamma level manually.

To calibrate the monitor manually:

1 In the Calibrate menu, enable Gamma, and then drag the Gamma slider.



Calibrating for Dual-Head Mode

If the video card on the Lustre system is set to Dual-Head mode, you can set different LUTs for each monitor. This is useful when using different display types, such as a monitor and a plasma screen.

To set different LUTs:

1 (Optional) Import a LUT into the LUT list. See Importing a LUT File (page 2027).

2 In the Calibrate menu, click 1 for the main monitor and select a 3D LUT for the main LUT.

NOTE A 3D LUT is processed directly on the video card and therefore ensures better performance in Lustre.



3 Click 2 and select any LUT for the second monitor.

NOTE This setup will not affect playback speed unless a 3D LUT is also chosen for the second monitor.

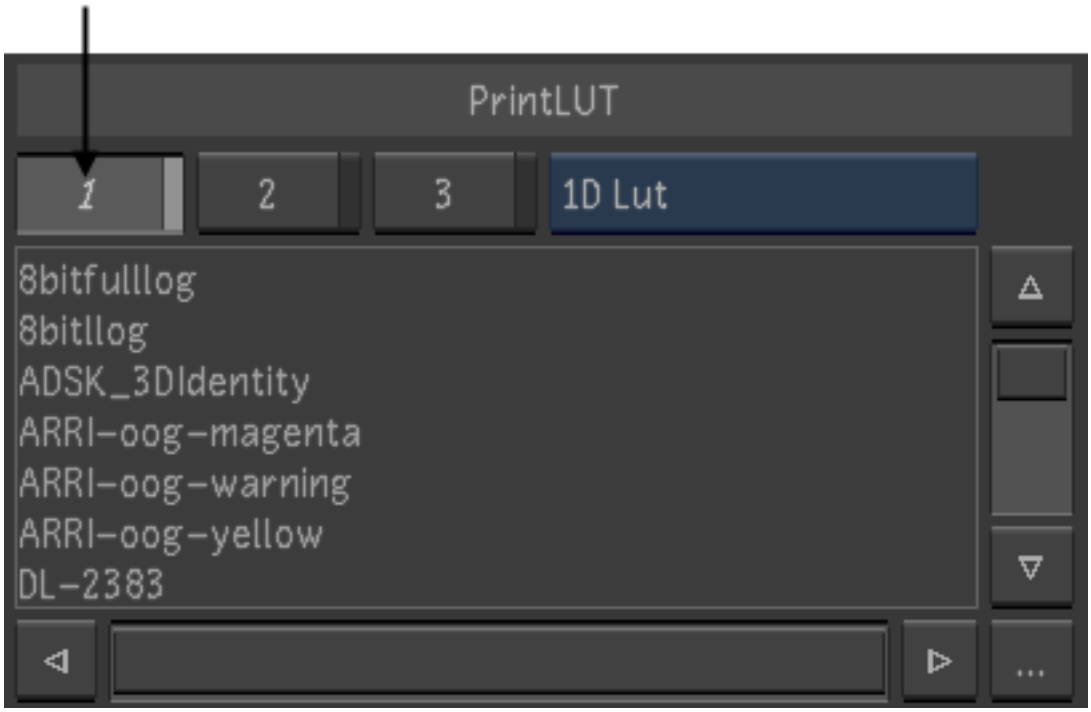
Applying LUTs for Viewing (Print LUTs)

Use viewing LUTs (Print LUTs) to apply preset gamma correction and other transforms to the displayed images. These LUTs allow you to preview images as they will appear on different devices or media without changing the underlying colour values.

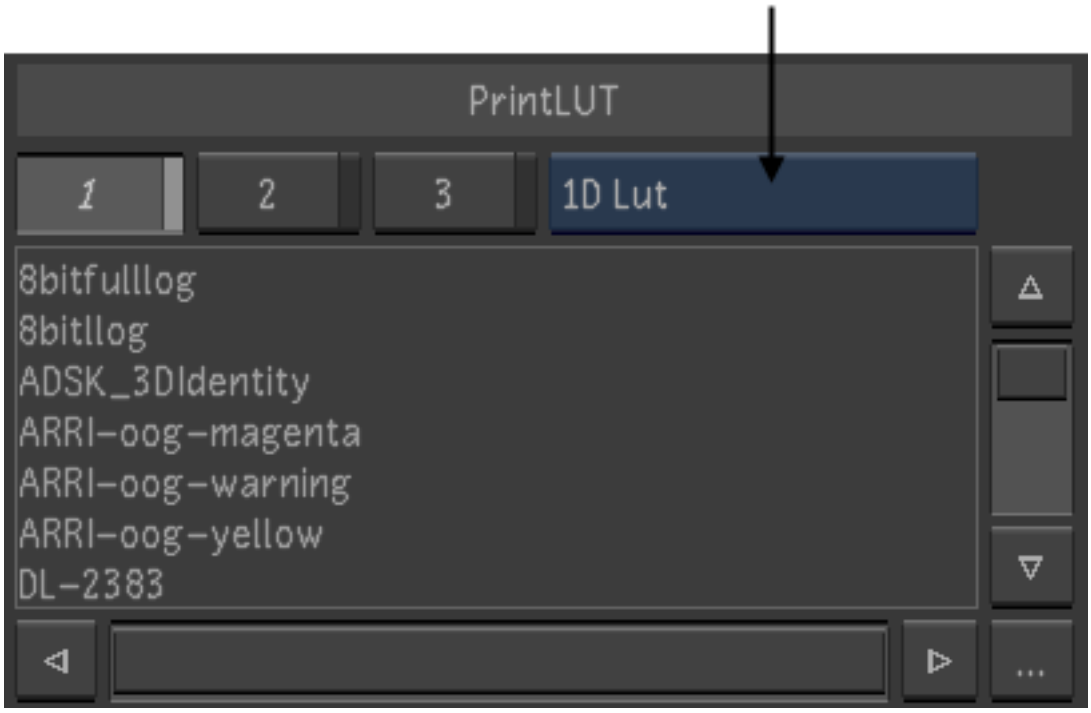
Defining Print LUTs for Viewing

You can define up to three viewing LUTs (Print LUTs) and switch between them for quick comparisons.

1 In the Setup / Calibrate menu, click the number of the Print LUT to assign.



2 Select the type: 1D Lut, 3D Lut, or Colour Transform.



3 Do one of the following:

- Select an item list.
- Click the Browse (...) button at the lower right to select a LUT or colour transform from your file system.

The Print LUT is assigned to to the corresponding number.



4 Repeat to assign LUTs or colour transforms to the other numbers.

Switching Between Print LUTs for Viewing

You can switch between viewing LUTs (Print LUTs) to change the display of your images anytime during a session.

1 Do one of the following:

- In the Calibrate menu, click 1, 2, or 3.
- Click the LUT button to toggle through the assigned Print LUTs.



The selected Print LUT is applied to the display.

Browsing for Footage

Accessing the file browser is one of the first steps of the grading process. Use the file browser to locate scanned or captured footage on your direct storage or shared networked storage. You can also use the file browser to access local or remote footage with the Wiretap Gateway or import Smoke timelines from a Wiretap server.

Supported File Formats

The following file formats are all natively supported in Lustre:

- DPX (8-bit, 10-bit, 16-bit)
- Cineon (8-bit, 10-bit)
- TGA (8-bit, 16-bit)
- TIFF (8-bit, 16-bit)
- SGI (8-bit, 16-bit)
- BMP (8-bit, 8-bit w/alpha)

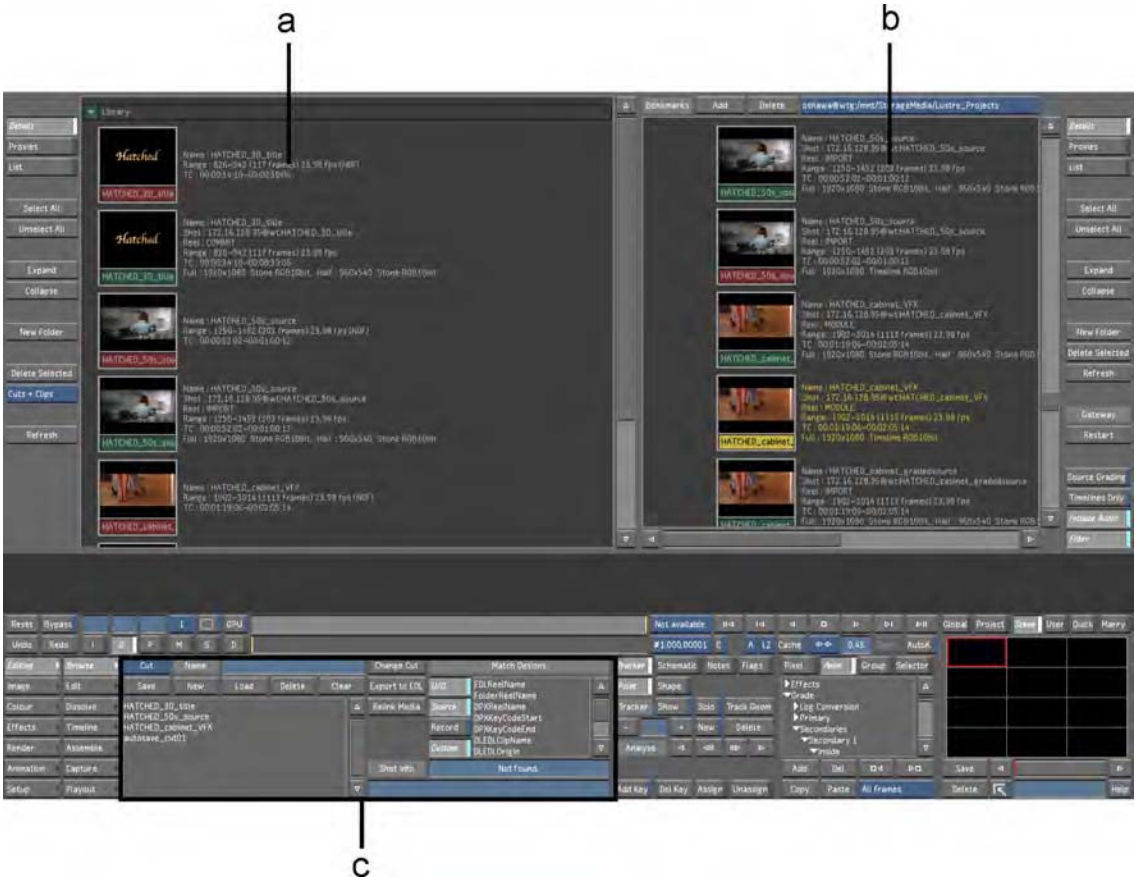
For more information on Wiretap Gateway supported media files, see Wiretap Gateway Supported Ingest File Formats (page 1865).

Accessing the File Browser

Use the file browser to locate shots and load them into the Library.

To access the file browser:

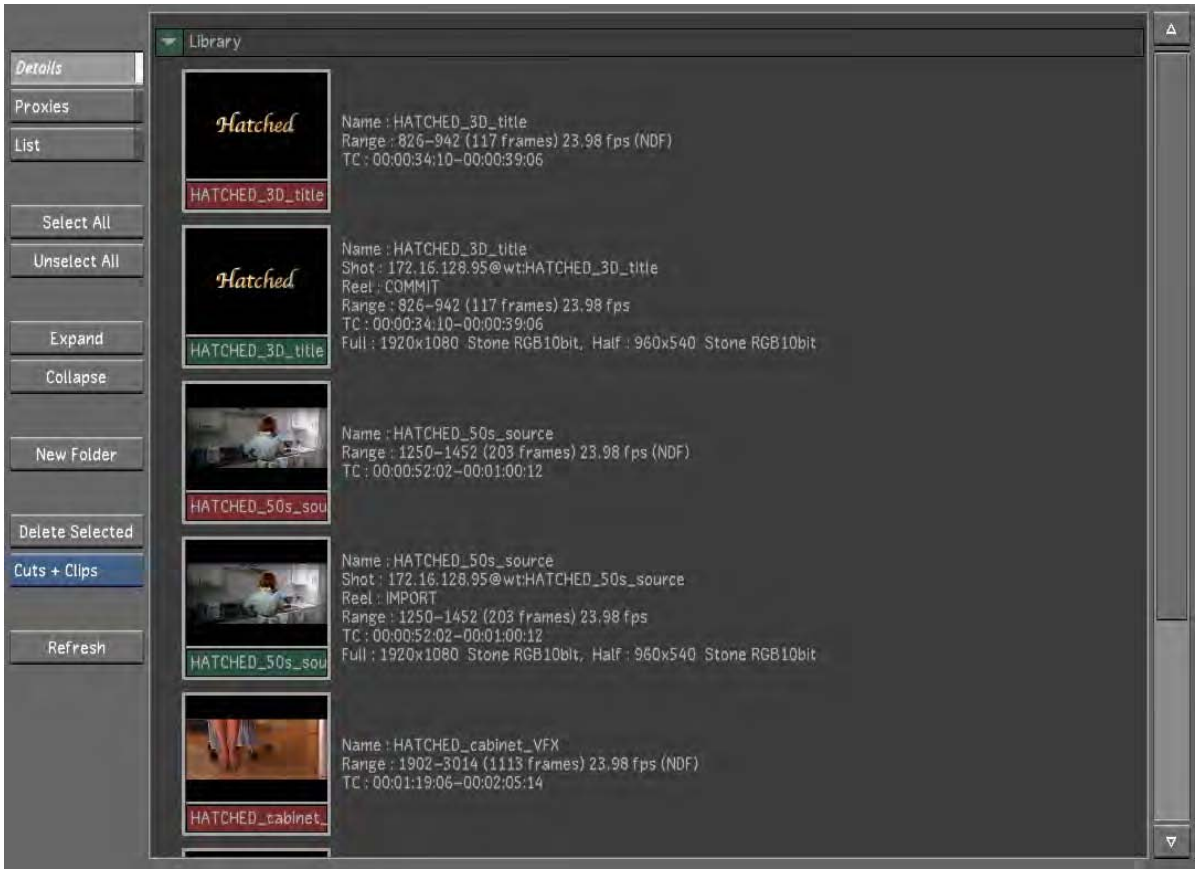
1. Click Editing, and then click Browse.
   The Browse menu and the file browser appear.



Footage courtesy of EVN PRODUCTIONS(a) Library (b) File browser (c) Browse menu

When you are in the Browse menu, the work area is divided into three areas.

Library Use to organize your cuts and clips.

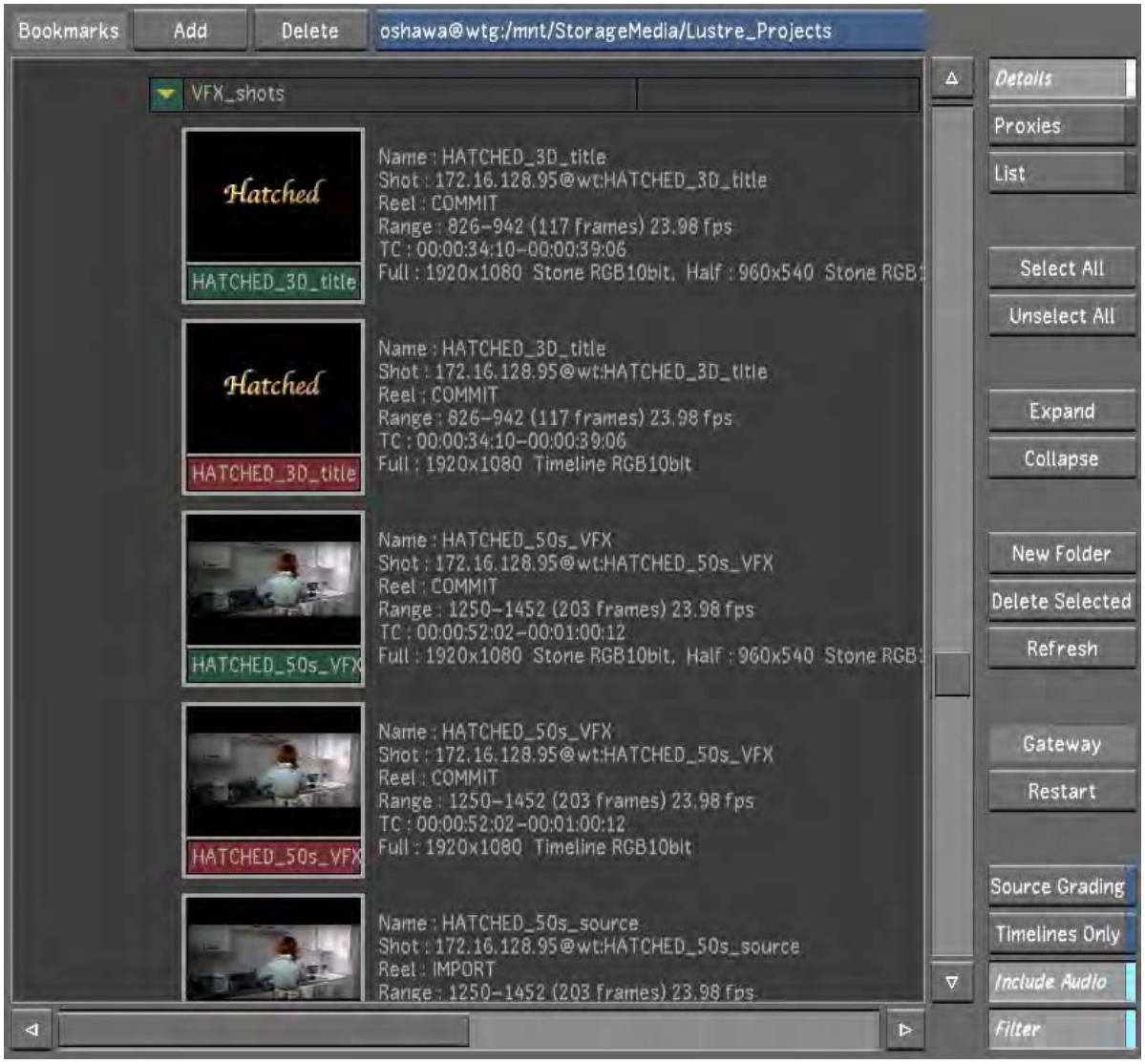


Footage courtesy of EVN PRODUCTIONS

Click:	**To:**
Details	Display clips in Thumbnail and Text view.
Proxies	Display clips in the Library as thumbnails.
List	Display clips in Text view.
Select All	Select all cuts and clips in the Library.
Unselect All	Deselect the selected cuts and clips in the Library.
Expand	Display the contents of the selected folder.
Collapse	Hide the contents of the selected folder.
New Folder	Create a new folder. First select the main folder under which the new one is to be created. Rename the folder by right-clicking it and entering a new name (the root, *Library*, cannot be renamed).
Delete Selected	Delete selected item.

Click:	To:
Cuts/Clips option box	Select the type of media to delete within the selection. The options are: Cuts, Clips, Cuts+Clips.

File browser Use to locate footage and bring it into the Library. When accessing footage from the local or network storage, always place the files into the Scans Full home directory. Footage located under this directory will show a <HOME> prefix before their path when imported in the library.



Footage courtesy of EVN PRODUCTIONS

Click:	To:
Add	Add the current folder location to the bookmark list in the Bookmarks box.
Delete	Remove the folder displayed in the Bookmarks box from the bookmark list.
Details	Display clips in Thumbnail and Text view.

Click:	To:
Proxies	Display clips in the Library as thumbnails.
List	Display clips in Text view.
Select All	Select all clips in the file browser.
Unselect All	Deselect the selected cuts and clips in the Library.
Expand	Display the contents of the selected folder.
Collapse	Hide the contents of the selected folder.
New Folder	Create a new folder in the currently selected folder. Rename the new folder by right-clicking it and entering a new name (the root folder cannot be renamed). Clips already in the file browser cannot be moved to the new folder.
Delete Selected	Delete selected shots.
Refresh	Rescan the file systems and update the file browser with up-to-date information.
Timelines Only	Display only the timelines in the File Browser. See Browsing for Remote Footage (page 2328).
Include Audio	Imports audio tracks from the Wiretap server. See Importing Audio From Wiretap (page 2308).
Filter	Hide specified folders (defined in the *init.config* file) from the file browser view.

Browse menu Use to create and manage the cuts in the project. See Managing Cuts (page 1773).

Viewing Media File Information

In the File Browser or the Library, you can view information about the media files that make up the clip.

You can view the files in different modes.

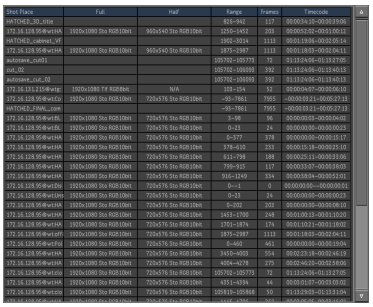
Footage courtesy of EVN PRODUCTIONS



Proxies



Details

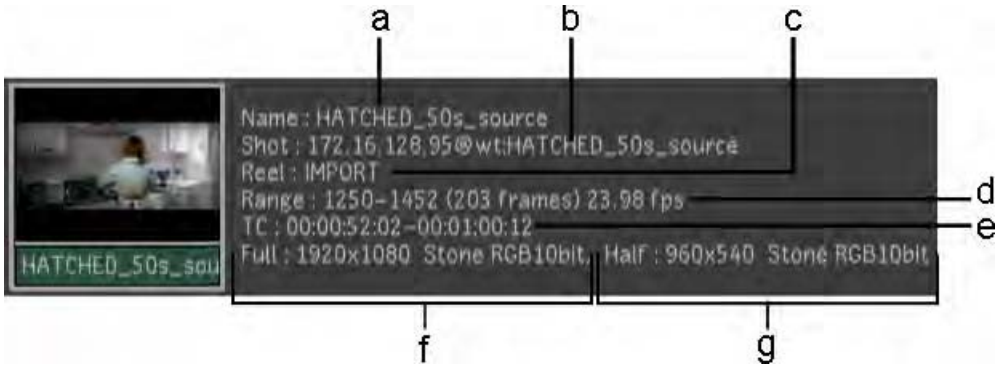


List

NOTE Make sure the cursor is in the Library or File Browser, and press B to switch between view modes in that area.

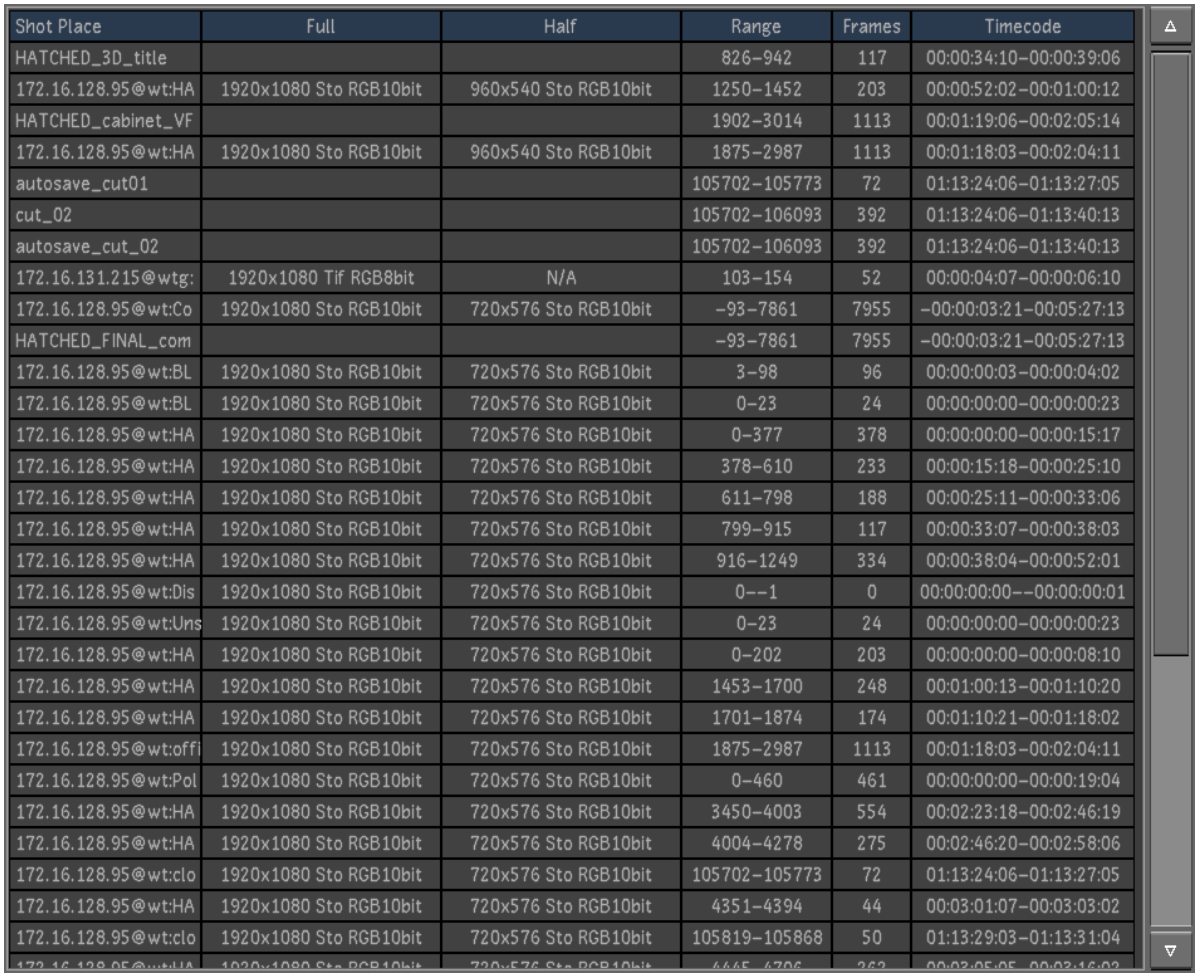
In Proxies view, the thumbnail is displayed and there is no text information.

In Details view, the following information is displayed.



Footage courtesy of EVN PRODUCTIONS (a) Clip name (b) Clip location (c) Reel name (d) Frame range, number of frames, and frame rate (e) Timecode range (f) Full resolution and file format (g) Half resolution and file format

In List view, the following information is displayed.



Shot Place	Full	Half	Range	Frames	Timecode
HATCHED_3D_title			826–942	117	00:00:34:10–00:00:39:06
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	960x540 Sto RGB10bit	1250–1452	203	00:00:52:02–00:01:00:12
HATCHED_cabinet_VF			1902–3014	1113	00:01:19:06–00:02:05:14
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	960x540 Sto RGB10bit	1875–2987	1113	00:01:18:03–00:02:04:11
autosave_cut01			105702–105773	72	01:13:24:06–01:13:27:05
cut_02			105702–106093	392	01:13:24:06–01:13:40:13
autosave_cut_02			105702–106093	392	01:13:24:06–01:13:40:13
172.16.131.215@wtg:	1920x1080 Tif RGB8bit	N/A	103–154	52	00:00:04:07–00:00:06:10
172.16.128.95@wt:Co	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	–93–7861	7955	–00:00:03:21–00:05:27:13
HATCHED_FINAL_com			–93–7861	7955	–00:00:03:21–00:05:27:13
172.16.128.95@wt:BL	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	3–98	96	00:00:00:03–00:00:04:02
172.16.128.95@wt:BL	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	0–23	24	00:00:00:00–00:00:00:23
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	0–377	378	00:00:00:00–00:00:15:17
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	378–610	233	00:00:15:18–00:00:25:10
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	611–798	188	00:00:25:11–00:00:33:06
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	799–915	117	00:00:33:07–00:00:38:03
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	916–1249	334	00:00:38:04–00:00:52:01
172.16.128.95@wt:Dis	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	0–-1	0	00:00:00:00–-00:00:00:01
172.16.128.95@wt:Uns	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	0–23	24	00:00:00:00–00:00:00:23
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	0–202	203	00:00:00:00–00:00:08:10
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	1453–1700	248	00:01:00:13–00:01:10:20
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	1701–1874	174	00:01:10:21–00:01:18:02
172.16.128.95@wt:offl	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	1875–2987	1113	00:01:18:03–00:02:04:11
172.16.128.95@wt:Pol	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	0–460	461	00:00:00:00–00:00:19:04
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	3450–4003	554	00:02:23:18–00:02:46:19
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	4004–4278	275	00:02:46:20–00:02:58:06
172.16.128.95@wt:clo	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	105702–105773	72	01:13:24:06–01:13:27:05
172.16.128.95@wt:HA	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	4351–4394	44	00:03:01:07–00:03:03:02
172.16.128.95@wt:clo	1920x1080 Sto RGB10bit	720x576 Sto RGB10bit	105819–105868	50	01:13:29:03–01:13:31:04

Footage courtesy of EVN PRODUCTIONS

The following information is displayed in List and Details view modes.

Shot place Points to the location of the media files that make up the clip.

Full Indicates the resolution and file format of the full-resolution images that make up the clip.

Half Indicates the resolution and file format of the half-resolution images that make up the clip.

Range Displays the range of frames in the shot sequence that make up the clip.

Frames Displays the number of frames that make up the clip.

Timecode Displays source in and out timecode values that make up the clip.

In the Browse menu, you can also view information about a selected shot in the Storyboard.

Bookmarking Folder Locations

You can quickly access file system folder locations using bookmarks. Bookmarks can be applied to local or remote file system folders, BrowseD, Wiretap and Wiretap Gateway server.



The Bookmarks box contains three default shortcuts.

- Scans Full Home navigates to the project Scans Full Home location.
- Current Shot Location navigates to the folder location of the source media for the current shot.
- Current Render Location navigates to the folder location of the rendered media for the current shot.

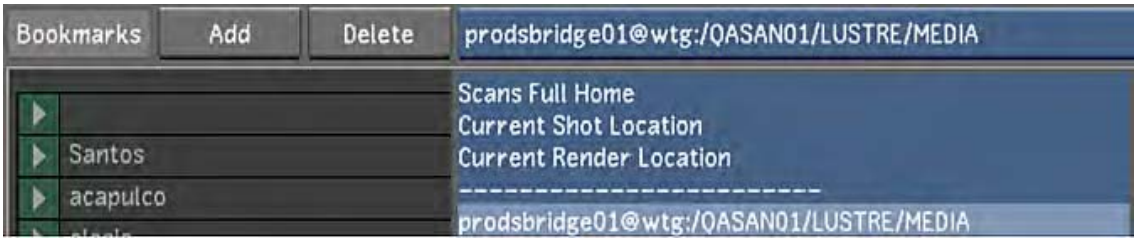
To add a folder:

1. Navigate to the folder you want to bookmark and select it.
2. If the folder contains subfolders, do one of the following:
   - Collapse the folder to mark only the main folder for bookmarking.
   - Expand the folder to mark the folder and all subfolders for bookmarking.
3. Click Add Bookmarks.

   A new entry is added to the Bookmarks box.

To navigate to a folder using bookmarks:

1. Click the Bookmarks box.



2. Select the bookmarks with the location where you want to navigate.

To delete a Bookmark from the list:

1. Select the Bookmark you want to delete.

   NOTE

   Only one bookmark can be deleted at a time.

2. Press Delete Bookmarks, and then confirm the deletion.

User-based Bookmark Data

Bookmark data is user-specific and saved in a file, located inside the User folder location (as defined in the *init.config* configuration file). The file named *UserContext.config* is XML, and it contains the following data:

```
<UserContext user="Marc" parser="1.0" version="2011.1.0.73">
 <BookmarkGroup>
  <Bookmark string="Z:\MEDIA" />
  <Bookmark string="upton@wtg:/QASAN01" />
  <Bookmark string="Z:\Design\cfg" />
 </BookmarkGroup>
</UserContext>
```

TIP Instead of navigating to a location, bookmarks can be added directly to *UserContext.config*.

Library and Reel Creation / Deletion

Library and Reel Creation

When browsing a Wiretap server, you can create Libraries and Library reels, using the New Folder option in the File Browser.

The nature of the newly created folder depends on what is selected:

- If you select a Project, clicking New Folder creates a Library folder within the Project.
- If you select a Library, clicking New Folder creates a Reel folder inside the selected Library.

NOTE If a Library is currently in use in Smoke or Flame, it is marked as Read Only and you cannot create new Libraries or Reels in it until it is no longer in use.

Library and Reel Deletion

When browsing a Wiretap server, it is now possible to delete content like Libraries, Library reels, timelines and source clips.

NOTE If a Library is currently in use in Smoke or Flame, it is marked as Read Only and cannot be deleted until it is no longer in use.

Compressed Media Support

The Wiretap Gateway allows you to import and grade compressed media files (e.g., RED, QuickTime, MXF files and OpenEXR) to Lustre.

The Transcode menu is specifically for compressed media files and is dynamically updated based on the format currently in use. The playback performance of these files varies depending upon the file format, resolution, codecs (see About Wiretap Gateway Supported Ingest File Formats (page 1865)), and configuration of the Wiretap Gateway. Since reading native compressed media files is CPU intensive, you can improve the playback of your media by transcoding your RED or ARRI media into a DPX file. If you are using a local Wiretap Gateway, you can also improve performance by increasing the number of Slaves. The default number of Slaves is four. See Transcode Workflow (page 1877).

For Wiretap Gateway configuration details, refer to the Autodesk WiretapCentral and Wiretap Gateway Installation and Configuration Guide

NOTE Since OpenEXR is a very flexible file format, there are some limitations regarding what is supported by Lustre:

- Mattes encoded in an OpenEXR file have to be RGB media. If not, the OpenEXR file will not be visible in the Media Browser or the non-RGB channels will display a red 'X' in the viewer.
- Channels must have the same resolution.
- OpenEXR playback requires fast CPUs and you may need to enable multiple Wiretap Gateway slaves to achieve acceptable playback performance. Also, when working with multiple channels, real-time playback may not be available and you may need to generate proxy media locally. In this case, make sure to enable the Project / Network Rendering / Wiretap / Local Proxy to be able to generate proxies for content coming from the Wiretap / Wiretap Gateway server.

WARNING Only RED media with firmware 16 and higher is supported in Lustre.

There are some limitations when it comes to assembling an EDL with compressed media. For example, not all QuickTime files contain a reel name and timecode data, therefore, only files with this data can be used to assemble an EDL. Also, only MXF files with timecode data can be used to assemble an EDL.

Browsing for Footage with the Wiretap Gateway

You can import compressed media footage files via the File Browser using the Wiretap Gateway. You can access a local or remote host that is running a Wiretap Gateway. Once imported, the compressed media can be transcoded. For more information about the Wiretap Gateway, refer to the *Autodesk WiretapCentral and Wiretap Gateway Installation and Configuration Guide*. To load the compressed media footage, see Transcode Workflow (page 1877).

NOTE The <HOME> prefix will not appear on media coming from the Wiretap Gateway or on managed media residing on the Standard FS.



About Wiretap Gateway Supported Ingest File Formats

An image sequence is a series of sequentially numbered files, traditionally the result of scanning film stock at high resolution to produce a digital intermediate. Each file contains the digital scan of an individual frame. Common formats include Cineon, DPX and Tiff. The file type is usually indicated by its extension.

In contrast, container formats, also called “wrapper” formats, can contain image sequences (commonly called streams or essences) and audio, compressed using a variety of compression algorithms (codecs) into a single file. Container formats do not impose specific video or audio codecs upon the media they contain. Rather, a container format defines only how the video, audio and other data is stored within the container itself. Unlike image sequences, it is not possible to tell by looking at the extension what kind of video or audio is inside a container format.

Supported image sequence formats

The Wiretap Gateway server supports ingest of the following image sequence file formats.

File Format	Bit Depth	Default Extension
Alias®	8-bit	als
Cineon®	10-bit	cin
DPX	8-bit, 10-bit, and 12-bit	dpx
Jpeg	8-bit	jpg
Macintosh® Pict	8-bit	pict

File Format	Bit Depth	Default Extension
OpenEXR	16-bit int, 16-bit float, 32-bit float	exr
Pixar	8-bit	picio
SGI®	8-bit and 16-bit	sgi
Softimage®	8-bit	pic
TARGA®	8-bit	tga
Tdi/Maya®	8-bit and 16-bit	iff
Tiff	8-bit and 16-bit	tif
Wavefront®	8-bit and 16-bit	rla

NOTE The Wiretap Gateway supports ingest of OpenEXR 16-bit RGB float only. Files with other channels, RGBA, for example, cannot be read in Lustre.

Supported Image Container Formats

Wiretap Gateway supports ingest of the container formats listed in the Related Topics.

Support for ARRI Files

The Wiretap Gateway supports the import of ARRIRAW™ media files produced by D-20, D-21, and ALEXA cameras (including the 4.3 and 4.4 SDK updates).

ARRIRAW files come in two native resolutions, 2880x2160 or 2880x1620, both at 12-bit depth.

Support for ARRIRAW Look Files

Version 4 of the ALEXA Software Update Packet enables a *look*-creation workflow. In this workflow, a look is created in the ARRI Look Creator application and either embedded in the source files during the shoot, or saved to a *look* file. In either case, the look data can be enabled or disabled from the Import Preferences menu.

NOTE The look file must be placed in the folder containing the source material and named *look.xml*.

When transcoding ARRIRAW files, you can decide how to handle the ARRIRAW look information.

To use a look created in the ARRI Look Creator application with imported ARRIRAW media:

1. Optional: If the look information is located in a look file, make sure the file is named *look.xml* and is located with the ARRIRAW media.
2. Drag the clip(s) you want to transcode to the Storyboard.
3. Navigate to Image / Transcode / Colour.

4 Set the ARRI Look options in the ARRI Look selection box. Options are:

Select:	To:
Do not apply Look	Not apply any Look information.
From File Header	Apply Look information contained in the ARRIRAW file header.
From Look File	Apply Look information contained in an XML Look file located in the same folder as the file to import.

Two additional options might need to be set to correctly use the look created with the ARRI Look Creator.

Apply Look LUT button Enable to apply to the clip the LUT information contained within the Look file.

Colour Rendering box When working with Look information, make sure the Colour Rendering to Rec709 or Lustre displays a red X instead of the clip, as ARRI Look Creator only supports images encoded with REC709.

Support for AVCHD Files

It is now possible to import AVCHD content in Lustre through the Wiretap Gateway. Video and audio media, and timecode tracks are supported.

NOTE Only PCM audio is supported. Some cameras can record AC-3 audio, but this format is not supported. Only the video portion of AVCHD media with AC-3 audio content is accessible from the application.

Support for GoPro HD MP4 Files

GoPro camera records H.264 files with an "mp4" extension. Files from this camera are supported by Wiretap Gateway Server.

The following frame rates and resolution are now supported (with audio):

- 1920x1080 (16x9, 29.97p and 25p)
- 1280 x 960 (4x3, 29.97, 25p)
- 1280 x 720 (16x9, 25p, 29.97p & 59.94p)
- 848x480 (16x9, 25p, 29.97p & 59.94p)

Due to limitation in the GoPro 59.94p files, the application assigns to those files the timing of the project. So if your project is set at 30fps, 59.94 GoPro files are imported at 30fps. To play back 59.94 GoPro files correctly, you must be using a 59.94 project.

HDR Format Support

The HDR file format (High Dynamic Range) is now supported by the Wiretap Gateway.

Like OpenEXR, before you import HDR images, you must set up a project-based float conversion LUT in the Calibration settings.

Multi-Channel PSD Import

When you import Photoshop files (PSD) via the Wiretap Gateway, Lustre creates a multi-channel clip. You can use the mattes for Secondary color grading.

Support for RED Files

Lustre can import RED® media (R3D files) encoded with codec versions 2 and 3.

Wiretap Gateway also supports the RED 4.3 and 4.4 SDKs and the RED-provided Magic Motion algorithm (a blending algorithm used to composite tracks A and X of an HDRx clip). Support also extends to RedColor 3 and RedGamma 3.These parameters are accessed in Image / Transcode / Format menu.

Lustre also reads the camera type from the R3D files metadata and displays it in the File Browser, allowing you to distinguish between RED Epic and RED Scarlet cameras.

See Transcode Format Settings (page 1879).

Detailed DNxHD MXF Support

The Wiretap Gateway supports the import of MXF files encoded with Avid® DNxHD® codec.

NOTE Material at 1080i encoded with the DNxHD 145 codec sometimes uses a thin raster frame format, denoted as 145-TR. It uses an frame size of 1440x1080 (NTSC) or 1280x1080 (PAL). When importing, enable Scale to Full HD in **MediHub ➤ Browse for Files ➤ Format Specific Options ➤ Image** to scale the material to the full 1920x1080 frames. Disable Scale to Full HD to import using the thin raster frame format.

Resolution	Frame Rate	Avid DNxHD CODEC	Bit Depth
1080i	59.94	DNxHD 220x	10
		DNxHD 220	8
		DNxHD 145	8
	50	DNxHD 185x	10
		DNxHD 185	8
		DNxHD 120	8
1080p	29.97	DNxHD 220x	10
		DNxHD 220	8
		DNxHD 145	8
		DNxHD 45	8

Resolution	Frame Rate	Avid DNxHD CODEC	Bit Depth
	25	DNxHD 185x	10
		DNxHD 185	8
		DNxHD 120	8
		DNxHD 36	8
	23.976 & 24	DNxHD 175x	10
		DNxHD 175	8
		DNxHD 115	8
		DNxHD 36	8
720p	59.94	DNxHD 220x	10
		DNxHD 220	8
		DNxHD 145	8
	50	DNxHD 175x	10
		DNxHD 175	8
		DNxHD 115	8
	29.97	DNxHD 110x	10
		DNxHD 110	8
		DNxHD 75	8
	23.976 & 25	DNxHD 90x	10
		DNxHD 90	8
		DNxHD 60	8

Import: supported P2 MXF CODECs

The Wiretap Gateway supports import of Panasonic P2 MXF files encoded with any of the following codecs.

P2 CODEC	CODEC Flag	Comment
AVC-Intra 50	AVC-I 50	Panasonic P2
AVC-Intra 100	AVC-I 100	Panasonic P2
DV 25	DV 25	
DVCPRO	DVCPRO	
DVCPRO 50	DVCPRO50	
DVCPRO HD	DVCPROHD	

Import: supported QuickTime CODECs

The Wiretap Gateway supports import of QuickTime® files encoded with any of the following codecs.

NOTE QuickTime files exported from QuickTime Pro in Drop-Frame mode may be improperly truncated. This will result in the video and audio frames not matching when imported into Lustre.

NOTE QuickTime PhotoJPEG files can only be imported. You cannot soft-import QuickTime PhotoJPEG files.

Broadcast CODEC	CODEC Flag	Comment
Component Y'CbCr 10-bit 4:4:4	v410	10-bit Packed
Component Y'CbCr 10-bit 4:2:2	v210	10-bit Packed Blackmagic or AJA-Kona compatible
Component Y'CbCr 8-bit 4:4:4	v308	8-bit Planar
Component Y'CbCrA 8-bit 4:4:4:4	v408	8-bit Planar
Component Video	yuv2	8-bit Packed 4:2:2 Blackmagic or AJA-Kona compatible
8-bit Packed YUV 4:2:2	2vuy	
DV 25 NTSC	dvc	Although the specifications allow the DV format to be field 1 or 2, the industry standard is "bottom first". Thus, before exporting to Lustre, ensure that the clip is Field 2. Reformat, if necessary.
DV 25 PAL	dvcp, dvpp	

Broadcast CODEC	CODEC Flag	Comment
DVCPRO 50 NTSC	dv5n	
DVCPRO 50 PAL	dv5p	
DVCPRO HD	dvh1	
DNxHD	avdn	8 bits: 36, 145, 220 (and variants) 10 bits: 220x (and variants)
IMX	mxn3, mxn4, mxn5	30, 40, 50

File CODEC	CODEC Flag	Comment
PhotoJPEG	RTJ0	RT PhotoJPEG compatible
MJPEG	MJPG, mjpg, mjpa, mjpb, JPEG, jpeg, dmb1, AVDJ	JPEG compatible
PNG	png	Portable Network Graphic sequence (alpha support only through Gateway library)
PNGA	pngalpha	Portable Network Graphic sequence (with alpha support)
RGB Uncompressed	raw	No alpha support
RGBA Uncompressed	rawalpha	With alpha support
TGA	tga	TARGA

Web CODEC	CODEC Flag	Comment
MPEG-1	mpg1, MPG1, pim1, PIM1	
MPEG-4	mp4v; DivX®; DIV1; div1; MP4S;M4S2; m4s2; xvid; XVID; XviD; DX50; dx50; DIVX; MP4V	
MSMpeg 4v3 (DivX)	DIV1, div1, MPG4, mpg4, DIV2, div2, MP42, mp42, DIV3, div3, DIV4, div4, DIV5, div5, DIV6, div6, MPG3, mpg3, MP43, mp43, AP41, ap41, MJPG	
Quicktime Planar RGB	8BPS	
Apple® Video	rpza	

Web CODEC	CODEC Flag	Comment
Apple Graphics	smc	
Apple Animation	rle	With alpha support
Cinepak	cvid	
H.264	avc1	

Audio CODEC	CODEC Flag	Comment
IMA 4:1	ima4	
Raw 8-bit audio	rawaudio	
Twos	twos	16-bit PCM (Big Endian)
Ulaw	ulaw	
Sowt	sowt	16-bit PCM (Little Endian)
A-law 2:1	alaw	
16-bit PCM	in16	
Linear PCM (QT 7)	lpcm	
Ogg Vorbis (qt4l compatible)	vorbis	
Ogg Vorbis (qtcomponents compatible)	vorbis_qt	
MPEG-2 Layer 2 Audio	mp2	
QDM2 Audio	qdm2	
Apple lossless	alac	
MS ADPCM	adpcm (ms)	
ADPCM ima WAV	ima adpcm (wav)	
Advanced Audio Codec (AAC)		Supported in mp4, m4v, and mov containers.

Import: supported XDCAM file CODECs

The Wiretap Gateway supports import of Sony™ XDCAM files encoded with any of the following codecs.

XDCAM CODEC	CODEC Flag	File Type	Comment
MPEG-2 IMX 30	IMX 30	MXF	XDCAM
MPEG-2 IMX 40	IMX 40	MXF	XDCAM
MPEG-2 IMX 50	IMX 50	MXF	XDCAM
MPEG-2 long-GOP	XDCAM HD	MXF	XDCAM HD (4:2:0)
MPEG-2 long-GOP	XDCAM HD422	MXF	XDCAM HD (4:2:2)
MPEG-2 long-GOP	XDCAM EX	MP4	XDCAM EX

Import: supported Apple ProRes files

The Smoke on Mac application imports ProRes files directly.

For applications running on Linux, you need to import the ProRes material through a Wiretap Gateway running on a Mac. The installation documentation explains how to install such a Wiretap Gateway. You can then connect through the Network Panel to Wiretap Gateway running on the Mac and import the ProRes files.

The following Apple ProRes codecs are supported.

Apple ProRes CODEC	CODEC Flag	Comment
ProRes 4444	ProRes4444	12-bit
ProRes 422 (HQ)	ProResHQ	10-bit
ProRes 422	ProResSD	10-bit
ProRes 422 (LT)	ProResLT	10-bit
ProRes 422 (Proxy)	ProResProxy	10-bit

Supported audio file formats

Wiretap Server and Wiretap Gateway now use 24-Bit MSB padding, rather than 24-Bit LSB padding for audio. Third-party Wiretap Client applications that assume LSB padding for 24-bit audio will therefore not be able to read clips with 24-bit audio from version 2011 of Visual Effects and Finishing applications. Until your

third-party Wiretap client application is updated to use the new method, you can work around this issue by performing a mix-down in the Visual Effects and Finishing application.

Audio file formats supported by Wiretap Gateway

File Format	Bit Depth	Default Extension
AIFF-C	16-bit, 24-bit, and 32-bit (float)	aifc
AIFF	16-bit and 24-bit	aiff
WAVE	16-bit, 24-bit, and 32-bit (float)	wav
Broadcast Wave	16-bit, 24-bit, and 32-bit (float)	wav

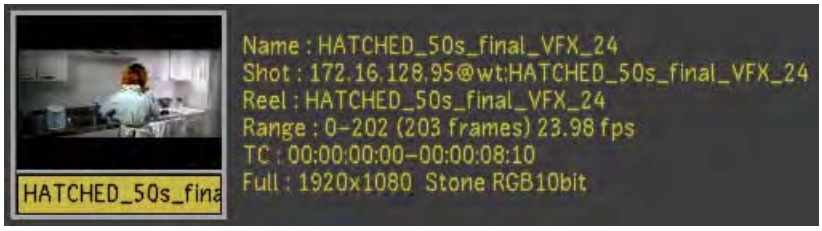
Loading Clips to the Library

Once you locate your clips in the file browser, you move them into the Library so that you can use them in your cuts. Clips in the Library are pointers to the actual media files on the file system. You can view information about the clips in both the file browser and the Library.

There are two ways to interact with clips and timelines in the File Browser and Library:

Select shots/timelines:

- Click on the lower portion of the proxy (red or green), to select the shot/timeline. The lower portion of the proxy becomes yellow. You can select multiple clips by clicking Ctrl as you select them.



Footage courtesy of EVN PRODUCTIONS

If you select the wrong clips/timelines, click any unused grey area in the User Interface to cancel your selection.

Pick shots/timelines

- Click on the image portion of the proxy, to pick the clip/timeline. You can pick multiple clips/timelines at once to move/load them simultaneously.



Footage courtesy of EVN PRODUCTIONS

If you pick up the wrong clips/timelines, click any unused grey area in the User Interface to cancel your selection.

To move shots or timelines to the Library:

1 Navigate to your clips or timeline.



Footage courtesy of EVN PRODUCTIONS

TIP You can open all of a folder's sub-directories by `Shift`-clicking the arrow to the left of the folder or by using the Expand button.

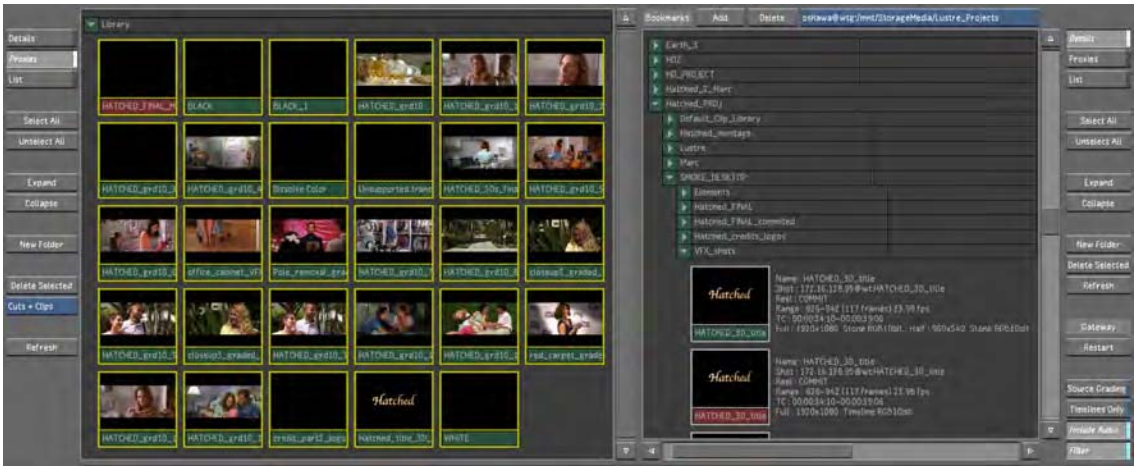
2 Select the clips you want to move to the Library.

TIP To select all the clips from a folder, select the folder.

3 Drag the selected clips to the Library, and then click to release the clips.

TIP You can drag clips directly from the file browser to the Storyboard. When you do this, the clips are also imported to the Library.

Newly added clips are highlighted. Clips already in the Library are not duplicated.



Footage courtesy of EVN PRODUCTIONS

Deleting Content From the Library

You can delete clips and cuts from the Library.

NOTE Deleting clips from the Library does not delete them from the Storyboard.

To delete selected clips from the Library:

1. Select the clips you want to delete.
2. Click Delete Selected, and then confirm the action.
   All the selected clips are deleted.

To delete all clips from the Library:

1. Click Delete All, and then confirm the action.
   All the clips are deleted.

Creating a Cut with Shots in the Library

Once the source clips are imported to the Library, you can assemble them using an EDL or by creating a timeline.

See:

Working with EDL, ALE and Cutlist Files (page 1892)

Adding Shots to the Storyboard (page 1915)

Adding Shots to the Timeline (page 1936)

Matching Shots

You can use Matchback to select a shot in the timeline and find it in the Library, or click a shot in the Library and see where it is used in the timeline.

To match a shot from the Timeline or Storyboard to the Library:

1. Select the shot by completing one of the following:
   - Click a shot in the Storyboard.
   - Place the Timeline positioner on a shot.
2. Press `Shift+Q`.

   The selected shot is highlighted in the Library.

To match a shot from the Library to the timeline:

1. `Alt`-click the shot in the Library.

   The selected shot is highlighted in the Storyboard and the positioner is updated in the Timeline.

   NOTE If you continue to `Alt-click` the shot in the Library, successive instances of the same shot are highlighted in the Storyboard and positioned in the Timeline.

Transcoding

The Wiretap Gateway server makes it possible for Lustre to easily ingest complex media formats. The Transcode menu displays information related to the format of the currently selected media. The full set of options, which include debayering and colour options are only available for RED, ARRIRAW and Sony F65 RAW files, whereas the LUT and resize options are available for any media file imported through Wiretap Gateway. Transcode parameters are greyed out in the Transcode menu when the file format does not support them.

NOTE

- When importing Sony F65 RAW files, a Scene Linear transformation is applied by default, to convert the 16-bit media to 16-bit half-float. To remove this transformation, select Native, from the Colour Space box in the Image / Transcode / Colour menu.
- Sony F65 RAW files are only available if you install the alternate Red Hat 5 build, which is available for HP Z800 and Z820 workstations.

The transcode settings are defined on a shot-by-shot basis. Some of the settings (e.g., Resize and LUT Options menus) can be applied to the media without having to click the Transcode button, and the other settings are not applied until you click the Transcode button. Once you click the Transcode button, the job is sent to the Backburner Manager where it is transcoded. When you begin transcoding, the transcoded media immediately replaces the current media. After the transcoding is complete, the cut is automatically saved and the media paths and processing options are stored in the cut file.

Once the transcode options are set, you can copy select settings from one shot to another using the Transcode Settings parameters found within the Selector. See Copying Parameters with the Selector (page 1816).

Transcode Workflow

Once you have imported the RED media and want to improve the playback of this file, follow the typical workflow to transcode your footage.

Step:	Refer to:
1. Set the high-level options.	Transcode Format Settings (page 1879).

Step:	Refer to:
2. Set the initial colour correction.	Transcode Colour Settings (page 1882) and Detail, OLPF Compensation, and Noise Reduction in Transcode Format Settings (page 1879).
3. Apply a LUT or a colour transform to the media.	Transcode LUT Options (page 1883).
4. Resize the media.	Transcode Resize Options (page 1886) .
5. Set the output options and transcode the media.	Transcode Output Settings (page 1887).
6. (Optional) Copy the transcode settings from one shot to another.	Copying Parameters with the Selector (page 1816).
7. (Optional) Save the transcode settings as a version.	Transcode Versioning (page 1889).
8. (Optional) Save the transcode settings as a template.	Transcoding Template (page 1891).

Transcode Options

The Transcode menu is divided into five submenus.

NOTE The Settings and Transcode options are available within the Format, Colour, LUT Options, Resize and Output settings tabs.



Format Sets the high-level options for the media decoding (e.g., debayering, resizing, filter, etc.). This submenu is exclusive to RED and ARRIRAW media. See Transcode Format Settings (page 1879).

Colour Sets the colour settings to be transcoded (e.g., colour and curve settings, etc.). This is useful to set the initial colour correction before grading your footage. This submenu is exclusive to RED and ARRIRAW media. See Transcode Colour Settings (page 1882).

LUT Options Applies a LUT or colour transform to the media to be transcoded. This is useful for colour space conversion, for example, before grading your footage. See Transcode LUT Options Settings (page 1883).

Resize Applies a resize to the media, without the need to transcode and generate new media. See Transcode Resize Options (page 1886).

Output Sets the file output settings (e.g., folder structure, timecode, and image format). See Transcode Output Settings (page 1887).

Once these values have been set, you can choose to save the settings to a transcoding template (similar to the project and user settings template option). See Transcoding Template (page 1891).

To enter the Transcode menu, click Image in the Main menu, and then Transcode.

Transcode Format Settings



(a) Fit Method option box (b) Selector

NOTE The Settings and Transcode panel options are also available in the Colour and Output settings tab.

Camera button Enable to load the original RED media values as set by the RED camera. When the RED media is initially imported, this button is automatically enabled. When you change one of the transcode options, this button is automatically disabled to show that these settings are no longer the original RED camera settings.

RSX/RMD button Enable to load the values from the latest version of the RSX/RMD file. When a clip is opened in RED ALERT!™, an RSX/RMD file is saved alongside the R3D file, using the same filename. It contains the additional metadata added to the R3D file within RED ALERT!.

Once you change one of the Lustre transcode options, this button is automatically disabled to show that these settings are no longer the original RSX file settings.

Scene button/Shot button Enable Scene or Shot to transcode either the entire scene or the shot that is selected in the Storyboard.

Heads & Tails button Enable to make sure the heads and tails from the RED media are kept when it is transcoded.

Handles button and Handles field Enable the Handles button when you want to transcode fewer heads and tails than what is currently showing in the footage. Enter a value in the Handles field to determine how many heads and tails to transcode. This option can only be used if the Heads & Tails option is enabled.

Transcode button When the format, colour, and output settings have been established, click this button to send the job to the Backburner Manager and transcode the RED media to a DPX file.

Resize button When enabled, the resize settings (i.e., the W and H fields) are implemented when the RED media is transcoded.

W/H field Enter the desired resize setting. Note that a resize setting that is not directly proportional to the size of the original media takes longer to process.

Fit Method option box To use a different aspect ratio during a resize, select one of the following fit method options to be applied to the exported clip.

Select:	To:
Fill	Fit the source, width, and height into the destination frame. If the source and destination frames do not have the same aspect ratio, the image can become distorted. This is the default option.

Select:	To:
Crop Edges	Fit one edge of the source into the destination frame without stretching or squashing the frame. Excess parts of the source frame after resizing are cropped. If the source, after the one edge is resized, is wider than the destination, its overhanging left and right edges are cropped. If the source is taller than the destination, the upper and lower edges are cropped.
Letterbox	Fit the source to the destination frame without squashing or stretching it, and without cropping the source. If the source is wider than the destination, black bars fill the top and bottom of the destination frame. If the source is narrower than the destination, black bars fill the right and left sides of the frame. If the source is narrower than the destination, black bars fill the right and left sides of the frame. In all cases, the entire source frame is contained within the destination frame.
Centre/Crop	Fit the source image, centred, over the destination frame. If the source is larger than the destination, it is cropped. If the source is smaller than the destination, it is surrounded by a black border.

Filter option box Select the filter option to determine the quality of the interpolated resize result.

Select:	To get:
Bicubic	Very good results for resizing soft-looking images. Use to sharpen the image. This is the default option.
Mitchell	Best results when resizing a clip to a higher resolution.
Triangle	Moderate results with little processing overhead.
Impulse	Quick, low-quality results.
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.
Shannon	Excellent results when resizing a clip to a lower resolution. Very similar to Lanczos, but results are a little softer.
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.

NOTE The following settings vary based on the type of media.

Include YUV Headroom Enable to include YUV headroom on import for Quicktime and MXF files.

Scale to Full HD Enable to scale MXF P2 & XDCAM HD narrow content to Full HD resolution on import.



Debayering Hires/Proxy option box Select the level of quality required from the debayering algorithm. Higher resolutions take more time to process. Select one of the following options for the full or proxy footage:

- Full
- Half Premium
- Half Good
- Quarter
- Eighth

Bit Depth option box Select the required bit depth. Your options are:

- 12 bit
- 16 bit fp

Detail option box Select the level of detail extraction required. Your options are:

- Low
- Medium
- High

OLPF Compensation option box Select the level of Optical Low Pass Filter compensation to use. The OLPF is a type of sharpening used to compensate for the optical anti-aliasing filter, which can induce softening of the image during recording.

Noise Reduction option box Select the level of noise reduction applied to the debayered shot.

HDRx Settings Select the HDRx Exposure options.

Exposure option box Select the exposure options of your HDRx media.

Select:	**To:**
Primary Exposure	Select the main exposure of the R3D file
Highlight Exposure	Select the second exposure of the R3D file.
Blend Exposure	Blend the values between the two exposures to create a single image for grading. This option uses both video tracks of their R3D file. Playback performance may be altered.
Magic Motion	The new HDRx merging option provided by RED. This blending algorithm is used to composite tracks A and X of an HDRx clip.
High Dynamic Range	Convert the two exposures into a 16-bit half-float image. When this option is selected, the Debayering / Bit Depth option is automatically set to 16-bit fp and the Gamma Curve setting is set to Scene Linear (16bit fp).

Select:	To:
RMD	Define the blending options from the RMD. If there is no RMD, the application defaults to using the A track.

Blend slider Set a value to blend the two exposures of a RED HDRx clip.With HDRx Settings set to Blend Exposures, this field behaves like the blend in REDCINE-X. The blend operation is a blend of the two exposures: -1 shows only the Highlight (X frame), 1 only the Primary (A Frame), and 0 a 50-50 mix. With HDRx Settings set to High Dynamic Range, Blend also attenuates the artefacts created by scene motions. Set to 1 unless you are troubleshooting motion artefacts.

Exposure Offset slider Set how much greater the Primary exposure was compared to the Highlight exposure, in units of stops. Only available when HDRx Settings box is set to High Dynamic Range. The Exposure Offset should be set to match the setting on the camera for how many stops separate the Primary and Highlight tracks. This is typically 2 or 3 stops. Enter the same value used to record the footage or you will get bad results.

Highlight Threshold slider Set the threshold when pixels from the Highlight exposure are used instead of the over-exposed pixels from the Primary exposure. Only available when HDRx Settings box is set to High Dynamic Range. Set Highlight Threshold last because import options such as ISO or FLUT lighten or darken the image. Too high and you get clipping (often including a magenta-coloured cast), too low and the midtones and shadows have noise leaking in from the Highlight track.

Offset From File button Enable to apply the exposure offset embedded in the file's metadata rather than setting a value manually.

Selector Select the parameters within the respective list to copy the settings from one shot to another. See Copying Parameters with the Selector (page 1816).

Transcode Colour Settings

Adjust the values in this Colour settings tab to set the preliminary colour correction to your RED media.



For information on the Settings and Transcode options, see Transcode Format Settings (page 1879).

Colour Space option box Displays the value of the native colour space of the images, as set in the camera. Overiding this value may produce unexpected results.

Gamma Curve option box Displays the value of the output gamma curve that is applied to the shots.

Colour Science option box

Displays the version of the RED codec currently selected (2.X or 3.X). Using the version 3 of the codec gives you access to the FLUT and the Shadow options in the Colour Settings menu, and enables the Lift/Gamma/Gain menu. It also enables the Curve Settings for you RGB channels.

ISO option box Displays the value of the linear gain operation. RED images are always shot at 320 ISO.

Kelvin slider Displays the perceived colour temperature of the image.

Shadow slider Sets the shadow level.

Flut slider Refines the ISO level. As FLUT units are in stops, a +1 FLUT value is the same as doubling the ISO.

Tint slider Adjust the level of green or magenta in the shadow areas of the shot.

DRX slider Adjust the setting for the Dynamix Range Extension (DRX), which sets how much pixel data is copied from non-saturated channels into saturated channels.

Saturation slider Adjust the intensity of the colours in the footage.

Brightness slider Adjust the red, green, and blue levels across the entire image.

Contrast slider Adjust the gradation between the light and dark areas of the footage.

Exposure slider Displays the exposure increments, which are equivalent to f-stops.

Legacy Gain slider Modifies the red, green, and blue within the shot.

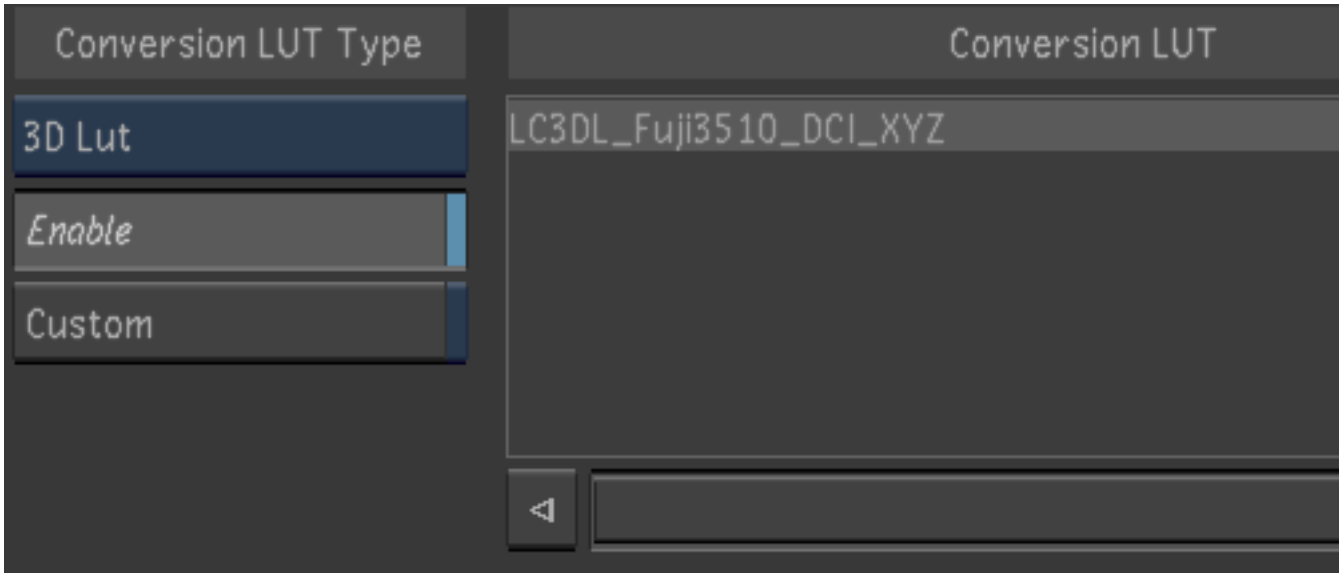
Curve Settings For more information on the curve settings, consult RED.

Transcode LUT Options Settings

From the Transcode / LUT Options menu, you can apply a LUT or colour transform on import to Gateway-imported media. Unlike Input LUTs (page 2026), the processing is performed in the Gateway rather than in Lustre.

NOTE If you are accessing media that was imported to a Smoke/Flame system with the Cache Source Media option enabled, that media is no longer considered Gateway media. It is seen by Lustre as managed media and hence the LUT and Resize on import options are not available. To work around this, you can perform a Flush Cache Media operation in Smoke / Flame, prior to importing in Lustre.

The LUT Options menu contains the following options:



Conversion LUT Type box Use to select the LUT Type you want to apply. Options are:

- 1D LUT
- 3D LUT
- Colour Transform

Enable button Activate to apply the selected LUT or colour transform.

Custom button Activate to build and apply a custom colour transform. See Creating a Custom Colour Transform (page 1885).

Conversion LUT list Click to select a LUT or colour transform to apply. Click again to deselect. This list is empty until you import LUTs or colour transforms using the Browse (...) button at the lower right. To remove an item from the list, select it and press the Delete key.

NOTE If you apply a colour transform that outputs floating-point data, the float conversion LUT is automatically applied after. See Working with Half Float Media (page 2037).

Applying a LUT or Colour Transform on Import

From the Image / Transcode / LUT Options menu, you can apply a LUT or colour transform to media imported using the Wiretap Gateway.

NOTE If you are accessing media that was imported to a Smoke/Flame system with the Cache Source Media option enabled, that media is no longer considered Gateway media. It is seen by Lustre as managed media and hence the LUT and Resize on import options are not available. To work around this, you can perform a Flush Cache Media operation in Smoke / Flame, prior to importing in Lustre.

1. From the Conversion LUT Type box, select 1D Lut, 3D Lut or Colour Transform.
2. Click Enable.
3. Select an item from the list, or click the Browse (...) button to locate and select a different LUT or colour transform. File paths are:
   - **1D LUTs:** /usr/discreet/flamepremium<version>/lut/

NOTE The Wiretap Gateway supports Flame-compatible 1D LUT files only. Lustre-compatible 1D LUT files are not supported.

- **3D LUTs:**
  - **Flame Premium:** /usr/discreet/flamepremium<version>/lut/Lustre_Color_3DLUTs/
  - **Lustre Standalone:** /usr/discreet/Lustre_Color/lut/Lustre_Color_3DLUTs/
- **Colour Transforms:** Click the Bookmark field to switch between the **Project**, **Shared**, and **Autodesk** locations. See Colour Management Files and Locations (page 1562) and Autodesk Colour Transform Types (page 1561).

  In addition to native .ctf files, you can also select files in several other supported LUT and transform formats.

Creating a Custom Colour Transform

You can create and apply a custom colour transform that is applied on import from the Image / Transcode / LUT Options menu. In addition, you can create and apply a custom colour transform that is applied when rendering from the Render / Output menu.

1. From the LUT Type box, select Colour Transform.
2. Click Enable.
3. Click Custom.
4. Click Add to insert an empty row.
5. Click the Type button and select a type:
   - **Shared:** The default location for shared custom colour transforms (available to all applications that use Autodesk Colour Management).
   - **Project:** The project transform folder (saved with the project).
   - **Autodesk:** The default location for preset colour transforms supplied with the application. See Autodesk Colour Transform Types (page 1561).
   - **Import:** Browse for colour transforms on your file system.
6. After you've selected a colour transform type, click the Transform button and select a transform. In addition to native .ctf files, you can also select files in several other supported LUT and transform formats.

   Information is displayed about the transform and the operations it contains.
7. To add more transforms to the end of the chain, repeat steps 4 to 6.

   You can also:
   - Select a row and click the Type or Transform buttons again to change a transform in the chain.
   - Select a row and click Delete to remove a transform from the chain.
   - Use the Up or Down buttons to reorder the chain. Moving a transform up applies it earlier in the chain, and moving it down applies it later.

   All transforms in the list are applied. The highlighted row indicates which transform will be affected by Up, Down, and other buttons. Information about the highlighted transform also appears in the metadata panel.

Exporting a Custom Colour Transform

Once you are satisfied with your custom colour transform, you can export it to file and recall it for later use.

1. Create a custom colour transform. See Creating a Custom Colour Transform (page 1885).
2. Click Export.

   The file browser appears.
3. Click the Bookmark field to select the destination location:
   - To store the transform with the project for archiving, save it to the Flame **Project** location (usr/discreet/<project>/synColor/transforms).
   - To make the transform available to other applications that use Autodesk Colour Management, save it to the **Shared** location. This location is stored in the synColor configuration file and can be set in the Flame LUT preferences. The default location is /opt/Autodesk/Synergy/SynColor/Shared/transforms.
   - Saving transforms to the **Autodesk** location is not recommended
4. Enter a file name.

   You can also enter a nickname and description. This information is included in the metadata that is displayed in the Description box in the Lut Options menu.
5. Click Export.

Transcode Resize Options

From the Image / Transcode / Resize menu, you can apply a resize to Wiretap Gateway imported media, without the need to transcode the media. The Resize is dynamically applied to the imported media. Adjust the resize values to define the destination resolution and the framing of the imported media.

NOTE If you are accessing media that was imported to a Smoke/Flame system with the Cache Source Media option enabled, that media is no longer considered Gateway media. It is seen by Lustre as managed media and hence the LUT and Resize on import options are not available. To work around this, you can perform a Flush Cache Media operation in Smoke / Flame, prior to importing in Lustre.



Output Resolution: Use to define the output resolution, aspect ratio, bit depth and scan mode of the transcoded media.

Resolution Field: Use to set custom resolution values.

Aspect Ratio Field: Select from a list of preset aspect ratios or set your own value.

Bit Depth Field: Use to define the bit depth of transcoded media.

Scan Mode Field: Use to define the scan mode of transcoded media.

Resize Options: Use to define the Fit Method, Resize Filter type, the Precision and the Crispness/Softness of the transcoded media.

Fit Method Box: Use to select the Fit Method applied to transcoded media. Options are:

- Fill (the default option)
- Crop Edges
- Letterbox
- Center/Crop

Resize Filter Type Box: Use to select the Resize Filter type applied to transcoded media. Options are:

- Impulse
- Triangle
- Mitchell
- Bicubic
- Quadratic
- Gaussian
- Shannon
- Lanczos

Aspect Button: Enable to use non-square pixel formats. This option is automatically enabled when Fit Method is set to Crop Edges or Letterbox.

Precision Field: Use to set the frequency cut-off point used during the Resize.

Crisp/Soft Field: Use to set the amount of blur used during the Resize.

Transcode Output Settings

Use the Output settings to set an output destination for your transcode job and to change how the timecode is used.



For information on the Settings and Transcode options, see .

Folders Structure option box Defines how the transcoded media is organized on the storage.

Select:	**To:**
Reel/Resolution	Establish a <reel name>/<resolution> folder structure. This is the default option.
Reel/Shot/Resolution	Establish a <reel name>/<file name>/<resolution> folder structure.

Timecode option box Select which RED media timecode to transcode. Note that when conforming an EDL, overiding the timecode data of the clips with a different timecode track will prevent relinking in the Autodesk Visual Effects and Finishing applications.

Select:	**To:**
Primary	Select either the Time of Day or Edgecode timecode. Only select this option if you know which timecode has been set as the primary.
Time of Day	Establish a timecode field that is set by the internal clock from the RED camera.
Edgecode	Establish a timecode field that is set by the camera operator.

Media Creation Order option box Defines the high-resolution and proxy media transcoding order.

Select:	**To:**
Proxy/Hires	Transcode the proxy media for all the shots first, and then the high-resolution media is transcoded.
Hires/Proxy	Transcode the high-resolution media for all the shots first, and then transcode the proxy media.
Hires only	Transcode only the high-resolution media. This is the default and recommended media creation option. This allows you to work on the native proxy as the high resolution is transcoded.

Select:	To:
Proxy only	Transcode only the proxy media.

Image Format list Select the type of file you want the RED footage to be transcoded to.

- Dpx
- OpenEXR

Bit Depth option box Select the bit depth to transcode your RED media to. The default is 10 bit.

Shot Media Hires/Proxy Transcoded button

Enable to view the Hires/Proxy transcoded media and disable to view the original RED media.

Transcoded Hires/Proxy fields Enter, or browse to, the location of the transcoded RED media directory. This directory must be an existing directory that is located on either a Wiretap server or a Wiretap Gateway. For Linux workstations, use the backward slash.

NOTE The Folders Structure, Media Creation Order, and Image Format options are not applied if you are transcoding to the Wiretap server.

Transcode Versioning box

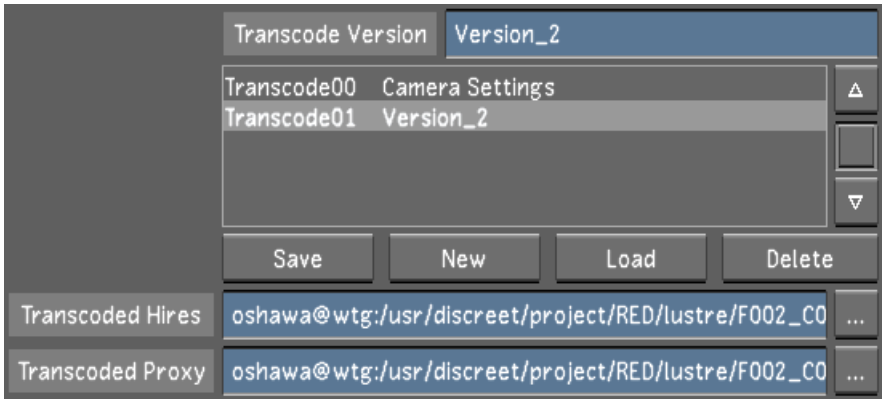
Use to save, update, load and delete Transcode versions to apply to your media. See Transcoding Versions.

Transcode to:	Type:
Wiretap Server	*<server_name>@wt:/stonefs/Test_transcode/Transcode*
Wiretap Gateway	*<server_name>@wtg:/mnt/md0/transcode*

Transcode Versioning

You can create and store multiple transcode settings for any given RED/ARRIRAW source. All of the transcode settings and their versions are stored in the Library source clip. Because the settings are stored in the Library source clip, you can update the transcode settings in a shot and share those settings with all shots originating from the same source clip. Each shot is still free to use its own version of the transcode settings. You can also share the transcoded media with other timelines that use the same shot, eliminating the need to transcode footage that has already been transcoded.

You can use the Transcode Version box to create, update, load and delete transcode versions.



To create a transcode version

1. Select a shot from your storyboard.
2. Type a name for your transcode version in the Comment field.
3. Click the New button.
4. Adjust your transcode settings.
5. Click the Save button.

NOTE

- You must click New before you start editing the settings, otherwise you will be editing the currently selected Transcode version.
- The default version of the transcode settings is labeled "Transcode00 Camera Settings". These settings are from the camera, as stored in the metadata of the R3D/ARRIRAW file. These settings can be recalled at any time by enabling the Camera button.

To load a Transcode version

1. Select a shot from your storyboard.
2. Select a Transcode version from the list.
3. Press the Load button.

To update a Transcode version

1. Select a shot from your storyboard.
2. Select a Transcode version from the list.
3. Adjust your Transcode settings.
4. Click the Save button.

NOTE When you modify the transcode settings of a shot, all shots originating from the same source clip are updated with the new settings and the changes will immediately be visible on all shots sharing the updated transcode settings.

To delete a Transcode version

1. Select a Transcode version from the list.
2. Click the Delete button.

Transcoding Template

You can save multiple transcode versions in a transcoding template. Once all settings have been established for your different transcoding versions, you can choose to save these settings as a template.You can create, load or delete a transcoding template.

Creating a New Transcoding Template

When you are working on a RED/ARRIRAW media project, you may want to apply the same transcoding settings to new footage as you import them. Being able to apply the settings from a previous session can ease the transcoding process.

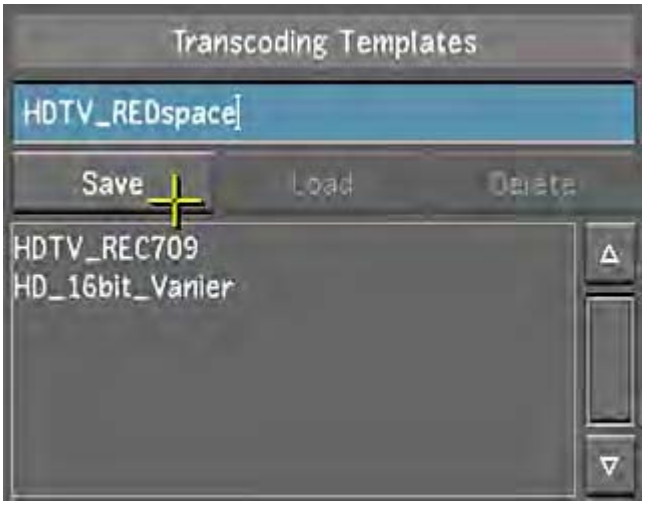
There are two ways to create a new template:

- By setting the values, saving your transcode versions and saving a new template.
- By loading a previously-saved template, making the necessary changes to the settings, and saving the new template.

To create a new transcoding template:

1. Set the values and save your transcode versions.
2. In the Transcoding Template field, enter a name and click Save.

   NOTE You should enter a name that will help you distinguish what type of settings you have selected.



To create a new transcoding template from an existing template:

1. From the Transcoding Template list, select the template to use as baseline.
2. Click Load.

   WARNING Rename the new template, or you will overwrite the original template with the new settings.

3. Click Save.

Your template is saved.

Loading a Transcoding Template

To load a transcoding template:

1. From the Transcoding Template list, select the template you want to apply.
2. Click Load.

Your template is loaded.

Deleting a Transcoding Template

To delete a transcoding template:

1. Select the template you want to remove from the Transcoding Template list.
2. Click Delete and then confirm.

Your template is deleted.

Conform

Working with EDL, ALE, and Cutlist Files

You can automatically recreate a timeline as it was edited during the offline editing stage of post-production provided you have all the source footage that was used, and a CMX3600-formatted EDL, ALE, or Cutlist file that describes how the source footage is arranged in the timeline. The process of rebuilding the timeline is called *assembling*.

When Lustre loads an EDL, ALE, or Cutlist file, it builds the timeline based on the following information:

- Source In/Source Out timecode or Record In/Record Out timecode.
- Tape/reel name. If the Use Reel Name button is enabled, Lustre verifies that the sources come from the correct tape. The tape name is part of the file path, and is always located one level above the resolution directory. For example, if the source files are located in *.../Scans/myclip/001/2058x1556/...* , then the tape name is 001.
- Source table. The source table enables the support of EDL reel names longer than eight characters.
- Dissolves. When Lustre finds a dissolve in the EDL, ALE, or Cutlist file, it places a dissolve of the specified duration at the appropriate transition.
- Speed changes. When a speed change is encountered in an EDL, ALE, or Cutlist file, the speed of the appropriate event is adjusted in the timeline using the Retime option.

To assemble an EDL, ALE, or Cutlist file, the system matches shot timecode values to source timecode in the EDL, ALE, or Cutlist file. Shot timecode values can be interpreted using two methods:

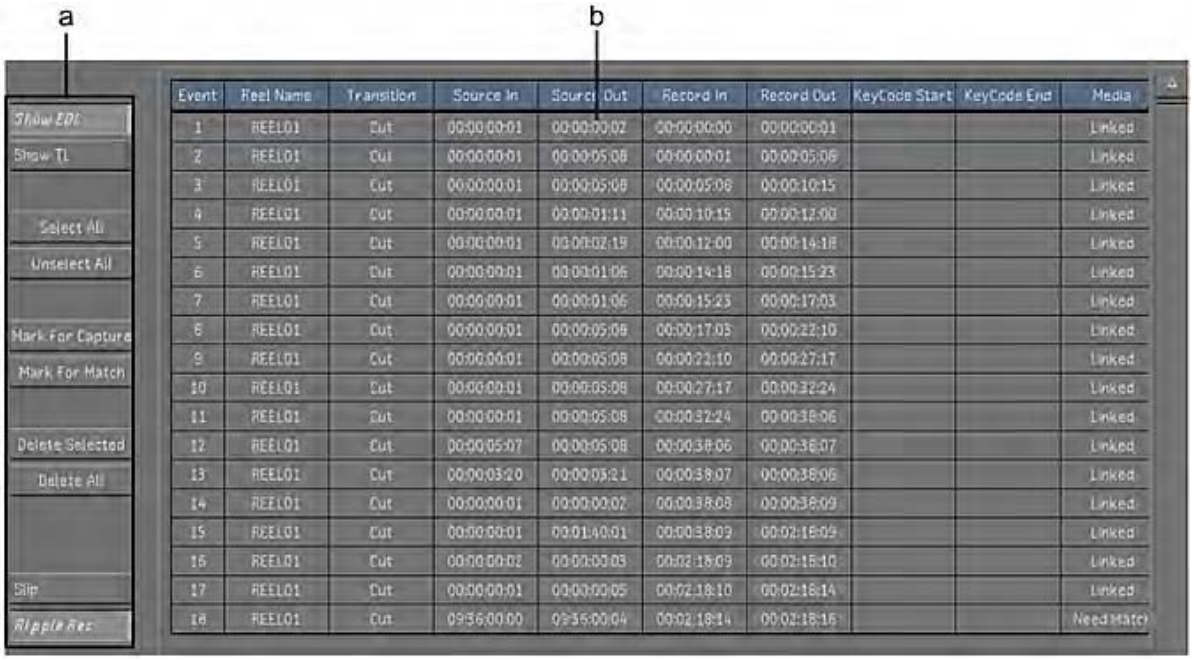
- By reading the timecode from the DPX file header. This method is intended for file formats (such as DPX files) that can contain embedded timecode values in the file header.
- By converting the shot's filename into timecode. The filename is established during the film scanning process.

The following table illustrates how a filename is converted into timecode.

Filename	Timecode 25 fps
00000.dpx	00:00:00:00
00024.dpx	00:00:00:24
00025.dpx	00:00:01:00
00060.dpx	00:00:02:10
00600.dpx	00:00:24:00
06000.dpx	00:04:00:00
06001.dpx	00:04:00:01

EDL Display and Functionality

When you load an EDL, ALE, or Cutlist file, its events appear in the Assembly window, located in the upper-left portion of the interface.



(a) EDL event buttons (b) EDL events

TIP Clicking a header will sort the EDL according to the selected data type.

Use the following buttons to facilitate EDL event management.

Click:	To:
Show EDL	View the currently loaded EDL.
Show TL	View timeline shot information. In order to display shot information for the currently loaded EDL, you must first assemble it.
Select All	Select everything in the EDL event list.
Unselect All	Deselect all selected events.
Mark For Capture	Flag the currently selected event for capture. See Metadata Maintained for Missing Shots (page 1905).
Mark For Match	Flag the currently selected event to be matched to a shot in the Library.
Delete Selected	Delete selected events. **NOTE** This button is only functional when the timeline is not displayed.
Delete All	Delete all events. **NOTE** This button is only functional when the timeline is not displayed.
Slip	Offset the shot's Source In/Out or Record In/Out timecodes without affecting the length of the shot. If you offset the Source In timecode, the Source Out timecode will be adjusted automatically, and vice versa. The same applies to Record In and Record Out timecodes.
Ripple Rec	Apply modifications made to a shot's Source In timecode to the Record Out timecodes for all subsequent events in the EDL.

EDL and Keycode Assembly Workflow

The recommended steps for EDL or Keycode assembly are as follows.

Step:	Refer to:
1. Access the file browser.	Accessing the File Browser (page 1856).
2. Load shots into the Library.	Loading Clips to the Library (page 1874).
3. Add CDL data to an EDL file to associate CDL data to individual events (shots).	Adding CDL Data to an EDL File (page 1895).
4. Copy the EDL into Lustre.	Importing an EDL, ALE, or Cutlist File (page 1897).

Step:	Refer to:
5. Assemble the EDL to automatically build your footage into a timeline.	Assembling an EDL (page 1898).
6. Optional: Trim or edit the shots in the timeline as needed. If you perform this step, you may have to export the timeline as an EDL.	Editing (page 1912), and Exporting Shots to an EDL (page 1903).
7. Save the assembled EDL as a cut.	Creating a New Cut (page 1774).

Example of a Possible CDL Workflow

To facilitate the transfer of colour intent from the production to the post-production world, many productions do on-set grading. This means that simple colour grading is done at the same time as production. Here is a simplified sequence of events explaining the process:

- Production is shooting content on film or digital.
- Film is processed.
- Film is scanned.
- DPX files are copied to FireWire® drive for production.
- Colourist on set uses third party application to grade DPX.
- CDL data is created (as EDL comment or XML files).
- Post facility receives CDL and starts dailies grading/preview or DI based on CDL (on-set) intention.
- Offline editorial receives dailies for editorial.
- EDL are generated for conform.
- Colour grading process starts.

Adding CDL Data to an EDL File

You can add CDL data to an EDL file either inline (CDL slope, offset, and power data embedded directly in the EDL), or through an XML reference.

An XML reference can be either a *.ccc* or *.cdl* file; both have one single XML file with multiple shot information. However, Lustre will only read the first SOP grouping (data for a single shot) of colour correction from either of these files. Therefore, users must have one XML file for each event.

The CDL specification defines CMX EDL comments as the transport mechanism. CDL data can be expressed as EDL data with the CDL values or as an XML file.

Storing Inline CDL Parameters as Notes in an EDL File

Add CDL data directly to an EDL file to transport simple primary grading data to the Lustre colour grading application.

The ASC_SOP note associates all nine ASC CDL colour values with the edit that precedes the note. The first three numbers are the R, G, and B values of the slope parameter. The second three numbers are the RGB values for offset. And, the last three numbers pertain to the RGB values for the power parameter.

Each parameter's three values are grouped by parentheses, and a space will separate the grouped parameters from each other and the Note ID string ("*ASC_SOP"). This note limits the parameter values to five digits of precision. This allows all nine parameters to fit into one 80- column note. An example of the inline CDL data added to the EDL file is as follows.

Each of the numbers that represents a colour value of a parameter should be in the form of a two to five digit number with a decimal point placed anywhere between the most and least significant digits (for example, from 0.0 or 0.0001 to 9999.9). This prevents the decimal from being missed or misinterpreted. The numbers that represent the offset parameters may have an optional leading minus sign. There is a leading space at the head of each number, except for the first value of each parameter (the Red value), which has a leading left parenthesis instead. The last value of each parameter (the Blue value) has a trailing right parenthesis. There is a space between the NOTE ID String ("*ASC_SOP") and the first left parenthesis.

To manually add CDL data to an EDL file (inline method):

1. Open your EDL file in a text editor.
2. Open the file with raw CDL data.
3. In the CDL data file, copy the SOP information for the first shot.
4. In the EDL file, enter the CDL data in a new line after the line identifying the clip name.

   NOTE Make sure you begin the new line with the *ASC_SOP note ID.

5. Perform this procedure for each shot for which you have CDL data.

Storing CDL Parameter References to ASC CDL XML Files or ASC CCC XML Files

The ASC_CC _XML note begins with the '*ASC_CC_XML' note ID string and is followed by the file name that associates that shot with a unique XML file. This XML file follows the XML implementation of the ASC CDL. An example of the reference line added to the EDL file is as follows.

```
010  216R05   V     C        05:40:12:18 05:40:14:09 01:00:29:16
01:00:31:07
FROM CLIP NAME: 0064
*ASC_CC_XML cc1009
```

NOTE In the previous example, the reference to 'cc1009' is a reference to the XML file cc1009.cdl (if it's a CDL XML file), or cc1009.ccc (if it's a CCC XML file).

NOTE The reference to the XML file must not include the '.cdl' or '.ccc' suffixes in the EDL file.

WARNING If you use both CDL and CCC files, make sure you do not assign the same name for any CDL file to a CCC file. If Lustre finds a CDL file with the name that is referenced in the EDL file, it will only read the CDL file, even if a CCC file with the same name exists.

To manually add CDL or CCC file references to an EDL file:

1. Open your EDL file.
2. In the EDL file, find the shot information for the shot corresponding to the XML file you wish to reference.
3. Enter the XML file reference information corresponding to the shot in a new line after the line identifying the clip name.

   NOTE Make sure you begin the new line with the *ASC_CC_XML note ID.

4. Perform this procedure for each shot for which you have a corresponding XML file (either a **.cdl* or **.ccc* file).

Importing an EDL, ALE, or Cutlist File

Before you can assemble an EDL, ALE, or Cutlist file, you have to copy the file into the current scene's *Library* directory.

NOTE The file must have a **.edl*, **.ale*, or **.ctl* file extension or it will not appear in the EDL list.

There are two ways to import EDLs into Lustre:

- Outside of Lustre, files can be copied manually to the *Library* directory of the current scene. If running the Windows version of Lustre, save the EDL to the *C:\Program Files\Autodesk\Lustre<version>\projects\<project_name>\Library* folder. If running the Linux version of Lustre, save EDLs in the */usr/autodesk/lustre_<version>/projects/<project name>/Library* directory.

  NOTE Press `Ctrl+R` to refresh the EDL list to see recently copied files.

- Within Lustre, copies of EDL files are imported using the Import file browser.

To import a copy of an EDL, ALE, or Cutlist file while in Lustre:

1. Click Editing, and then click Assemble.
   The Assemble menu appears.



(a) EDL list (b) Browse button

2. In the EDL List, click the Browse button.
   The Import file browser appears.



3 Navigate to the appropriate source folder and select one or more EDL files (You can also use bookmarks (page 1862) to retain a list of folder locations). Traditional methods of multi-selecting (`Shift`-click, `Ctrl`-click) are enabled.

4 Click Import.

WARNING If an EDL with the same filename is already in the EDL list, the Import button will turn red. You must click again to confirm that you want to overwrite the file.

TIP If no files are selected, you can also press `Enter` to cancel file selection and close the Import file browser.

A copy of the file is imported into the *Library* folder, and appears in the EDL list.

Assembling an EDL

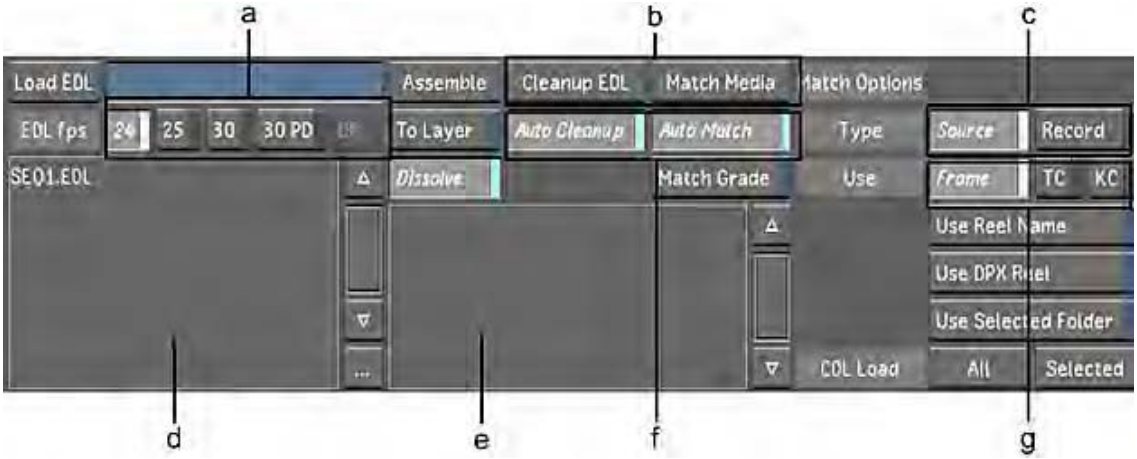
Use an EDL to rebuild a timeline using original source footage that matches the edited timeline from the offline editing stage of post-production. It is also possible to add the assembled EDL to a new layer of the current timeline.

To assemble an EDL:

1 Load the shots that you want to assemble into the Library.

2 Click Editing, and then click Assemble.

The Assemble menu appears.



(a) Frame Rate buttons (b) Manual options (c) Base of assembly buttons (d) EDL list (e) Log window (f) Assemble options (g) Timecode buttons

3 (Optional) Import an EDL into the EDL list. See Importing an EDL, ALE, or Cutlist File (page 1897).

4 Select the EDL you want to load from the EDL list.

5 Determine the base of assembly by enabling one of the following.

Source Recreates a timeline according to how the source footage was arranged.

Record Assembles the timeline using the timecode of the final edited clip.

NOTE If you selected Record as your base of assembly and your shots include dissolves, enable First-Last Key Anim Type on the Setup page of the Animation menu. See Synchronizing Keyframe Functionality (page 2188).

6 Click the Frame Rate button that corresponds to your EDL's frame rate. The following frame rates are supported.

Enable:	To:
24	Load a 23.98 or 24 fps EDL.
25	Load a 25 fps (PAL) EDL.
30	Load a 29.97 or 30 fps (NTSC) EDL.
30 PD	Convert an NTSC EDL to 24 fps and load the converted EDL.
DF	Load a 29.97 fps drop frame mode (NTSC) EDL.

7 Enable Match Name to match the timecode and tape name in the EDL to your shots.

Lustre tries to detect the Tape name in the EDL to the shots' directory path. This path is fixed—the correct directory should be one level above the resolution directory. For example, if the tape name in the EDL is BTE164, the matching shot is at:

<HOME>\scans\BTE164\2048x1556\0002487.dpx

TIP To match the EDL reelname to the reelname in the DPX header, enable the Use Reel Name button in the System & Menu page of the user configuration. See System & Menu Settings (page 1762).

8 Enable Dissolve to include any dissolves in the EDL.

NOTE Only dissolve transitions are supported when loading an EDL. Other transitions are replaced by cuts.

WARNING Continuing this process creates a new cut from the EDL. If you have not saved your work, it will be lost. Therefore, it is a good idea to start from an empty cut before loading an EDL.

9 Enable either Frame or Timecode depending on how you want Lustre to read the shot's timecode when assembling the EDL.

- Enable Frame to convert the shot filename to a timecode value.
- Enable Timecode to use the timecode value stored in the file header.

NOTE For information on Keycode assembly, see Keycode Assembly (page 1904).

10 Click Load EDL.

The system checks to make sure that the EDL is a valid CMX3600-formatted EDL. If the EDL is not valid, a message appears in the Load log and the EDL is not loaded.

NOTE ALE and Cutlist files are also valid. See Keycode Assembly (page 1904).

11 To perform an EDL cleanup, do one of the following:

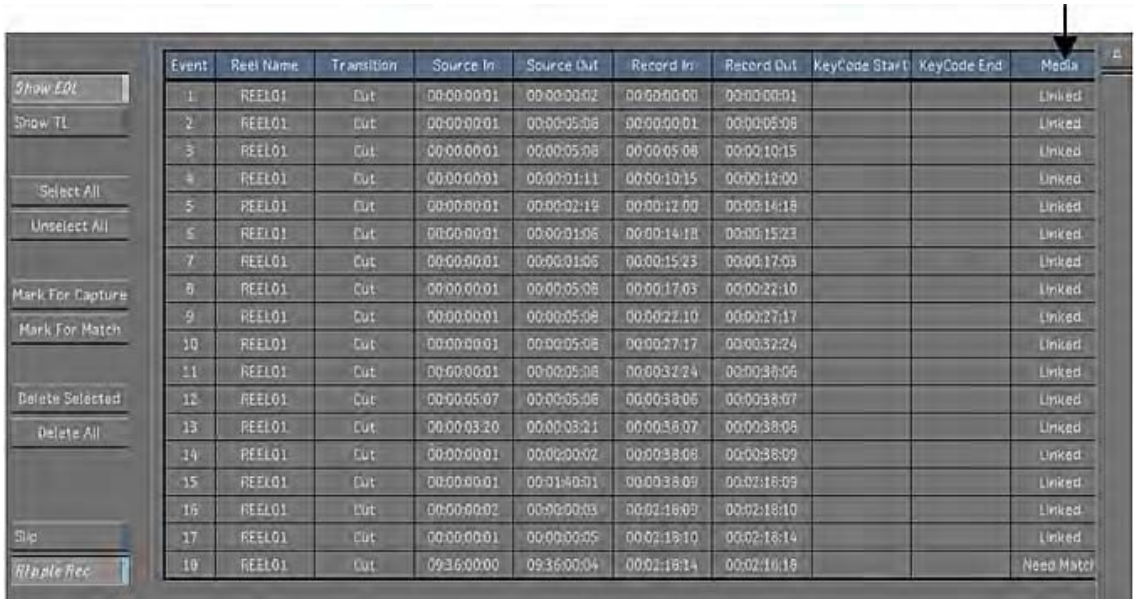
- Click EDL Cleanup to perform cleanup prior to assembling.
- Enable Auto Cleanup to automatically perform cleanup during assembly.

When Lustre performs an EDL cleanup, it arranges EDL events according to record timecode and deletes problematic events. For example, if an EDL has two events with conflicting timecodes, the event with the lower number is deleted during cleanup.

12 To match EDL event timecodes to shot timecodes in the Library, do one of the following:

- Click Match Media to match EDL event timecodes to available shot timecodes in the Library prior to assembling.
- Enable Auto Match to automatically match EDL event timecodes to available shot timecodes in the Library during assembly.

The matching process links EDL event timecodes to media timecodes in the Library. The match status is indicated in the Media column of the Assembly window. Successful matches are indicated by a Linked status, whereas unsuccessful matches generate a Need Match status.



NOTE It is not necessary for all shots to be available prior to conforming. For example, shots with a Need Match status can be captured after the initial conform and assembled with the rest of the EDL later. In the meantime, media metadata will be maintained and black frames will be added where the shots would have appeared. You can also update shots after conforming. See Metadata Maintained for Missing Shots (page 1905).

13 Enable or disable the To Layer depending on how you want to assemble the new cut.
- Enable To Layer to assemble a cut to a new layer above the existing layer(s) in the currently loaded cut.
- Disable To Layer to overwrite the currently loaded cut with the newly assembled cut.

14 Click Assemble to conform the EDL.

NOTE If a timeline has not been loaded, then a cut is automatically created with the same name as the EDL.

NOTE When a timecode gap occurs between EDL events, black frames are inserted between the events to fill the gap.

15 (Optional) To manually match shots in the Library to EDL events, use the Replace Shot tool to add the correct footage into the blank shots. See Replacing Shots (page 1926).

After working on a previously assembled EDL, to ensure the same grading is applied to the newly assembled EDL, you need to apply the match grade option. See Match Grade (page 1901).

NOTE You can also assemble an EDL with CDL data. See Assembling an EDL with CDL Data (page 1911).

To assemble an EDL to the currently opened Timeline:

1 In the Editing menu, click Assemble.
2 Enable To Layer.
3 Load the EDL.

The EDL is now assembled to a new layer in the current Timeline.

NOTE The record timecode of the EDL is used as the in point when assembling an EDL to a layer of an existing Timeline.

To load the EDL as a new Timeline:

1 In the Editing menu, disable To Layer.
2 Load the EDL.

The EDL is loaded to a new Timeline. If there is already an existing cut, it replaces the current cut with the newly assembled EDL.

Match Grade

The Match Grade feature allows you to apply current grade settings to a new assembled timeline without having to create a new cut file and use the Change Cut option. For information about the Change Cut option, see Changing a Cut (page 1779).

When you enable Match Grade during a timeline assembly, the function applies the appropriate grade to any shots that are also included in the current cut. Match options are also available to allow you to specify the basis of the match.

To apply the current grade settings to a new assembled timeline:

1. Make sure a graded cut is already loaded into the timeline.
2. To specify the basis of the match, select one or more Match Options from the Browse menu.



UID Performs matching based on the shot's unique ID.

NOTE UIDs are regenerated every time a new cut file is created in Lustre.

Source Performs matching based on source data, such as a shot's source timecode, ID, and tape/reel name.

Record Performs matching based on record data, such as an EDL's record timecode.

Custom Performs matching based on the keyword that is selected in the custom list.

If the Custom option is enabled, there is a list of keywords to which you can associate the custom option.

Select:	To:
EDLReelName	Match the reel name of the EDL.
FolderReelName	Match the folder reel name.
DPXReelName	Match the DPX reel name.
DLEDLClipName	Match the clip name of the shot coming from the Wiretap server.
DLEDLOrigin	Match the image import path and file name of the timeline coming from the Wiretap server.
DLEDLSourceId	Match the media source's unique ID of the timeline coming from the Wiretap server.
DLEDLSegmentId	Match the segment's unique ID of the timeline coming from the Wiretap server.
DLEDLStartTc	Match the start source timecode of the timeline coming from the Wiretap server.
DPXKeycodeStart	Match the DPX header keycode start.
DPXKeycodeEnd	Match the DPX header keycode end.

Select:	To:
EDLComment	Match the unique comment that is applied to the original cut from another application.

3 In the Assemble menu, enable Match Grade.

4 Assemble the EDL, ALE, or CTL file.

5 Click Browse to display the Browse menu.

6 Save the new cut file.

7 Click Setup, and then click Grade to display the Grade menu.

8 Click New Version to save the grade with the newly associated cut.

When working with a multi-layer timeline and Solo mode is disabled, the layers are flattened and only the grade settings from the shots that are visible in the Storyboard are applied to the newly assembled timeline. When working with a multi-layer timeline and Solo mode is enabled, only the grade settings from the active layer are applied to the newly assembled timeline. If the new timeline has multiple layers, then the grade setting is applied to each layer.

Exporting Shots to an EDL

You can export your cut file as a CMX3600 EDL. The exported EDL is saved with the name of the currently loaded cut, plus a **.edl* file extension, and is placed in the scene's *Library* directory.

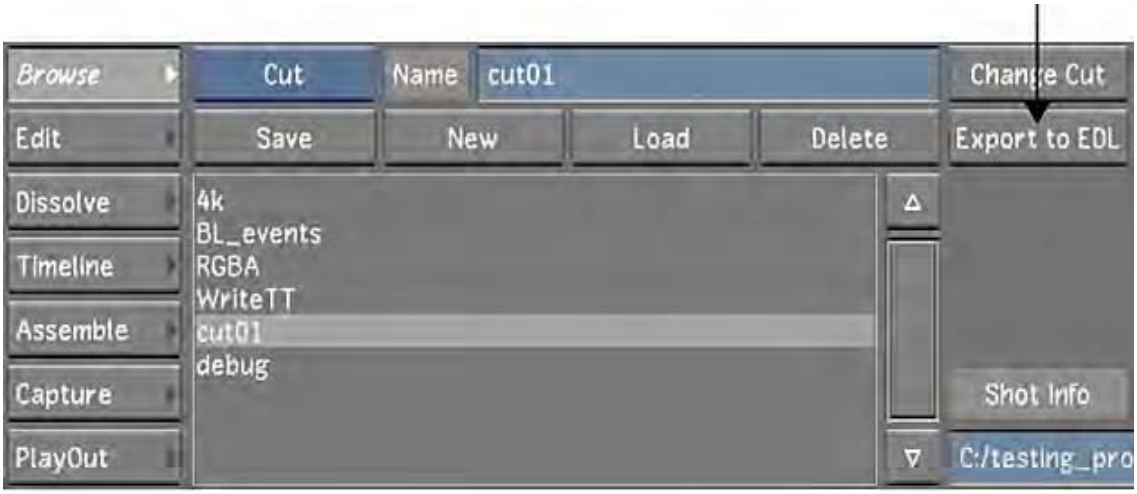
When exporting a multi-layer timeline and Solo mode is disabled, the layers are flattened and only the shots that are visible in the Storyboard are exported. When exporting a multi-layer timeline with Solo mode enabled, only the shots in the soloed layer are exported.

To export a timeline as an EDL:

1 Click Editing, and then click Browse to display the Browse menu.

2 If it is not already loaded, load the cut you want to export. See Managing Cuts (page 1773).

3 Click Export to EDL.



A CMX3600 EDL is created from the shots in the current timeline and placed in the scene's *Library* directory.

Keycode Assembly

You can load ALE (**.ale*) and Cutlist files (**.ctl*) as EDLs in the Assemble menu. Once loaded, you can use their keycode information for media assembly.

Most keycode types are fully supported in ALE and Cutlist file assembly. However, if Lustre encounters an unknown keycode type, it cannot parse its information. In these cases, add the missing keycode type name and value manually to the *init.config* file, located in *C:\Program Files\Autodesk\<version number>* on the Windows version of Lustre, or in */usr/autodesk/<version number>* on the Linux version of Lustre. For example, if the missing type is FN and the keycode value is 123456, you would enter the following in the *init.config* file:

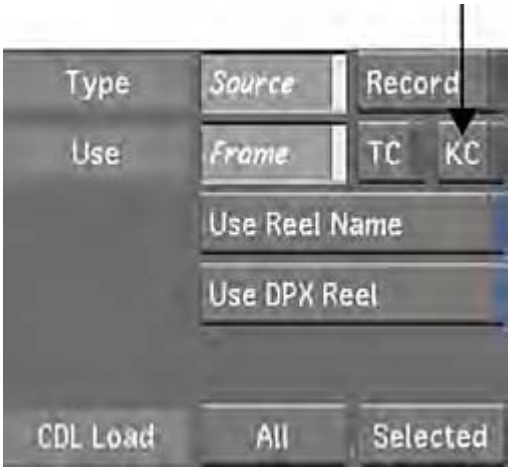
FN 123456

To load and assemble ALE and CTL files:

1. Load the shots that you want to assemble into the Library.
2. Click Editing, and then click Assemble.
3. (Optional) Import an EDL into the EDL list. See Importing an EDL, ALE, or Cutlist File (page 1897).
4. Select the ALE or Cutlist file from the list.



5. Select the appropriate EDL Frame Rate.
6. Click KC to assemble media according to the file's keycode information.



7 Click Load EDL.

The system checks to make sure it is a valid ALE or Cutlist file. If not, the file is not loaded.

8 To perform a cleanup on the ALE or Cutlist file, do one of the following:

- Click EDL Cleanup to perform the cleanup prior to assembling.
- Enable Auto Cleanup to automatically perform cleanup during assembly.

When Lustre performs a cleanup on an ALE or Cutlist file, it arranges events according to record timecode and deletes problematic events. For example, if an ALE file has two events with conflicting timecodes, the event with the lower number is deleted during cleanup.

9 To match ALE or CTL event timecode/keycode to shot timecode/keycode in the Library, do one of the following:

- Click Match Media to match event timecode/keycode to available shot timecode/keycode in the Library prior to assembling.
- Enable Auto Match to automatically match event timecode/keycode to available shot timecode/keycode in the Library during assembly.

The match status is indicated in the Media column of the Assembly window. Successful matches are indicated by a Linked status, whereas unsuccessful matches generate a Need Match status.

NOTE It is not necessary for all shots to be available prior to conforming. See Metadata Maintained for Missing Shots (page 1905).

10 To manually match shots in the Library to ALE or CTL events, use the Replace Shot tool to add the correct footage into the blank shots. See Replacing Shots (page 1926).

11 Click Assemble to conform the ALE or CTL file and build the cut.

Metadata Maintained for Missing Shots

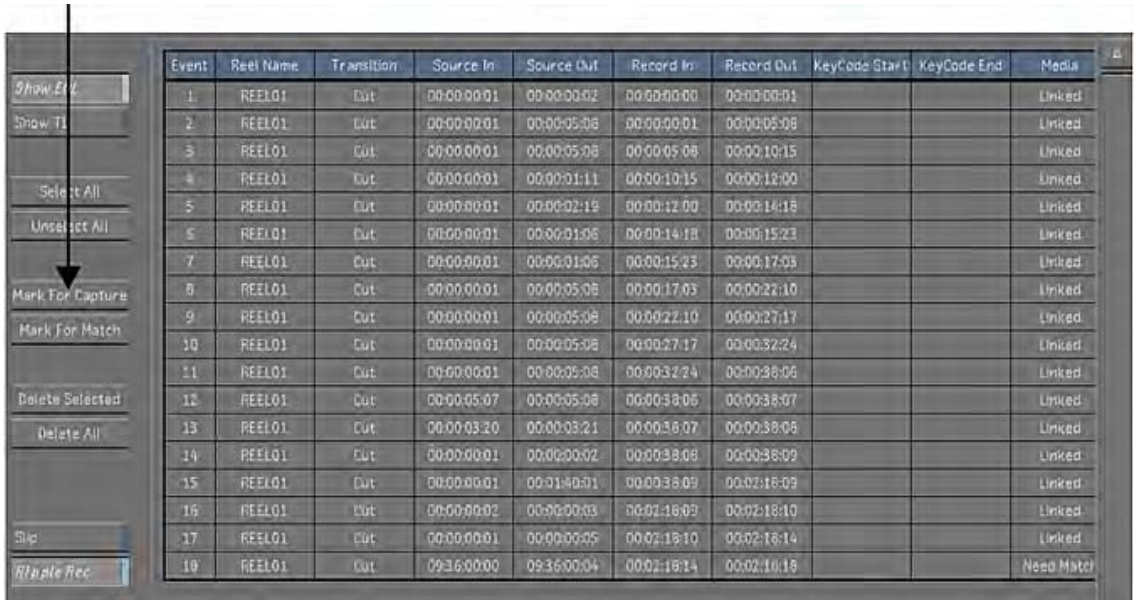
If certain shots in the EDL, ALE or Cutlist file are not yet available, or if you would like to update shots that are already linked to events with new shots, you can mark them for capture. Shots can be individually captured either before or after the initial conform. Metadata for missing shots will be maintained for inclusion at a later time, and black frames will be inserted in the timeline where the shot should be. In the meantime, work can begin on the available shots.

To mark missing shots for capture:

1 In the Editing menu, click Assemble to display the Assemble menu.

2 In the Assembly window, `Ctrl`-click each event to be marked for capture.

3 Click Mark For Capture.



4 In Editing, click Capture to display the Capture menu.

5 In the Capture Format group, select the video file format you want to create during the capture operation. See Capture Menu Options (page 2258).



6 In the Naming group, enter values in the Capture, Tape, and Resolution fields.

7 Click EDL, and then confirm the action.



The capturing session begins.

The tape currently loaded in the VTR is ejected. You are prompted to enter the first tape needed in the EDL. Once you enter the tape, Lustre begins capturing automatically. The words "Capture from tape..." appear in the Player. There is no video displayed in the Player during capture.

In the Log window, information is displayed about the capture process. If more than one tape is required, you are prompted when it is time to enter the next tape.

8 In Editing, click Match Media.

9 To assemble the EDL to include the newly-captured shots, click Assemble.

Colour Decision List (CDL)

The American Society of Cinematographers Colour Decision List (ASC CDL) colour correction specification defines a common language format for primary colour correction. It is becoming a standard defined by the ASC and has been adopted by most major manufacturers.

CDL is expressed by nine numbers (three RGB triplets) representing predefined colour functions: offset, slope, and power. The format of CDL is standardized so results in one colour corrector can be duplicated in another corrector, so long as they both support the ASC CDL format. Currently, ASC CDL only supports primary colour correction, with the exception of chrominance (saturation change).

CDL Menu

The CDL menu allows you to:

- Change the four basic transfer functions (slope, offset, power, and saturation).
- Select whether you would like the transfer functions to be applied before, or after, an Input LUT is applied.
- Enable or disable the transfer functions you have set.

To access the CDL menu:

1 In the main menu, click Image.

2 Click CDL.

The CDL menu is displayed.



NOTE The CDL panel is not accessible from the ACS or the CP-100.

TIP You can increase or decrease the value quickly by holding `Shift` and dragging the slider, or slowly by holding `Alt` and dragging the slider.

Slope

When you change the slope transfer function, it allows you to change the lift and gain of the image without shifting the black level that is established by the offset. The following is a graph displaying how the slope value affects your image.

The following are the values to set the slope:

- Default value is 1.00 (i.e., no change).
- Minimum value is 0.00.
- Maximum value is 5.00.

Offset

The offset transfer function either raises or lowers the overall brightness of the component, while keeping the slope constant.

The following are the values to set the offset:

- Default value is 0.00 (i.e., no change).
- Minimum value is -5.00.
- Maximum value is 5.00.

Power

The power transfer function changes the intermediate shape and is the only non-linear function.

The following are the values to set the power:

- Default value is 1.00 (i.e., no change).
- Minimum value is 0.00.
- Maximum value is 5.00.

Saturation

The saturation transfer function adjusts the intensity of the colours.

The following are the values to set the saturation:

- Default value is 1.00 (i.e., no change).

- Minimum value is 0.00.
- Maximum value is 5.00.

Pre/Post Input LUT

You can decide to apply the CDL data either before (Pre) or after (Post) applying an Input LUT.

NOTE Applying CDL data after applying an Input LUT will work only if the Input LUT is enabled within the Input LUT module.

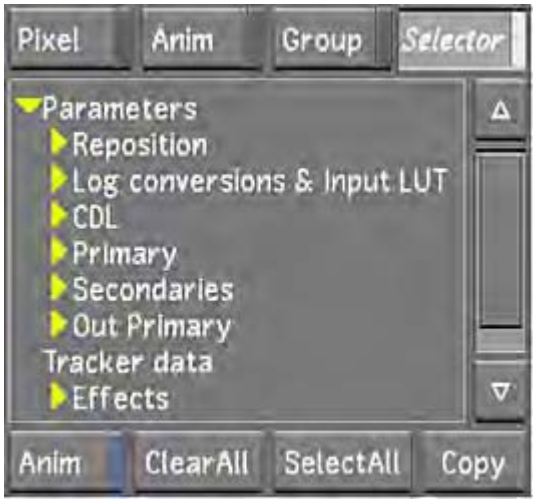
CDL On/Off

The CDL On/Off toggle allows you to either enable or disable CDL data. Since the Bypass All function does not bypass both the Input LUT and CDL, you would use the CDL On/Off button to 'mute' the effect of the CDL.

CDL Parameter in the Selector

The Selector contains the following CDL parameters:

- Slope (RGB sliders)
- Offset (RGB sliders)
- Power (RGB sliders)
- Pre/Post Input LUT
- CDL On/Off



You can use the Selector to:

- Select the CDL section you want to copy/paste from shot to shot.
- Selectively load CDL data saved in the individual grades that are saved in the Grade bin.

Copying Parameters with the Selector

Lustre allows you to copy CDL parameters with the Selector in the same way as they would copy grading parameters. The parameters the Selector can copy are as follows:

- Colour grading parameters

- Input LUT selections
- CDL parameters
- Repositioning values
- Tracker data
- Effects
- Render flag settings

Loading CDL Data in the EDL Panel

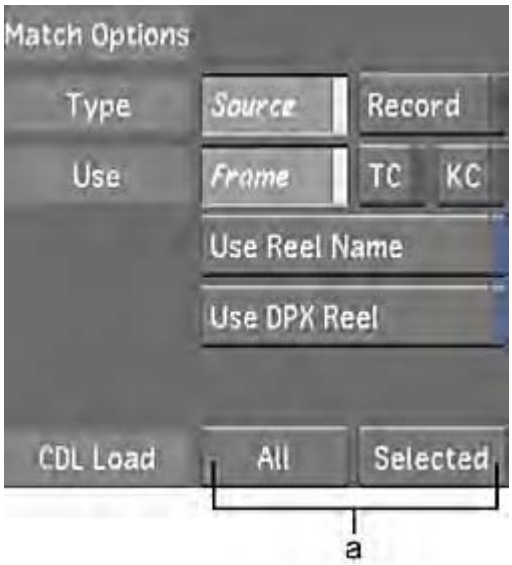
The EDL panel contains an interface to support loading CDL data.

When you load an EDL into Lustre, the CDL data is not converted at that time. You can conform and assemble the EDL as you normally would do. However, if you want to use the CDL data of the currently loaded EDL, you need to use the CDL Load options to apply the CDL data to the cut file.

CDL Load Options

When you are ready to use CDL data, you can select one of two actions listed in the CDL Load list:

- All: Clicking All reads all CDL data from all the EDL events in the EDL Editor and applies the converted data to the events on the timeline by matching the record timecode (REC TC) value.
- Selected: Based on the selected events in the EDL Editor, the Selected action reads the CDL data of the selected events, then converts and applies each one of them to the events on the timeline by matching the REC TC value.



(a) CDL Load options

NOTE Loading CDL data does not erase the current grade applied to the shots. Loading CDL data only loads the data into the CDL panel. If the CDL data is already loaded, it will be overwritten with the newly loaded CDL data.

You can use a different EDL to conform and a different EDL to import CDL data. To load data from another EDL, `Ctrl`-click a CDL Load button. The data will be loaded using the Match Options from a cut selected in the Cut Name list. See Changing a Cut (page 1779).

Assembling an EDL with CDL Data

The following procedure assembles an EDL with CDL data.

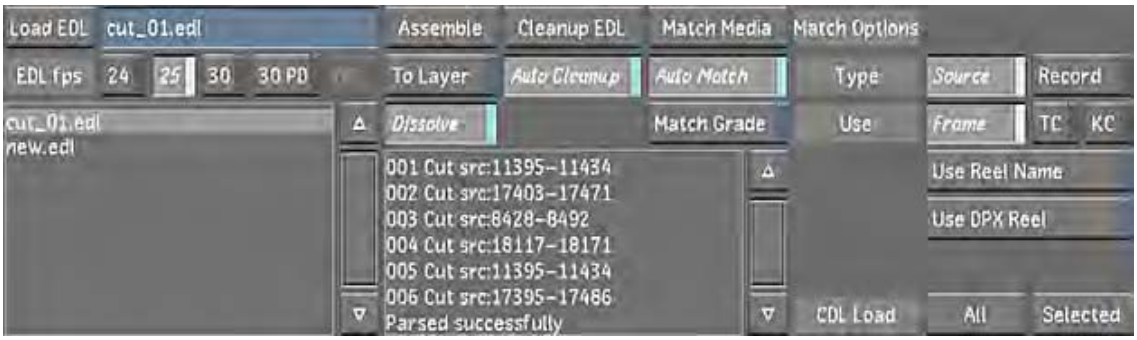
To assemble an EDL with CDL data:

1. Drag the shots that you want to assemble into the Library.
2. Click Editing, and then click Assemble.
   The Assemble menu appears.



(a) EDL list

3. Select the EDL that contains the CDL data that you want to load from the EDL list.
4. Click Load EDL.
   The selected EDL loads.



NOTE For more information on assembling an EDL, see Assembling an EDL (page 1898).

5. Click Match Media to match the EDL event timecodes to the available shot timecodes in the Library prior to assembling.
6. Click Assemble to conform the EDL and build the cut.
   The EDL events in the Assembly window are cleared.
7. Select the same EDL and click Load EDL.
8. Click CDL Load All or CDL Load Selected. See CDL Load Options (page 1910).
   The CDL data is now part of the current cut.

CDL EDL Comment

CDL data for a given shot is represented as a comment in the EDL file.

```
0001 VT029     V     C        09:11:16:20 09:11:22:22 01:00:13:05 01:00:19:07
*ASC_SOP (1.0 1.0 1.0) (0.0 0.0 0.0) (1.0 1.0 1.0)
```

There is one CDL comment line per event. See Adding CDL Data to an EDL File (page 1895).

CDL XML File

CDL data for a given shot is represented as a reference in the EDL file to an XML file.

```
0003 VT032     V     C        12:11:43:21 12:11:52:07 01:00:27:05 01:00:35:15
*ASC_CC_XML cdn03345
```

NOTE The ASC CDL specification allows more than one colour correction data set per XML file. Lustre only supports a single colour correction data set per CDL and XML file. See ASC CDL specifications for more detail.

See Adding CDL Data to an EDL File (page 1895).

CDL XML Files Location

To be able to use XML files, place them in an import sub-folder located in the project's Library folder:

<my project>/<scene 1>/<Library>/<import>

Editing

Use the Editing tools in Lustre to make modifications to your cuts, for example, to move, trim, cut, delete or replace shots. You can also add dissolves between shots, perform a confidence check, or use scene detection. Typically, you perform editing operations before colour grading. In this way, the colourist works on only the frames that are destined for the final master.

Perform a confidence check to detect changes in scenes. A confidence check is a process in which you manually compare the frames in an assembled EDL to frames captured from an offline digital cut. With a confidence check, you ensure that the scanned film frames match the frames that were edited together during the offline edit. This is an important step in guaranteeing that the final colour-graded timeline is in sync with the audio that appears in the final edited master.

With scene detection, you can take a long captured shot that spans multiple scenes and automatically introduce splices each time the scene changes. You can then grade the new shots separately. Do this, for example, when you are capturing a final edited HD master into Lustre for colour grading.

Representing the Timeline

The sequence of shots in a cut file is referred to as the timeline. Lustre has two visual ways of representing the timeline: the Storyboard and the multi-layer timeline.

When there is only one layer in the timeline, the Storyboard is simply the thumbnail view of the sequence of shots in the cut. When there are multiple layers in the timeline, the Storyboard is the thumbnail view of the flattened timeline, taking into consideration shots in a cut that represents a single-layer view of the timeline (see Top Vertical Priority (page 1958)). This single layer corresponds with one of the following, depending on the soloing and muting status:

- The active layer of the multi-layer timeline, where either the active layer is soloed or the remaining layers are muted
- The default (flattened) view of the multi-layer timeline, where soloing is disabled

The multi-layer timeline organizes the cut into a multiple layer format. The elements appear as a series of rectangles on a time-proportional grid, with a positioner at the location of the current frame. Multiple layers are useful for grade versioning, roughing out edit sequences, and for multi-layer editing. This allows you to try various edits and colour grading versions on your shots before choosing the one which is used in the final master.

For more information about:

- The Storyboard, see Editing with the Storyboard (page 1914).
- The multi-layer timeline, see Multi-Layer Timeline (page 1932).
- Solo mode, see Solo Mode (page 1963).
- Mute mode, see Muted Layers (page 1959).

Accessing the Edit Menu

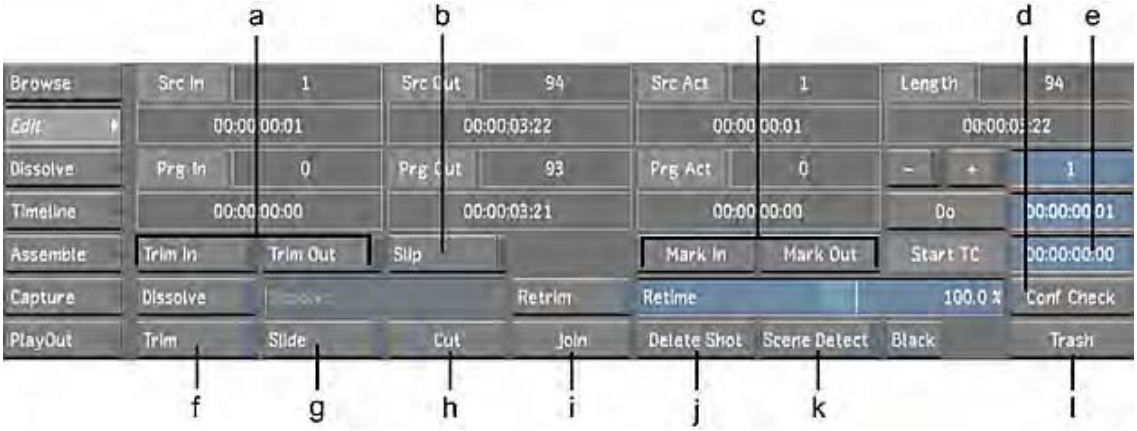
To modify shots in the timeline, you use the Edit menu.

NOTE If a layer is soloed, the Edit tools (Trash, Delete, Trim In/Trim Out, and Mark In/Out) default to 'Ripple End' behaviour, regardless of the Ripple Mode setting. If no layer is soloed, however, the Edit tools behave according to the individual Ripple Mode selected (Ripple Off, Ripple Start, or Ripple End) and apply to the active or visible layer that contains the focus point. For more information about Ripple Modes, see Trimming Ripple Mode (page 1954).

To access the Edit menu:

1. In the Main menu, click Editing.
2. Click Edit.

   The Edit menu appears.



(a) Trim In/Out buttons (b) Slip button (c) Mark In/Out buttons (d) Confidence Check button (e) Start TC field (f) Trim button (g) Slide button (h) Cut button (i) Join button (j) Delete Shot button (k) Scene Detect button (l) Trash button

The Edit menu is made up of the following buttons.

Trim In/Trim Out Allows you to enter the number of frames to trim at the in or out point for the current shot. See Trimming the Head and Tail of a Shot (page 1918).

Slip Moves the source frames within a shot without adjusting the shot's position in the timeline. See Slipping a Shot (page 1921).

Mark In/Mark Out Sets the in or out point of the current shot at the Current Frame positioner in the Shot timebar. See Changing In and Out Points for a Shot (page 1922).

Start TC Sets the frame number or timecode at which you would like the timeline to begin. If you enter a frame number, it is converted into the corresponding timecode value and vice versa. See Modifying the Timeline's Starting Record Timecode (page 1964).

Trim Interactively sets the shot's in and out points when you drag left and right in the viewers. Use the left viewer to trim the in point and the right viewer to trim the out point. See Trimming Interactively (page 1922).

Slide Slides a cut point forward or backward. See Sliding the Cut Point between Shots (page 1923).

Cut Cuts one shot into two at the Current Frame positioner. See Inserting Cuts (page 1924).

Join Joins two shots coming from the same source clip with consecutive source timecode. See Joining Shots (page 1925).

Delete Shot Deletes a shot from the timeline.

Scene Detect Starts a scene detection process. When changes in your footage are above a threshold, a cut is introduced. See Using Scene Detection (page 1966).

Confidence Check Allows you to compare your current timeline with a clip coming from a digital cut of your offline edit. See Performing a Confidence Check (page 1965).

Trash Deletes shots from the Library or Storyboard when you drag them to the Trash button.

Global Access to Editing Hotkeys

When you enable the Editing While Grading button in the User settings, you can access the following editing hotkeys from any Lustre menu except the colourist multi-layer timeline.

Press:	To:
`Shift+C`	Cut the shot in two at the current frame.
`Shift+J`	Join the shot that was cut in two.
`Shift+T`	Start interactive visual trimming.
`Shift+R`	Start interactive visual sliding between the current and the previous shot.
`Shift+S`	Start a scene detection.

For more information about the Editing While Grading button, see System & Menu Settings (page 1762).

Editing with the Storyboard

The Storyboard is a thumbnail representation of the shots in a cut and provides a Storyboard view of the active timeline layer. For a timeline with multiple layers, if Solo mode is disabled (this is the default), the Storyboard always displays the topmost layer. If Solo mode is enabled, the Storyboard displays the layer according to the location of the focus point. Lustre includes an assortment of tools for adding shots to, and rearranging shots in, your Storyboard. Once you place a shot in the Storyboard, you can modify it.

Adding Shots to the Storyboard

You can add shots or timelines to the Storyboard by dragging them from the Library. You can also drag shots or timelines directly from the file browser and drop them into the Storyboard. In this case, the shots are also placed into the Library (if they are not already there).

TIP To add shots to the Storyboard where the insertion marker is positioned, enable Solo mode in the Timeline menu. If you do not do this, added shots are automatically appended to the end of the timeline. See Multi-Layer Timeline (page 1932).

To add shots to the Storyboard:

1. Select shots in the Library.

   The order in which you click shots in the Library determines their order in the Storyboard.

   TIP If you select the wrong shots, click any empty grey area under the Storyboard. This releases the selected shots from the cursor.

2. Drag the shots to the Storyboard.

   A yellow insertion marker appears on the Storyboard.

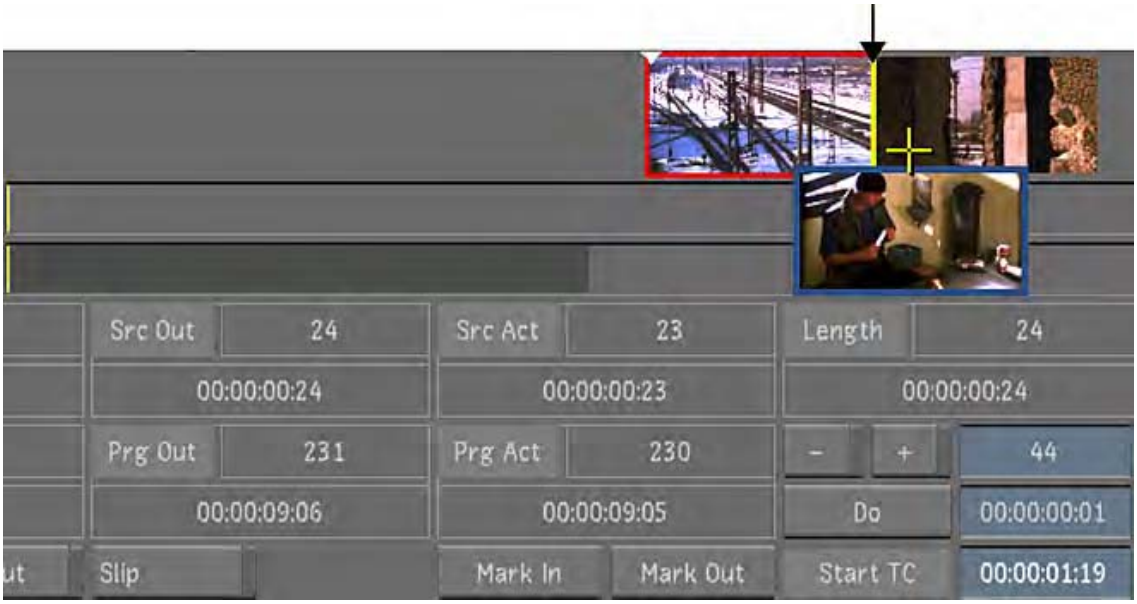


Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

   As you move the cursor back and forth along the Storyboard, the insertion marker moves.

3. Click to add the shots to the Storyboard.

   The shots are added to the end of the Storyboard if Solo mode is disabled or to the location of the insertion marker if Solo mode is enabled.

Reordering Shots in the Storyboard

You can reorder the shots in the Storyboard.

NOTE Solo mode in the Timeline menu must be enabled.

To reorder a shot in the Storyboard:

1. In the main menu, click Editing.
2. Click Timeline.
   The Timeline menu appears.
3. Enable Solo.
4. Drag the shot you want to move.
   The shot becomes partially transparent.
5. Drag the shot elsewhere in the Storyboard.
   A yellow insertion marker shows where the shot will be placed.
6. Release the mouse, then click it again, to complete the edit.

   TIP If you select the wrong shots, click the grey area to the left of the Trash button. This releases the selected shots from the cursor.

Duplicating Shots in the Storyboard

You can duplicate shots in the Storyboard.

NOTE Solo mode in the Timeline menu must be enabled.

To duplicate shots in the Storyboard:

1. After adding a shot to the Storyboard, drag the shot from the Library a second time to create a duplicate.

   TIP If you select the wrong shots, click the grey area to the left of the Trash button. This releases the selected shots from the cursor.

   TIP To insert a duplicated shot to the insertion marker location of the Storyboard, you must enable Solo mode in the Timeline menu. See Multi-Layer Timeline (page 1932). Otherwise, the duplicated shot is inserted at the end of the Storyboard.

Deleting Shots

You can delete shots from the Library, Storyboard, or multi-layer timeline (see Deleting Elements (page 1941)). Deleting shots from the Library does not delete them from the Storyboard or multi-layer timeline, and vice versa.

Be aware that Timeline Solo and Ripple mode affects how the shots are deleted in the Edit menu. If Solo is enabled, the Trash and Delete tools default to Ripple End behaviour. If Solo is disabled, the Trash and Delete tools behave according to the Timeline Ripple mode (see Trimming Ripple Mode (page 1954)) and apply to the layer that the focus point is on.



(a) Delete Shot button (b) Trash button

To delete a shot from the Storyboard:

1. Click Editing, and then click Edit to display the Edit menu.
2. Select the shot, and then do one of the following:
   - Click Delete Shot and then confirm the deletion.
   - Click and drag the shot to the Trash button, and then confirm the deletion.
   - Press `Shift+D`.

   The selected shot is deleted.

Trimming from the Edit Menu

Use the Trimming tools to adjust the frames used in your edits. When you trim, you are either adding frames to or removing frames from the head or tail of the shots in your cut.

Be aware that timeline Solo and Ripple mode affects how the shots are trimmed in the Edit menu. If Solo is enabled, the Trim In/Trim Out and Mark In/Out tools default to Ripple End behaviour. If Solo is disabled, the Trim In/Trim Out and Mark In/Out tools behave according to the Timeline Ripple mode (see Trimming Ripple Mode (page 1954) and Slip & Slide Ripple Mode (page 1955)) and apply to the layer that the focus point is on.

You can trim by entering exact frame values, by dragging frames interactively, or by marking in and out points. You can also trim by sliding the cut point between shots or slipping the frames used in a shot.

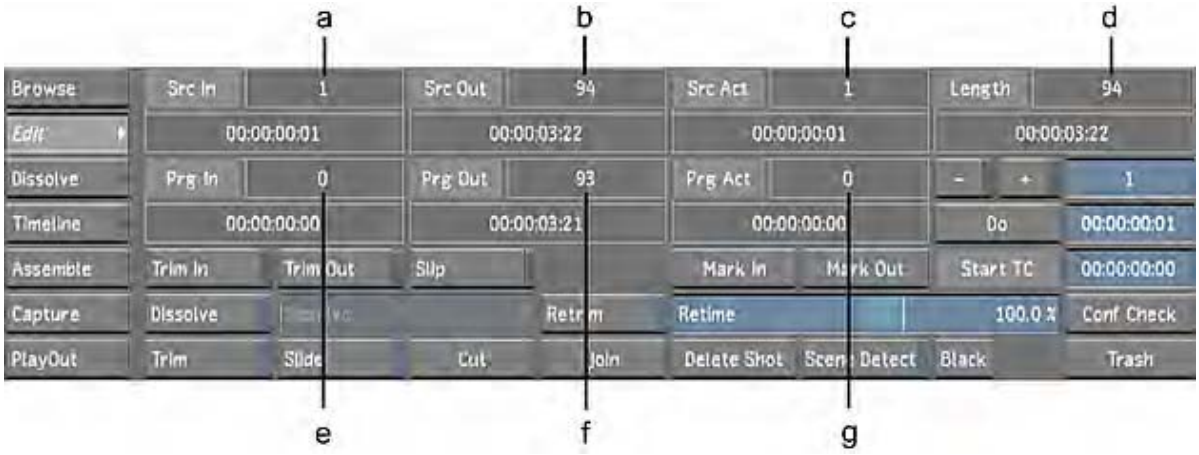
Viewing Shot Information

When trimming, you will occasionally need information about the frames and timecode in your shots and cuts.

You can find shot information in the Edit menu.

For example, refer to the Src In field to determine if you can add a few frames to the start of the shot, or check whether you changed the length of the timeline by checking the value in the Length field.

You cannot enter values directly in these fields. They are updated when you scrub through the shot or Storyboard. For each field, the actual frame number is displayed, as well as the corresponding timecode value.



(a) Src In field (b) Src Out field (c) Src Act field (d) Length field (e) Prg In field (f) Prg Out field (g) Prg Act field

Use the following fields to view shot information.

Src In Displays the selected shot's Mark In frame number and timecode.

Src Out Displays the selected shot's Mark Out frame number and timecode.

Src Act Displays the absolute frame number of the current frame in the source material.

Length Displays the total length of the current shot in the timeline in frames and timecode.

Prg In Displays the program in point for the selected shot.

Prg Out Displays the program out point for the selected shot.

Prg Act Displays the current program frame number of the timeline.

NOTE The appropriate fields are highlighted when you enter an editing mode, indicating where editing operations affect your frames.

Trimming the Head and Tail of a Shot

You can trim frames from the head or tail of one or more shots. The head is the first source frame that you specify to use in the shot. The tail is the last source frame that you use in the shot.

You specify the number of frames to trim from either the head or tail. If you do not want to change the overall length of your timeline, when you add frames to, or subtract frames from, one end of a shot, you should subtract from or add the same number of frames to the other end.

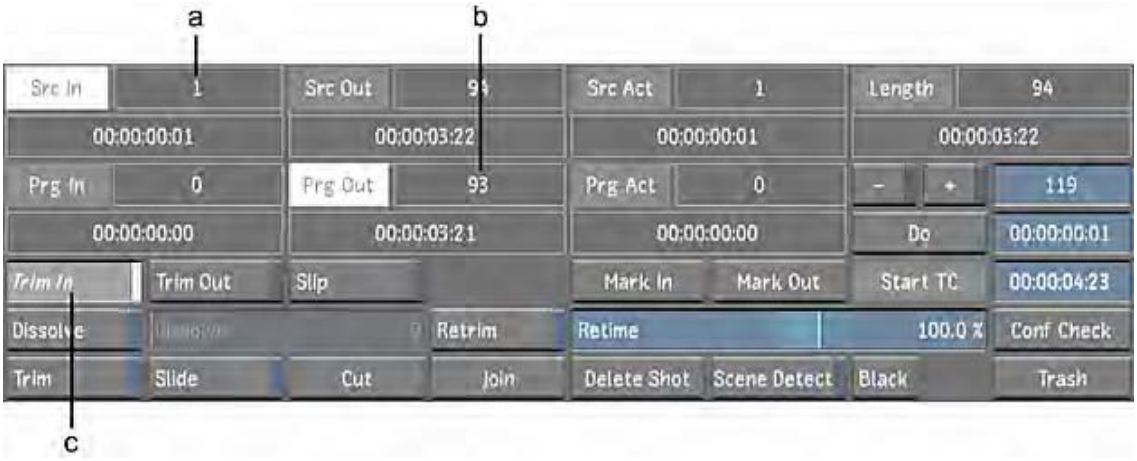
Remember that you cannot add more frames to a shot than the available number of handles. Also, you cannot trim a shot completely out of existence. If you want to delete a shot, use the Delete Shot button. See Deleting Shots (page 1916).

To trim frames at the in point for one or more selected shots:

1. Select the shot you want to trim in the Storyboard. The selected shot's border turns red. To select multiple shots on the Storyboard, right-click each shot. The borders turn aqua.

   NOTE You can only add frames if your shots have available handles.

2. Click Editing in the main menu, and then click Edit.
   The Edit menu appears.



(a) Src In field (b) Prg Out field (c) Trim In button

3 Click Trim In to go into Trim In mode.

Src In and Prg Out are highlighted because these values are changed by the Trim In operation.

4 Add frames to or remove frames from the mark in (or head):

- Click + to add frames.
- Click - to subtract frames.



(a) - (minus) button (b) + (plus) button (c) Frame field (d) Do field (e) Do button

NOTE If you do not click either + or -, the trim operation removes an absolute number of frames.

5 Do one of the following:

- Click the Do field, then click the frame field and enter the number of frames to add or subtract by using the numeric keypad. The frame number is converted into the corresponding timecode value.
- Click the Do field and enter the timecode for the number of frames to add or subtract. The timecode is converted into the corresponding frame number.

  NOTE `Ctrl`-click the field to clear it.

6 Click Do.

The specified number of frames is added to or subtracted from the shot(s). The Length field is updated to display the number of frames now used.

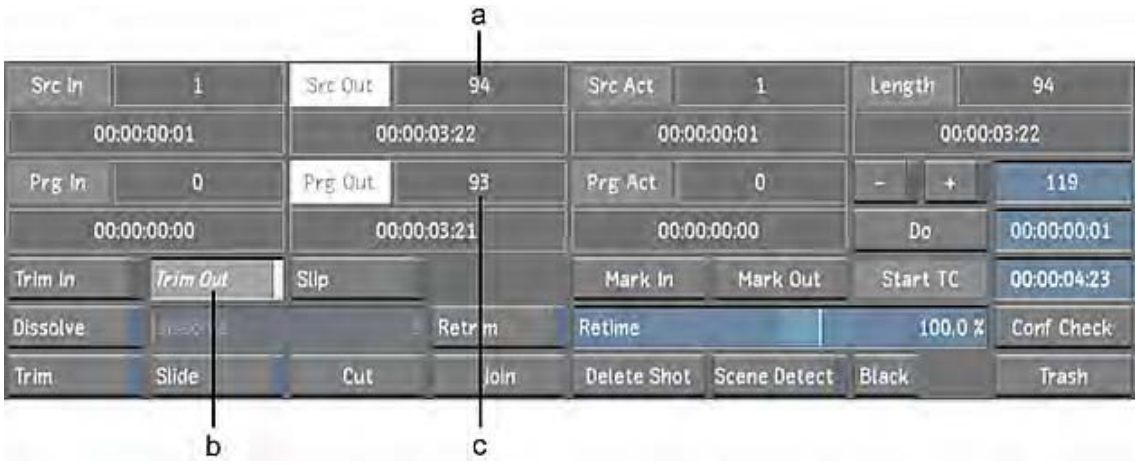
To trim frames at the out point for one or more selected shots:

1 Select the shot you want to trim in the Storyboard. The selected shot's border turns red. To select multiple shots on the Storyboard, right-click each shot. The borders turn aqua.

NOTE You can only add frames if your shots have available handles.

2 Click Editing in the main menu, and then click Edit.

The Edit menu appears.



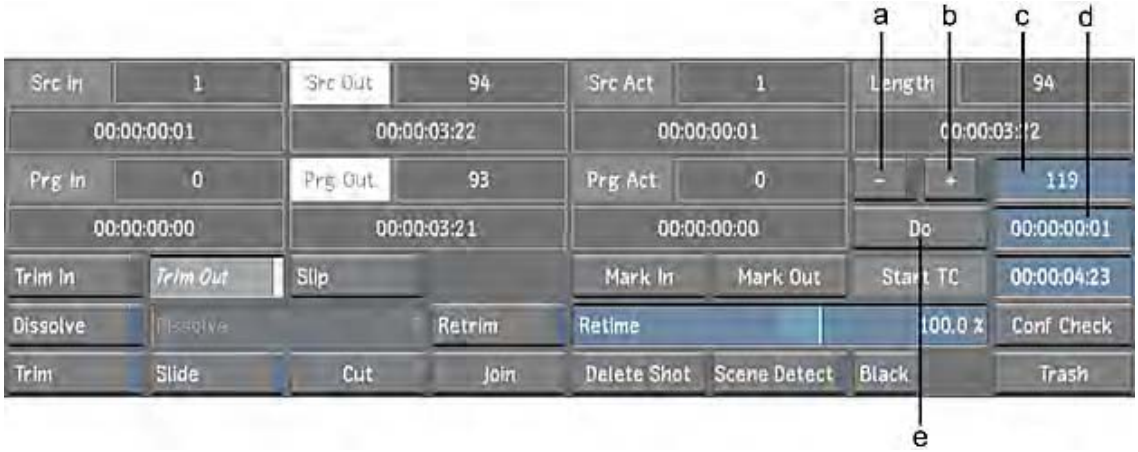
(a) Src Out field (b) Trim Out button (c) Prg Out field

3 Click Trim Out to go into Trim Out mode.

Src Out and Prg Out are highlighted because these values are changed by the Trim Out operation.

4 Add frames to or remove frames from the mark out (or tail):

- Click + to add frames.
- Click - to subtract frames.



(a) - (minus) button (b) + (plus) button (c) Frame field (d) Do field (e) Do button

NOTE If you do not click either + or -, the trim operation removes an absolute number of frames.

5 Do one of the following:

- Click the Do field, then click the frame field and enter the number of frames to add or subtract by using the numeric keypad. The frame number is converted into the corresponding timecode value.
- Click the Do field and enter the timecode for the number of frames to add or subtract. The timecode is converted into the corresponding frame number.

NOTE Ctrl-click the field to clear it.

6 Click Do.

The specified number of frames are added to or subtracted from the shot(s). The Length field is updated to display the number of frames now used in the shot.

Slipping a Shot

You can slip the frames used in one shot without adjusting the shot's position in the timeline. You may need to slip your shot to synchronize action in the picture to the audio or because unwanted frames are appearing at either the head or tail.

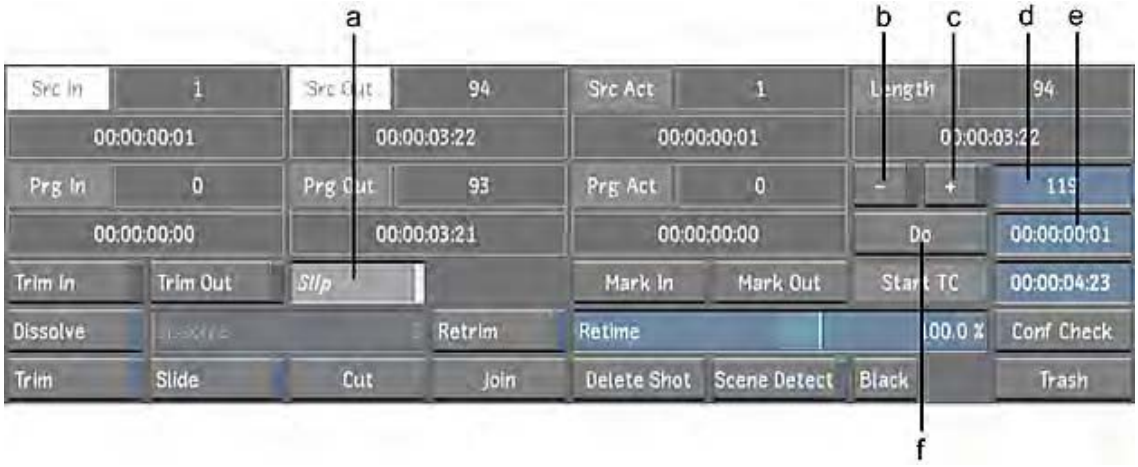
To slip one or more shots:

1 Select the shot you want to slip in the Storyboard. The selected shot's border turns red. To select multiple shots on the Storyboard, right-click each shot. The borders turn aqua.

NOTE You must have available handles to slip a shot.

2 Click Editing in the main menu, and then click Edit.

The Edit menu appears.



(a) Slip button (b) - (minus) button (c) + (plus) button (d) Frame field (e) Do field (f) Do button

3 Click Slip to go into Slip mode.

Src In and Src Out are highlighted.

4 Slip forward or backward:

- Click + to slip forward.
- Click - to slip backward.

5 Do one of the following:

- Click the Do field, then click the frame field and enter the number of frames you want to offset the shot by. The frame number is converted into the corresponding timecode value.
- Click the Do field and enter the timecode for the number of frames to offset the shot by. The timecode is converted into the corresponding frame number.

NOTE Ctrl-click the field to clear it.

6 Click Do.

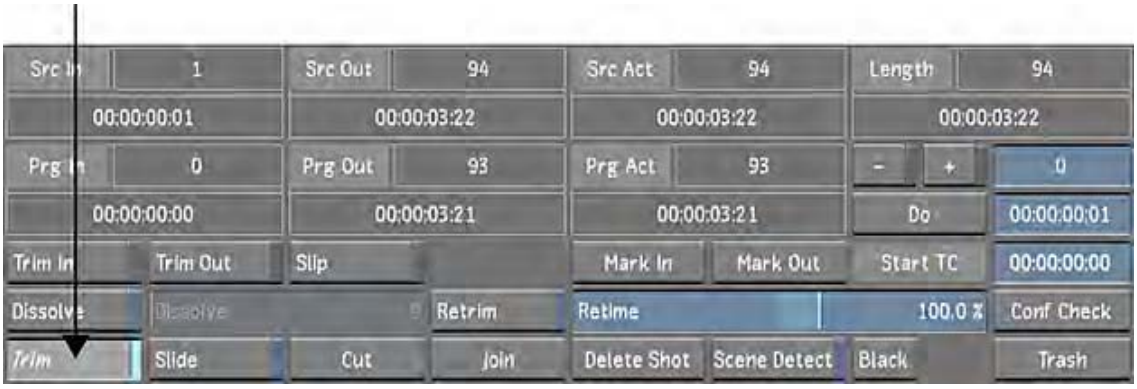
The shot or shots are offset by the specified number of frames. The Src In and Src Out fields are updated to display the frames used.

Trimming Interactively

You can trim frames interactively. When you trim interactively, you scrub the Player to add frames to, or remove frames from, the in or out point.

To trim interactively:

1. In the Storyboard, select the shot you want to trim.
2. Click Editing in the main menu, and then click Edit to display the Edit menu.
3. Enable Trim.



TIP If the Editing While Grading button is enabled in the User settings, you can start interactively trimming shots from any menu by pressing `Shift+T`. For more information about the Editing While Grading button, see System & Menu Settings (page 1762).

In Trim mode, you see the first frame of the shot in the left viewer, and the last frame in the right viewer.

4. Do one of the following:
   - To trim frames at the head, click the left viewer and then drag the cursor left to add frames or right to subtract frames.
   - To trim frames at the tail, click the right viewer and then drag the cursor right to add frames or left to subtract frames.
5. For greater control of the trim operation, middle-click as you drag to decrease the start frame by one frame or right-click to increase the offset by one frame.
6. To exit the Interactive Trim mode, do one of the following:
   - Click Trim.
   - Press the `Esc` key.

Changing In and Out Points for a Shot

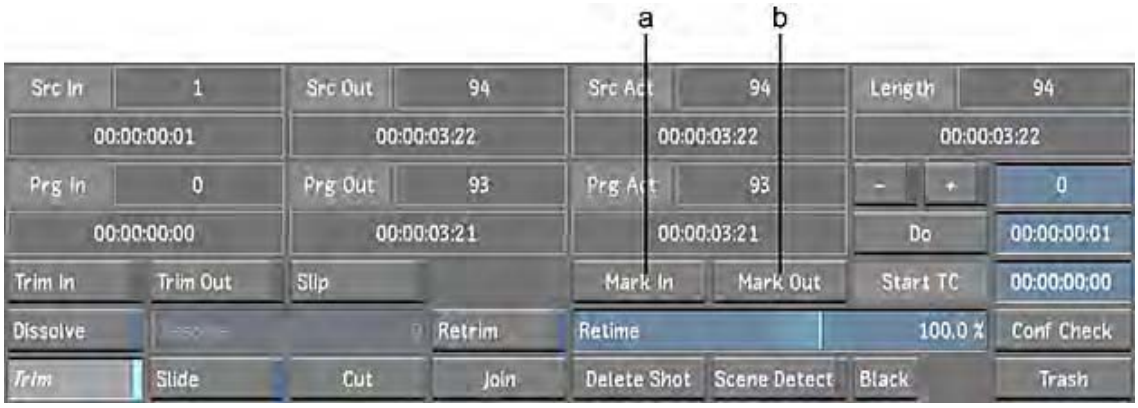
You can change the in and out points for a shot. This is like trimming, except you can only shorten a shot by changing the in or out points.

The in point is the first frame used in a shot. The out point is the last frame. Setting a new in or out point shortens the total length of a shot. The frames that you remove by changing the in or out point become available handles that you can use in later trimming operations.

To change the in or out point for a shot:

1. In the Storyboard, select the shot you want to modify, and then scrub to the frame you want to mark as the in point.
2. Click Editing, and then click Edit.

   The Edit menu appears.



(a) Mark In button (b) Mark Out button

3. Click Mark In.

   The extra frames are trimmed from the head of the shot.
4. Scrub to the frame you want to mark as the out point.
5. Click Mark Out.

   Extra frames are trimmed from the tail of the shot.

   NOTE You must go into one of the Trim modes to add frames back to the head or tail of the shot.

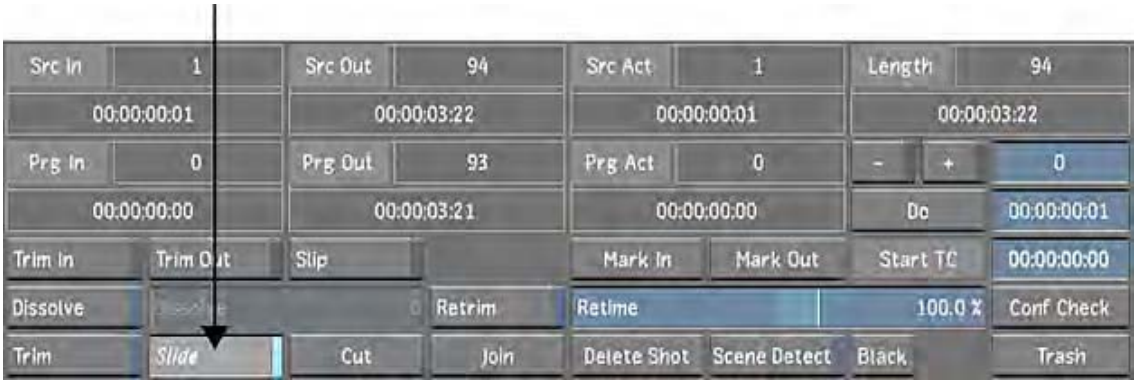
Sliding the Cut Point between Shots

You can interactively move the cut point between two shots forward or backward, if handles are available. Using Slide does not adjust the overall length of your timeline. By removing frames from one side of the cut, you automatically add frames to the other side.

You may want to slide the cut point to synchronize the cut to audio or simply to use different frames in your shot.

To slide a cut point:

1. In the Storyboard, select the shot to the right of a cut point.
2. Click Editing, and then click Edit to display the Edit menu.
3. Click Slide.



TIP If the Editing While Grading button is enabled in the User settings, you can start interactive visual sliding between the current and the previous shot from any menu by pressing `Shift+R`. For more information about the Editing While Grading button, see System & Menu Settings (page 1762).

In Slide mode, you see the outgoing frame before the cut in the left viewer and the incoming frame after the cut in the right viewer.

4 Drag in either viewer to move the cut point.

If both clips have available handles, drag to the left or right to move the cut point.

Frames are added to one side of the cut and subtracted from the other to keep the total timeline duration the same. When you run out of handles, you can no longer continue sliding in the same direction.

5 For greater control of the slide operation, middle-click the mouse to decrease the cut point by one frame or right-click to increase the cut point by one frame.

6 To exit the Interactive Slide mode, do one of the following:

- Click Slide.
- Press the `Esc` key.

The timeline is updated when you exit Slide mode.

Inserting Cuts

You can cut a shot into two shots. This is useful when you want to insert cuts between scenes, or if you need to apply separate grades within a shot.

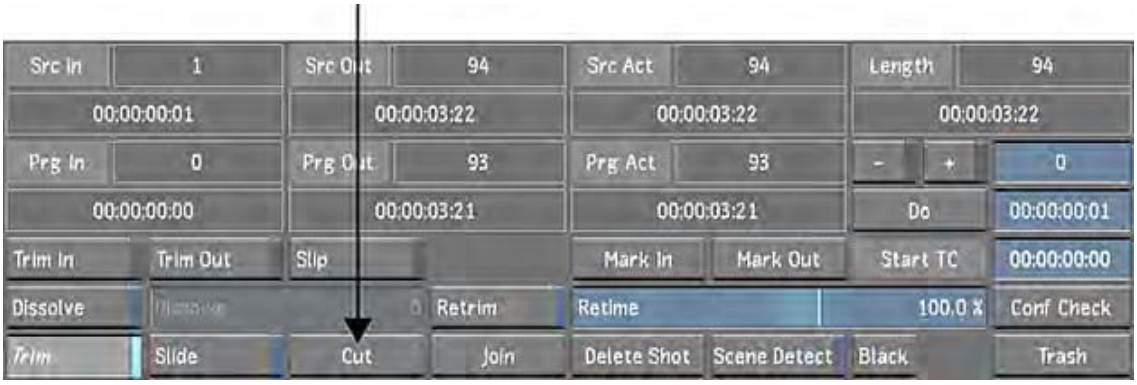
NOTE If working on field-based material, you cannot apply a cut between two fields of an image.

To insert a cut in a shot:

1 In the Storyboard, scrub to the position where you want to introduce the cut.

2 Click Editing, and then click Edit to display the Edit menu.

3 Click Cut.



The shot is cut into two separate shots.

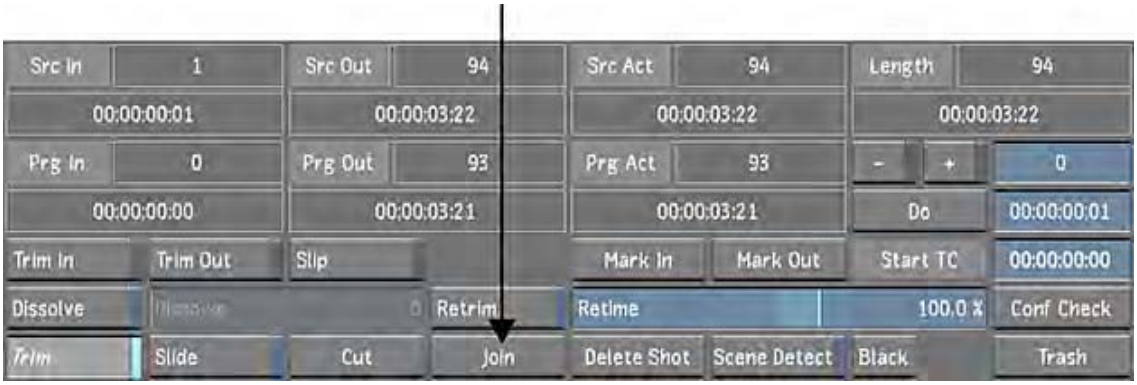
TIP If the Editing While Grading button is enabled in the User settings, you can insert a cut from any menu by pressing `Shift+C`. For more information about the Editing While Grading button, see System & Menu Settings (page 1762).

Joining Shots

You can rejoin a shot that is cut. The shots have to come from the same original source, must be adjacent in the Storyboard, and must be incrementing timecode. Join recreates the original sequence, even if there is material missing from the current Storyboard. For example, if you create a cut and trim out something in the middle of the original shot, you can join the shots back together and the missing frames are put back.

To join shots together:

1. In the Storyboard, select the right-most (also known as the B-side) of the previously cut shots.
2. Click Editing, and then click Edit to display the Edit menu.
3. Click Join.



The shots are joined together.

TIP If the Editing While Grading button is enabled in the User settings, you can join shots from any menu (except the colourist multi-layer timeline) by pressing `Shift+J`. For more information about the Editing While Grading button, see System & Menu Settings (page 1762).

Changing Playback Speed

Use the Retime option to change the playback speed of your shots. When you change the playback speed of a shot, Lustre duplicates or removes frames as needed so that the shot plays at the specified constant frame rate. Changing the speed changes the length of the shot, so you may need to trim the shot after you change the speed.

To change a shot's playback speed:

1. Select the shot in the Storyboard that you want to adjust.
2. Click Editing, and then click Edit.



(a) Retrim button (b) Retime slider (c) Retime field

3. Enable Retrim to trim the shot so that the same source in and out points are used. You can also trim the shot using the tools in the Edit menu.

   If you do not enable Retrim, the source out point changes shots. This is especially noticeable when you speed up a shot, since you may end up with black frames at the end of the adjusted shot. You can remove the black frames using the trimming tools. See Trimming from the Edit Menu (page 1917).
4. Drag the Retime slider to the new playback speed or right-click on the Retime slider to enter a retime value in the Retime field.
5. Play back the shot to see the effect of the speed change.

Replacing Shots

You can take a shot from the Library (the source shot) and swap it with another shot in the Storyboard (the destination shot). You can replace the destination shot with a source shot starting at the first frame of the source, or you can replace the destination shot with a source for which you have offset the starting frame.

NOTE You can also replace a shot in the Timeline menu using the Library view. See Replacing Shots in the Timeline (page 1938).

When replacing shots, you do not modify the overall duration of the timeline.

To replace a shot in the Storyboard without offsetting the source shot:

1. `Shift`-click the shot in the Library (the source shot) that you want to use to replace the shot in the Storyboard (the destination shot).
2. Move the source shot over the destination shot in the Storyboard.

   The destination shot is highlighted with an orange box.

3 Click the destination shot.

The destination shot is replaced by the source shot.

To replace a shot in the Storyboard with an offset source shot:

1 `Shift`-click the shot in the Library (the source shot) that you want to use to replace the shot in the Storyboard (the destination shot).

2 Move the source shot over the destination shot in the Storyboard.

The destination shot is highlighted with an orange box.

3 `Shift`-click the destination shot.

A dual view appears. In the left viewer, you see the original shot and in the right viewer you see the new shot.

4 Drag in the right viewer.

The start frame of the first shot begins to change. You can see by how much you are offsetting the start frame by referring to the numbers in the upper-left corner of the screen.

5 For greater granularity, middle-click and drag to decrease the start frame by one frame and right-click and drag to increase the offset by one frame.

You can also use the numeric keypad to change the offset. Specify a negative number and then press `Enter` to decrease the offset or specify a positive number and then press `Enter` to increase the offset.

6 Press `Enter` on the numeric keypad to complete the replace shot operation.

NOTE If the source shot is shorter than the destination shot, or if you offset the source starting point too much, part of the destination shot will not contain any images. Instead, an X appears in the viewer for these frames.

Creating Dissolves

Use dissolves to create a gradual blend between two shots. A dissolve is a transition that causes the outgoing shot to gradually disappear at the same time as the incoming clip gradually appears. Depending on the number of available handles, you can use dissolves to create long, gradual changes, or you can create dissolves with short durations (called soft cuts).

NOTE Centre dissolves are created automatically if you import an EDL that contains dissolves. However, other transitions, like wipes, are replaced by straight cuts.

Adding a Dissolve

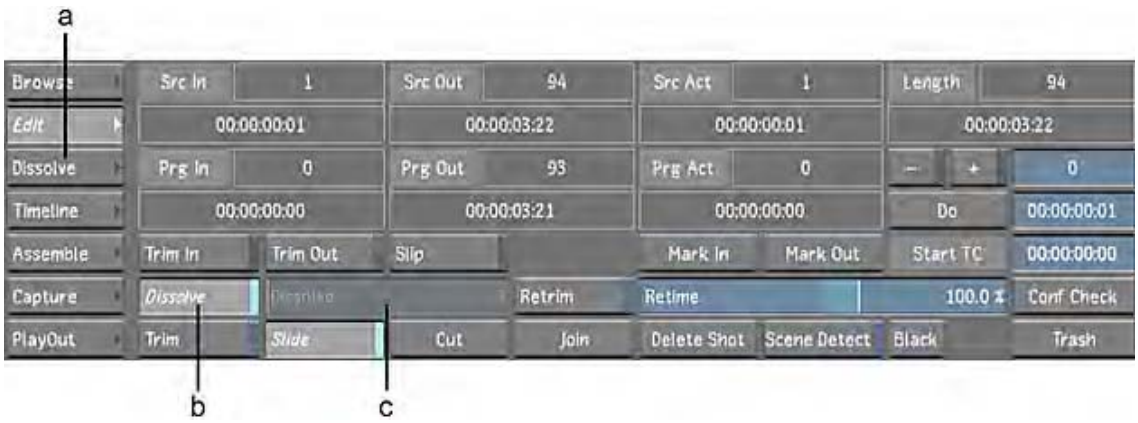
You can add a dissolve between two shots provided that there are a sufficient number of underlying handles.

NOTE You must have sufficient handles to add a dissolve. If you do not, use the Retrim tool. For more information, see Creating Additional Handles for Dissolves (page 1928).

To add a dissolve between two shots:

1 Click Editing, and then click Edit.

The Edit menu appears.



(a) Dissolve menu button (b) Dissolve button (c) Duration slider

2 Place the positioner on or near the transition to which you want to add a dissolve.

NOTE You can also add a fade-in at the beginning of the timeline and a fade-out at the end. Also, when adding a fade-in or fade-out to a gap on Layer 1, Lustre automatically fills the gap with a virtual colour source clip.

3 Enable Dissolve, and then drag the Dissolve slider to set the dissolve duration.

4 To edit the shape of the Dissolve curve, see Editing Dissolves (page 1929).

Changing Dissolve Duration

You can change the duration of a dissolve at any time.

To change dissolve duration:

1 Select the shot to the left of the dissolve.

2 In the Edit menu, drag the Dissolve slider to adjust the dissolve duration or right-click the slider for the pop-up entry.



Creating Additional Handles for Dissolves

You can trim handles from outgoing and incoming shots and use them to add a dissolve.

To create handles for a dissolve:

1. In the Storyboard, select the shot to the left of the cut point to which you want to add a dissolve.
2. In the Edit menu, enable Retrim.



3. Enable Dissolve, and then drag the Dissolve slider to set the duration or right-click the slider for the pop-up entry.

   Frames are automatically trimmed from the tail of the outgoing clip and the head of the incoming clip.

Editing Dissolves

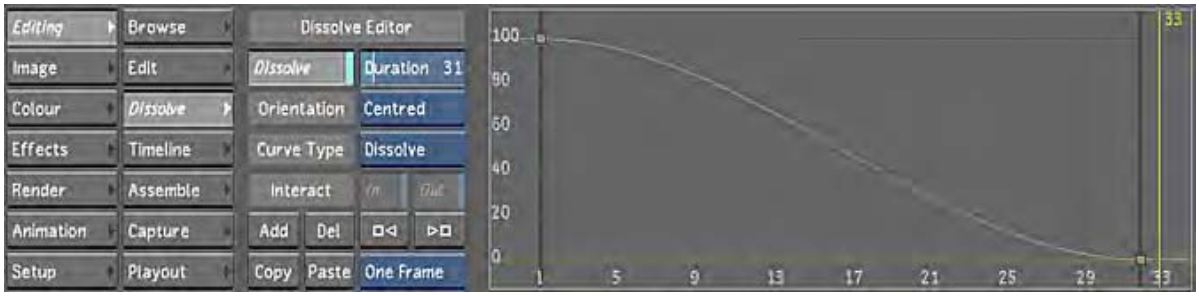
The Dissolve menu allows you to animate your dissolve. Use the controls located in the lower left side of the Dissolve editor to add, delete, copy, paste and navigate between keyframes. Navigate the timeline to edit any transition using the Dissolve menu's editor and curve window. Use the navigation buttons, the hotkeys or the Autodesk Control Surface to navigate the timeline.

TIP

To enter the Dissolve editor from the timeline, you can double-click on the Dissolve icon.

Right-click to zoom the Dissolve curve and middle-click to pan. To reset the Dissolve curve, press the Home key on the keyboard.

Keyframe controls are located on the lower left side of the Dissolve curve window. On the right side of the Dissolve curve window, you will find the Animation Controls and the Animation options. See Animation (page 2171).



Dissolve button Enable the dissolve located at the current transition.

WARNING

Re-enabling a disabled dissolve will restore dissolve metadata (e.g., duration, orientation) for the current session only. Saving the cutfile with the Dissolve button disabled does not save the original dissolve settings.

Duration field Set the length of the dissolve. The default value is the defined in the User settings. See Tools Settings (page 1765).

Orientation box Define the orientation of the dissolve.

Select:	To:
Centred	Place the dissolve in the middle of the transition.
From Cut	Start the dissolve at the cut point.
Up To Cut	End the dissolve at the cut point.

Curve Type box Select the type of dissolve curve.

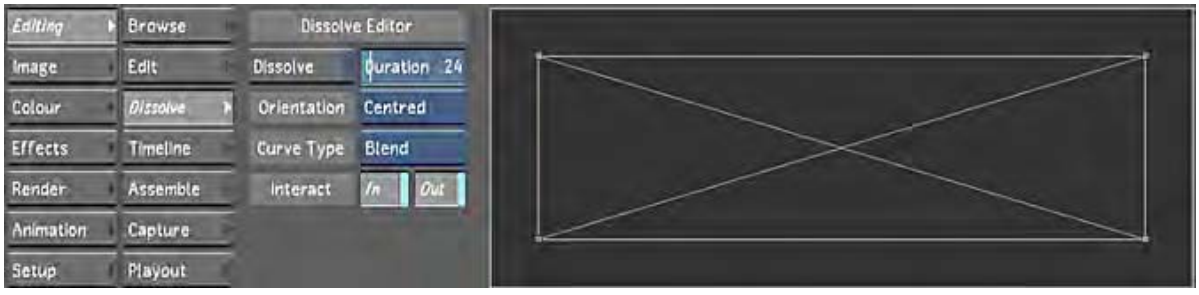
Select:	To:
Blend	Display two curves to blend between the outgoing and incoming shots.
Dissolve	Display a dissolve curve similar to that found in Smoke and Flame. Display buttons in the Dissolve editor to edit and navigate between keyframes. **NOTE** Importing a timeline with dissolves from Wiretap server 2011 Extension 1 or higher will include dissolve curves.

In and Out buttons Enable to blend between the outgoing and incoming shots. Enabled when the Curve Type is set to Blend.

Use the windows in the Dissolve menu to edit the rate at which the dissolve changes.

The Blend Curves window displays two curves: one represents the outgoing shot and the other represents the incoming shot. These curves are plotted on a graph that shows time vs. percent visible. When curves are at the top of the window, the frames at that point in time are 100% visible. Curves near the bottom correspond to 0% visible frames.

By default, each curve progresses in a linear fashion from one corner of the window to the opposite corner.



NOTE

Cutfiles from versions prior to Lustre 2011 Extension 1 will display the Blend curve.

To modify the Dissolve curve:

1 In the Editing menu, click Dissolve to display the Dissolve menu.

2 In the Curve Type box, select Dissolve.

To modify the Blend curve:

1 In the Editing menu, click Dissolve to display the Dissolve menu.
2 In the Curve Type box, select Blend.
3 Enable Interact In or Out to specify whether you are modifying the outgoing or incoming frames of the dissolve. You can also work on both curves simultaneously.

You can change the start or end visibility to any value between 0-100%; however, you cannot adjust the time at which a dissolve starts or ends in the Dissolve Curves window. To do this, return to the Edit menu and use the Dissolve slider.

Turning Dissolves Off

Use the Dissolve button in the Edit menu or the Dissolve menu to toggle a dissolve on and off.

Resetting Dissolves

To reset the Dissolve curve to its default value (i.e., two keyframes), press the Reset button in the Lustre user interface.

Adding a Virtual Black Clip using the Storyboard

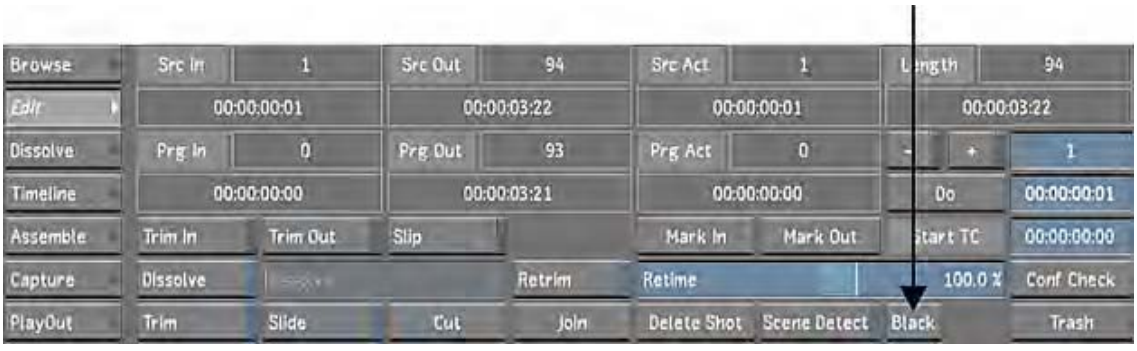
Virtual black clips can be added anywhere on the timeline. The default duration of each clip is one second, according to the current project frame rate. See Adding Black Media to the Timeline (page 1940).

To add a virtual black clip using the Storyboard:

1 In the Main menu, click Editing, and then Edit.

NOTE Solo mode must be enabled if you want to add the black clip to a specified position. See Multi-Layer Timeline (page 1932). If Solo mode is disabled, the black clip is inserted at the end of the timeline.

2 Click Black.



3 Move the clip to the Storyboard.
A yellow insertion marker appears at the cut where the black clip is entered.

NOTE The status of Ripple and Snap modes is taken into account when adding black media.

4 Insert the clip on the Storyboard.

NOTE You can change the virtual clip's colour in the Virtual Black Clip Colour group in the Project settings. See Engineering Settings (page 1749).

Multi-Layer Timeline

The multi-layer timeline view organizes footage or elements into a multi-layer format. You can scroll, pan, and zoom the multi-layer timeline, as well as rearrange elements on the timeline by dragging and dropping.

The multi-layer timeline in Lustre supports Video Tracks and Video Layers. However, vertical compositing is not supported in Lustre.

Multi-layer timeline editing is useful for roughing out edit sequences and grade versioning. In addition, shot selections are now linked between the timeline and the Storyboard.

You can also copy the grading information in the multi-layer timeline or colourist multi-layer timeline. See Copying Grading Information in the Colourist Timeline (page 2055).

The multi-layer timeline allows you to display footage, play out to tape, and render in accordance with rules that prioritize some shots or layers over others. See Displaying the Multi-Layer Timeline (page 1958).

Timeline Menu

There are various areas to work with when you access the Timeline menu:

- Storyboard. See Storyboard Viewing Options (page 1979).
- File browser. See Accessing the File Browser (page 1856).
- Library. See Accessing the File Browser (page 1856).
- Player and full-screen Player. See The Player (page 1803).

NOTE From the Timeline menu, you can press `Enter` to toggle through these different views.

To access the Timeline menu and multi-layer timeline:

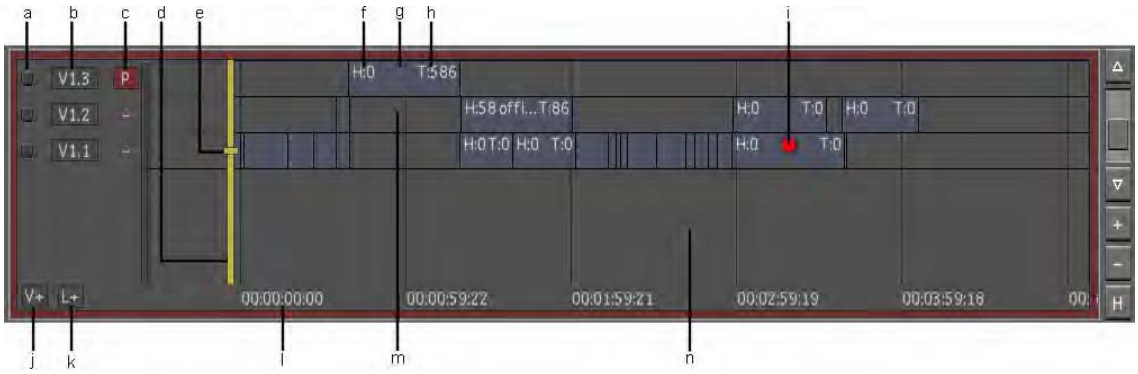
1 In the Main menu, click Editing.

2 Click Timeline.

The Timeline menu and the multi-layer timeline canvas appear.



The multi-layer timeline is made up of the following components.



(a) Layer selector (b) Track/Layer name (c) Primary Track/Layer indicator (d) Positioner(e) Track/Layer selector(f) Element Head (g) Element (h) Element Tail(i) Shot Priority indicator (j) Add Video Track button (k) Add Video Layer button (l) Starting timecode (m) Gap (n) Canvas

Layer selector Selects the entire layer in preparation for an operation.

Track/Layer name The name of the layer, where V stands for Video and L for Layer. Adding a layer generates V1L2, V1L3, etc.

Primary Track/Layer indicator Indicates which Track/Layer is set to Primary. Click the left side of the track (-) to set the track as Primary (P).

Positioner The play head for the Player. The frame directly beneath the positioner is displayed in the Player and is the current location for an edit. The positioner's location is reflected in the Source Timecode or Timeline Information field. See Monitoring the Location of the Current Frame (page 1977).

Focus point When Solo mode is enabled, the focus point indicates the current layer and which layer is displayed in the Storyboard. When Solo mode is disabled, the top layer acts as the primary layer and is displayed in the Player and Storyboard. The focus point also determines to which layer a cut or dissolve is added. Together, the position of the focus point and the positioner is reflected in the Source Timecode or Timeline Information field. See Monitoring the Location of the Current Frame (page 1977).

Element Head Number of available frames for trimming, succeeding a transition.

Element The media in the timeline.

Element Tail Number of available frames for trimming, preceding a transition.

Shot Priority indicator Graphical representation of a shot with priority. See Shot Priority (page 1960).

Add Video Track button Adds a new Video Track to the timeline.

Add Video Layer button Adds a new Video Layer to the timeline.

Gap The absence of media. In a multi-layer context and when Solo mode is disabled, gaps are also transparent.

Canvas Click the canvas to zoom in/out and move the timeline.

The Timeline menu is made up of the following buttons.



Solo When Solo mode is enabled, Lustre displays, plays out to tape, or renders only the layer on which the focus point is positioned.

NOTE The Solo state is not saved in the grade file.

Cut Adds a splice at the positioner location on the focused layer.

Black Generates one second of black media that can be added to the timeline.

Copy Copies the current selected elements to the clip board. Copied elements are displayed in red. Drag and drop the copied elements to the destination timecode or layer.

Trim Trims the head and/or tail of an element.

Ripple mode Determines whether the length of the edit sequence is affected when new material is inserted into or removed from the timeline.

Snap mode Affects how elements are placed when gesturally moved or copied to the Timeline.

Ctnr Adds a container to the timeline.

Join Removes the splice on selected elements coming from the same source clip with consecutive source timecode.

Diss Adds a dissolve at the positioner location on the focused layer.

New Copies the current element to the next available top layer with grade data.

Del Deletes the current selection.

Timeline Navigation Controls

The following is a list of hotkeys and mouse gestures commonly used to navigate within the Timeline menu.

Press:	To:
Right `Alt`	Go to the previous frame.
Right `Ctrl`	Go to the next frame.
Left arrow	Go to the previous element.
Right arrow	Go to the next element.
Up arrow	Move the focus point to the layer above.
Down arrow	Move the focus point to the layer below.
Middle mouse button and drag	Move the multi-layer timeline.
Right mouse button and drag	Zoom into or out of the multi-layer timeline.
Left mouse button and drag the positioner	Move the positioner.

Additional multi-layer timeline navigation controls are shown below.



(a) Scroll timeline canvas up (b) Vertical scroll bar (c) Scroll timeline canvas down (d) Vertical zoom control (e) Home/selected Focus view (f) Scrub area

Scroll timeline canvas up Displays the top layers located outside of the canvas view.

Vertical scroll bar Quickly moves the multi-layer timeline view up or down.

Scroll timeline canvas down Displays the bottom layers located outside of the canvas view.

Vertical zoom control Enlarges or reduces the size of the layers.

Home/selected Focus view Click this button to reset the vertical/horizontal zoom and position of the canvas view to display the entire timeline. `Alt` + press this button to zoom into the selected shot(s).

Scrub area Click or drag in this area to quickly move to any point in the timeline.

Viewing Element Information

It is possible to vertically expand the size of a layer to see more information about the element.

To vertically expand the layer:

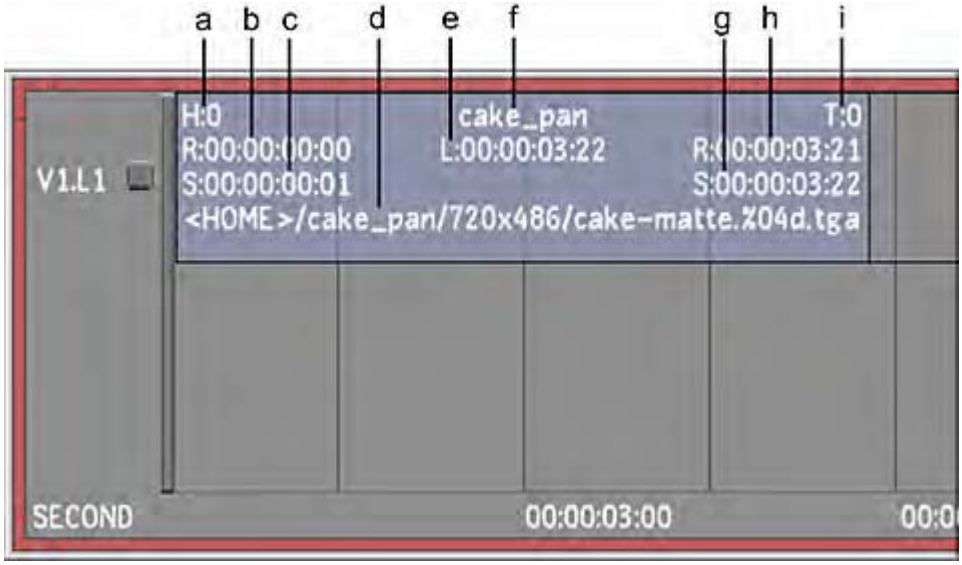
1 Click +.
More information about the element is revealed.

NOTE Each time you press +, more information is revealed.

To minimize the layer:

1 Click -.
Other layers become visible.

The following information is displayed about an element.



(a) Head (b) Start record timecode (c) Start source timecode (d) Shot location (e) Length of element (f) Name of element (g) End source timecode (h) End record timecode (i) Tail

Editing in the Timeline

The following sections describe how to perform various basic timeline operations.

Creating a Timeline

When you add shots to the Storyboard view or assemble an EDL, Lustre creates a timeline. You can then perform editing operations or colour grade the sequence. A timeline can also be called a cut file.

Adding Shots to the Timeline

You can add shots by dragging and dropping from the Library or Browser to the multi-layer timeline or to the Storyboard. See Adding Shots to the Storyboard (page 1915).

NOTE Snap and Ripple mode affects how shots are added to the timeline.

Interchange with Flame or Smoke

When dropping content, coming from a Smoke or Flame project to a Lustre timeline, the sequence or the clip is inserted as a layer above the current video layer or track, at the content's record timecode. This functionality is useful when importing revised editorial content from Smoke & Flame into the currently opened Lustre timeline.

NOTE In order to drop imported content from Wiretap server at its native record timecode, the selected sequence must be dropped directly from Media Browser to the current sequence. Dropping the selected sequence to the Media Library first, and then dropping this sequence into the current sequence will use the timeline Snap options (Off, Positionner or Transition) rather than its native record timecode.

Importing Timelines as Source Clips

Drag and drop timelines (clips with a red background) from the library to the Lustre timeline to build a timeline containing multiple conformed EDLs (for example, multiple scenes of a project).

You can drag and drop the timeline clips to the Storyboard or Timeline view:

- Storyboard: inserts the incoming timeline to the end of the current timeline on the primary track or layer, while preserving the video tracks and layers.
- Timeline: inserts the incoming timeline at the drop point or according to the defined Snap option. The layer where you drop the timeline(s) defines where a multi-layer timeline is created.

NOTE Grading metadata is not part of the timeline structure. Use Marry Grade to share grading data between timelines.

Snapping Elements

Snap mode affects how elements are placed when gesturally moved or copied to the timeline. Snap mode consists of the following three options.

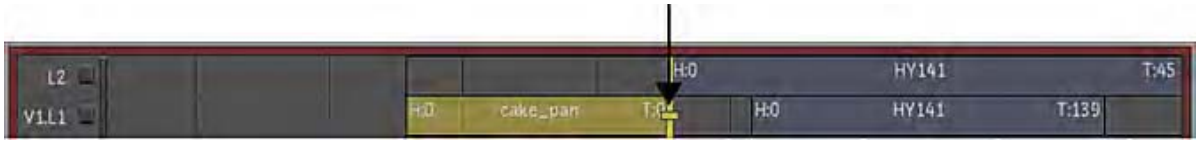
Snap Off Drops the element at the location that you choose.

NOTE An element can be overwritten if you drop one element over another.

Snap Trn Snaps the element to the closest transition.



Snap Pos Snaps the element to the positioner.



Selecting Elements

You can select multiple elements to move, copy, or delete. Use the following modifier keys to create a selection.

To select elements in a continuous selection:

1. Click the first element in the selection.
2. Press `Shift` and click the last element.
   All elements between the first element and the last element you clicked are selected.

   NOTE This function only works within the same layer.

To select elements in a broken selection:

1. Press `Ctrl`.
2. Click the elements you want to select.

   Various elements in broken sequence are now selected.

   NOTE To remove elements from a multiple selection, press `Ctrl` and select the elements you want to remove from the selection.

TIP You can use `Shift` to define a selection on a layer and then hold `Ctrl` and start a new selection on another layer (with `Shift` or `Ctrl`).

Moving Elements

You can move an element or a selection of elements within the same layer or to another layer.

To move an element or a selection of elements within the same layer or to another layer:

NOTE The status of Ripple and Snap modes are taken into account when dropping elements. See Trimming With Ripple Mode (page 1953) and Snapping Elements (page 1937).

1. In the multi-layer timeline, select the element you want to move.

   The selected element becomes yellow.
2. Drag the element and drop it at the destination.

Replacing Elements in the Timeline

You can now replace shots and select a timeline segment as the destination. The replaced shot starts at the location of the original shot and gap or tail handles appear to fill the original duration.

1. Load the timeline you want to modify.
2. In the library, press SHIFT key and select a clip. Release SHIFT key.
3. Move the selection over the timeline segment you want to replace and click on the segment.



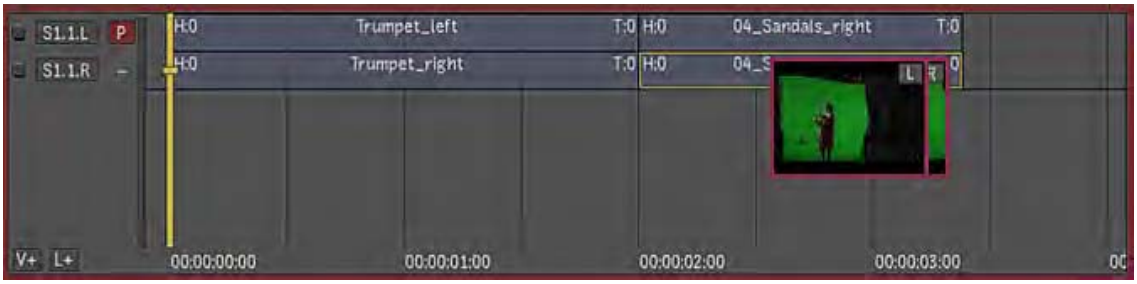
   The shot is replaced and the grading on the original shot is preserved. Tail media or Gap is seen in the timeline if the shots don't have the same duration.

Replacing Shots in a Stereo Timeline

When working on a Stereoscopic project, you can perform a shot replace that applies to both eyes at the same time.

NOTE In Stereo, you must replace the shot for both eyes. It is not possible to use Replace Shot for only one eye.

1. Load the timeline you want to modify.
2. In the library, hold SHIFT key and select the Left clip. A "L" icon appears on the top right side of the thumbnail.
3. Navigate library to find the Right clip, hold SHIFT key and select the Right clip. A "R" icon appears on the thumbnail. Release SHIFT key.
4. Move the selection over the Storyboard or Timeline segment you want to replace and click on the thumbnail.



Both Left and Right shots have been replaced with the selection and the grading on the original shots is preserved.

Swapping Elements

You can swap any two shot elements by selecting them in the timeline or Storyboard. The shots can be in different layers.

To swap shots within the same layer or between two layers:

NOTE When swapping elements, the status of the Ripple mode is always Ripple Off.

1. In the multi-layer timeline, select two shots. See Selecting Elements (page 1937).

   The selected elements become yellow.

   WARNING If the shots do not have the same duration, the shot with the longer duration is trimmed, and the shot with the shorter duration is followed by a gap.

2. Press `Shift+'`.

   The shot positions are interchanged.

Copying Elements

You can copy one or more elements within the same layer, or from one layer to another. In addition, when copying across layers, you can maintain a vertical lock on the shots you are copying to preserve their 'horizontal' place in the timeline.

NOTE The status of Ripple and Snap modes are taken into account when dropping elements. See Trimming With Ripple Mode (page 1953) and Snapping Elements (page 1937).

To copy elements within a layer:

1. In the multi-layer timeline, select the element(s) you want to copy. See Selecting Elements (page 1937)
2. Click Copy.
   The copied element(s) become red.
3. Drag the elements to a new position in the timeline.
4. Click the mouse button to complete the copy.

To copy a single element from one layer to another layer:

1. In the multi-layer timeline, select the element you want to copy.
2. Click Copy.
   The copied element becomes red.
3. Position the cursor over the element you wish to copy and click and hold the left mouse button.

   NOTE To maintain a vertical lock on the copied element, hold down the `Shift` key.

4. Drag the element to the desired layer and horizontal time position.
5. Release the mouse button (and the `Shift` key, if required) to complete the copy.

To copy multiple elements from one layer to another layer:

1. In the multi-layer timeline, click once on the far-left element you want to copy in the layer.
2. Select the remaining elements you want to copy. See Selecting Elements (page 1937).
3. Click Copy.
   The copied elements become red.
4. Position the cursor over any of the elements you wish to copy and click and hold the left mouse button.

   NOTE To maintain a vertical lock on the copied elements, hold down the `Shift` key.

5. Drag the elements to the desired layer and horizontal time position on the Timeline.
6. Release the mouse button (and the `Shift` key, if required) to complete the copy.

Adding Black Media to the Timeline

You can add black media to the timeline to create a space (or fades) between elements. By default, a black clip is one second in duration, based on the project's frame rate.

To add black media to the timeline using the Storyboard, see Adding a Virtual Black Clip using the Storyboard (page 1931).

To create black media:

1. In the Timeline menu, click Black.
   A black shot appears attached to the mouse cursor.
2. In the timeline, position the cursor of the black shot where you want the black media to appear.

   NOTE The status of Ripple and Snap modes is taken into account when adding black media.

3. Click the mouse button to drop the shot.
   The black media is now in place.

Deleting Elements

You can delete selected elements in the timeline in numerous ways.

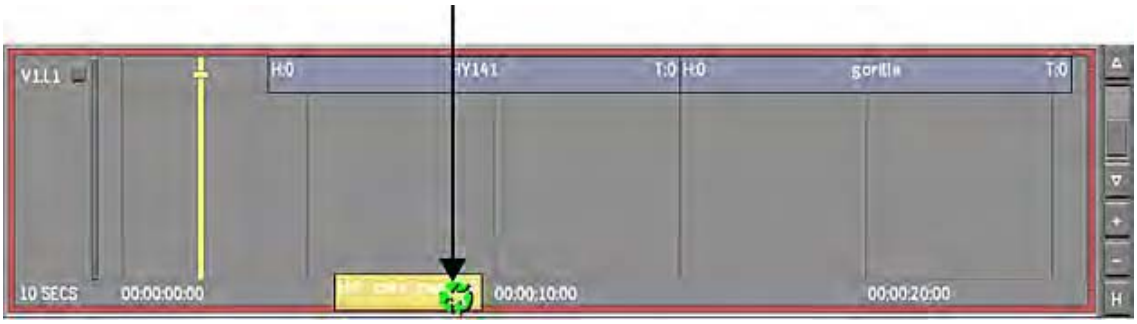
NOTE Deleting an element is affected by the Ripple mode. For example, if Ripple Off is enabled, deleting a shot leaves a gap.

To delete the element(s) from the timeline:

1. Select the element(s). See Selecting Elements (page 1937).
2. Do one of the following:
   - Click Del, and then confirm the deletion.

     If no selection is made, the element with focus is deleted.
   - Press `Shift+D`.

     If no selection is made, the element with focus is deleted.
   - Drag the selection outside the red border of the Timeline. When the recycle symbol appears, drop the element(s) to delete it.



NOTE You can also delete shots in the Storyboard view. See Deleting Shots (page 1916). The Timeline immediately reflects the changes.

Gaps in the Timeline

A gap is an empty space in the timeline. On a single layer, a gap is displayed as a hole in the multi-layer timeline. A red X is displayed in the Player when the positioner is located on a gap. Gaps are also displayed as 'GAP' in the Source Timecode field.

When working with a multi-layer timeline, a gap located on the top layer shows the elements located below (Solo mode disabled only).

Gaps can be modified like other clips. They can be trimmed by using Ripple Start and Ripple End mode (see Trimming Ripple Mode (page 1954)). Their length can also be increased or decreased by modifying the clips around them. Gaps can be selected, copied, or deleted.

Working With Containers, Matte Containers and Multi-channel Clips

Containers, matte containers and multi-channel clips are a convenient way of grouping elements on the timeline. They can each be represented in the timeline as a single element.

- A container (page 1942) is composed of multiple versions of the same shot with different grades or multiple shots.
- A matte container (page 1944) is composed of a primary track and one or more external mattes.
- A multi-channel clip (page 1947) is a source clip that contains one or multiple matte channels.

Note the following functionality while working with containers, matte containers, and multichannel clips:

- Containers, matte containers and multi-channel clips can be part of a sorted selection. When a timeline is sorted, it is not possible to enter inside the containers.
- You can apply retime effects on containers, matte containers and multi-channel clips. All shots inside a container get the same retime effects. When you drag and drop clips to a container, the shots you add also get the same retime effect.
- You can collapse a selection of shots, containers, matte containers and multi-channel clips. When the timeline is collapsed, the matte media in matte containers and multi-channel clips are still available for grading, but you cannot enter inside these timeline segments when they are collapsed.
- You can remove containers and matte containers on multiple selections.

Matte Workflow

You can import mattes like any other source clip. Use one of the following workflows with externally generated mattes:

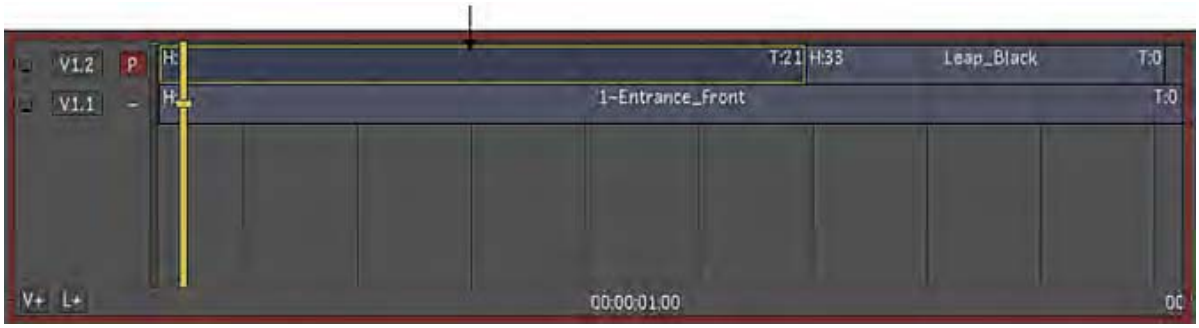
- Use a matte container created in Lustre and import a matte as a source clip.
- Use multi-channel clips when you want to add matte channels to a source clip and share this footage with multiple timelines in Lustre. These clips can be created from Lustre or generated outside the application using the Autodesk Wiretap Gateway XML schema.
- Use a matte container created in Smoke and Flame through Wiretap.

NOTE The legacy external matte workflow that required the manual modification of matte files to match the source footage file name and location is now obsolete. You cannot load mattes that were created prior to Lustre 2012. You must recreate them.

About Containers

Use containers to store multiple versions of the same shot with different grades or multiple shots. Grading is applied on the shots inside the container and not on the result of the container.

Containers on the timeline are dark grey in colour:



On the storyboard, a dark grey box on the left corner of the thumbnail indicates a container:



Inside a container you can work with video tracks and layers:

- When working on layers inside the container, you can perform any editorial task, add versions and use Shot Priority to define the output of the Container.

NOTE Do not use Solo on layers inside a Container to define the container output because Solo is not saved with the grading data. Use Shot Priority instead.

To create a container:

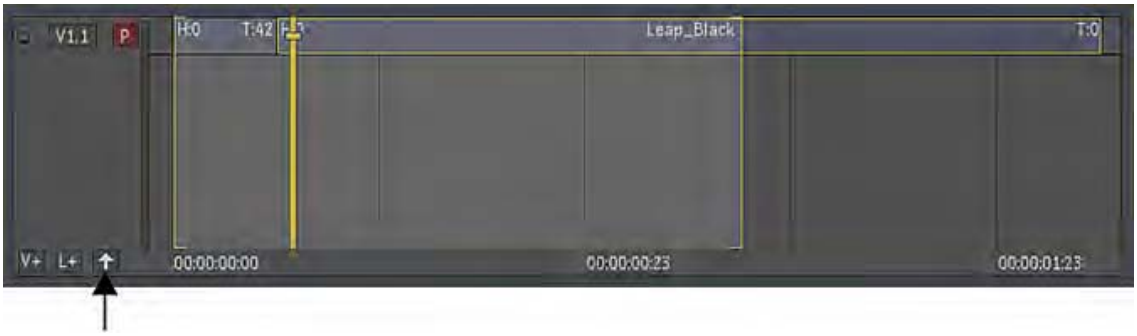
1. Select one or more timeline segments.
2. Click the Ctnr button.
   Container segments/elements appear dark grey in the record timeline.

To enter a container:

1. Double-click the container segment.
   The contents of the container are displayed.

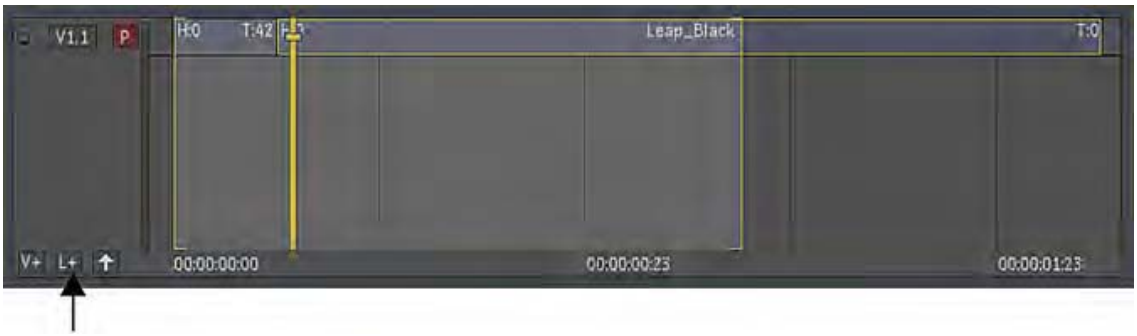
To exit a container:

1. Click the Up Arrow button (next to L+ button, located at the bottom left side of the timeline).
   The contents of the container are collapsed into a single segment.



To add a clip to a layer in a container:

1. Double-click the container to open it.
2. Click L+ to add a new layer.



3. Select the clips from Library.
4. Drag the clips to the layer in the container.

Use containers to hold multiple versions of grading on a shot.

To create a new version of the grading without entering the container:

1. Hold down the `Ctrl` key and click the New button.
   A new version on vertical layer appears inside the container.

To remove a container:

1. Place the positioner on the container, or select the container.
   The selection overrides the positioner.
2. Hold down the `Alt` key and click the Ctnr button.
   The container is removed and the container contents are inserted on the current timeline.

About Matte Containers

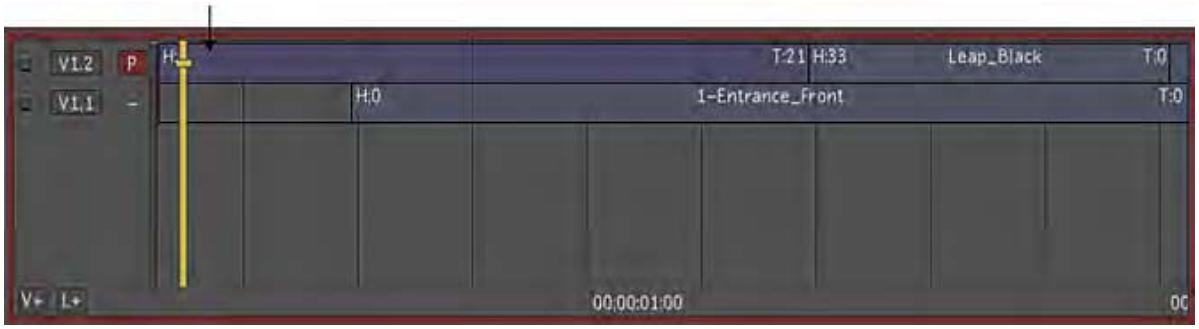
Use matte containers to perform secondary colour grading and effects with externally generated mattes.

Matte containers are composed of timeline segments that contain two video tracks:

- The primary video track contains the shot.
- The secondary video track contains the matte. Use the matte for secondary colour grading and effects.

It is possible to build complex multi-layered content inside a matte container. However, only the source media of the matte track is used for secondary colour grading. Effects created on the matte, including dissolves, are not used in secondary colour grading.

Matte containers on the timeline are purple:



On the storyboard, a purple box on the left corner of the thumbnail indicates a matte container:



If there is a gap or virtual colour clip on a matte track and an external matte is enabled (R/G/B/A buttons), the matte is seen as opaque (black) so any correction will not be seen on the graded image. This behavior is also true for multi-channel clips where no matte media is available (displayed as gap).

You can create matte containers from both monoscopic and stereoscopic media.

To create a matte container:

1. Select one or more timeline segments.
2. Press Ctrl and click the Ctnr button.
   The segment turns purple to indicate that you have created a matte container.

To change a video track inside a matte container from Primary to Secondary:

1. Do one of the following:
   - Click the left side of the video track (-) to make it Primary (P).
   - Middle-mouse-click on the left side of the track or channel (-) to make it Secondary (S).

Enter, exit and remove a matte container in the same manner as a container.

Adding a Matte to a Matte Container

There are two ways to add a matte to a matte container.

- By entering the matte container and adding the matte to a video track.
- Adding a clip from the library to the timeline without entering the matte container directly.

The matte clip must have the same resolution as the front clip in a matte container.

To enter a matte container and add a matte to it:

1. Display the timeline and the library.
2. On the timeline, double-click the matte container to enter it.
3. Click V+ to add a new track.
4. Drag and drop a clip from the library to the track on the timeline.

5 (Optional) Set the matte as the secondary.

To add a matte to a matte container without entering it:

1 Display the timeline and the library.
2 Select the matte you want to add to the container from the library.
3 Press C trl, drag and drop a matte onto the matte container segment on the timeline to insert the matte at the first frame of the first shot inside the matte container.
4 (Optional) You can drag and drop a matte onto the matte container while holding Ctrl+Alt to insert the matte at the first frame of the In point of your timeline segment rather than the first frame of the source clip. This is useful when working with scene detected footage requiring mattes.

NOTE The Ctrl+Alt option is exclusive to Lustre 2012 Extension 1.

You can drag and drop multiple mattes into a matte container. The first matte you add is the secondary track. You can apply any matte in your container to a separate secondary colour grading layer in the Colour | Secondaries menu. See Selecting a Matte to Grade Secondary Layers (page 1946).

To add a matte to a stereoscopic matte container, you must drop mattes one at a time to add them to the matte container. You cannot select both a left and right matte at the same time and drop them on a clip to create a stereo matte container.

To add a matte to a stereoscopic matte container:

1 Display the timeline and the library.
2 Hold down Ctrl or Ctrl+Alt and drag and drop a matte onto the matte container segment that corresponds to the stereo eye.
3 Hold down Ctrl or Ctrl+Alt and drag and drop a second matte onto the matte container segment that corresponds to the other stereo eye.

NOTE The Ctrl+Alt option is exclusive to Lustre 2012 Extension 1.

Selecting a Matte to Grade Secondary Layers

By default, the Secondary track in a matte container or multi-channel clip is selected for grading.

You can select a different matte track or channel for each secondary layer of colour grading. This is useful when you want to grade the image based on the values in its colour channels. You can make this change from the Colour | Secondaries menu, rather than going into the matte container on the timeline.

Here is a matte container with 3 matte tracks, V2.1 is set as the secondary.



In the Secondaries menu, you can enable a different matte for each secondary layer. Enable a secondary. Then click the Matte button and select the matte track to apply to the secondary.

See Working with Mattes (page 2137).

About Multi-channel Clips

A multi-channel clip is a source clip that contains one or multiple matte channels. Add matte media to source clips in the Lustre timeline to create a multi-channel clip and use the matte for secondary colour grading and effects. See Working with Mattes (page 2137).

On the timeline, the multi-channel clip is labelled Multi.



On the storyboard, the multi-channel clip is labelled Multi.



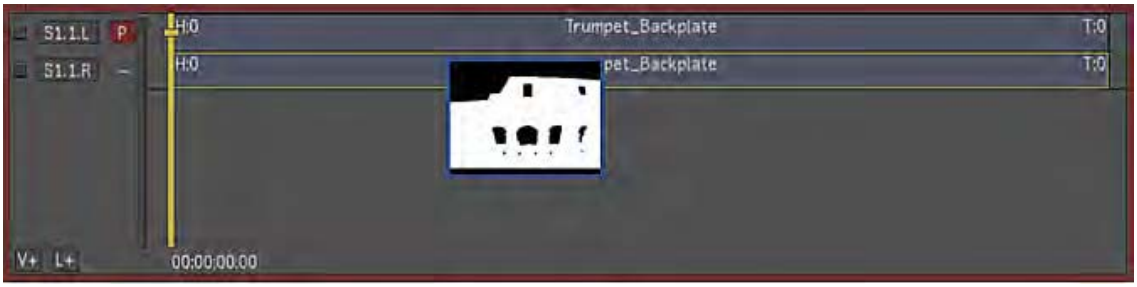
When you create a multi-channel clip, the source clip is modified in the library. Next time you use the source clip, the matte will be available. Any modification done inside the multi-channel clip is replicated in the library source clip.

NOTE When rendering a timeline that contains multi-channel clips from Lustre to Smoke or Flame, a matte container is created.

The front and the matte clip can be different file types or bit depth, but they must have the same resolution to create a multi-channel clip.

To create a stereoscopic multi-channel clip:

1. Add the left and right source media to the stereo timeline.
2. Hold Ctrl and drag the matte clip from the library and drop it on the source media on the timeline that corresponds to the stereo eye.



TIP Dragging and dropping while holding Ctrl drops the matte at the first frame of the source clip. Holding Ctrl+Alt while dragging and dropping inserts the matte at the first frame of the In point of the desired timeline segment.

NOTE The Ctrl+Alt option is exclusive to Lustre 2012 Extension 1.

Dragging and dropping while holding Ctrl drops the matte at the first frame of the source clip. Holding Ctrl+Alt while dragging and dropping inserts the matte at the first frame of the In point of the desired timeline segment. The Ctrl+Alt option is exclusive to Lustre 2012 Extension 1.

A "multi" icon appears on the Stereo timeline segments, but only one of the stereo sources is updated in the Library.



If you open the multi-channel clip, you see that the channels for the other eye are empty.



3 Hold Ctrl and drag the other matte clip from the library and drop it on the source media on the timeline that corresponds to the other stereo eye.

A "multi" icon appears for the other stereo source in the library.



If you open the multi-channel clip now, you see that the front and matte clips for both.



NOTE When creating a stereoscopic multi-channel clip, you must drag and drop a left clip to the left eye and a right clip to the right eye in the timeline so that each segment is identified as multi-channel. Lustre does not support stereoscopic eyes that are in different states.

To enter a multi-channel clip:

1. Double-click the multi-channel timeline segment.
   The matte media is located on the Secondary video channel (C2).

To convert a multi-channel clip to a standard source clip:

1. Delete all channels from C2 and up.
2. Exit the multi-channel clip.

Edit the Timing of a Matte in a Multi-Channel Clip to Match the Source Clip

Timing modifications done inside a multi-channel segment are saved in the source clip in the library. All instances of that source clip are updated to use the new timing.

You can only slip a shot to adjust its timing. You cannot trim.

To protect the structure of the source footage, it is not possible to add dissolves or layers inside the multi-channel clip. Use matte containers for more complex editing.

To create a temporary grade to help you set the timing of a matte in a multi-channel clip:

1. Exit the multi-channel clip on the timeline.
2. Enable a Secondary.
3. Enable one of the External Matte channels (R/G/B/A).
4. Modify the grading so that you can see the matte media over the image.
5. From the Timeline menu, enter the multi-channel clip.
6. Slide the clip in the timeline to change the timing. Use the display of the matte over the shot to help you find the right starting frame.
7. Click the Up arrow to exit the multi-channel segment. You are now ready to grade your shot.

Multi-Channel Clip Management

Automatic Creation of Multi-Channel Clips with Matte / MultiMatte Folder Structure

You can create folder structures that automatically generate Multi-Channel clips and automatically add and sync the matte media located in the *matte*/*multimattes* folder to the source clip. If there is a *matte*/*multimattes* folder located within the resolution folder of the source media files, Lustre displays the source clip as a Multi-Channel clip.

NOTE Grade files created with pre-2012 versions of Lustre and using external mattes are converted to Multi-Channel clips and can be imported in Lustre 2013.

Matte media files must have the following attributes:

- The index (frame numbers), in the source/mattes file names have to match. You can have any number of characters before the frame number value, as long as the frame numbers at the end of the filename match, they will be compatible. For example, *shot1_10000.dpx* is compatible with *000010000.tif* as well as with *lustrematte10000.tif*.
- The matte media files must be of the same resolution as the clip.
- The mattes must be either RGB or RGBA files.

Single Matte Media Structure

Here are some examples of media structures that Lustre uses for automatic Multi-Channel clip creation with a single matte.

Simple source clip (one clip in one folder):

- /scans/project/0001/1920x1080/[10000-10100].dpx
- /scans/project/0001/1920x1080/matte/[10000-10100].tif

The resulting clip in Lustre displays as a Multi-Channel clip that has a matte channel spanning all the frames of the source clip.





NOTE When Lustre creates a Multi-Channel clip, the matte channel has the same duration as the source clip. If the matte media does not cover the full duration of the source clip, the portions without matte media is seen as opaque when in the "O" view. As a result of this, the secondary grading cannot be displayed.

Simple source clip (one clip in one folder) with partial matte media:

- /scans/project/0001/1920x1080/[10000-10100].dpx
- /scans/project/0001/1920x1080/matte/[10040-10055].tif

The resulting clip in Lustre displays as a Multi-Channel clip that has a matte channel spanning frames 40 to 55. The matte channel still covers the entire duration of the source clip but secondary grading is not visible on the portions of the clip where there is no matte media even if the RGBA Matte buttons are enabled.

Multiple source clips located in a single folder:

- /scans/project/0001/1920x1080/[10000-10100].dpx
- /scans/project/0001/1920x1080/[20000-20100].dpx
- /scans/project/0001/1920x1080/[30000-30100].dpx
- /scans/project/0001/1920x1080/matte/[10000-10100].tif

In this source clip structure, multiple source clips are located inside a single folder along with a *matte* folder but it only contains matte files for the first sequence that has the frame range of 100000-10100.

The media files that do not have matte files with a corresponding frame range are read as standard source clips.

Multiple Mattes Media Structure

You can create a folder structure to store multiple matte clips by creating a *multimattes* folder instead of *matte*. The *multimattes* folder must contain a subfolder for each matte sequence to be included in the multi-channel clip, as in the example below:

Simple source clip (one clip in one folder) with multiple matte media:

- /scans/project/0001/1920x1080/[10000-10100].dpx
- /scans/project/0001/1920x1080/multimattes/matte1/[10000-10100].tif
- /scans/project/0001/1920x1080/multimattes/matte2/[10000-10100].tif
- /scans/project/0001/1920x1080/multimattes/matte3/[10000-10100].tif

This structure creates a 4-channel Multi-Channel clip (media + 3 mattes) with a matte on channels 2, 3 and 4.

NOTE The *multimattes* subfolders do not have to conform to any particular naming convention. You can name the *multimattes* subfolders as you like. Lustre sorts the subfolders and the mattes in alphanumeric order.

Simple source clip (one clip in one folder) with "matte" and "multimattes" structure:

You can also mix and match the *matte* and *multimattes* structure, as in the example below:

- /scans/project/0001/1920x1080/[10000-10100].dpx
- /scans/project/0001/1920x1080/matte/[10000-10100].tif
- /scans/project/0001/1920x1080/multimattes/matte1/[10000-10100].tif
- /scans/project/0001/1920x1080/multimattes/matte2/[10000-10100].tif
- /scans/project/0001/1920x1080/multimattes/matte3/[10000-10100].tif

This structure creates a 5-channel Multi-Channel clip (media + 4 mattes) with a matte on channels 2, 3, 4 and 5.

Synchronizing an Existing Cut and Source Clips with Newly Added / Removed Matte Media

When working on a project, it may happen that the matte media files located in *matte* and *multimattes* folder are modifed (edited, added or deleted). In this case, Lustre can synchronize the cut or grade file and the source clips with the external matte files.

To synchronize the cut or grade file and source clips with the external matte files:

1. Select the Cut to update from the Cut List or the Grade to update from the Grade List.
2. Hold the Shift key.
3. Click Load.

Lustre synchronizes the shots in the Timeline and the clips in the Library with the content of the *matte* / *multimattes* folders. It then updates the clips, converting them to multi-channel when mattes are present or leaving them as single-channel when no mattes are found.

OpenEXR and External Mattes

You can create multi-channel OpenEXR files (multiple channels embedded in the OpenEXR file) that are displayed as multi-channel clips in Lustre. You can also use OpenEXR files with a *matte* / *multimattes* folder structure to automatically generate a multi-channel clip. When working with multi-channel OpenEXR files with an external *matte* / *multimatte* folder, you must always manually resync the cut / grade to have access to the matte channels. To do this:

1. Import your multi-channel OpenEXR clip.

NOTE At this stage, only the embedded RGB mattes are available.

2. Save your cut/grade.
3. Press Shift on the keyboard and load your cut or your grade.

All embedded and external mattes are now available.

Trimming Elements

Trimming adds head or tail frames to the element or subtracts head or tail frames from the element. You can trim using the trim cursors in the timeline canvas. For a specific trim cursor to appear, you must place the cursor over the required position within the element (i.e., head, tail, or middle section of the element). For example, when you place the cursor over the start of the element (the head), the trim head cursor appears.

The following editing changes are automatically updated in the Player:

- trimming
- slipping
- sliding

NOTE You can also access some of the multi-layer timeline features through the Colourist's Timeline. See Colour Grading Basics (page 2038).

Trimming Cursors

The following cursors are used when trimming or slipping and sliding an element.

Cursor	Cursor name
	Trim head cursor
	Trim tail cursor
	Slip cursor
	Slide cursor
	Slip & slide cursor

Trimming With Ripple Mode

Whether Ripple mode is enabled or disabled it determines how the elements are trimmed. When Ripple mode is disabled, the overall duration of the sequence does not change when trimming. When you remove frames from the head or tail, it creates a gap (see Gaps in the Timeline (page 1941)). If there is an element located before of after the trimmed element, only a head or tail trim is possible. To slip & slide an element with Ripple mode disabled can only be done if there are handles at the head and/or tail of the shot. See Slip & Slide (page 1955).

When Ripple mode is enabled, the overall duration of the sequence changes when trimming. Removing frames from the head or tail does not create a gap and the elements located before or after the trimmed element move in time (based upon the Ripple Start or Ripple End mode). You can slip & slide an element with Ripple mode enabled on any element when there are available handles.

NOTE If a layer is soloed, the Edit tools (Trash, Delete, Trim In/Trim Out, and Mark In/Out) default to 'Ripple End' behaviour, regardless of the Ripple Mode setting. If no layer is soloed, however, the Edit tools behave according to the individual Ripple Mode selected (Ripple Off, Ripple Start, or Ripple End) and apply to the active or visible layer that contains the focus point. For more information about Ripple Modes, see Trimming Ripple Mode (page 1954).

To trim an element:

1. Enable Trim.
2. Select a Ripple mode (i.e., Ripple Off, Ripple Start, or Ripple End). See Trimming Ripple Mode (page 1954).
3. Set the cursor on either the head or the tail of the element and the corresponding head or tail cursor appears.

4 Click and drag the element.
   - Dragging the element in the same direction of the head or tail cursor adds frames to the element.
   - Dragging the element in the opposite direction of the head or tail cursor removes frames from the element.

Trimming Ripple Mode

There are three types of Ripple mode when trimming.

Ripple Off Trimming does not affect the duration of the timeline.

Ripple Start Trimming moves in time the element located before the trimmed element.

Ripple End Trimming moves in time the element located after the trimmed element.

Slipping an Element

Slipping simultaneously trims the head and tail of the element without changing its position in the sequence. The duration of the shot is not altered. Other elements in the edit sequence are not affected and the total duration of the sequence does not change. You can slip a single element or a selection.

NOTE Slipping is a non-rippling action.

To slip an element:

1 Enable Trim.

2 Position the cursor on the middle of the element that you want to slip.
The slip & slide cursor appears.



3 Press the , (comma) key.
The slip & slide cursor changes to a slip cursor.



4 Click and drag the element right or left.



Sliding an Element

Sliding changes a shot's position in the edit sequence without changing the head or the tail frames. It simultaneously slides the shot under the cursor and trims the head and tail of the surrounding gaps.

NOTE Sliding can only occur if there is a gap before and/or after the element.

You can slide an element or a transition. See Transitions (page 1956).

NOTE When you slide the first or last segment of the element, it changes the total length of the sequence. When you slide the middle of the element, the sequence does not ripple.

To slide an element:

1. Disable Trim.
2. Position the cursor on the middle of the element that you want to slide.



3. Click the element.
   The slide cursor appears.



4. Drag the element right or left.



Slip & Slide

Slip and slide changes a shot's position in the edit sequence, as well as the head and tail frames. The media of the element stays at the same position in the sequence, but the location of the element changes.

NOTE Slipping and sliding can only occur if the element has available handles.

To slip and slide an element:

1. Click Trim.
   This enables the Trim mode.
2. Position the cursor on the middle of the element that you want to slip & slide.
   The slip & slide cursor appears.
3. Click and drag the element right or left.

Slip & Slide Ripple Mode

There are three types of Ripple mode when you slip and slide.

Ripple Off With Ripple Off, only the first and last shot of the Timeline and any shot surrounded by a gap, can slip and slide. The slip & slide operation adds a gap and changes the length of the edit sequence.



Ripple Start The slip & slide operation moves in time the element located before the trimmed element and inserts a gap after the slip & slide element.



Ripple End The slip and slide operation moves in time the element located after the trimmed element and inserts a gap before the slip & slide element.



Transitions

A transition is a way to go from one element to another. There are two types of transitions:

- Splices
- Dissolves

Splices

When you add a shot to the Timeline, it automatically creates a splice. A splice is where one element ends and the following element begins, creating a hard transition. It is also possible to add a splice to an element in the Timeline to divide the element into two sections. You can remove a splice from an element, creating a continuous element.

To add a splice:

1. Click and drag the focus point to the layer you want to splice.
2. Move the positioner to the exact location of the element you want to splice.



3. Click Cut.
   A splice is added.



To remove a splice:

1. Place the positioner on the right-most (also known as the B) side of the previously cut shot.
2. Click Join.

   NOTE You cannot join elements coming from two different shots. You can join two shots coming from the same source clip with consecutive source timecode.

Dissolves

A dissolve is the blending of two elements over a defined period of time. By default, Lustre adds a dissolve centred over the transition. You can slide the dissolve to make it start before or after the transition.

Also by default, a dissolve is set to be one second, based upon the project's frame rate (e.g., 24, 25, or 30 frames per second). You can change the duration by trimming the dissolve or by using the dissolve duration field in the Edit menu.

When you are working with multiple layers, elements located on the top layer can dissolve to a gap that reveals the content located on the bottom layer, therefore creating the equivalent of a dissolve between two shots.

Adding a Dissolve

There are two ways to add a dissolve to the upcoming transition of the layer under Focus.

To add a dissolve:

1. Place the focus point and positioner on top of the A-side of the transition where you would like to create your dissolve.
2. Do one of the following:
   - Click Diss.
   - Press `D`.

     A dissolve is centred over the transition.



WARNING When building Timelines with multiple layers and transitions from layer to layer, avoid dissolving from a shot to a dissolve located on another layer. Dissolve only from a shot to a shot, otherwise, Lustre displays a red 'X'.

Removing a Dissolve

You can remove a dissolve from the Timeline.

To remove a dissolve:

1. Select the dissolve, or multiple dissolves.

   The dissolve turns yellow.
2. Do one of the following:
   - Press `Shift +D`.
   - Click Del.

Editing a Dissolve

There are a couple of ways to change the duration of a dissolve.

To change the duration of the dissolve:

1. Right-click and drag the timeline to make sure the right dissolve is showing.

2 Place the positioner on the dissolve you want to modify.

NOTE The positioner needs to be located before the dissolve.

3 Click Edit.

The Edit menu appears.

4 Drag the Dissolve slider to set the dissolve duration. See Changing Dissolve Duration (page 1928).

To change the duration of the dissolve using Trim mode:

1 Enable Trim.

2 Click and drag the dissolve using the Trim Head or Trim Tail cursor.

Changing the Orientation of a Dissolve

By default, Lustre adds a dissolve centred over the transition, but you can change the orientation of the dissolve. See Editing a Dissolve (page 1957).

To change the start/end point of the dissolve:

1 Enable Trim.

2 Move the cursor over the dissolve you want to modify. The slip & slide cursor appears.

3 Click and drag the dissolve to its new position.

NOTE When you add a dissolve, make sure there are enough handles for the duration of the transition. If you add a dissolve over a transition between two elements that do not have head/tail frames, the dissolve goes from the outgoing element to black and from black to the upcoming element. Trimming or slipping the elements may help solve the problem.

Displaying the Multi-Layer Timeline

When there are multiple layers and you display footage in the Player, or play out to tape, or render, Lustre's default behaviour (called Top Vertical Priority) is to display the visible shots from a top-down view of the flattened timeline. This behaviour is further customized by muting layers and prioritizing individual shots. It is also possible to override Top Vertical Priority by soloing a single layer, which forces Lustre to display, play out to tape, and render only that layer.

When selecting segments in the multi-layer timeline, the Storyboard view shows the same selection. See Selecting Shots in the Storyboard (page 1980).

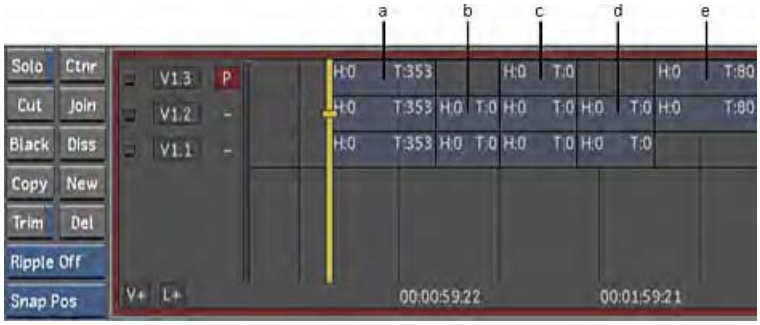
Top Vertical Priority

With exceptions, at any given point in a timeline with multiple layers, the top shot in a vertical stack of layers is the shot that is displayed, rendered, or played out to tape. This is the Top Vertical Priority rule and it is the default behaviour of the multi-layer timeline when Solo mode is disabled (See Solo Mode (page 1963)).

When there is a gap in the top layer, this gap is transparent when looking down on the timeline; Lustre looks in progressively deeper layers (going from top to bottom in a vertical stack) until it finds the top vertical shot to display. To visualize this, it may help to imagine looking down on the multi-layer timeline so that you would have to see through the top layers in order to see the bottom layers. Your view from the top is a 'flattened' view of the timeline.

The only exception to the Top Vertical Priority rule is layer soloing (see Solo Mode (page 1963)). There are a couple of features which alter the way the Top Vertical Priority rule works; layer muting (see Muted Layers (page 1959)), and shot priority (see Shot Priority (page 1960)).

The following example illustrates how top vertical priority determines which shots are displayed, played out to tape, and rendered. It also illustrates how the displayed shots come from various layers throughout the course of the timeline. For simplicity's sake, in this example, there are no muted layers and no shots assigned with shot priority.



(a) Shot 1 (L3) (b) Shot 2 (L2) (c) Shot 3 (L3) (d) Shot 4 (L2) (e) Shot 5 (L3)

You can use the timeline information field to verify the layer and shot being displayed at any given point in the timeline. See Monitoring the Location of the Current Frame (page 1977). For example, in the following screen capture, the shot being displayed is shot 3 on Layer 2, and the current frame is frame 183.

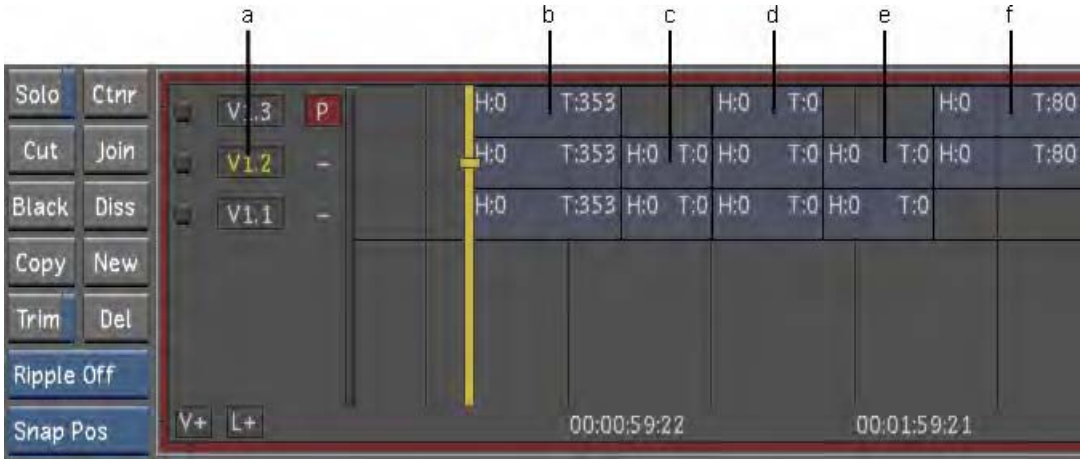


Muted Layers

When a layer is muted, Lustre ignores this layer when applying Top Vertical Priority. That is, at a given point in the timeline, if a muted layer has Top Vertical Priority, Lustre looks at progressively lower layers in the vertical stack until it finds a shot in an unmuted layer, and then displays that shot.

WARNING If you are using the mute function as a method to create editorial or grade versioning, please keep in mind that the status is not part of the grade file data structure. Therefore, this state is not saved upon exiting Lustre.

The following example illustrates how a muted layer is used within a multi-layer timeline to determine what shots are displayed in the Storyboard and Player when the Top Vertical Priority rule is applied.



(a) Muted layer (b) Shot 1 (L3) (c) Shot 2 (L1) (d) Shot 3 (L3) (e) Shot 4 (L1) (f) Shot 5 (L3)

In this example, Layer 2 is muted. Therefore, only the shots in Layer 1 and Layer 3 are active.

To mute a layer:

1 Do one of the following:
- `Ctrl`-click the name of the layer you want to mute (e.g., V1.L1).
- `Shift+M` to mute the layer with focus.

The layer name indicator turns yellow.

To unmute a layer:

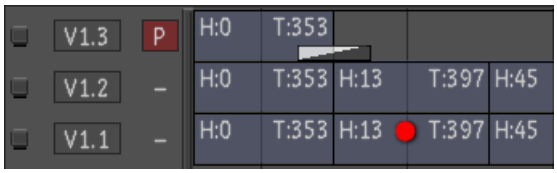
1 Do one of the following:
- `Ctrl`-click the name of the layer you want to unmute (e.g., V1.L1).
- `Shift+M` to unmute the layer with focus.

Shot Priority

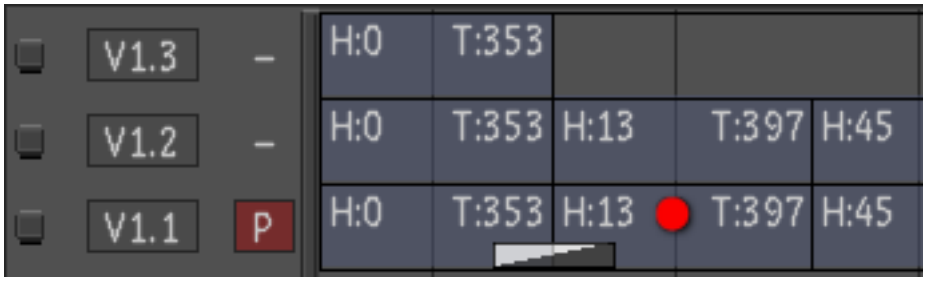
When you work in the Top Vertical Priority mode (Solo is disabled), you can assign priority on a shot-by-shot basis. A 'priority shot' in a given layer is displayed even if there are shots in the layers above it in the timeline. In the case where there are multiple priority shots in a vertical stack, the priority shot with Top Vertical Priority is displayed or updated accordingly in the Player. Furthermore, the Storyboard thumbnails are updated according to priority after an initial refresh. Lustre represents a priority shot in the multi-layer timeline as a shot with a red dot.

The following behaviours are expected when using the shot priority function:
- Shot priority information is saved to the cut file.
- A priority shot survives trimming, slipping, and sliding operations.
- When using the New button to create a new layer, the reproduced shot in the new layer preserves shot priority.
- When using the Copy button to copy a shot, the reproduced shot does not preserve shot priority.
- You can assign shot priority to a virtual black clip.
- You cannot assign shot priority to a gap.
- When performing a change cut operation with a cut that contains multiple versions of the same shot but with different grades, the shot priority feature allows you to designate the version of grading you would like to be transferred to the same shot in the new cut. However, if a muted layer has a priority shot, the grading metadata for this shot is not transferred to the new cut.
- In the case where a shot dissolves into another priority shot on a different layer within the vertical stack of shots, then the first shot does not dissolve seamlessly into the priority shot. For a seamless dissolve into a priority shot, you must create a dissolve on the layer of the priority shot.

  In the following example, there is a dissolve on Layer 3 between shot 1 and shot 2. The dissolve begins near the end of shot 1, but once shot 2 on Layer 1 begins, the dissolve does not continue. Instead, shot 2 begins without showing the second half of the dissolve effect.



  In the following example, the dissolve works as expected because the dissolve is on the same layer as the priority shot.



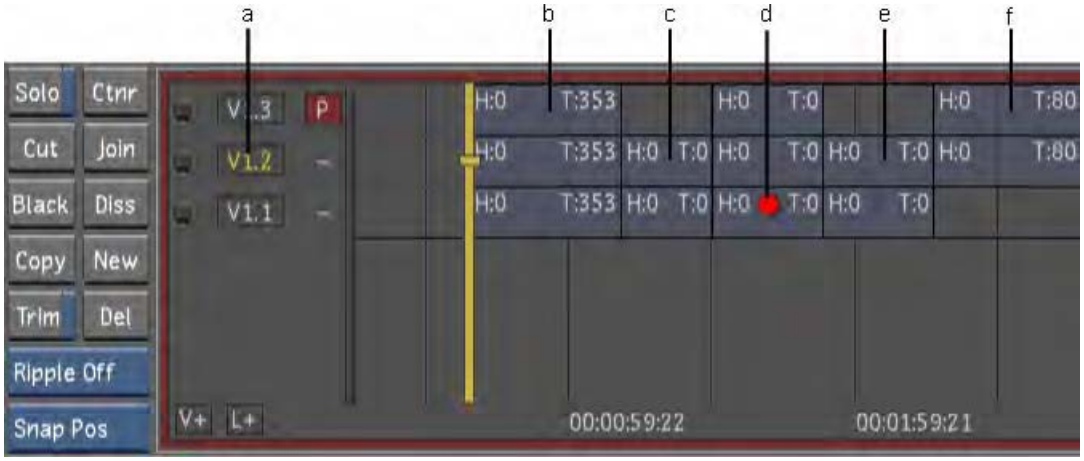
- When one priority shot overlaps another in a different layer, the priority shot with Top Vertical Priority is displayed. In overlap situations, this means that one or more shots are displayed that begin part way through or end part way through the shot.

  In the following example, the first Layer 3 shot is displayed completely and then the Layer 2 shot is displayed starting somewhere in the middle of the shot.



WARNING If you mute a layer that has a priority shot, that shot is not displayed, played out to tape, nor rendered.

The following example illustrates shot priority in the multi-layer timeline.



(a) Muted layer (b) Shot 1 (L3) (c) Shot 2 (L1) (d) Shot 3 (L1) (e) Shot 4 (L1) (f) Shot 5 (L3)

In this example, Layer 2 is muted and Shot 3 in Layer 1 is a priority shot.

To assign shot priority to a shot:

1. In the multi-layer timeline, `Shift`+right-click the shot (in the desired layer) to which you would like to assign priority.

   A red dot appears on the shot.

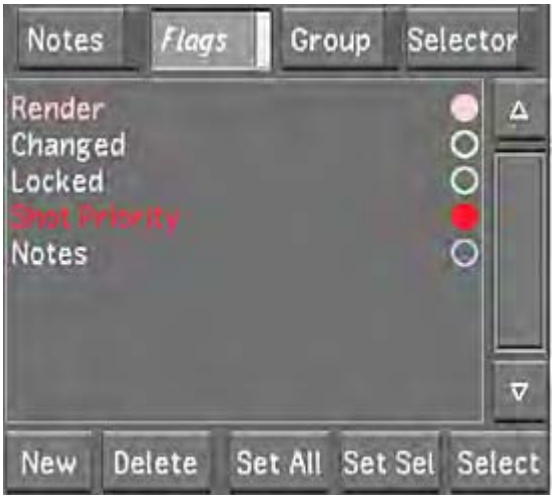
To assign shot priority to the shot in focus:

1. In the multi-layer timeline, make sure the positioner and its focus point are focused on the shot to which you apply priority.
2. Press `Shift+\`.

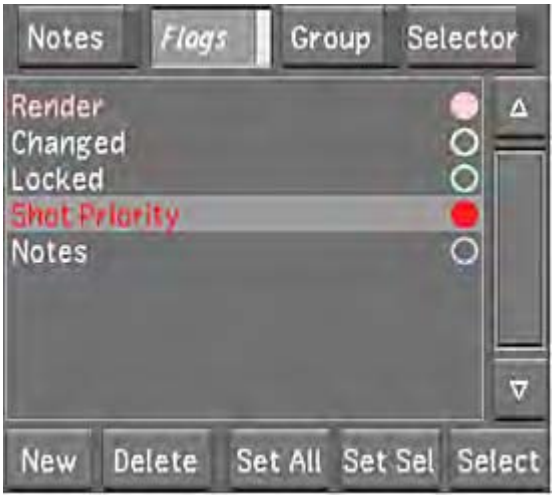
A red dot appears on the shot in focus.

To assign shot priority to several selected shots with the Shot Priority flag:

1. In the multi-layer timeline, select all the shots to which you wish to assign shot priority. See Selecting Elements (page 1937).
2. `Shift`+right-click one of the selected shots to assign shot priority to that shot.
   The Shot Priority flag is enabled.



3. In the Flags menu, left-click the Shot Priority flag.
   The Shot Priority flag is highlighted.



NOTE The indicator is enabled only if the positioner and focus point is over a priority shot.

4. Click Set Sel.
   All the selected shots are assigned shot priority.

To assign shot priority to the current shot of the currently active layer:

1. In the Timeline menu, enable the Flags button.
2. Right-click the Shot Priority flag.

A red dot appears on the current shot of the currently active layer.

To remove shot priority from a shot:

1. In the multi-layer timeline, `Shift`+right-click the shot with priority.
   The red dot is removed from the shot.

To select all shots that are assigned shot priority:

1. In the Timeline menu, enable the Flags button.
2. In the Flags menu, left-click the Shot Priority flag.
   The Shot Priority flag is highlighted.
3. Click Select.
   All priority shots are selected.

Solo Mode

When you enable Solo mode in the multi-layer timeline and place the focus point over a layer, Lustre displays only this layer (including its gaps) in the Storyboard and Player. In addition, this is the only layer that is played out to tape and rendered locally.

A soloed layer in Lustre overrides the effects of layer muting and shot priority (see Muted Layers (page 1959) and Shot Priority (page 1960)). That is, even when a layer is muted, if you subsequently solo this layer, this layer alone is displayed. Likewise, when there are shots with shot priority in several layers, a soloed layer is still the only layer displayed.

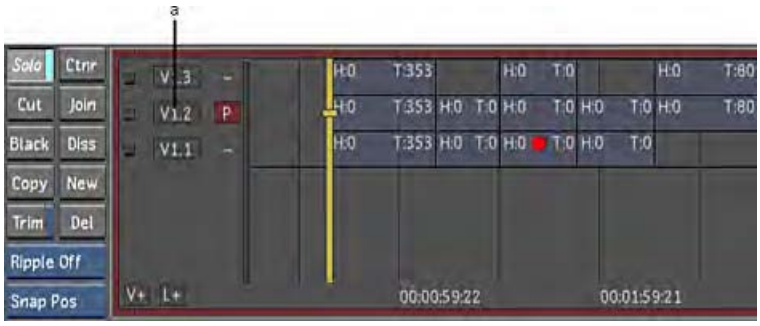
NOTE If you are using the solo function as a method to create editorial or grade versioning, please keep in mind that the status is not part of the grade file data structure. Therefore, this state is not saved upon exiting Lustre.

The following example illustrates how a soloed layer overrides other layer and shot prioritizing factors.

When Solo mode is enabled, the Storyboard is updated to display shots from only the soloed layer. When Solo mode is disabled, what Lustre displays is governed by what layers are muted, the presence of shots assigned with priority, and typical Top Vertical Priority behaviour.

In the following example, Layer 2 is soloed. In this case, only Layer 2 is displayed in the Player and Storyboard, played out to tape, and rendered. It should be noted that although there is a shot assigned with shot priority in Layer 1, only Layer 2 shots are displayed because Solo mode takes priority.

WARNING If a layer is soloed, the Edit tools (Trash, Delete, Trim In/Trim Out, and Mark In/Out) default to 'Ripple End' behaviour, regardless of the Ripple Mode setting. If no layer is soloed, however, the Edit tools behave according to the individual Ripple Mode selected (Ripple Off, Ripple Start, or Ripple End) and apply to the active or visible layer that contains the focus point. For more information about Ripple Modes, see Trimming With Ripple Mode (page 1953).



(a) Soloed layer

To solo a layer:

1. Move the focus point of the positioner to the layer you wish to solo.
2. Enable the Solo button.

To un-solo a layer:

1. Disable the Solo button.

   NOTE This operation un-solos the layer regardless of where the focus point is.

Rendering the Timeline

There are numerous scenarios when you render a timeline. See Rendering (page 2215).

Modifying the Timeline's Starting Record Timecode

You can change the timeline's start timecode by entering either a start timecode or the number of frames you want to start from. Modifying the start timecode does not delete frames that precede the new start frame.

To modify a shot's starting timecode:

1. Click Editing in the main menu, and then click Edit to display the Edit menu.
2. Do one of the following:
   - To enter a timecode, click in the Start TC field and use the numeric keypad to enter the timecode at which you would like the timeline to begin. This value is converted into the corresponding frame number (as shown in the frame field).
   - To enter a frame number, click the Start TC field, then click the frame field and use the numeric keypad to enter the frame number at which you would like the timeline to begin. The frame number is converted into the corresponding timecode value.

   NOTE `Ctrl`-click either the frame or Start TC field to clear it.



(a) Frame field (b) Start TC field

3 Press `Enter` on the numeric keypad.

Performing a Confidence Check

A confidence check is a process in which you manually check the contents of your timeline (which you created by assembling shots from the scanned film frames) against a digital reference cut captured directly from an offline edit video tape or other medium. This is an important step in ensuring that the assembled timeline contains the frames specified by the offline editor. If, for any reason, there is an offset between your current timeline and the digital cut, your picture could end up out of sync with the sound.

NOTE The confidence checking function allows you to synchronize the start frames of the timeline and digital reference cuts and does not involve a synchronization of timecodes.

To perform a confidence check:

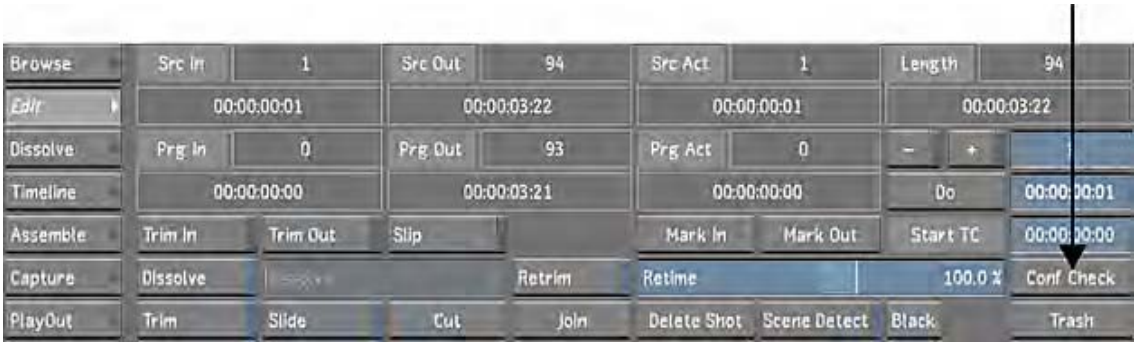
1 Load or create the timeline by doing one of the following:
- Loading a current timeline. See Importing Timelines as Source Clips (page 1937).
- Creating a newly assembled timeline from an EDL. See Working with EDL, ALE, and Cutlist Files (page 1892).

2 Save the current cut, create a new cut, and load it.

3 Capture the footage you want to use as a point of comparison (this is the offline digital cut). See Capturing Material (page 2264).

4 Click Editing, and then click Edit to display the Edit menu.

5 Drag the digital reference cut from the Library to the Confidence Check button.



A split screen display appears. The current first frame is loaded in the left viewer and the digital cut is loaded in the right viewer with the first frame displayed.

6 Navigate between shots to make sure each timeline shot starts on the exact same frame as in the reference cut. You must check the two versions by eye in search of discrepancies. If, for any shot, you determine the two start frames (timeline and reference) do not start on the same frame, synchronize the start frames.

   Synchronize the reference cut to the timeline by offsetting the reference cut in the right viewer:

   - Left-click the reference image (right-side image) and drag left to scrub in reverse.
   - Left-click the reference image and drag right to scrub forward.
   - Middle-click the reference image to go one frame forward.
   - Right-click the reference image to go one frame backward.

7 Navigate within each shot by stepping through the cut points frame-by-frame to make sure there are no shots that are offset.

   If you determine there are shots that are offset, you can use the Editing tools (trim, slip, and slide) to adjust the shots to the correct frames. This is usually performed in the timeline cut and not the reference cut.

   To trim from the Edit menu, see Trimming from the Edit Menu (page 1917).

   To trim from the Timeline menu, see Trimming Elements (page 1952).

It is important to note that when matching your timeline to a reference clip, several editing changes are automatically updated in the Player without the need to release the mouse button:

- trimming
- slipping
- sliding

8 Save the current cut.

   The confidence check reference is saved with the cut. When using a clip from the Library for a Confidence Check, the reference clip, its timing offset and the viewing mode (Single or Dual and Wipe position) are saved with the cut information. When reloading a cut, you do not need to manually reload the Confidence Check clip and set the proper offset value. Simply clicking the Conf Check button (located in the Editing/Edit menu) displays the confidence check view with the reference clip and the defined offset value.

   NOTE Only source clips can be used as a Confidence Check reference. Sequences are not supported.

Using Scene Detection

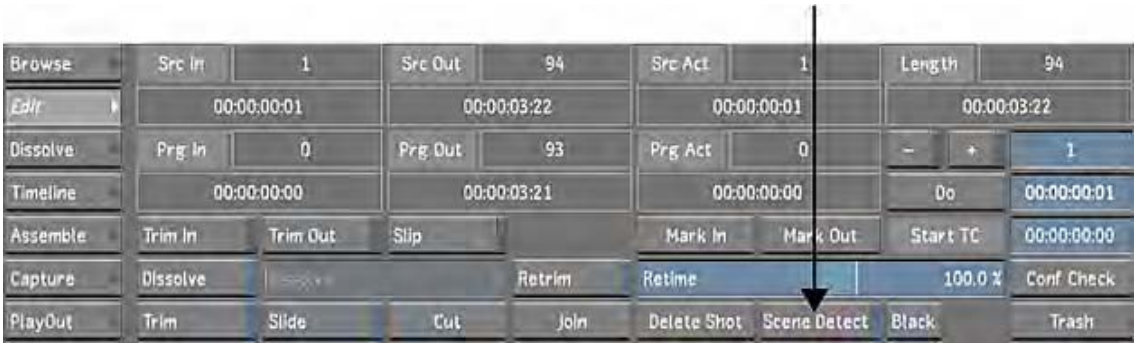
Use scene detection when you are working on a single shot made up of multiple scenes. Scene detection locates the frames when scenes change and splices the shot at those points. When you apply scene detection to a shot in the Storyboard, it is broken up into numerous smaller pieces that you can then colour grade independently.

For example, you may have a finished edited HD master tape that needs to be colour graded before it is ready for broadcasting. You can capture the entire tape as one shot, and then use scene detection to cut it into separate shots. You can then colour grade each shot one-by-one.

NOTE Lustre uses the scene's Src In number to assign unique ID numbers during scene detection. This ensures that any scene captured more than once has the same number assigned to it, which is useful when scenes are transferred from Lustre to other effects applications, and back again. By using the Src In number as the basis for assigning numbers, the grading metadata applied to a scene that has been captured and put through scene detection more than once, is maintained.

To use scene detection:

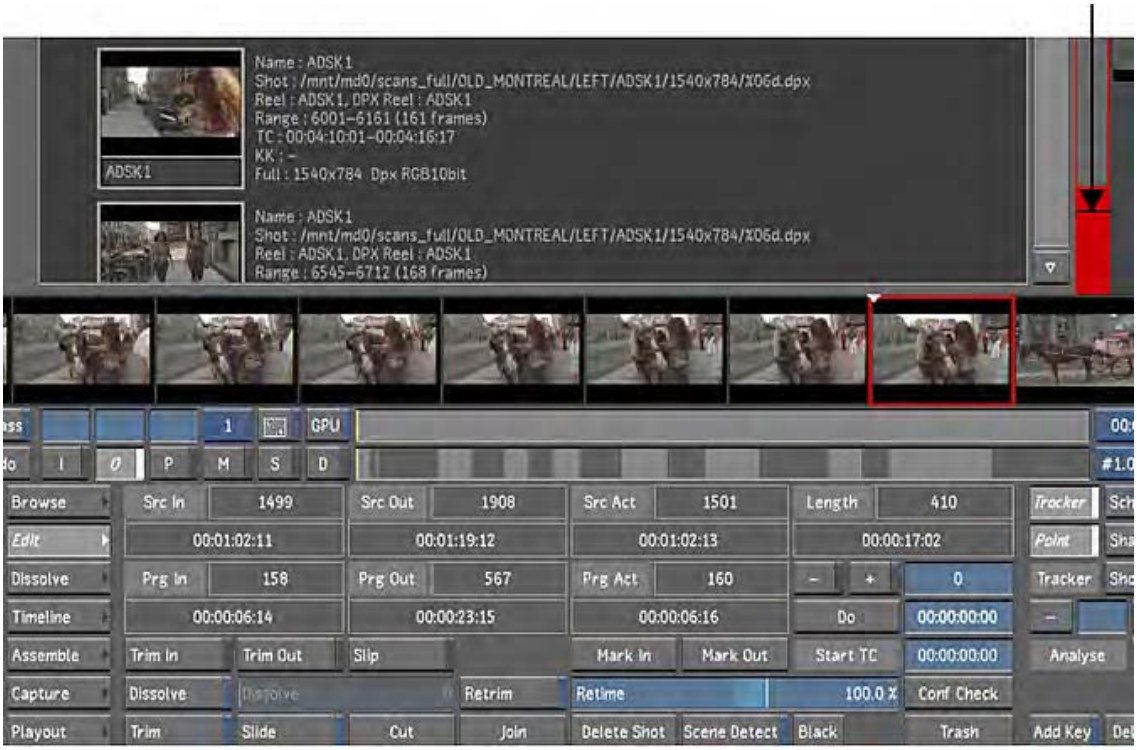
1. Capture from an edited video tape. See Capturing Material (page 2264).
2. Create a new empty cut. See Managing Cuts (page 1773).
3. Drag the digitized shot from the Library to the Storyboard.
4. Click Editing, and then click Edit to display the Edit menu.
5. Enable Scene Detect.



TIP If the Editing While Grading button is enabled in the User settings, you can start a scene detection from any menu by pressing `Shift+S`. For more information about the Editing While Grading button, see System & Menu Settings (page 1762).

A threshold indicator bar appears to the right of the Library. When a scene change is detected, the indicator jumps and turns red, and a splice is introduced into the Storyboard, Timeline, and timebar at that frame. When the process is completed, a series of shots is created in the Storyboard.

6. You can change the threshold by dragging the thin black horizontal line in the threshold indicator up or down. When you lower the threshold value, a small amount of change is enough to trigger a splice. If the value is too low, splices appear in the middle of a scene. If the value is too high, some changes may not be detected. Generally, the default threshold value gives a good result.



7 Press `Spacebar` to pause the scene detection process.

NOTE To continue a paused scene detection, press `Spacebar` again. To stop scene detection, hold down your left mouse button.

8 If, for any reason, a scene change is not detected, you can add a cut manually by clicking Cut. See Inserting Cuts (page 1924).

Playing, Viewing, and Sorting Shots

Lustre offers multiple options for playing, viewing, and sorting shots during the creative process. Use the Player to play one or more shots with your choice of view options, such as proxy resolution with the red colour channel displayed. Move through shots on the timeline using one of several navigation methods. Sort the shots in the timeline so grading can be completed with more ease. Use a reference shot to facilitate tasks such as colour matching. View the reference shot in a separate window or in split-screen with the current shot. If the addition of multiple effects are slowing down playback time, you can cache frames to bring the playback speed closer to real-time.

Playing Shots

You can play shots in the Player from any menu in Lustre using either playback controls or hotkeys. To play shots, you must first load them into the Storyboard. See Creating a Cut with Shots in the Library (page 1876).

You can also use a full-screen Player to view shots without the clutter of menus. This Player is particularly useful for viewing 2K footage at full resolution.

When you play shots in the Player, you can do the following:

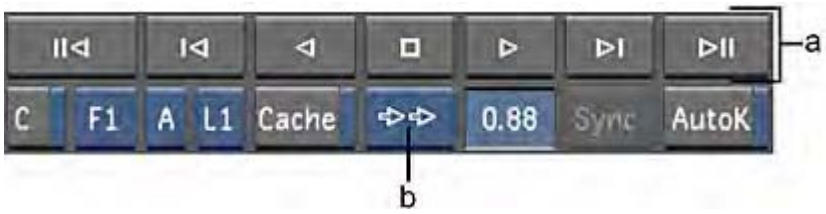
- Loop the entire Storyboard.

- Loop the current shot in the Storyboard.
- Loop between in and out points.

NOTE For information on playing shots with audio, see Using Audio (page 1844).

To play shots in a cut:

1. Load your shots into the timeline. See Adding Shots to the Timeline (page 1936).
2. Click the Play Mode button to cycle through playback range options.



(a) Playback controls (b) Play Mode button

Select:	To specify a looped playback range:
	Of the entire timeline.
	Of the current shot.
	Between in and out points.

3. To set in and out points, go to the start and end frame and mark the in and out points.

Press:	To:
`Shift+I`	Mark an in point.
`Shift+O`	Mark an out point.
`Shift+L`	Clear in and out points.

4. Use the following playback controls or hotkeys to play your shot forward or backward.

Click:	Or press:	To:
	`Up Arrow`	Play the cut forward.
	`Down Arrow`	Play the cut backward.
	N/A	Stop playback.

Click:	Or press:	To:
N/A	Spacebar	Start or stop playback in the direction last played.

NOTE For information on navigating to a particular frame, see Navigating through Shots (page 1975).

Using the Full-Screen Player

The full-screen Player allocates the entire screen to the image. Use it when you want to maximize screen space for viewing shots, for example, while colour grading.

There are no playback controls available in the full-screen Player. Instead, you use hotkeys and mouse gestures to play shots, access both playheads, display two or more shots at the same time, and pan and zoom the image. You can also display the Storyboard, vectorscope, histogram, and waveform monitor. See Viewing Colour Distribution (page 2049).

NOTE The histogram, vectorscope, and waveform monitors do not dynamically update when GPU is enabled. They retain the colour distribution of the image prior to GPU being enabled.

The full-screen Player is available from all menus except the Editing menus.

To use the full-screen Player:

1. From any menu in Lustre except the Editing menus, press Enter.

   The full-screen Player appears.
2. Use hotkeys or mouse gestures to play the shot, zoom in or out, pan the image, change the view, or switch playheads. You can also display the Storyboard, vectorscope, histogram, and waveform monitor. See Player Hotkeys (page 1996) for a list of the hotkeys and mouse gestures commonly used in the full-screen Player.
3. To exit the full-screen Player, press Enter again.

Viewing Reference Images

The Player has two playheads, A and B, which allow you to quickly access two different areas of your cut. Lustre keeps track of the location of the current frame on each playhead as you navigate through the cut, so you can instantly return to that location by switching playheads.

You can use the playheads to view reference images while you work. Reference images are useful when you want to create continuity grades, apply input LUTs, or create effects with Sparks. You can also switch between the current shot and a reference shot if you do not need to view them simultaneously.

It is recommended that you work on your shot in Playhead A and load the reference images into Playhead B. For example, in Playhead B, you can view a reference image from the shot adjacent to the one you are currently working on in Playhead A. Visually compare the two images to match the saturation, whites and blacks, and the colours of the shot you are working on to the image in the reference buffer. The Playhead button indicates which playhead is active.

TIP After you have assigned shots to the playheads, you can view them together in Dual View. See Using Dual View (page 1986).

To view a reference image loaded from a cut:

1. Press F1 or select A from the Playhead button and then select the shot in the Storyboard.



   The shot you are working on is assigned to Playhead A.
2. In the Storyboard, go to the frame you want to use for the reference image.
3. Press K to store the frame in the reference buffer.
   The current frame overwrites the last frame loaded into the reference buffer.
4. Press F2 or select B from the Playhead button.
   The shot is assigned to Playhead B.
5. Press L or click the S button to display the reference image in Playhead B.



   The reference image appears in the Player.

   TIP At any time, press L to toggle between the original shot (O button) and the reference image (S button).

   The current shot on each playhead is indicated by the border colour in the Storyboard.

Border colour:	**Indicates the current shot on:**
Bright red	Playhead A (active).
Bright purple	Playhead B (active).
Dimmed red	Playhead A (inactive).
Dimmed purple	Playhead B (inactive).

6. To work on the shot you assigned to Playhead A, press F1 or select A from the Playhead button.
   The shot you are working on is selected. You can move back and forth between the shots in Playhead A and Playhead B by pressing F1 and F2, respectively.

To view a reference image loaded from a Grade bin:

1. Press F1 or select A from the Playhead button and then select the shot in the Storyboard.
   The shot you are working on is assigned to Playhead A.
2. In the Grade bin, right-click the frame you want to use for the reference image.
   The frame from the Grade bin overwrites the last frame you loaded into the frame buffer.
3. Press F2 or select B from the Playhead button.

The shot is assigned to Playhead B.

4 Press L or click the S button to display the reference image in Playhead B.

The reference image appears in the Player.

TIP At any time, press L to toggle between the original shot (O button) and the reference image (S button).

5 To work on the shot you assigned to Playhead A, press F1 or select A from the Playhead button.

The shot you are working on is selected. You can move back and forth between the shots in Playhead A and Playhead B by pressing F1 and F2, respectively.

To switch between a reference image and the shot you are working on:

1 Load the reference image to the frame buffer. Do one of the following:
- Press K to load the current frame in the Storyboard to the frame buffer.
- Right-click a Grade bin thumbnail.

2 In the Storyboard, select the shot you are working on.

The shot appears in the Player.

3 To toggle between the reference image and the shot you are working on, press L.

Panning and Zooming the Image

You can pan and zoom the image from any menu in Lustre except the Editing menus. You can pan and zoom in both the Player and the full-screen Player.

To pan the image:

1 Place the cursor over the image, press and hold the middle mouse button, and drag in the direction you want to pan.

To zoom the image:

1 Do one of the following:
- Place the cursor over the image, press and hold the middle and right mouse buttons, and drag right to zoom in or left to zoom out.
- Press and hold the mouse button in the Zoom field and drag right to zoom in or left to zoom out. Click once in the field to reset it to 1.00 (Player only).
- Right-click in the Zoom field to display the calculator, and then use it to enter a zoom value.



TIP Zooming normally occurs in increments of 0.1. However, to quickly jump between a zoom factor of 1.00 and 2.00, press F10.

Setting Image View Options

Lustre has several view options. As you work on a grade, you can:

- Display various versions of the image (for example, before or after secondary colour grading).
- Toggle between full resolution and proxy resolution.
- Display a specific channel of the image.
- Switch between fields when working with interlaced footage.
- Switch between viewing LUTs.

Setting the View Mode

Use the View Mode buttons to view different versions of the image as you work on your grade.

To set the view mode:

1 Click one of the View Mode buttons.



Click:	Or press:	To view:
I	I	The image with colour corrections made in the Grading and Curves menus only. **NOTE** You cannot select this mode if no corrections have been made in the Secondaries menu.
O	O	The output image, which contains the result of all corrections applied. This is the default viewing mode.
P	P	The printed, or rendered image. See Rendering (page 2215). If the frame has not been rendered, a red X appears in the image window. **NOTE** If you rendered a shot and still see a red X in the image window, press Shift+P to force Lustre to re-detect all rendered files.
M	F11	The matte on a secondary correction. Press F11 to toggle between Secondary (Overlay) and Matte (Greyscale) view modes for the matte. See Creating a Secondary by Keying a Range of Colours (page 2116).
S	L	The stored reference image. See Viewing Reference Images (page 1970).

The image changes to the selected view mode.

Selecting the Resolution

While working, you can display the full-resolution version of shots or half-resolution proxies in both the Player and full-screen Player. Displaying proxies speeds up interaction and playback on shots with many effects added to them. Half-resolution proxies are also useful for playing to projectors and for saving disk space—they take up only 1/4 of the space of the original footage. For example, you can store proxies on the server and full-resolution footage on the main storage device.

You can switch between full and half resolution any time. Generating and displaying proxies does not affect your original footage in any way. The resolution is independent of the grade file data.

To display proxies, you must first generate them. See Generating and Viewing Proxies (page 2227).

To set the resolution:

1 In any menu, click the Full/Half Resolution button.



The resolution toggles between full and half (proxy) resolution. An ungenerated proxy appears as a red X with a black background.

NOTE Buttons for switching between full and half resolution are also found in the Grade and Render menus.

Setting the Active Field for Display in the Player

Interlaced video formats divide frames into two line-based fields. One field consists of the first and subsequent odd lines in the frame, and the other field consists of the second and subsequent even lines in the frame. During the recording process, the images that make up each field are recorded at slightly different moments. For example, when recording NTSC, field 1 is recorded 1/60th of a second before field 2.

While in Output view mode, Lustre can only display one field at a time. The field that is displayed is the active field. When you render the footage, both fields are processed.

NOTE Setting the active field affects preview display only and does not impact the final output. Rendering is performed according to field dominance grade settings selected during project creation.

To set the active field:

1 Toggle the F1/F2 button.

NOTE You need to select Interlaced in the Setup > Grade menu in order to enable this button.

This switches the view in the Player between field 1 and field 2.



Setting the Viewing LUT

Use the LUT button to display the image using one of three Print LUTs selected for viewing purposes. See Defining Print LUTs for Viewing (page 1854).

To cycle between LUTs:

1 Click the LUT button one or more times.



The button label changes to L1, L2 or L3, indicating the currently loaded LUT.

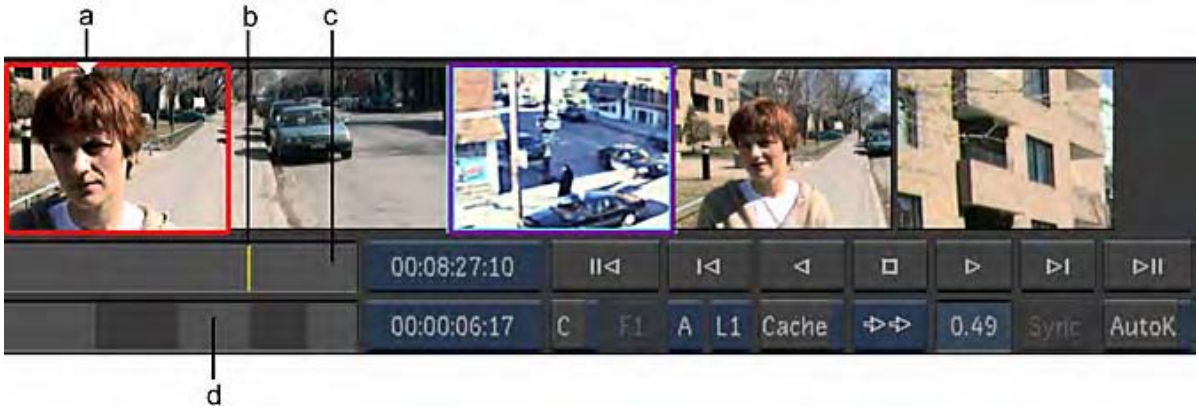
Navigating through Shots

Various methods are available for navigating through the shots in your cut. You can navigate using:

- Storyboard and the shot and scene timebars.
- Playback controls and hotkeys.
- Timecode field.
- Timecode calculator.

Navigating Using the Timebar and Storyboard

Lustre has two timebars to help you navigate through your cut. The Shot timebar represents the current shot, and the Scene timebar represents all the shots in the cut. Use either timebar to move to another frame.



(a) Storyboard position arrow (b) Shot positioner (c) Shot timebar (d) Scene timebar

In the Scene timebar, shots are represented by grey bars in alternating shades of grey, which makes them more visible. The length of each bar is proportional to the length of the shot. Each timebar has a positioner that indicates the location of the current frame. In the Storyboard, the position arrow also indicates the position of the current frame.

To move to a new location in the cut:

1 Click either the Scene or Shot timebar.
The positioner moves to the frame corresponding to the location you clicked.

To scrub through one or more shots:

1 Drag the positioner of either timebar to the left or right.

To pan the Storyboard:

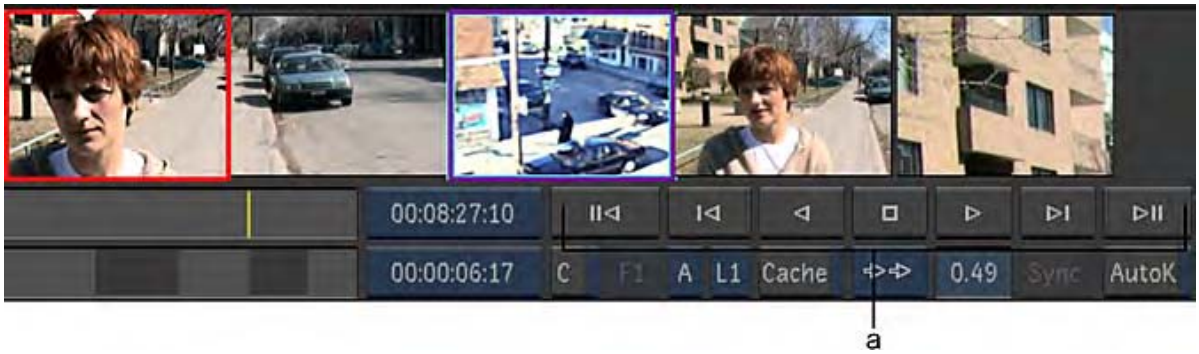
1 Place the cursor over any thumbnail, press the middle mouse button, and drag to the left or right.

To go to the first frame in a shot:

1 Click the shot thumbnail in the Storyboard.

Navigating with the Playback Controls and Hotkeys

You can use playback controls to navigate through shots. There are hotkey equivalents for several of these controls.



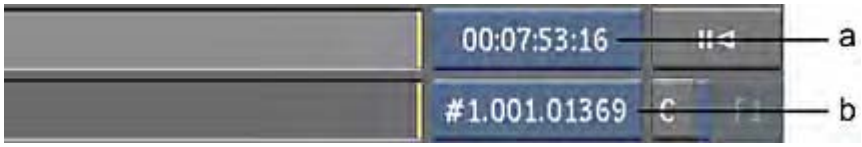
(a) Playback controls

Playback Control	Hotkey	Description
⏮	N/A	Goes to the first frame in the cut. To step one frame back, right-click the button.
⏮	`Left Arrow`	Goes to the first frame in the shot. Click or press repeatedly to move backward from shot to shot.
◁	`Down Arrow`	Plays the cut backward.
□	`Spacebar`	Stops playback.
▷	`Up Arrow`	Plays the cut forward.

Playback Control	Hotkey	Description
	Right Arrow	The playback control goes to the last frame in the shot. The hotkey goes to the first frame in the next shot. Click or press repeatedly to move forward from shot to shot.
	N/A	Goes to the last frame in the cut. To step one frame forward, right-click the button.

Monitoring the Location of the Current Frame

The Source Timecode field indicates the source timecode for the current frame. The timeline information field indicates the location of the current frame, as well as other information pertaining to the shot.



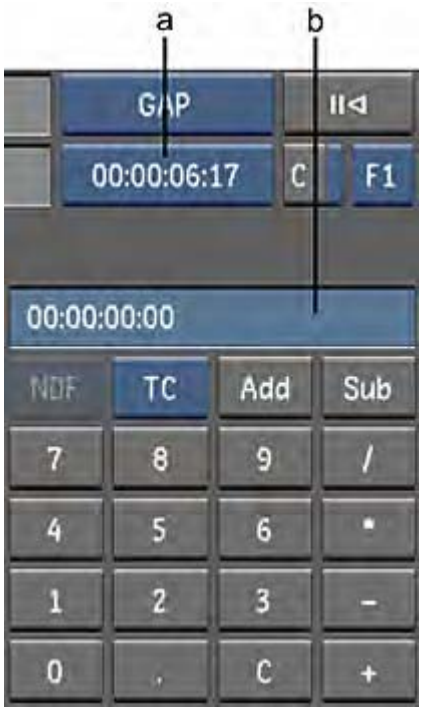
(a) Source Timecode field (b) Timeline Information field

You can click the timeline information field one or more times to display other information, as described in the following table.

Information:	Indicates:
	Frame number of the current frame.
	Frame rate (when playing shots).
	Timecode of the current frame.
	Frame number in film feet/frames.
	Layer, Shot, and Frame numbers of current frame in the following format: #LL.SS.FF (LL-Layer, SS-Shot, FF-Frame)

Navigating With the Timecode Calculator

The timecode calculator is displayed when you right-click in the timeline information field with either the frame number or timecode of the current frame displayed.



(a) Timeline Information field (b) Timecode calculator

You can use the timecode calculator to do the following:

- Jump to a specific frame on the Storyboard.
- Calculate timecodes from frame count and vice versa.
- Switch between drop frame and non-drop frame timecodes (NTSC only).
- Convert drop frame timecodes to non-drop frame timecodes and vice versa.

The timecode calculator operates much like the standard Lustre calculator, but with some differences.

Click:	**To:**
The calculator's numeric field	Jump to the frame number that corresponds with the value entered in the calculator's numeric field.
Add	Jump ahead the number of frames entered in the calculator's numeric field.
Sub	Jump back the number of frames entered in the calculator's numeric field.
. (period)	Insert two zeroes in the calculator's numeric field.
TC/FRM	Switch between the timecode and frame number. This button indicates the current state. TC indicates timecode, and FRM indicates frame.
DF/NDF	Switch between drop frame and non-drop frame timecodes. This button indicates the current state. DF indicates Drop Frame mode. NDF indicates Non-Drop Frame mode. This button is only enabled if 30 or DF is set as the project frame rate in the Setup Project menu.

NOTE If you enter a timecode or frame number that exceeds the shot or Storyboard limit, nothing will happen.

To calculate timecodes from frame counts:

1. Click the timeline information field until the frame number is displayed.
2. Right-click the timeline information field to display the timecode calculator.
3. Using the timecode calculator's numeric keypad, enter a frame number.
4. Click FRM. The corresponding timecode is displayed.

Storyboard Viewing Options

The Storyboard is a thumbnail representation of your timeline. Each individual thumbnail represents a single shot. The Storyboard has multiple viewing options. You can:

- Show or hide the Storyboard.
- Display the Storyboard in regular or large view.
- Select one or more shots.
- Identify selected shots by means of their selection colours.
- Refresh Storyboard thumbnails.
- Collapse the Storyboard thumbnails.
- Automatically update the Storyboard thumbnails.

Showing and Hiding the Storyboard

You can hide the Storyboard to increase the space available for the image you are working on. You can do this from all menus except the Editing menus — the Storyboard is always visible from the Editing menus.

To show or hide the Storyboard:

1. Press `Tab`.

Setting the Storyboard View

You can display the Storyboard in regular or large Storyboard view. In regular Storyboard view, it appears below the image window. In large Storyboard view, it appears to the left of the browser, where the Library is usually displayed. Large Storyboard view displays more shots at once in the Storyboard.

You can do all operations, such as selecting or deleting shots, from either Storyboard view. You can display the large Storyboard view from all menus except the Editing menus, in which case, the Library always appears to the left of the browser.

To toggle between regular and large Storyboard view:

1. Press `Q`.

   The Storyboard switches views. When you switch to large Storyboard view and are viewing 2K images, the image changes to half resolution to accommodate the Storyboard.

Updating the Storyboard Thumbnails

As you are making changes to the shots within your timeline, you may want to automatically see the changes within the Storyboard thumbnail. The update Storyboard option allows this to happen.

To enable or disable the automatic Storyboard thumbnail update option:

1 Do one of the following:

- Within the Setup > Interface menu, click the Update Storyboard button.



- Press `Shift+1`.
- Within the Display & Interface menu of the user configuration settings, click Update Storyboard. See Display & Interface Settings (page 1759).Refer to “Display & Interface Settings” in the “Project Management” chapter in the Lustre User Guide.

If the Update Storyboard feature is enabled and you make changes to any shot that is not currently in the Player, a grey dot will appear on the bottom-right corner of the thumbnail. The grey dot signifies that a change has been made to that shot, but the thumbnail has not been automatically updated.



When you apply changes to a group of shots, only the thumbnail of the current shot within the Player is automatically updated. To update the other thumbnails, click the Storyboard thumbnail with the grey dot.

Selecting Shots in the Storyboard

You can select a single shot or multiple shots in the Storyboard. Multi-select shots to copy parameters, delete multiple shots, and create groups.

When selecting Storyboard thumbnails, the multi-layer timeline shows the same selection. Likewise, when selecting multi-layer timeline segments, the Storyboard shows the same selection. It should be noted that:

- The Storyboard view shows segments as being selected provided these segments are the topmost shot or have shot priority.
- The Storyboard view shows all segments in the multi-layer timeline if Solo mode is enabled.

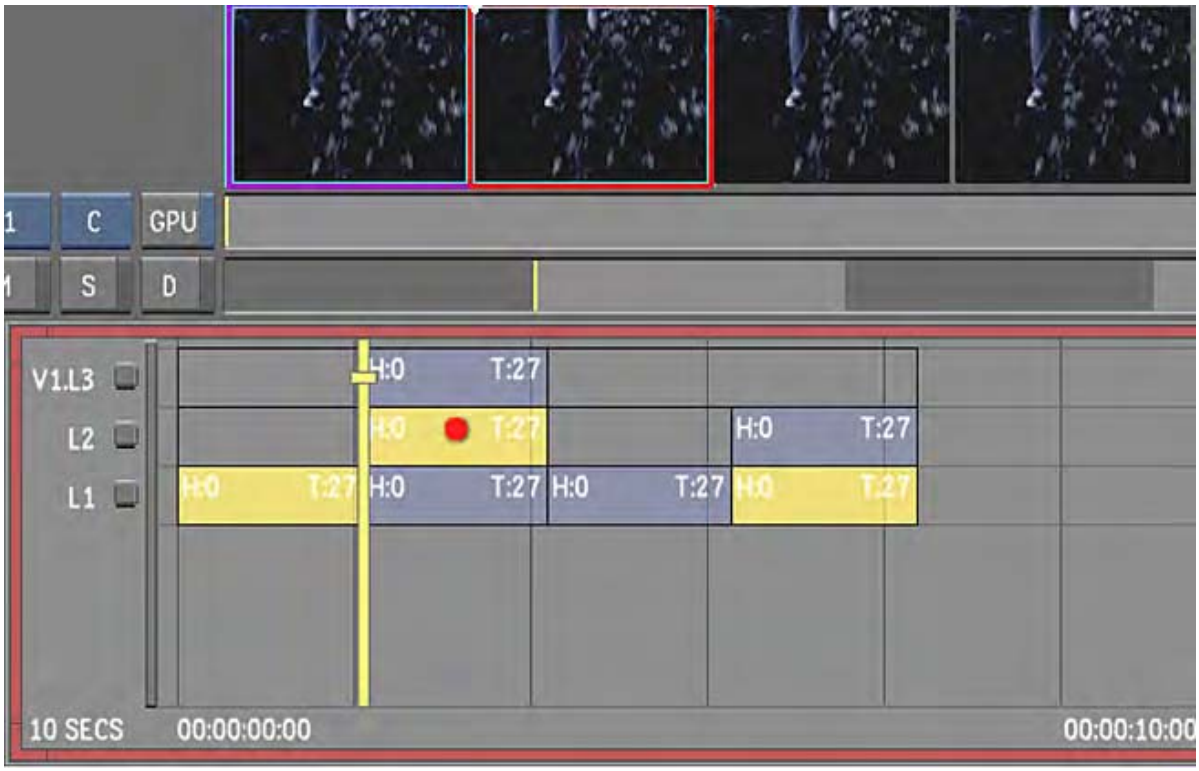


Image courtesy of Moviworld / UK File & TV Company / Videolab

In the above example, all the selected shots in the multi-layer timeline are displayed and selected in the Storyboard view except for the far-right Layer 1 shot. The Layer 1 shot on the far right is neither displayed nor selected in the Storyboard. For more information about top vertical priority and shot priority, see Top Vertical Priority (page 1958).

To select a single shot:

1. Click the thumbnail.
   The border changes to red, indicating it is the current shot.

To select multiple shots:

1. Click the first shot you want to select.
   The border changes to red, indicating it is the current shot.
2. Right-click the subsequent shots you want to select.
   The border of each subsequent selected shot changes to aqua.

To select a range of shots:

1. Right-click the first shot you want to select and drag across the range of shots while holding down the mouse button.

To deselect multiple selected shots:

1 Right-click the selected shots.

Storyboard Selection Colours

As you work with shots in the Storyboard, the border colours indicate the selection state, as described in the following table.

Colour:	Indicates:
Red	The current shot on Playhead A.
Pink	The source shot picked up in the Library for a Shot Replace operation.
Aqua	Multiple selected shots (except for the current shot).
Yellow	Shots included in a group (in Gang mode).
Purple	The current shot on Playhead B.
Orange	A target shot in a Shot Replace operation.
Blue	A shot picked up from the Library.
Green	A shot picked up from the file browser.

Refreshing Storyboard Thumbnails

As you work on your material, you must refresh the Storyboard thumbnails so that they will reflect the changes made to the image. You can refresh them anytime. You can refresh the current shot, or refresh multiple thumbnails at once.

To refresh a single Storyboard thumbnail:

1 Select the thumbnail you want to refresh. You can be on Playhead A or B.
The thumbnail border turns red if you are on Playhead A, or purple if you are on Playhead B.
2 Press . (period).

To update multiple Storyboard thumbnails simultaneously:

1 Press Shift+. (period).
The thumbnails from the current shot onwards are updated.
2 To stop the update process, click any mouse button.

Displaying Only Selected Shots in the Storyboard

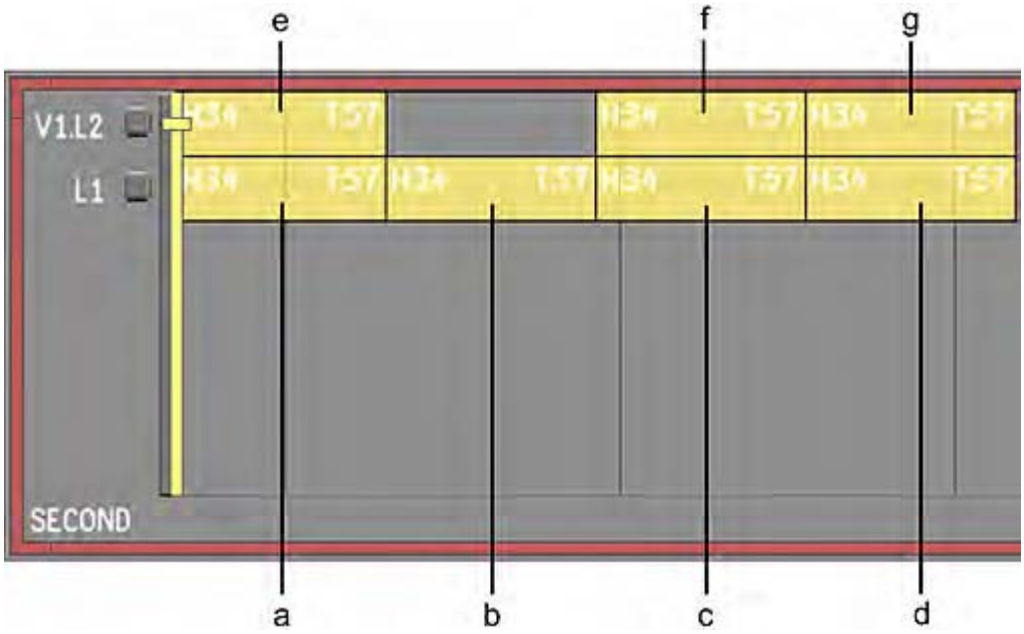
In the multi-layer timeline you can select shots on multiple layers and collapse the Storyboard so that only selected shots are visible. This is useful to isolate certain types of shots for grading, such as outdoor scenes. In addition, in the multi-layer timeline, you can select all the shots on all layers and use the collapse function to align all layers sequentially in the Storyboard.

NOTE Another way to present a multi-layer timeline selection in the Storyboard for grading purposes is to select either A-Mode or C-Mode for sorting the timeline. For more information about the Timeline Sort feature, see Timeline Sort Mode (page 1990).

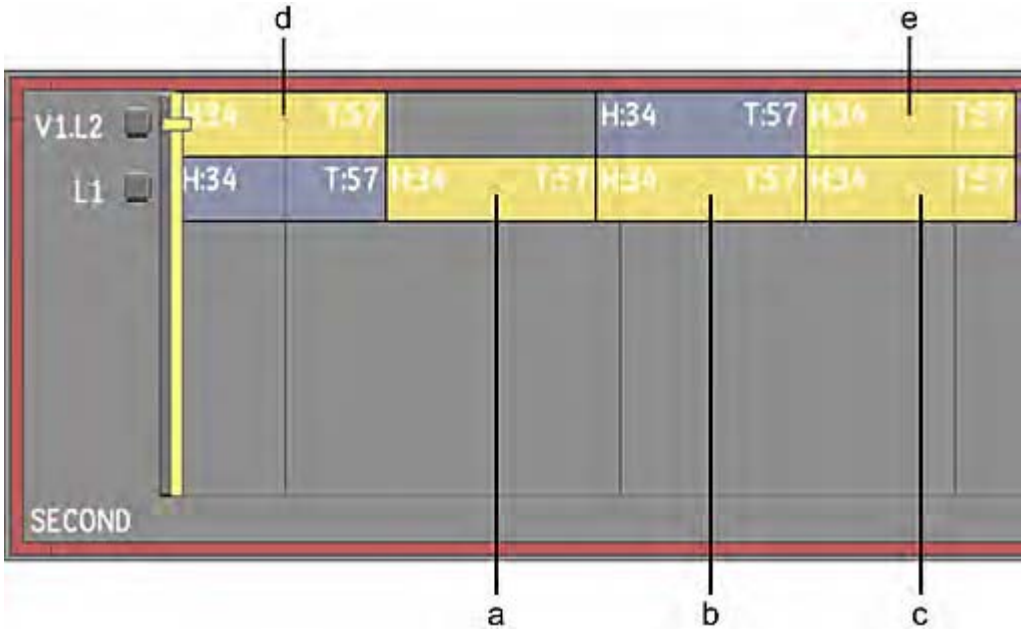
When you enable the collapse function, Lustre disables the Edit, Dissolve, Editing > Timeline, and Colour > Timeline menus. If you are in one of these menus when you enable the collapse function, Lustre takes you to the Colour > Grading menu. This behaviour guards against accidentally modifying the editorial structure on footage not being displayed while in the collapsed view.

The viewing order of shots in the collapsed view of the Storyboard starts from shot 1 in Layer 1 and goes to the last shot in Layer 1 before starting on the first shot of Layer 2.

In the following example, all shots on all layers are selected. The order in which shots are displayed in the Storyboard when the collapsed view is enabled is from a to g.



In the following example, several shots are selected and some are not. The order in which shots are displayed in the Storyboard when the collapsed view is enabled is from a to e.



To align all shots on all layers into a single Storyboard sequence:

1. In the multi-layer timeline, select all shots on all layers. See Selecting Elements (page 1937).
2. Press F on the keyboard, or enable the C button.



All shots on all layers are displayed in the Storyboard.

NOTE To disable the collapse function, press F on the keyboard again, or click the C button again.

To align a selection of shots from one or more layers into a single Storyboard sequence:

1. In the multi-layer timeline, select the desired shots. See Selecting Elements (page 1937).
2. Press F on the keyboard, or enable the C button.
   All selected shots are displayed in the Storyboard.

NOTE To disable the collapse function, press F on the keyboard again, or click the C button again.

Setting the Viewing Options

Lustre has several ways to view and work with single or multiple shots in the Player or the SDI monitor.

Within the Viewing Options, you can select the following:

- **Colour View** You can display all of the colour channels of an image, or individual channels in greyscale. The greyscale versions of colour channels represent, in values of grey, the amount of that colour channel found in each part of the image. The darker the grey, the greater the amount of colour present.

- **Single View** This is the default viewer. You can select which playhead you want to display in the Player (i.e., Playhead A or B).
- **Dual View** You can display the image of Playhead A and Playhead B at the same time. See Using Dual View (page 1986).

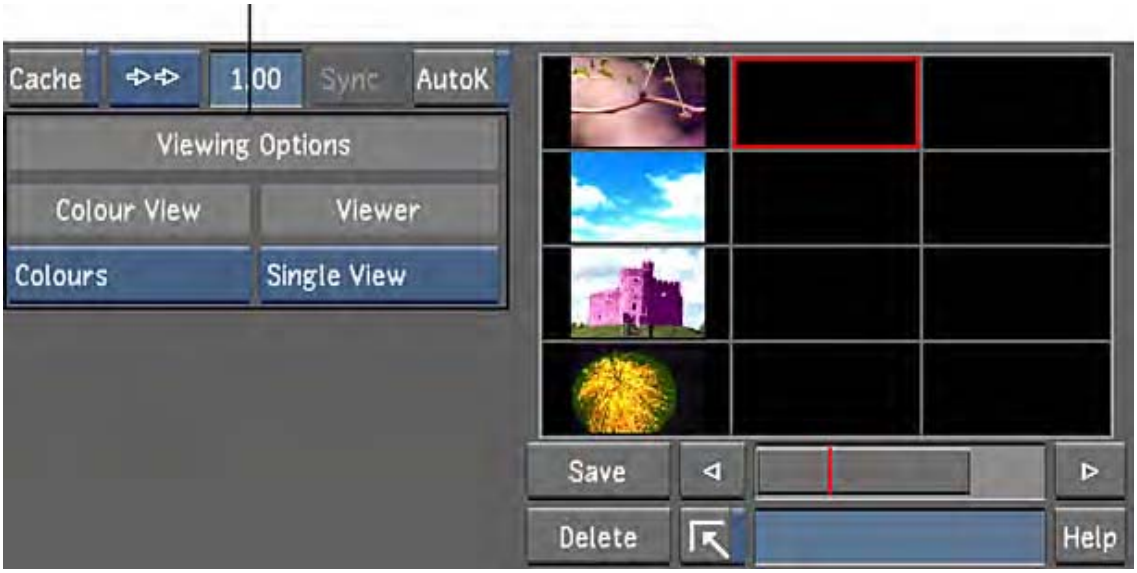
  If you are working in a stereoscopic project, you can also use this mode to display the image for the Left and Right Eye of the current frame at the same time. See Viewing Stereoscopic Footage in Dual View (page 2212) .
- **Multi View** You can display 2, 4, 9, or 16 shots. See Using Multi View (page 1988).

To enable the viewing options:

1 Enable the Viewing Options button. The look of this button depends on the viewing option that is currently selected.



The Viewing Options panel appears to the left of the Grade bin.



2 Select the colour view option.

Cycle through the following options:

- Colours
- Red Channel
- Green Channel
- Blue Channel
- Luminance

3 Select the number of shots to view using the Viewer option box.

Select:	To display:
Single View	The default viewer. The viewer displays the image of the current playhead.
Dual View	The images of Playhead A and Playhead B. See Using Dual View (page 1986). For a stereoscopic project, it displays either Playhead A and Playhead B, or the Left Eye and the Right Eye. See Viewing Stereoscopic Footage in Dual View (page 2212).
Multi View	2, 4, 9, or 16 shots. See Using Multi View (page 1988).

The Viewing Options are also accessible through the Autodesk Control Surface panel. See: Player Viewing Options (page 2437) in "The Autodesk Control Surface" chapter of this User Guide.

To return to the Single view:

1 Do one of the following:
- Press F5.
- Right-click the Viewing Options button.

This returns you to the default Single view.

NOTE Disabling the Viewing Options button does not disable the viewing options you have selected.

Using Dual View

The two images displayed in Dual view are the current frames on Playheads A and B. Dual view is useful for matching two shots or for copying colour correction parameters from one shot to another. You can view images side by side, or use a wipe. With a wipe (i.e., horizontal or vertical), you see a portion of each image. You can control the position of the wipe bar by adjusting the Value slider.

In a stereoscopic project, you can view the image of the Left Eye in one viewer and the image at the same frame of the Right Eye in the other. See Viewing Stereoscopic Footage in Dual View (page 2212).Refer to "Viewing Stereoscopic Footage in Dual View" in the "Stereoscopy" chapter of the *Lustre User Guide*.

To display Dual view:

1 Do one of the following:
- Enable the Viewing Options button, and in the Viewer option box, select Dual View.
- With the Viewing Options button enabled, press F3 or click the Viewer option box to cycle through the viewer options until Dual View is displayed.
- If you are in Single view, and the previous mode was Dual view, press F5 to toggle between the two views.

  The last used mode in Dual view is displayed.

2 Press F4 to cycle through the display modes.
- **2-up** The current frames on Playheads A and B are displayed side by side.



- **Horizontal Wipe** The images assigned to Playheads A and B appear on the top and bottom, respectively.



- **Vertical Wipe**The images assigned to Playheads A and B appear on the left and right, respectively.



TIP When applying a wipe, use the Value slider to edit the percentage of the viewer that displays the image on Playhead B.

NOTE If you are working in a stereoscopic project, you can blend the Right Eye image over the Left Eye. See Viewing Stereoscopic Footage in Dual View (page 2212).

Using Multi View

Multi view is useful for viewing your sequence in a Storyboard format while developing continuity between shots. With Multi view, you see the entire image, and you can cycle through viewing 2, 4, 9, or 16 shots.

To view multiple shots simultaneously in the Player:

1. (Optional) Select up to 16 shots. See Selecting Shots in the Storyboard (page 1980).

   NOTE If you do not select any shots, you will see the first shots in the sequence in Multi view.

2. Do one of the following:
   - Enable the Viewing Options button, and in the Viewer option box, select Multi View.
   - With the Viewing Options button enabled, press `F3` or click the Viewer option box to cycle through the viewer options until Multi View is displayed.
   - If you are in Single view, and the previous mode was Multi view, press `F5` to toggle between the two views.

     The last used Multi view mode is displayed.

3. Press `Ctrl+F5` to cycle through the display modes.



The first two shots displayed are the current frames on Playheads A and B. The rest of the shots displayed are the selected shots, displayed sequentially from left to right in the order of selection.

NOTE If no shots are selected, after the first two shots, shots starting at the third shot in the cut are displayed sequentially from left to right.

4 (Optional) Pan the displayed shots by holding and dragging the middle mouse button within the Player, or scale the displayed shots by holding and dragging the middle and right mouse buttons within the Player.

You can make a new selection of shots while you are in Multi view. After you have selected the new shots, press F6 to update the Multi view display with the new shots.

Viewing Multiple Shots on the SDI Monitor

You can use Dual view and Multi view in the SDI monitor.

To view multiple shots on the SDI monitor:

1 Click Editing in the Main menu, and then click Playout.
The Playout menu is displayed.

2 Make sure GFX/SDI is selected in the Video/Graphics Raster option box.



3 From the Raster list, select the video resolution and scan mode that corresponds with your footage.

4 Press F7 to activate the SDI output mode.

5 Complete one of the following:

- To view two frames on the SDI monitor, perform the procedure for viewing two frames. See Using Dual View (page 1986).
- To view multiple shots on the SDI monitor, perform the procedure for viewing multiple shots. See Using Multi View (page 1988).

NOTE You cannot view horizontal wipes, vertical wipes, or blends on the SDI monitor.

Timeline Sort Mode

Sometimes it may be easier for you to grade a timeline if the shots are in a different order than the final sequence. You can rearrange the order of the shots so common shots are grouped together, graded, and then returned to their original order. The timeline sort mode allows you to do this. You can modify the order of the shots in your assembled EDL and then apply grading to those shots. Once you have finished grading the sorted shots, you can output the shots to a VTR (via the write telecine tape feature), save the shots as a new cut, or return the shots to their original order. You can sort the shots either by the record timecode or by the reel name and source timecode.

By default, all the shots are sorted into a single layer when you use the timeline sort mode (whether you are sorting a single-layered timeline or a multi-layered timeline). You can also choose to sort only the selected shots within the timeline. If you are working with a multi-layered timeline, be aware of layers that are muted or if a layer is soloed. If a layer is soloed, then only the shots in that layer can be sorted. If any layers are muted, the shots within those layers cannot be sorted. As well, the same shot can appear in numerous layers. These shots are sorted from the bottom to top layer.

When the timeline is sorted, only the source media is shown, therefore, you cannot modify or collapse the timeline and dissolves and retimes are not visible. The Edit and Dissolve menus are greyed out and all the options within the Timeline menu and the Colourist Timeline menu are also greyed out. You also cannot add or delete shots while the shots are sorted.

Accessing the Timeline Sort Feature

To sort the shots within your timeline, you can access the Timeline Sort section in the Assemble menu.

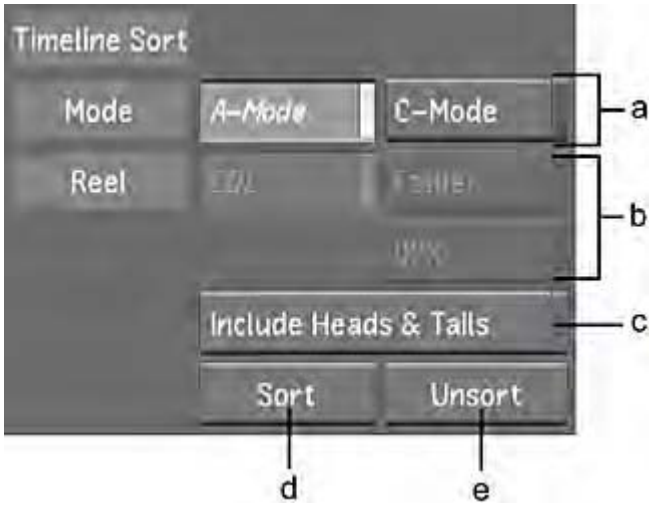
To access the Timeline Sort feature:

1. In the Main menu, click Editing.
2. Click Assemble.
   The Timeline Sort section appears.



(a) Timeline Sort mode buttons

The Timeline Sort section is made up of the following buttons.



(a) Sort mode options (b) Reel options (c) Heads & Tails button (d) Sort button (e) Unsort button

A-Mode button When enabled, this sorts either the entire timeline or selected shots by the order of the shots in the sequence. If you have a multi-layer timeline, the shots are organized by their time location in the sequence. See Sorting with A-Mode (page 1991).

C-Mode button This mode sorts either the entire timeline or the selected shots by the reel name and the source timecode. The Reel options (e.g., EDL, Folder, and digital picture exchange (DPX)) are only available for a C-mode sort. See Sorting with C-Mode (page 1993).

EDL button The EDL option sorts the shots by the EDL reel name (in ascending alphanumerical order) and by source timecode.

Folder button This option sorts the shots according to the folder name (or directory if you are using Linux) your scans or footage are saved into and by source timecode.

DPX button Shots are sorted by DPX reel name and source timecode.

Include Heads & Tails button When enabled, the sorted shots' heads and tails portions are expanded to let you see and work on the entire shot. It also allows you to work on other parameters (e.g., animation, tracking feature, etc.) without disturbing the initial edits (such as dissolves). This option is available for both A-mode and C-mode and is disabled by default.

Sort button Once a sort mode, a reel option (if applicable), and the heads and tails option have been selected, click Sort to organize your shots according to those criteria. The editing tools are disabled and a green border appears around the timeline canvas.

Unsort button Click Unsort to return the timeline back to its original sequence. The editing tools are enabled.

Sorting with A-Mode

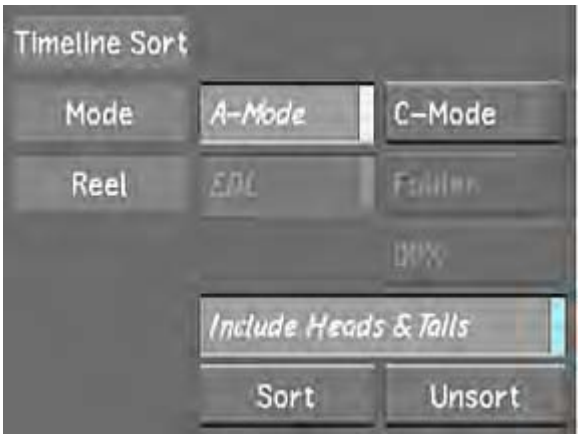
When you use A-mode to sort your timeline, your shots are organized according to their time location within the sequence (i.e., the record timecode). The shots are displayed one after another on a single layer and any dissolves and retimes are not displayed.

To sort the shots using A-mode sort:

1. Click Editing and then click Assemble to display the Timeline Sort section.
2. Select the shots you want to sort. If you want to sort the entire timeline, you do not need to select any shots.



3 Enable the A-Mode button and choose whether you want the head and tail frames to be exposed.



NOTE If any of your shots have a retime value with heads and tails, the Retrim button (in the Edit menu) must be enabled so the sort function can calculate the handles properly.

4 Click Sort.

The shots are now sorted by the record timecode. A green border appears around the timeline canvas to indicate that you are working in a sorted timeline.

NOTE You cannot do any editorial work or delete and add shots while you are in the sorting mode. As well, undo actions are disabled.

Once you have completed grading the shots, you can do one of the following:

- Create a new cut list with the sorted shots. See Creating a New Cut (page 1774).
- Render the sorted shots. See Rendering Shots (page 2216).
- Play out the sorted shots to a VTR. See Playing Out to a VTR (page 2280) and Writing Telecine-Style Tape (page 2290).
- Return the sorted shots to their original order.

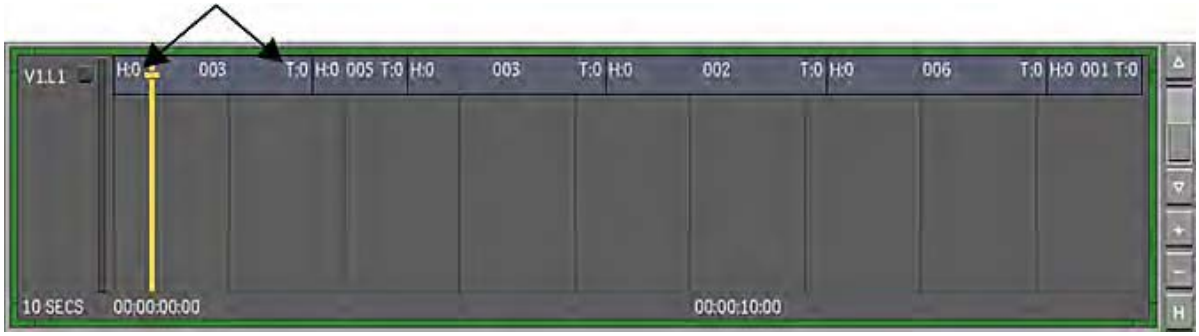
To return the sorted shots to their original order:

1 Do one of the following:

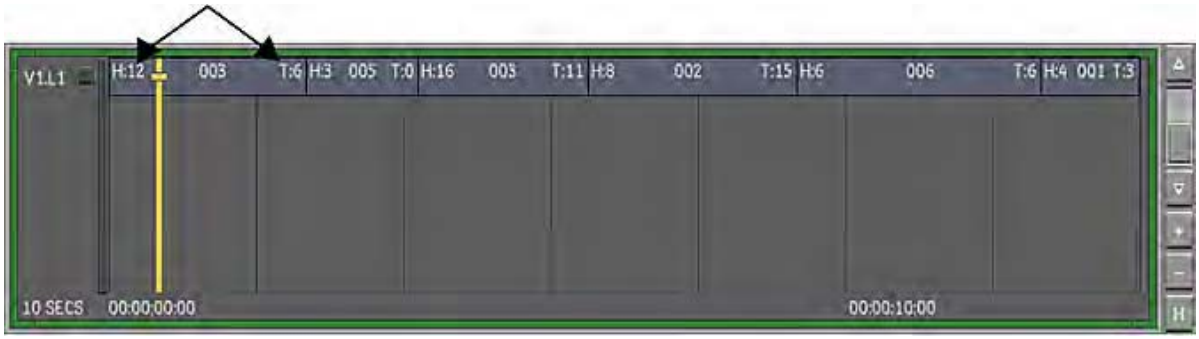
- Click Unsort.
- Press `Alt+F8`.

A-Mode Sort Using Hotkeys

You can also sort the shots, without accessing the Assemble menu, by using hotkeys. Press `Alt+F9` to sort the shots in A-mode and enable the heads and tails. The heads and tails are showing 0 (zero) so you can work on every single frame within the shot.



Press `Alt+F10` to sort the shots in A-mode and for head and tail frames to remain unexposed (disabled).



Sorting with C-Mode

When you sort the timeline with C-mode, you are organizing the shots based on the reel name and the source timecode. If the shot does not have a reel/folder name, then the source timecode is used for sorting. The shots are displayed one after another on a single layer and dissolves and retimes are not displayed. You can choose to have your timeline sorted by EDL reel name, folder name, or DPX reel name.

You want to sort the shots by EDL if you are working with a cut that has been assembled from an EDL, or a cut that is based on a Wiretap EDL. When you sort the timeline using the EDL option, the shots are arranged by the EDL reel name in alphanumerical order (numerical before alphabetical), and then by source timecode in ascending order (e.g., from 00:00:00:00 to 23:59:59:29). This data is read from the cut file EDLReelName XML tag.

If you are sorting the timeline with the Folder option, the shots are arranged in alphanumerical order (numerical before alphabetical) by the folder or directory name where you are storing your scans or captured footage (see Recommended Directory Structure for Projects (page 1734)). You can view the folder name in the file browser. This data is retrieved from the cut file AliasName XML tag. This tag is based on the name of the reel folder or directory when you are working with imported scans or captured footage.

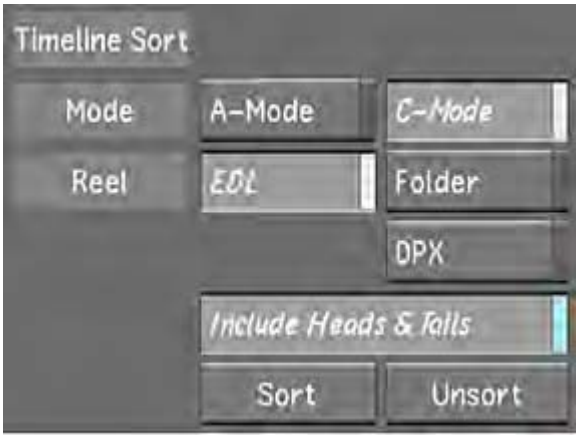
Lastly, you can sort the timeline of DPX files by their reel name. This data is read from the XML tag DPXReelName within the cut file. This XML tag is based on the data in the DPX header of the imported film scans. For more information on XML tags, see XML in Lustre (page 2362).

To sort the shots using C-mode sort:

1. Click Editing and then click Assemble to display the Timeline Sort section.
2. Select the shots you want to sort. If you want to sort the entire timeline, you do not need to select any shots.



3. Enable the C-Mode button, choose a Reel option, and choose whether you want the head and tail frames to be exposed.



NOTE If any of your shots have a retime value with heads and tails, the Retrim button (in the Edit menu) must be enabled so the sort function can calculate the handles properly.

4. Click Sort.

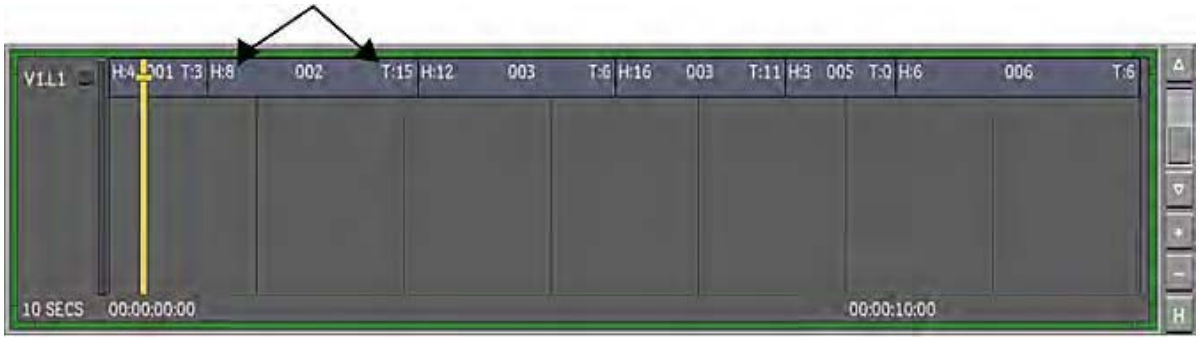
   The shots are now sorted according to your criteria. A green border appears around the timeline canvas to indicate that you are working in a sorted timeline.

   NOTE You cannot do any editorial work or delete and add shots while you are in the sorting mode. As well, undo actions are disabled.

The following image is a C-mode sort using the EDL option with heads and tails enabled. The heads and tails are showing 0 (zero), so you can work on every single frame within the shot.



This image shows the timeline being sorted by the C-mode EDL option and the heads and tails are disabled.



Once you have completed grading the shots, you can do one of the following:

- Create a new cut list with the sorted shots. See Creating a New Cut (page 1774).
- Render the sorted shots. See Rendering Shots (page 2216).
- Play out the sorted shots to a VTR. See Playing Out to a VTR (page 2280) and Writing Telecine-Style Tape (page 2290).
- Return the sorted shots to their original order.

To return the sorted shots to the original EDL:

1. Do one of the following:
   - Click Unsort.
   - Press `Alt+F8`.

C-Mode Sort Using Hotkeys

You can also sort the shots without accessing the Assemble menu. You can sort the shots by using the hotkeys.

Press:	To:
`Alt+F11`	Sort the timeline in C-mode by EDL reel name and heads and tails are enabled.
`Alt+F12`	Sort the timeline in C-mode by EDL reel name and heads and tails are disabled.
`Ctrl+Alt+F11`	Sort the timeline in C-mode by folder name and heads and tails are enabled.
`Ctrl+Alt+F12`	Sort the timeline in C-mode by folder name and heads and tails are disabled.

Press:	To:
Ctrl+Alt+Shift+F11	Sort the timeline in C-mode by DPX reel name and heads and tails are enabled.
Ctrl+Alt+Shift+F12	Sort the timeline in C-mode by DPX reel name and heads and tails are disabled.

Caching Frames to Memory for Real-Time Playback

After applying many effects to a shot, you may no longer get real-time playback. To play shots with many effects in real time, you can render and temporarily cache the rendered frames into memory.

You can also navigate through shots with the control surface.

NOTE You can also obtain real-time playback by first rendering the shot from the Render Local menu, or with background rendering, and then using the P view mode to play the shot. See Rendering (page 2215) and Setting the View Mode (page 1973).

To view shots in real time with memory caching:

1. Go to the frame where you want to start viewing in real time.
2. Specify looped playback of the current shot, the entire Storyboard, or between in and out points by clicking the Play Mode button. See Playing Shots (page 1968).
3. Enable Cache.



   The application renders the current frame and caches the result in memory. It continues forward through the shot or cut (depending on the state of the Play Mode button), caching each frame to memory.
4. Click the Stop playback control at the frame where you want to stop caching. Otherwise, caching continues to the end of the shot or cut and then looped playback starts.
5. If you stopped caching, press the Play control to start looped real-time playback of the cached frames.
6. Use the playback controls to view the cached frames as needed, keeping the Cache button enabled. The cached frames remain in memory until you disable Cache.
7. When you have finished viewing the cut, disable Cache by clicking it again or press N to disable memory caching.

Player Hotkeys

Following is a list of hotkeys and mouse gestures commonly used in the Player and the full-screen Player.

Press:	To:
Enter	Toggle full-screen Player mode.

Press:	To:
Spacebar	Start or stop playback in the direction last played.
Up Arrow	Start forward playback.
Down Arrow	Start backward playback.
Right Ctrl (right hand side of keyboard)	Frame step forward.
Right Alt (right hand side of keyboard)	Frame step backward.
Shift+I	Mark an in point for loop play.
Shift+O	Mark an out point for loop play.
Shift+L	Clear in and out points.
. (decimal point on the numeric keypad)	Toggle through play modes (loop timeline, loop shot, and loop in and out points).
Shift+Spacebar	Switch to P (Print) view mode, open the full-screen Player and start playback.
Middle mouse button and drag	Pan the image.
Middle and right mouse buttons and drag	Zoom into or out of the image.
F1	Display the current frame on Playhead A.
F2	Display the current frame on Playhead B.
F5	Toggle between the current view and Single View.
F3	Cycle between Single View, Dual View and Multi View.
F4	Activate Dual View. In Dual View, cycle between Dual View options.
F6	Update multi-view with the new selection.
F7	Display the Timeline, Player, grading tools, and whichever keyer (HLS or Diamond) is enabled. Also activate the SDI out if the GFX SDI is enabled.
F9	Toggle between full- and half-resolution in the Player.
Tab	Display and hide the Storyboard.
Q	Toggle between regular and large Storyboard view.

Press:	To:
M	Enable memory caching.
N	Disable memory caching.
Alt+1	Display R, G, and B colour channels separately in the waveform monitor.
Alt+2	Display the histogram.
Alt+3	Display R, G, and B colour channels together in the waveform monitor.
Alt+4	Display the vectorscope.

Removing Dust

Use the Dust Removal tools in Lustre to remove dust, hair, speckles, and other artefacts on your shots. Lustre removes artefacts for you by cloning pixels over them. The cloned pixels may come from the identical location on the previous and next frames, or from the area surrounding the artefact on the current frame, depending on the repair method that you use. You can locate artefacts on each frame yourself, or use an automatic method of artefact detection over one or more shots.

Dust removal can be done at any time in the project workflow. Also, you can do more than one dust removal pass on the footage. For example, you may choose to perform the main dust removal early on in the project, render the results, and then do a final pass (quality check) after all colour grading is complete.

You can perform dust removal on the original 2K shots or on 1K generated proxies. Working on the 2K version may provide more accurate removal. However, the process will be completed more quickly when using 1K proxies, particularly in the case of automatic removal.

Dust metadata is saved independently of the colour grading data, in a text file stored with the footage. This makes it possible to remove dust on one system while grading on another. You can then combine the metadata into one completed project. Dust removal is non-destructive to the original footage. If you decide to replace the original footage with cleaned up frames, a copy of the original frame is saved and given a unique name. This file can be restored if needed.

Preparing for Dust Removal

Before removing dust artefacts from your shots, there are several things you should do:

- Identify the best repair type for your footage.
- Determine if you should be using the manual or automated method.
- Optimize the dust removal options for the characteristics of your shots.

Deciding on a Repair Type

To remove artefacts, you can use either the Motion Estimation or Single Frame repair type.

With Motion Estimation, Lustre looks for colour differences between the current frame, the previous frame, and the next frame within a specified area, and differentiates between motion in the shot and artefacts using

motion estimation calculations based on a specified search area. Large search areas are needed for shots with fast motion since the colour differences between frames will span a larger area than would be required for shots with slower motion. If a colour difference is found between adjacent frames, and is not identified as motion in the shot, it is assumed to be dust. To remove an artefact, Lustre creates a mask for it using a calculation that uses the pixel values on the previous and next frames at the identical location as the artefact.

The Single Frame repair type removes artefacts by cloning pixels adjacent to the artefact on the current frame rather than using pixel values from the previous and next frames.

Generally speaking, the Motion Estimation repair type does a better job than Single Frame, and it should be used when possible. Try Single Frame if the characteristics of the shot or the artefacts prevent the Motion Estimation repair type from successfully identifying and removing artefacts. For example, you will need to use Single Repair if an artefact appears on more than one frame.

Deciding on Manual or Automated Dust Removal

There are two methods you can use to remove artefacts—manual or automated. With the manual method, you visually identify artefacts frame by frame and specify the area where you want to apply a mask by drawing boxes around them as you go along. With the automated method, Lustre searches for artefacts and creates the boxes automatically, based on search criteria that you specify.

These methods of dust removal only apply to motion estimation.

The method you choose depends on the nature of your footage—how much dust is on it and the complexity of the motion in it. Automatic removal is faster, but you may experience problems with it if the footage has a lot of fast motion or highlights in the motion or elsewhere in the footage. In this kind of footage, the automatic method may mistakenly identify motion as dust, and the clean up job afterwards may be more effort than using manual removal. As a general rule, choose manual removal for footage with complex motion and little dust (most normal feature work), and automatic removal for footage with less motion and large amounts of dust.

Optimizing Dust Detection

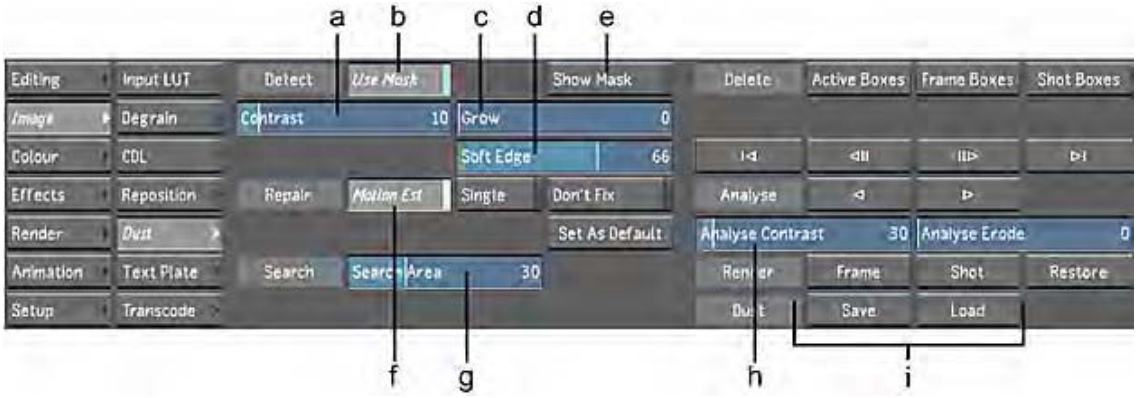
For most shots, when you set dust removal parameters for one dust artefact, the settings will work well for other artefacts in the shot. Prior to removing dust either manually or automatically, set parameters in the Dust menu to the best settings for the amount of motion and the colour composition of your shot. When the settings are optimized, save them as default settings so they will be applied to all newly created mask boxes.

The optimization settings apply only to dust removal methods that are based on motion estimation.

To optimize dust removal settings:

1. In the Main menu, click Image.
2. Click Dust.

   The Dust menu appears.



(a) Contrast field (b) Use Mask button (c) Grow field (d) Soft Edge field (e) Show Mask button (f) Motion Estimation button (g) Search Area field (h) Analyse Contrast slider (i) Dust Save and Load buttons

3 Enable Use Mask.

Lustre will create masks for any artefacts it finds inside dust removal boxes, based on the Contrast and Grow parameters.

4 Enable Motion Estimation.

This repair type uses a motion estimation calculation to identify motion in successive frames.

5 Set the Search Area.

This parameter controls the size of the area (in pixels) around each pixel in which the motion estimation calculation searches for motion. This enables Lustre to differentiate between motion in the shot and dust artefacts. Estimate the approximate search area required for the amount of motion-caused colour displacement between frames. Set a large area for shots with fast motion, and a small area for shots with slower motion.

NOTE If the area set is too small for the speed of the motion, the calculation may mistakenly define colour value differences between frames caused by motion as dust artefacts. However, larger search areas take longer to analyse.

6 Adjust the Analyse Contrast slider to set the amount of contrast used to find dust during automatic analysis.

Low contrast values cause the algorithm to accept low colour value differences between corresponding pixels on the current, previous, and next frames. Therefore, low values result in more complete removal of the artefact. However, if the contrast is set lower than actually needed, adjacent pixels are modified more than actually required to remove the dust.

7 On the shot you are working on, find a frame containing an artefact that is typical of the artefacts on the footage. Use the Next Frame and Previous Frame buttons or `Ctrl` and `Alt` hotkeys to move forward and backward through the shot.

Zoom in closely on a dust artefact by pressing the middle and right mouse buttons and dragging on the image. You should be close enough to see pixels.

8 Draw a box around the dust artefact by dragging the cursor diagonally across the artefact and then releasing the cursor. Surround the artefact closely, but make the box large enough to leave a radius of 5 to 10 pixels around the artefact. See Working with Boxes (page 2004).

The artefact is repaired according to the current settings. It should be fainter or have disappeared completely.

9 Enable Show Mask.

Red pixels identify those areas affected by the removal process.

10 Adjust the Contrast value.

Low values result in more complete removal of the artefact. However, if the contrast is set lower than actually needed, adjacent pixels are modified more than actually required to remove the dust. Increase (or decrease) the Contrast value until right before the dust starts to reappear.

11 Adjust the Grow parameter and the Soft Edge parameter until the red pixels exactly cover the dust particle.

The Grow parameter controls the number of pixels around the pixels detected by Lustre. The Soft Edge parameter softens the stroke of the pixels specified by the Grow parameter in order to replicate a natural paint stroke.

12 Turn off Show Mask.

13 Click Set As Default.

The values you just set for all parameters are now default values that will be applied to any new box created either manually or automatically.

You have now set Dust Removal parameters to their optimum values for the shot and you are now ready to remove dust from the rest of the shot using either the manual or automatic method.

Removing Dust Manually

To remove dust manually, you scan each frame visually for dust artefacts and draw boxes around any artefacts that you find.

To remove dust manually:

1 Follow all steps in Preparing for Dust Removal (page 1998).

2 Go to the first frame of the shot where you want to find and remove dust.

3 On the image window, drag a box around each dust artefact that you find. Make the box as small as possible—to just surround the dust and leave five to ten pixels of space around the artefact. This reduces the portion of the image that is affected by the removal process. See Working with Boxes (page 2004).

4 If needed, tweak the Contrast, Grow, and Soft Edge parameters. Enable or disable Show Mask as needed. Any changes you make are applied only to the artefact in the currently selected box.

5 Go to the next frame and check for artefacts, removing them as needed.

6 Continue the process frame by frame until the shot is cleared of artefacts.

7 If you are unable to successfully remove any artefacts, see Single Frame Dust Removal (page 2003).

Removing Dust Automatically

You can have Lustre automatically remove dust from each frame in the sequence by analysing the frames based on specified parameter values. Once the analysis is complete, you will need to verify that each box created is around a legitimate dust artefact, and you may need to tweak the settings for some boxes.

Analysing the Cut

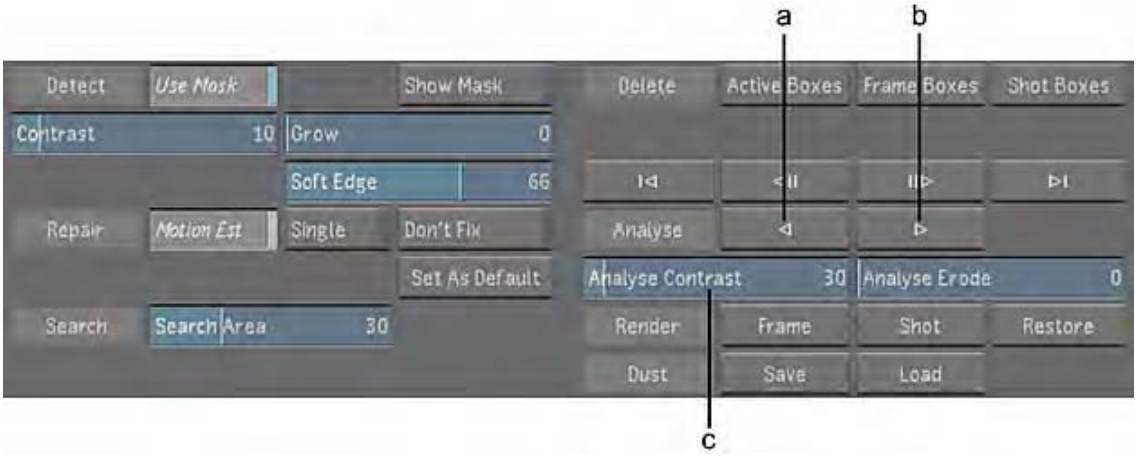
Analysis starts at the current frame and continues from shot to shot until you stop it. You can analyse forward or backward through the cut.

To analyse a cut:

1 Optimize dust removal settings. See Optimizing Dust Detection (page 1999).

2 Go to the frame where you want to start analysing.

3 Adjust the Analyse Contrast slider as needed.



(a) Backward Analyse button (b) Forward Analyse button (c) Analyse Contrast slider

Low contrast values cause the algorithm to accept low colour value differences between corresponding pixels on the current, previous, and next frames. Therefore, low values result in more complete removal of the artefact. However, if the contrast is set lower than actually needed, adjacent pixels are modified more than actually required to remove the dust.

4 Click the Forward Analyse or Backward Analyse button.

Lustre searches each frame for artefacts based on the parameters you set. It places an active box around each one it encounters and removes the artefact.

To stop the analysis:

1 Click anywhere in the grey area of the menu.

Lustre finishes analysing the frame currently being analysed then stops.

Post-Analysis Clean-Up

Verify the analysis and tweak the results if needed. You can delete any boxes that are not surrounding legitimate artefacts. You can also tweak the contrast setting on individual boxes if needed.

TIP If many boxes were created as a result of motion in the shot rather than dust particles, click Shot boxes to delete all boxes, increase the search area, click Set as Default again, and re-analyse the shot. You may need to experiment a bit to find the best value.

To tweak the analysis:

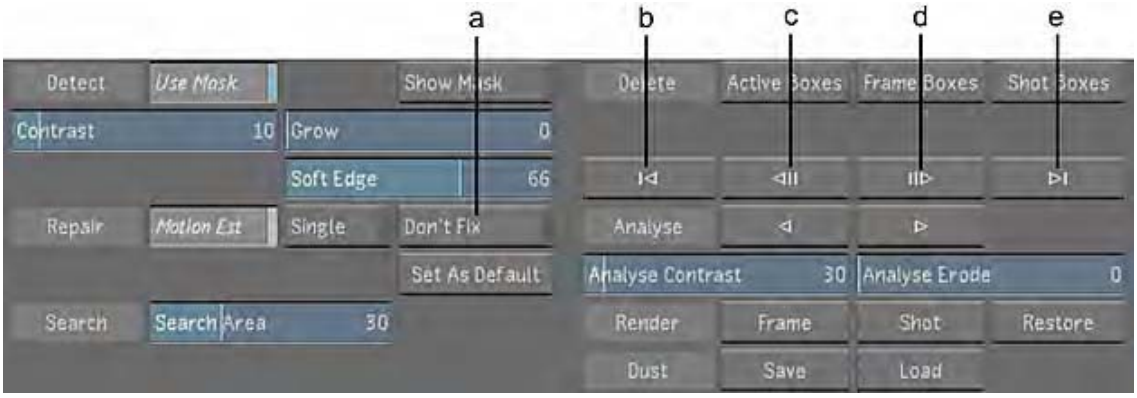
1 Visually scan the first analysed frame.

TIP To jump to the first or last frame in a shot where an artefact has been identified, use the First Analysed Frame button or the Last Analysed Frame button.

2 If you see a box that is not surrounding an artefact, or that you want to tweak, click inside it to select it.

It becomes the only active box on the frame.

TIP Click the Previous and Next Box buttons to move from box to box.



(a) Don't Fix button (b) First Analysed Frame button (c) Previous Box button (d) Next Box button (e) Last Analysed Frame button

3 If you need to see what has been removed, click Don't Fix. This shows you the contents of the box with no removal applied. If the artefact is genuine, click Motion Est again to reapply the Motion Estimation repair type.

4 Do one of the following as needed:

- If the removed object is not an artefact, delete the box by clicking the Active boxes button.
- If the object is an artefact but is not removed properly, tweak the Contrast, Grow and Soft Edge parameters. Enable or disable Show Mask as needed. When Show Mask is on, red pixels identify those areas affected by the removal process. Any changes you make are applied only to the artefact in the currently selected box.

5 If you are still unable to successfully remove the artefact, see Single Frame Dust Removal (page 2003).

Single Frame Dust Removal

In some cases, for example, when a dust artefact occurs in an area of very fast motion, the Motion Estimation repair cannot remove a dust artefact. When this happens, you have two methods you can use to remove the artefact. You can simply instruct the system to repair the entire contents of a box, or you can use a Single Frame repair to clone pixels in the area surrounding the problem.

If neither of these work, you can flag the artefact for later removal with the Can't Fix repair type. When you set a box to Can't Fix, no dust removal process is applied to the particle, and the x and y co-ordinates of the box are saved with the dust removal metadata. The removal of the artefact can then be performed using other methods. For example, a simple script could be written that would list the frames with “Don't Fix” boxes on them, and then send the files to a paint program.

To fix problem dust artefacts:

1 In the Main menu, click Image, and then click Dust to display the Dust menu.

2 Select the box surrounding the problem artefact.

All other active boxes on the frame are deselected.

3 Turn off Use Mask.

If the artefact is still visible, turn on Use Mask again and continue to step 4.

4 Click Single.



(a) Contrast field (b) Single button (c) Grow field

The box border changes from blue to green, indicating the applied Single Frame repair type.

5 Set the Contrast, Grow, and Soft Edge parameters to the optimum values. Use Show Mask as needed.

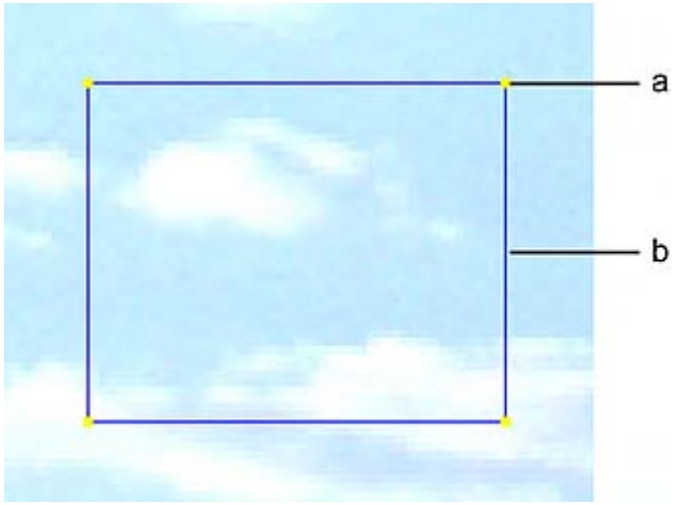
6 (Optional) If the artefact is still visible, set the repair type to Don't Fix.

The box border changes to red, indicating the applied Don't Fix repair type.

TIP To navigate between boxes set to Don't Fix, right-click the Previous and Next Box buttons.

Working with Boxes

Whether you choose the manual or automatic method of identifying dust artefacts, you will probably need to adjust the position or size of some boxes, or delete them. Following are basic procedures for working with boxes.



(a) Vertex (b) Border

To create a box:

1 Hold down the left mouse button and drag the cursor diagonally on the image.

2 Release the mouse button.

To select a box:

1 Click inside the box.

The box vertices turn yellow, indicating the box is active (selected). You can only select one box at a time.

To de-select a box:

1 Right-click inside the box.

To move a box:

1 Click inside the box and drag it.

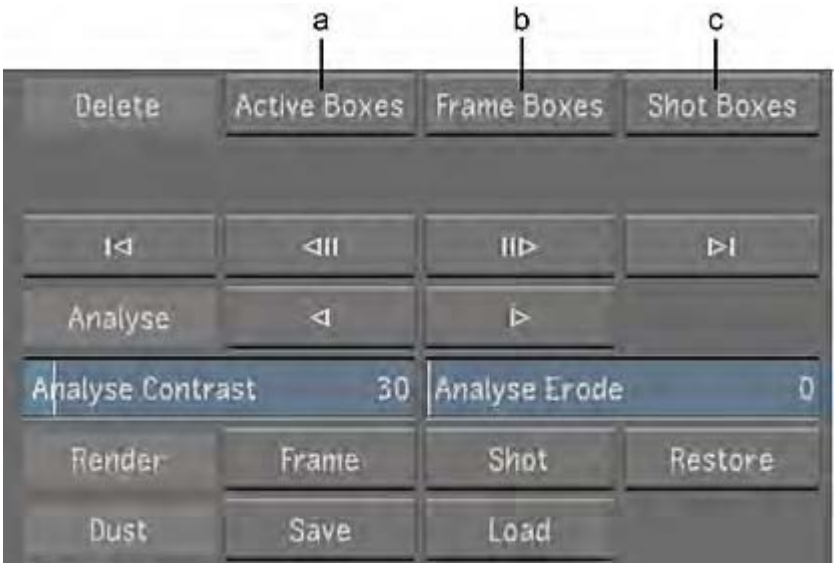
To re-size a box:

1 Click on a vertex and drag.

To delete one or more boxes:

1 Do one of the following.

To delete:	Do this:
The single active box	Select the box and press `Backspace` or click the Active boxes button.
All boxes on the current frame, whether active or not	Click the Frame boxes button.
All boxes on all frames in the current shot	Click the Shot boxes button.



(a) Active boxes button (b) Frame boxes button (c) Shot boxes button

Infrared Channel Dust Detection

The infrared (IR) channel is available in DPX and Cineon 10-bit files that contain the SGI byte order. This IR channel is collected during the film scanning process at the same time as the visible colour channels (red, green, and blue). Whereas film is mostly transparent to infrared radiation, dust and scratches are not and

therefore appear in the IR channel. Once imported into Lustre, the information contained in the IR channel can be used to detect the appearance of dust and scratches to a higher degree than is possible using standard dust detection techniques.

NOTE IR dust detection is only available for the Master Station and the Lustre Station.

To perform IR dust detection:

1 With the clip loaded on the timeline, click the Analyse forward button while holding down `Ctrl`.

The IR dust analysis begins, and boxes are placed around detected problems.

2 To reduce the amount of boxes that appear, increase the value in the Analyse Erode slider.

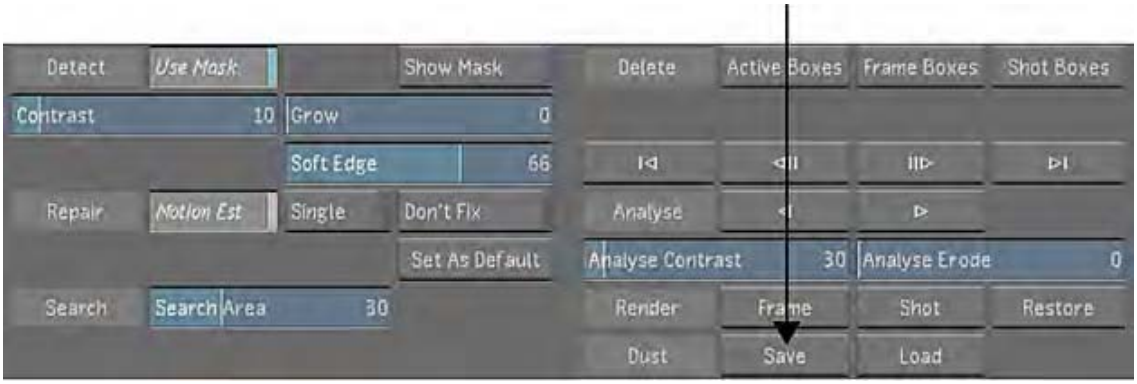
Options for Saving and Loading Dust Removal Metadata

You can save dust removal metadata manually, or automatically when you save the grade. In either case, the metadata is not saved in the grade file itself, but rather as a separate file called *dustbust.data*, located in the same directory as the original footage. See Loading Grades for a Cut (page 1778).

When you save dust removal metadata manually, only the dust removal metadata is saved. If your timeline is long and has complex grading applied to it, saving dust metadata separately can save you time. You can also improve system performance by configuring Lustre to load grades without the corresponding dust removal metadata. If required, you can load the dust removal metadata later.

To save dust removal metadata manually:

1 In the Dust menu, click Save.



To ensure that dust removal metadata is not automatically loaded with the grade:

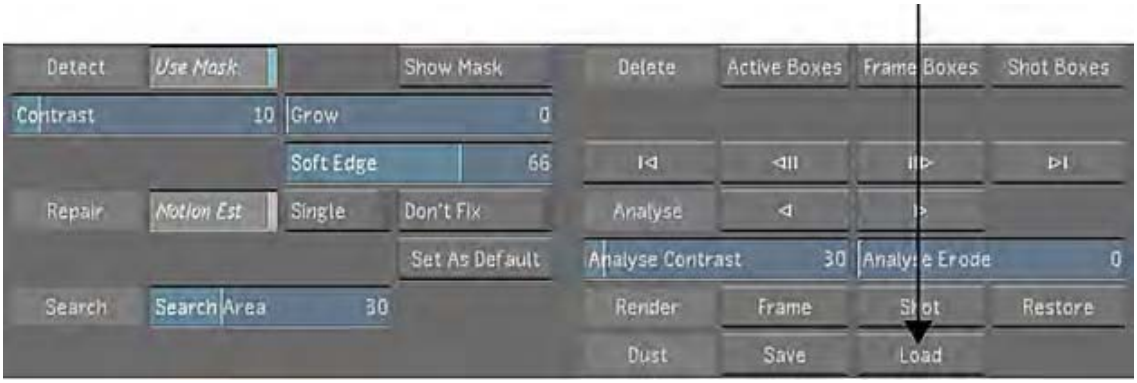
1 In the Setup > Settings menu, select Edit for the user settings.

2 In the Tools page, disable the Load Dust Data button.

Disabling this button ensures that dust removal metadata is not automatically loaded when you load the grade. This can save you time if you need to load a grade and do not mind seeing artefacts on your footage.

To load dust removal metadata after you have already loaded the grade:

1 In the Dust menu, click Load.



Rendering the Result

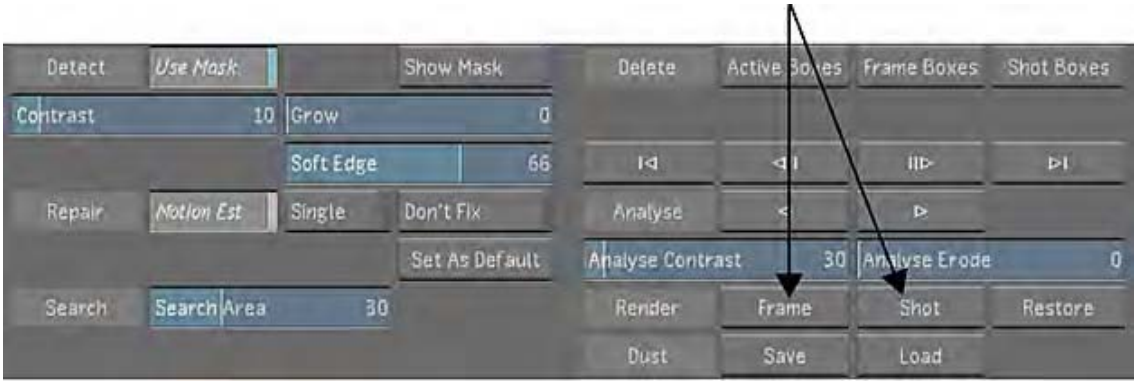
When you are satisfied with the dust removal results, you can render your material. Two rendering options are available depending on whether you render using the Render menu or the Dust menu:

- When you render from the Render menu, you have the option of turning dust metadata on or off. When dust metadata is on, you render dust removal changes made to the grade. Otherwise, dust removal is not included in the render files. Render files are created for the entire region selected for render and saved to a separate directory. See Rendering Shots (page 2216) and Rendering Dust Metadata (page 2223).
- When you render from the Dust menu, the render files replace the original files, and the originals are backed up to a sub-directory called *Original*. You can therefore work with dust-free frames while continuing to point to the original footage directories during the course of the project. You can render the current frame or the shot. Only those frames containing changes are rendered. If you change your mind and want to revert to the original shot, you can restore it.

  NOTE When you render from the Dust menu, the rendered result is visible from all menus except the Dust menu. In this menu, boxes and their settings remain in place, giving you the option to tweak them further and re-render if needed.

To render from the Dust menu:

1. Navigate to the shot containing frames you want to render.
2. Do one of the following:
   - To render the current frame, click Frame.
   - To render all frames in the shot that have been modified, click Shot.
   - To render the entire timeline, `Shift`-click the Shot button.



The frames are rendered.

To restore the original shot:

1. Navigate to the shot.
2. Click Restore and confirm by clicking it a second time.
   The entire rendered shot is replaced with the original.

   NOTE The boxes are retained. Therefore, if you want to start from scratch, you must delete the boxes first.

Removing Grain

Use the Degrain tool in Lustre to remove film grain, video noise, and other compression noise from your shots while minimizing damage to the image. Grain removal can be done at any time during the project workflow, but it is typical to perform an initial degrain operation on the source image before primary colour grading begins.

A recommended grain removal workflow consists of the following basic steps. You can view the result as you work. See Viewing the Result (page 2009).

Step:	Refer to:
1. Define the degrain cache location.	Project Settings (page 1738) and Rendering Settings (page 1743).
2. Analyse the grain profile of the input image.	Analysing the Frame (page 2008).
3. Perform post-analysis cleanup.	Post-Analysis Clean-Up (page 2011).
4. Cache the filtered images.	Caching the Result (page 2013).
5. Apply input and secondary grading.	Primary Colour Grading (page 2058) and Secondary Colour Grading (page 2083).
6. (Optional) Analyse the grain profile of secondaries to de-grain isolated parts of the image.	Removing Grain with Secondaries (page 2015).
7. Render the result.	Rendering (page 2215).

Analysing the Frame

Before processing the grain filter, the image must be analysed. Use the Grain Analysis panel to perform an analysis on a selected frame. The grain profile for the shot is recalculated each time an analysis is performed. If you navigate to a different frame in the shot and analyze it, the grain profile will be based on the new data.



NOTE It is recommended that you perform an analysis on the entire image, then analyse multiple regions of the image, if required.

To analyse the grain profile of an image:

1. Navigate to the frame you want to analyse.
2. Do one of the following:
   - If you want to degrain the entire image, click Degrain in the Image menu.
   - If you want to degrain a region of interest, click Degrain in the Colour menu.
3. (Optional) Enable the Sub-Region Analysis button and define an area to analyse. For best results, define multiple regions of uniform colour with no features. See Working with Sub-Region Boxes (page 2010).
4. Click the Analyse button to determine the grain structure.

Viewing the Result

After the frame is analysed, you can view the result of applying the degrain filter on all or a portion (sub-region) of the image. For increased performance, it is recommended that you select the sub-region where you want to view the filtered result.



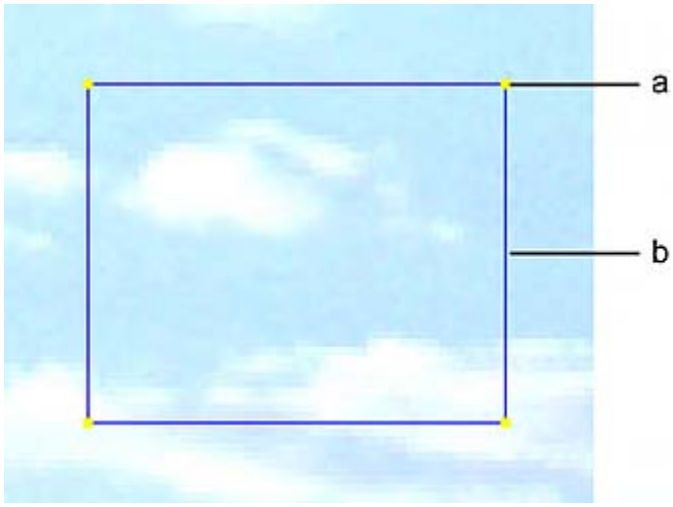
To apply the degrain filter:

1. Navigate to the frame you want to view.
2. Do one of the following:
   - If you want to degrain the entire image, click Degrain in the Image menu.
   - If you want to degrain a region of interest, click Degrain in the Colour menu.
3. (Optional) Enable the Sub-Region Filtering button and drag to define an area to view with the result. See Working with Sub-Region Boxes (page 2010).

   NOTE Unlike sub-region analysis boxes, the sub-region filtering box cannot be deleted, only disabled.
4. Enable the Apply Degrain button to view the filter on the image.

Working with Sub-Region Boxes

To analyse specific regions of the image's grain structure, you need to use sub-region analysis boxes. If you are not satisfied with an analysis of the entire image, you can use sub-regions to constrain the analysis to focus on regions of interest. You will probably need to adjust the position or size of some boxes, or delete them. Following are basic procedures for working with boxes.



(a) Vertex (b) Border

To create a box:

1. Enable Sub-Region Analysis.
2. Hold down the left mouse button and drag the cursor diagonally on the image.
3. Release the mouse button.

To select a box:

1. Do one of the following:
   - Click inside the box.

     The current box is active and all other sub-region analysis boxes are inactive.
   - Right-click inside the box to change its state, without affecting the state of other sub-region analysis boxes.

   The vertices of active boxes are yellow (selected). The vertices of inactive boxes are red (disabled).

To move a box:

1. Click inside the box and drag it.

To re-size a box:

1. Click on a vertex and drag.

To display or hide boxes:

1. Press Z.

To delete one or more boxes:

NOTE The sub-region filtering box cannot be deleted, only disabled.

1 Do one of the following:
- Press `Backspace` to delete all active boxes.
- Using the Delete Boxes panel, do one of the following.

Click:	To delete:
Active	The currently selected box (highlighted in yellow).
Frame	All boxes on the current frame, whether active or not.
Shot	All boxes on all frames in the current shot.

Locating Sub-Region Analysis Boxes Within a Shot

Use the navigation buttons in the Reference panel to easily locate and navigate between shots that use sub-region boxes for degrain analysis.

To navigate between sub-region analysis boxes:

1 Click a button in the Reference panel.

Click:	To display:
	The first frame in the shot that has a sub-region box.
	The previous frame in the shot that has a sub-region box.
	The reference frame on which the most recent analysis was performed.
	The next frame in the shot that has a sub-region box.
	The last frame in the shot that has a sub-region box.

Post-Analysis Clean-Up

Once you have analysed the grain profile, you can make adjustments to the profile. You can adjust spatial degrain parameters, or apply a temporal degrain that can include previous or subsequent frames in the shot. A curve is displayed for each colour channel. You can manually adjust a response curve for the gain of each colour channel in the Curve Editor.

Adjusting Spatial Degrain Settings

You can fine-tune grain removal by changing the spatial degrain parameters in the Degrain Settings panel.



The Degrain Settings menu is made up of the following elements.

Red slider Sets the gain for red channel colour values.

Green slider Sets the gain for green channel colour values.

Blue slider Sets the gain for blue channel colour values. For film scans, the grain is often greater in this channel.

Link button Click to change gain values proportionally for all three colour channels.

Grain Size slider Sets a value in pixels that is proportional to the size of the grain. The default value is 3, but may be higher for 4K images.

Smoothing Radius slider Sets the blur radius. For smoother results, a higher value will add more pixels to the blur, but increase processing time.

Detail slider Sets the amount of detail to preserve when the Smoothing Radius is set to a high value. It is recommended you enter 0.05 to 0.15 as an initial value.

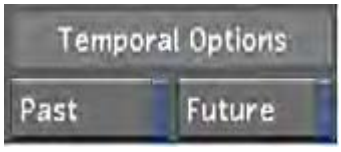
Opacity slider Sets the level of opacity between the source image and the output with the applied grain filter.

Applying Temporal Degrain Settings

If you degrain the source image (using the Degrain menu in the Input menu), you can include temporal degrain operations to the grain filter, by comparing the pixel data in past or future frames. These options yield best results when frames include many static or slowly moving objects in the image. Temporal options will improve grain removal results, but it will also increase processing time.

NOTE When temporal options are used on cached images, the result must be re-cached.

NOTE Incinerator does not process temporal degrain. Render to view these results.



To use temporal degrain:

1 In the Temporal Options panel, enable one or both states.

Enable:	To include pixel data from:
Past	Previous frames in temporal processing.
Future	Subsequent frames in temporal processing.

Adjusting the Response Curves

Response curves represent the noise intensity for each colour channel. Modify the red, green, or blue response curves to fine-tune the degrain. You can add vertices to a curve to refine your adjustments.

To modify the red, green, or blue curve:

1 To display the response curve, click Show Curves.

2 Show the response curve you want to modify by clicking the button corresponding to the colour of the curve.

The selected curve is highlighted.

3 Sample the colour you want to use as a reference for your adjustments.

Red, green, and blue vertical lines representing the colour channels are plotted on the curve.

4 Do one of the following:

- To add a vertex to the curve, place the mouse cursor over the area on the curve where you want to add the vertex and press `A`.
- To lock the curve in place while adding a vertex, press `Shift+A` on the curve.

5 Click a vertex to display its tangent handle and then drag the handle to adjust the curve. You can also adjust the curve by dragging the vertex.

6 Modify the curve until you are satisfied with the results:

- To select a vertex, click it. To select several vertices, draw a selection box around them.
- To move one or several selected vertices, drag a selected vertex. To restrict vertex movement to the Y axis, press `Shift` while moving the mouse. To restrict vertex movement to the X axis, press `Shift+Alt` while moving the mouse.
- To delete vertices, select them and then press `D`.

Caching the Result

Degrain operations use caching of input images to manage the intensive processing requirements imposed on Lustre. If the Apply Degrain button is enabled, a degrained shot is automatically cached when the shot is initially played back after a Degrain operation. You can also use the Slave Renderer and Burn to cache frames. See Caching Frames Remotely (page 2014).

The Degrain menu does not have to be visible for Lustre to continue caching frames.

You can cache the filtered frames created using the CPU. Once cached, Lustre can recall cached frames during future playback in both CPU processing and GPU acceleration mode. See GPU Acceleration (page 2043). (If any parameters are changed, GPU acceleration must be disabled and images must be reanalysed in CPU mode to update and re-cache the frames.)

When configuring your project in Lustre, you also define the location of cached frames using the project settings. Degrained frames can be cached in a sub-directory of the Scans folder or a user-defined degrain cache file location. See Project Settings (page 1738) and Rendering Settings (page 1743).

Only one instance of a frame can be cached in a shot, even if it is used in multiple cuts. If the cached media is saved as a preset, then you can load the cached media into the shot in a different cut. See Saving and Loading Degrain Presets (page 2016).

Caching Frames from Wiretap

You can cache frames from media imported from Wiretap and Wiretap Gateway. Cached frames are saved in the user-defined location set in the Degrain Cache field found in Project settings. Media is saved in this location even if, in the Rendering menu, the Degrain File Location option box is set to Save with Scan.

Media that is soft-imported from Wiretap use source files that can be accessed using a file path structure, rather than a Wiretap or Wiretap Gateway address. As a result, soft-imported frames can be saved in Scans Home or the user-defined degrain cache location.

Caching Frames Remotely

You can use a Slave Rendering machine to cache frames in the background as you work, or Burn to submit a caching job.

NOTE Degrained frames cannot be cached using Incinerator.

If you want to cache remotely to a user-defined location, you must enter a BrowseD file path in the Degrain Cache field. See Project Settings (page 1738). For example, in Linux, a valid BrowseD file path can have the following structure: *<IP address>:/mnt/storage/<project_name>*. In Windows, it can be *<IP address>:\F:\storage\<project_name>*.

In Linux, to cache remotely using absolute file paths for a user-defined degrain cache location, you must create a shared NFS mount point on the Slave Renderer or Burn. In Windows, you use a UNC path for the Windows Slave Rendering machine, or use a Samba path to create a mount point on a Burn node.

To perform background caching using the Slave Renderer:

1. Display the SlaveRender menu. See Rendering Shots as You Work (page 2231).
2. In the SlaveRender panel, toggle the On/Off button to On.
3. In the Degrain Only panel, toggle the On/Off button to On.

   When the Apply Degrain button is enabled in the Degrain menu, degrained frames are cached after you navigate to another shot.

NOTE To render degrained frames using the Slave Renderer, set the Degrain Only panel On/Off button to Off. Graded frames with Degrain will be rendered in the Renders Full Home destination.

To cache using a Burn node:

1. Display the Backburner menu. See Submitting a Remote Render Job (page 2234).
2. Enable the Degrain Only button.
3. Click Burn.

   When the Apply Degrain button is enabled in the Degrain menu, the degrained frames are cached.

NOTE To render degrained frames using a Burn node, disable the Degrain Only button. Graded frames with Degrain will be rendered in the Renders Full Home destination.

Clearing the Cache

You can clear the cached frames for the current shot or clear the *degrain_cache* folder for the entire cut. The files that will be removed are dependent upon where you choose to save your files.

When degrain cache frames are saved in the Scans directory, the cache for the current shot in the Scans directory can be cleared. When degrain cache frames are saved in a user-defined location, all files stored at that location will be removed.

To clear the cache for the current shot:

NOTE This operation only applies when degrain cache files are saved in the Scans directory.

1. In the Rendering menu of the project configuration settings, set the Degrain Cache File Location option box to Saved with Scan.
2. In the Image menu, click Degrain.
3. Navigate to the shot with the frames you want to clear.
4. Click Clear Cache.

To clear the cache for the project:

NOTE This operation only applies when degrain cache files are saved in a user-defined location.

WARNING Lustre will delete all files stored in the user-defined location. This includes all degrained frames stored in this location for the entire project, even if it includes files from a different scene. It is strongly recommended that you store cached frames for different projects separately. See Configuring File Locations in Project Home (page 1735).

1. In the Rendering menu of the project configuration settings, set the Degrain Cache File Location option box to Saved with Degrain.
2. Do one of the following:
   - In the Image menu, click Degrain.
   - In the Project settings menu, click Rendering.
3. Click Clear Cache.

Removing Grain with Secondaries

After you have applied degrain operations to the input image, you can isolate parts of the image to degrain using secondaries. Degrain operations are cumulative: after initial noise removal, multiple passes using secondaries on the same area will be added to the final result.

When applying a degrain filter to a secondary layer, you cannot use temporal options and cache degrained frames.

To degrain a secondary layer:

1. In the Colour menu, click Degrain.
2. Select a secondary layer. See Secondary Colour Grading (page 2083).

3 (Optional) Add a geometry to constrain the area of degrain analysis and filtering.

4 Click Analyse.

5 To view results, click Apply Degrain.

Copying Degrain Information

You can copy all or a selection of the Degrain menu parameters from a shot using the Selector. These parameters include the option to have degrain enabled or disabled on the destination shots. See Copying Parameters with the Selector (page 1816).

Saving and Loading Degrain Presets

Once you create a grain profile, you can save it as a preset and reload it for later use or in other shots. A preset saved for an input image can be loaded on a secondary, and vice versa.

See Saving and Loading Presets Using the Presets Lists (page 2051).

Repositioning Images

Use the Reposition tools to reframe your shots. Generally, when you reframe a shot, you are adjusting the viewable area of your frames within a specified smaller area.

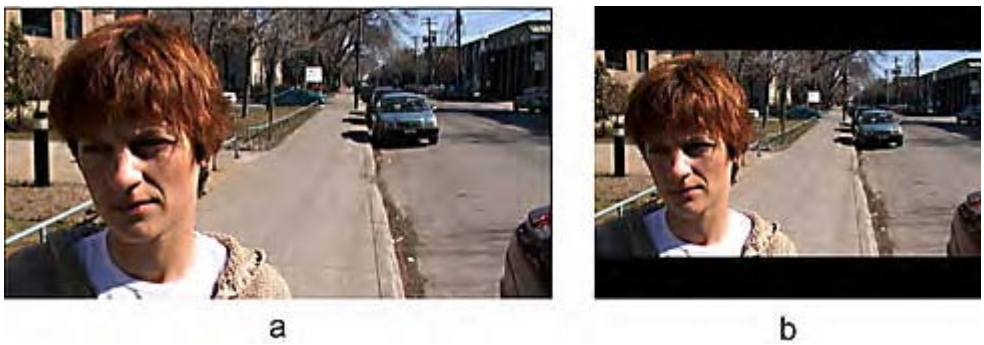
In a stereoscopic project, you may want to reposition images to realign shots for the Left Eye and Right Eye so that they converge correctly. See Repositioning the Left Eye and Right Eye (page 2210).

NOTE Support for repositioning depends on the version of the graphics card installed on your system. See your release notes for details.

For example, if you are shooting in a widescreen film format and you intend to project the finished film on a television, you have to make certain decisions about how to deliver the content. You can scale and letterbox the film frames, or you can reposition the images in such a way that the important subjects of the scene are visible on screen (at the expense of “unimportant” elements).

If you shot your film in 35mm, but you want to show it in a widescreen format, you will need to crop the frames and adjust the racking—or the vertical positioning—of the image to modify the image's composition.

With letterbox images, you are scaling down a widescreen frame to fit horizontally on a television screen. This process adds black bars across the top and bottom of the screen and reduces the size of the images. However, letterboxing has the advantage of showing the entire image, thus remaining true to the vision of the artists.



(a) Original widescreen frame (b) Letterboxed frame

When you reposition frames, the actual source images are larger than the viewable area. To compensate, you make adjustments to the pan and scan of the frames to emphasize the key elements of a scene. For example, if the original frames show two people facing each other and talking, you may pan back and forth to see the person who is talking. The other person would then be partially or even completely off screen.



(a) Original widescreen frame (b) Cropped and repositioned frame (c) Final frame

Use racking when you want to deliver your footage in a wider format than it was shot in. For example, when shooting on 35mm film, you may compose the scene so that the important elements of the image are contained in a narrow horizontal portion of the frame. In such a case, you set a cropping gate, and then adjust the vertical position (scan) of the shot in order to compose the scene as needed.



(a) Original 35mm with racking on (b) Cropped frame

Use the Reposition tools to animate the position, rotation, scaling, and aspect ratio of your shots. Do this, for example, to simulate camera movement, which is useful when you want to make a still shot more dynamic.

You can also use racking and repositioning in conjunction with tracking to stabilize shots with too much motion.

Finally, you can modify shot composition by flipping the shots vertically or flopping them horizontally.

TIP For real-time playback of repositioned shots, enable GPU processing by clicking the GPU button. See GPU Acceleration (page 2043).

Adjusting the Frame Aspect Ratio

Before you begin repositioning your shots, make sure you have set the aspect ratio of the output frame.

You adjust the aspect ratio when your source material was shot in a format that differs from the intended delivery format, for example, if you shot in 35mm film (1.33 aspect ratio), but will deliver in a widescreen-format aspect ratio of 1.85. Or, you may have shot in a widescreen format at a 2.35 aspect ratio but intend to output a format for television (1.33 aspect ratio).

For information on adjusting the aspect ratio for a cut, see Setting the Aspect Ratio (page 1788).

Accessing the Reposition Menu

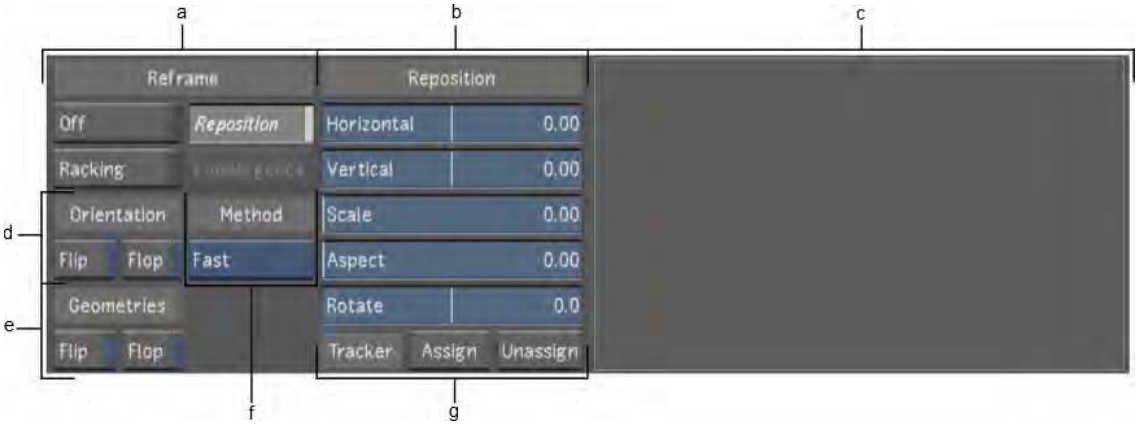
Once you select the shot you want to adjust or modify, you can access the Reposition menu. This menu provides the commands you need to reframe an image by:

- Repositioning
- Racking
- Changing the orientation
- Tracking a repositioned or racked shot

To access the Reposition menu:

1. Make sure you have set the aspect ratio to match the intended output format. See Setting the Aspect Ratio (page 1788).
2. Select the shot to adjust.
3. Click Image, and then click the Reposition menu button.

   The Reposition menu appears.



(a) Reframe panel (b) Reposition sliders (c) Preview window (d) Orientation panel (e) Geometries panel (f) Rendering Method box (g)Tracker Method panel

Reframe panel Enables a reframing tool to reposition, rack or converge an image. See Reframing an Image (page 2019).

Reposition sliders Adjust the position of the shot by changing its vertical or horizontal position, rescaling, changing its aspect ratio, rotating or a combination of these repositioning options. See Repositioning an Image (page 2020).

Preview window Displays the the relationship of the visible area (red rectangle) to the original image (blue rectangle). By default, the red reference rectangle is hidden by the blue one: they are equal in orientation and scale.

Orientation panel Displays the tools to apply a horizontal or vertical translation to the image. See Flipping or Flopping a Shot (page 2021).

Geometries panel Displays the tools to apply a horizontal or vertical translation to the geometries associated to a secondary, independently of the orientation of the image. See Flipping or Flopping a Shot (page 2021).

Rendering Method box Contains the rendering quality levels of the resulting repositioned image. See Setting Rendering Quality (page 2025).

Tracker panel Controls whether point tracking is enabled or disabled for a repositions image. See Tracking Repositioned Shots (page 2022).

Reframing an Image

Depending on how the original film was shot, you may have to reframe your footage at some point to account for the final aspect ratio of the delivery format. It may also be necessary to use the reframe functions to output film frames to common SD formats or DVD.

To adjust the reframing of an image, you can:

- Use racking to adjust vertical position only.
- Use repositioning to zoom and adjust the position.
- In a stereoscopic project, use convergence to reposition a shot in the assigned eye and apply the equal and opposite repositioning operation to the other eye. See Repositioning the Left Eye and Right Eye (page 2210).

Racking an Image

Racking involves adjusting the vertical position of a shot. You need to rack your images after you adjust the aspect ratio. By racking, you ensure that the important element of the shot composition is framed correctly in the final images.

To adjust the racking of an image:

1. Display the Reposition menu. See Accessing the Reposition Menu (page 2018).
2. Enable Racking.



3. Use the Vertical slider to adjust the vertical position of the image.

4 You can animate the vertical position using the Animation controls. See Animating Reposition Values (page 2024).

Repositioning an Image

Repositioning involves scaling, rotating, and horizontally and vertically moving images to ensure that the correct elements are visible in the frame. You need to do this after adjusting the project aspect ratio. If you do not, key elements of the shot composition may be cropped or even completely off screen.

You can zoom into the image and offset the position in the horizontal and vertical directions.

To reposition an image:

1 Display the Reposition menu. See Accessing the Reposition Menu (page 2018).

2 Enable Reposition.



3 In the Reposition panel, use the sliders to adjust the image scale and positioning as needed.

4 You can animate Reposition parameters using the Animation controls. See Animating Reposition Values (page 2024).

Turning Off Reframing

You can remove Reframing settings from your shots at any time.

To turn off reframing:

1 Display the Reposition menu. See Accessing the Reposition Menu (page 2018).

2 Enable Off.



Flipping or Flopping a Shot

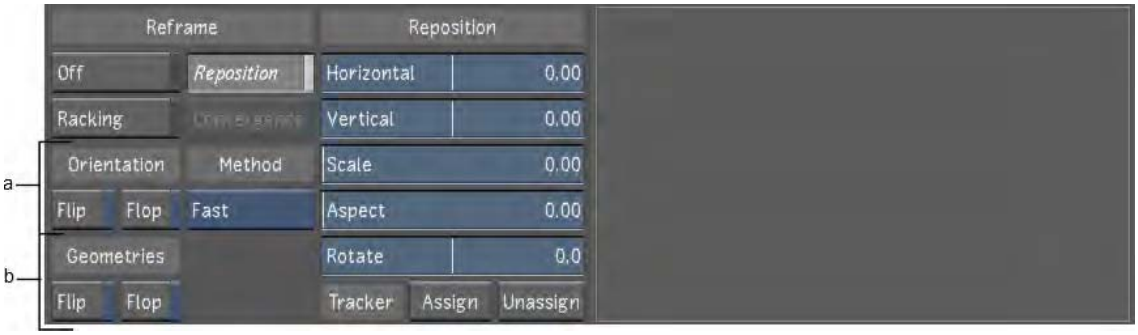
Use the Flip or Flop tool to change your shot's orientation.



Original Image | Flipped image | Flopped image

To change the orientation of a shot:

1. Display the Reposition menu. See Accessing the Reposition Menu (page 2018).
2. Enable one or both orientation tools.



(a) Orientation tools (b) Geometries tools

Enable:	To:
Flip	Apply a horizontal translation to the image.
Flop	Apply a vertical translation to the image.

If you have geometries associated to secondaries, you can apply a horizontal or vertical translation to the geometries, independently of the image's orientation, using the Geometries tools. By default, the geometries will follow the image's orientation.

Enable:	To:
Flip	Apply a horizontal translation to the geometries.
Flop	Apply a vertical translation to the geometries.

NOTE When applying a flip or a flop to Stereo3D content from inverted camera rigs, it is important to also enable to corresponding Geometries Flip / Flop to avoid issue when drawing shapes in Sync mode. Flipped or flopped Stereo3D content should always have the same Geometry orientation, regardless of whether or not there are geometries in the shot to avoid issues if you add geometries to the shot later.

Tracking Repositioned Shots

Use the Point Tracker to stabilize a repositioned shot so that a selected point or shape in the source frame remains fixed in the same spot in the Player. You can use the Point Tracker in conjunction with reframing to stabilize jittery shots, or to automate the repositioning process. For example, if the original widescreen shot contains a pan across a landscape, but you want to fix the view on a house (for television viewing), you track the house, and then assign the tracking data to the Reposition tool. This sets pan and scan values at each frame. The result is that the house remains in the same spot in the repositioned image.

NOTE You can use exported stabilizer data from other Autodesk applications to stabilize your shots. See Loading Tracker and Stabilizer Data from Autodesk Applications (page 2154).

This section is intended to provide an overview of the stabilizing process. See Animating the Point Tracker (page 2144).

Using the Point Tracker

You can use the Point Tracker to stabilize a repositioned shot.

To assign tracking information to the pan and scan settings of an image:

1. Make sure the aspect ratio value is set up as needed. See Setting the Aspect Ratio (page 1788).
2. Select the shot to track.
3. Click Colour, and then click Secondaries to display the Secondaries menu.
4. Select the tracking feature by clicking Tracker.
5. Select the Point Tracker by clicking Point.
6. Enable a new tracker by clicking New.



(a) Tracker button (b) Point button (c) New button

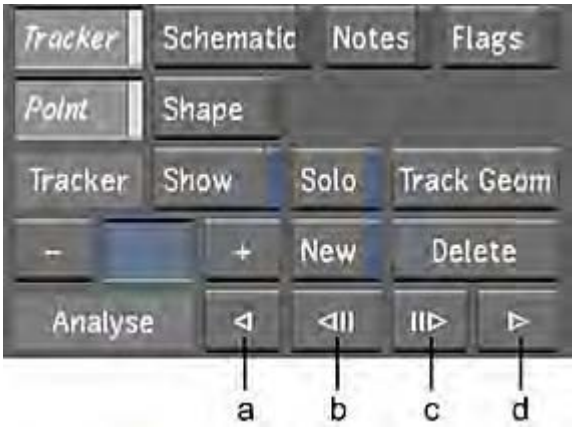
7 Click the tracking point in the image.

8 Click Image, and then click Reposition to display the Reposition menu.

A tracking box appears. You can adjust the box as needed to improve the track.

9 Click one of the Analyse buttons to begin the tracking process.

The system tracks the tracking point. After tracking is complete, you may need to make slight adjustments to the tracking data. See Animating the Point Tracker (page 2144).



(a) Analyse backwards (b) Analyse back one frame (c) Analyse ahead one frame (d) Analyse forward

10 Enable either Racking or Reposition.

11 If you enabled Reposition, scale the image as needed.

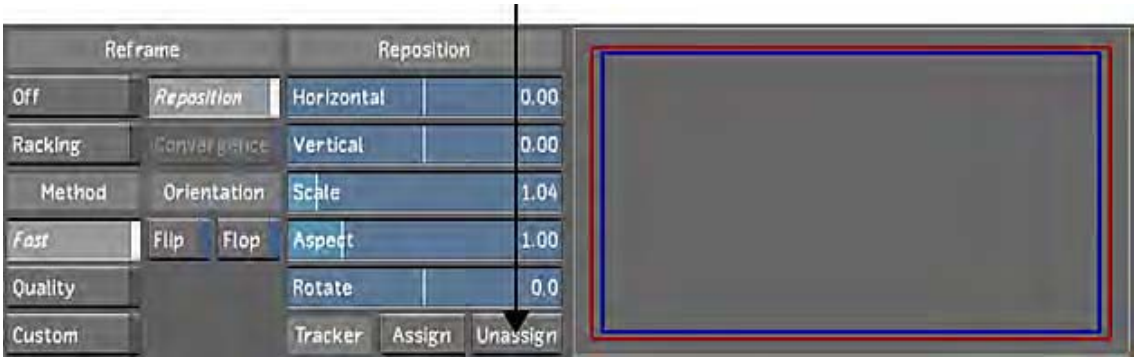
12 Click Assign to apply the tracking data to the shot.

If you enabled Racking, the tracking point remains fixed vertically. If you enabled Reposition, the tracking point remains fixed both horizontally and vertically.

To turn off tracking data for a shot:

1 Display the Reposition menu. See Accessing the Reposition Menu (page 2018).

2 Click Unassign.



Animating Reposition Values

Use the Animation controls in conjunction with the Reposition tools to animate the position, rotation, scaling, and aspect ratio of your shots. You can use this feature to simulate camera movement, which is useful when you want to make a still shot more dynamic. You can animate manually or using Autokey.

To animate reposition values manually:

1. Display the Reposition menu. See Accessing the Reposition Menu (page 2018).
2. Enable Racking or Reposition.
3. Click Anim to display the Animation controls.



4. In the channel hierarchy, select the Reposition directory.
5. Move to the first frame of the shot, click Add to add a keyframe, and then adjust either the rotation, scaling, aspect ratio, horizontal or vertical position as needed.
6. Move to the last frame of the animation.
7. Set the reposition values and then click Add.

   Keyframes are added to the selected channels. The values in between the keyframes are interpolated using Bezier interpolation.
8. Scrub to view the animation.

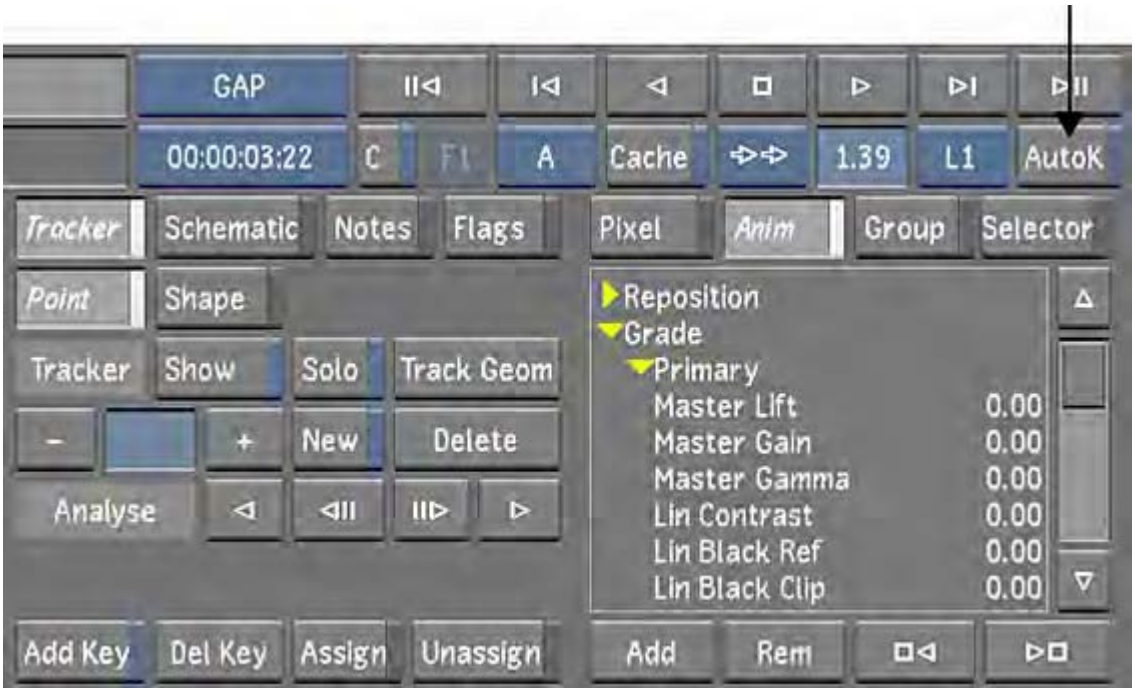
9 To modify the animation curve, use the tools in the Animation menu. See Accessing the Channel Editor (page 2171).

To animate position and scaling with Autokey:

1 Display the Reposition menu. See Accessing the Reposition Menu (page 2018).

2 Enable Racking or Reposition.

3 Enable AutoK (Autokey).



4 Move to the first frame of the shot and then adjust the scaling or the positioning as needed.

Keyframes are automatically added at the current frame to the channel parameter you change.

5 Move to the last frame of the animation.

6 Set the scaling and the position.

Keyframes are added to the adjusted channels. The values in between the keyframes are interpolated using Bezier interpolation.

7 Scrub to view the animation.

8 To modify the animation curve, use the tools in the Animation menu. See Accessing the Channel Editor (page 2171).

Setting Rendering Quality

There are several options you can use to modify the way the repositioned images are rendered.

To set reposition rendering quality:

1 Display the Reposition menu. See Accessing the Reposition Menu (page 2018).

2 From the Rendering Method box, select a rendering method.



Enable:	To:
Fast	Use bilinear rendering. This gives fast display at a lower quality. Generally, you should work in Fast mode.
Quality	Use bicubic rendering. This gives higher quality but requires longer to display the frames. Use this option during MemCache rendering to show exactly how the image scaling will be performed in the final render.
Custom	Use the highest rendering quality. You should use this when performing the final render. This is the default method. It uses a Lanczos resize filter.

NOTE In Real-Time Deliverable mode, you can select additional custom filters. See Creating Real-Time Deliverables from the Timeline or Rendered Material (page 2291).

Input LUTs

Images originating from different media and devices, such as digital cameras or scanned film, have different colour characteristics. On import, you can apply an Input LUT to convert them all to the grading working space prior to the grading process. Unlike LUTs applied for viewing, Input LUTs change the underlying colour values of the pixels.

NOTE If you are working media imported using the Wiretap Gateway, you can use the Transcode LUT options instead of an Input LUT. The Transcode LUT options allow you to apply colour transforms based on the Academy/ASC XML colour transform format as well as 3D LUTs and Flame-compatible 1D LUTs, which provides a great amount of flexibility. See Transcode LUT Options Settings (page 1883).

If running the Windows version of Lustre, save print LUTs in the *C:\Program Files\Autodesk\Lustre <version_number>\lut* folder. If running the Linux version of Lustre, save print LUTs in the */usr/autodesk/lustre_<version_number>/lut* directory. Lustre provides industry-standard 3D LUTs which are located in *C:\Program Files\Autodesk\Lustre Color <version_number>* in the Windows version and*/usr/discreet/Lustre_Color* in the Linux version.

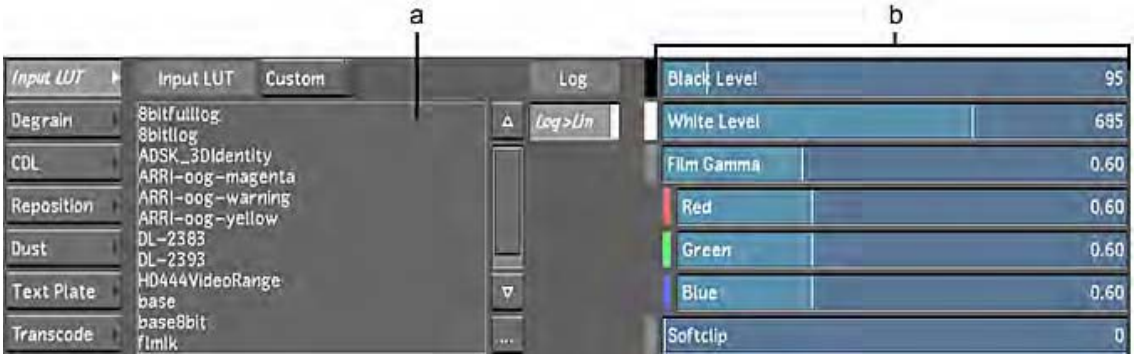
Accessing the Input LUT Menu

Use the Input LUT menu to apply existing input LUTs to shots. In Linear mode, you can also use the LUT Editor to create Log to Lin conversion LUTs for shots containing logarithmic data.

To access the Input LUT menu:

1. In the Main menu, click Image.
2. Click Input LUT.

   The Input LUT menu appears. Both 1D LUTs and 3D LUTs are listed in the Input LUT list.



(a) Input LUT list (b) LUT Editor (Linear mode)

Importing a LUT File

Lustre can use the LUT files that you import into the application folder's *lut* directory. The LUT filename is displayed when the LUT list is displayed.

There are two ways to import LUTs into Lustre:

- Outside of Lustre, files can be copied manually to the application *lut* folder. If running the Windows version of Lustre, save print LUTs in the *C:\Program Files\Autodesk\Lustre<version>\lut* folder. If running the Linux version of Lustre, save print LUTs in the */usr/Autodesk/lustre_<version>/lut* directory.
- Within Lustre, copies of LUT files are imported using the Import file browser.

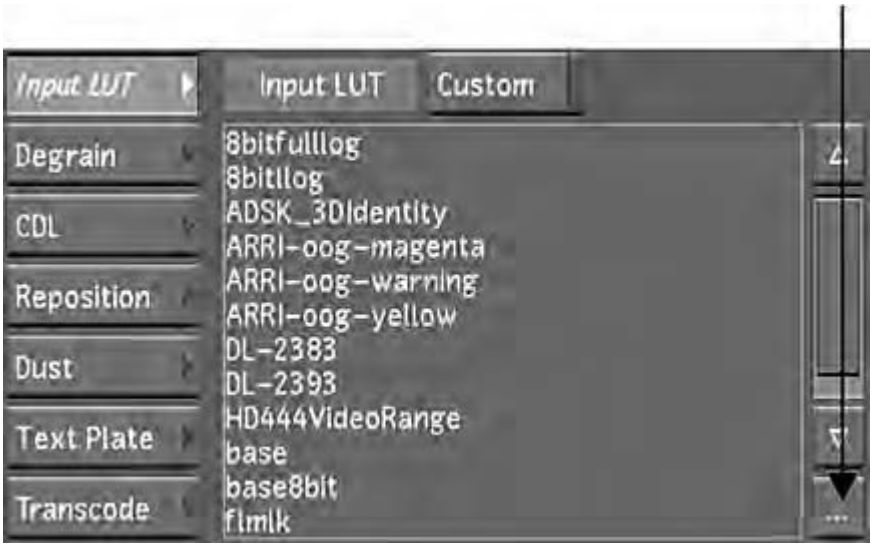
TIP Lustre provides industry-standard 3D LUTs which are located in *C:\Program Files\Autodesk\Lustre Color <version>\3D LUTs* in the Windows version and*/usr/discreet/Lustre_Color* in the Linux version.

NOTE 1D LUTS, 3D LUTs and floating conversion LUTs must have the naming convention, *<name.lut>*, *<name.3dl>*, and *<name.fclut>*, respectively, or they will not appear in the LUT list.

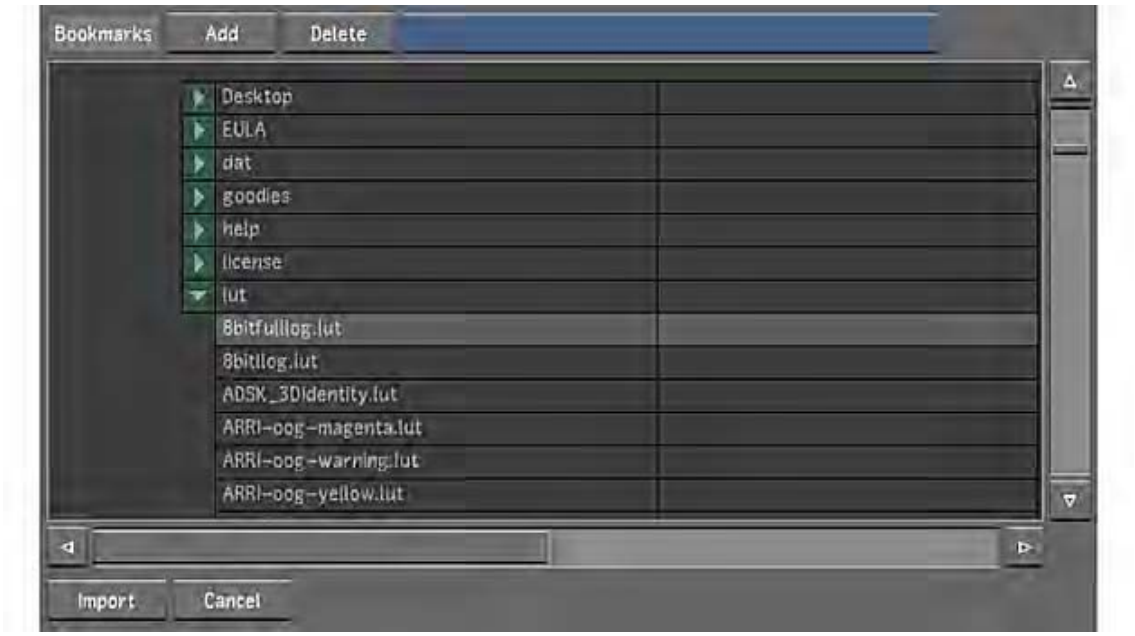
All LUT lists contain 1D and 3D LUTs, except the Float Conversion LUT list. Files with the *.fclut* extension can be imported to the Float Conversion LUT list are exclusive to that list. See Calibration Settings (page 1741).

To import a file while in Lustre:

1. Open the Input LUT menu. See Accessing the Input LUT Menu (page 2027).
2. In the LUT List, click the Browse button.



The Import file browser appears.



3 Navigate to the appropriate source folder and select one or more LUT files (You can also use bookmarks (page 1862) to retain a list of folder locations). Traditional methods of multi-selecting (`Shift`-click, `Ctrl`-click) are enabled.

4 Click Import.

WARNING If a LUT with the same filename is already in the LUT list, the Import button will turn red. You must click again to confirm that you want to overwrite the file.

TIP If no files are selected, you can also press `Enter` to cancel LUT file selection and close the Import file browser.

A copy of the LUT file is imported into the *lut* folder, and appears in the LUT list.

Applying an Existing Input LUT

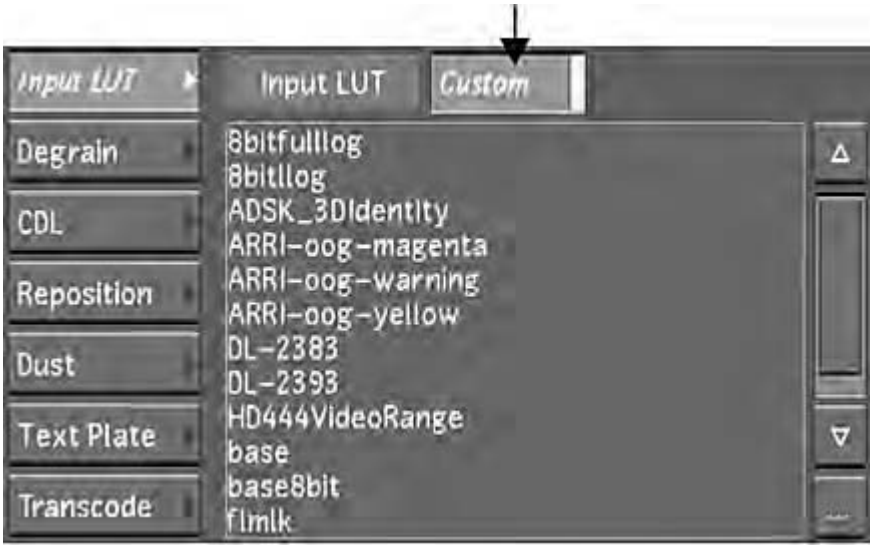
An input LUT can be used to convert and gamma correct the logarithmic or linear data. Apply an input LUT to a shot to remap the colour values. Input LUTs are applied on a shot-by-shot basis and can be toggled on and off.

The settings in the Input LUT menu are saved in grade files. This applies to both 1D LUTs and 3D LUTs. When you use a grade from another station that contains an input LUT that does not exist on your station, the input LUT will appear in the Input LUT list, under Extra LUT from Grade. You can apply this LUT to other shots in the timeline.

NOTE When a LUT is imported into Lustre, it is automatically embedded in the grade file and can therefore be rendered on remote rendering systems without the original LUT file being present.

To apply an existing input LUT:

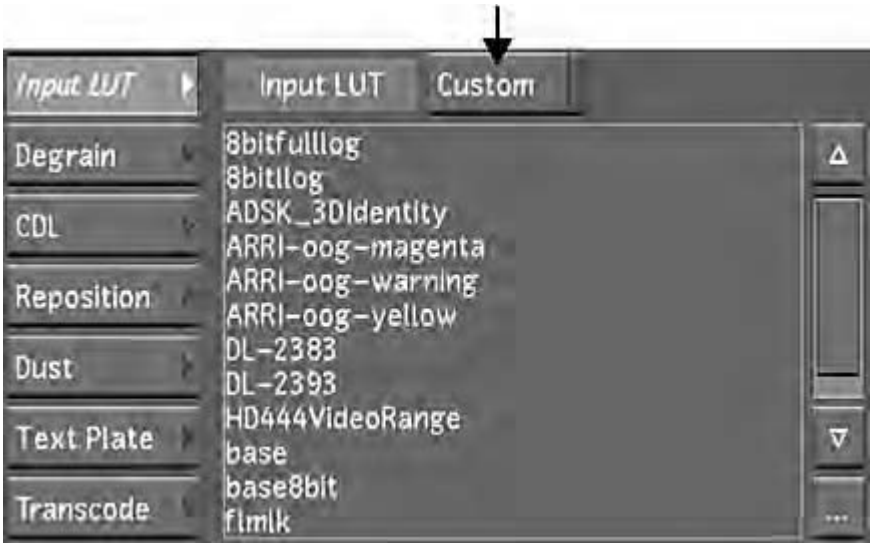
1. Select a shot in the Storyboard to which you want to apply an input LUT.
2. In the Input LUT menu, select either a 1D input LUT or a 3D input LUT from the Input LUT list and then enable Custom.



   The input LUT is applied to the current shot.
3. To select a LUT originating from another station, expand Extra LUT from Grade in the Input LUT list, and then select the LUT.

To turn off an applied input LUT:

1. Select a shot in the Storyboard with an input LUT.
2. In the Input LUT menu, disable Custom.



The input LUT is no longer applied to the shot.

3 Click the input LUT in the input LUT list to remove it completely.

The input LUT is deselected.

Linear Mode: Creating Conversion LUTs

Different film stocks produce various results. Shoots can also occur under many conditions, creating colour variations in the image. To create continuity between shots, you can apply unique Log to Lin conversion LUTs to different shots. Use the LUT Editor to create Log to Lin conversion LUTs. Conversion LUTs are used to gamma correct the shot and convert logarithmic data to linear data.



(a) LUT Editor (b) Pixel Analyser

To create a Log to Lin conversion LUT, the following steps are usually involved:

1 Set up the Player to monitor the shot you are converting and a reference image.

2 Use the Pixel Analyser to sample corresponding colours from the original and converted shot, as well as a reference image.

3 Modify the default conversion settings using the LUT Editor.

Viewing Reference Images

While creating Log to Lin conversion LUTs, it is important to view reference images. For example, view other shots in the cut to ensure continuity.

You can also view a frame from the shot you are working on to see how it looked on film. To do so, save a reference image to a Grade bin in Log mode, and then load it to the frame buffer in Linear mode. For information on saving shots to a Grade bin, see Using Grade Bins (page 1821).

When you view reference images, you can switch between the reference image and the current shot, or you can view them both simultaneously.

To switch between a reference image and the shot you are working on:

1. Load the reference image to the frame buffer by doing one of the following:
   - Go to the frame you want to use in the Storyboard and then press `K` to load it to the frame buffer.
   - Right-click a Grade bin thumbnail.



2. In the Storyboard, select the shot you are working on.

   The shot appears in the Player.
3. To toggle between the reference image and the shot you are working on, press `L`.

To display the reference image and the current shot simultaneously:

1. Assign the shot you want to work on to Playhead A. Press `F1` or select A from the Playhead button, and then select the shot in the Storyboard.



2 Press F2 or select B from the Playhead button.

The shot assigned to Playhead B appears.

3 Load the reference image to the frame buffer by doing one of the following:

- Press K to load the current frame in the Storyboard to the frame buffer.
- Right-click a Grade bin thumbnail.

4 Press L or click S to assign the reference image to Playhead B.



The reference image appears in the Player. At any time, press L to toggle between the original shot (O button) assigned to Playhead B and the reference image (S button).

TIP If you want to use a shot from the Storyboard for reference, you can use the shot assigned to Playhead B. In this case, you do not need to load a reference image to the frame buffer.

5 Press F4 to enable Dual View.



6 Press F4 until you display Horizontal Wipe or Vertical Wipe mode.

The images assigned to Playhead A and Playhead B are displayed. Playhead A is currently selected and displays the shot you are working on. Playhead B displays the reference image.

TIP Press F4 until you are in 2-up node to view your reference image (Playhead B) and the shot you are working on (Playhead A) in separate viewers.

7 To work on the shot you assigned to Playhead A, press F1 or select A from the Playhead button.

The shot you are working on is selected. You can move back and forth between the shots in Playhead A and Playhead B by pressing `F1` and `F2` respectively.

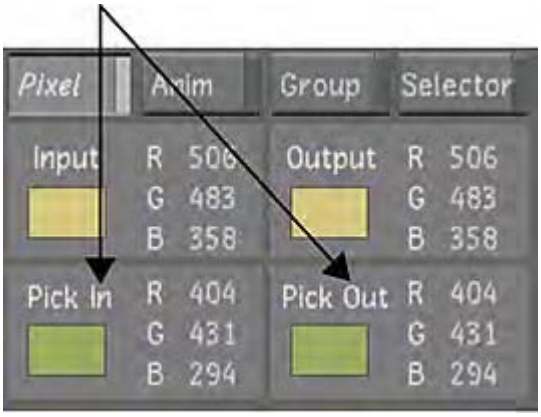
Comparing Colour Values

While creating the Log to Lin conversion LUT, you can use the Pixel Analyser to sample the shot and compare the original and resulting colour values.

To compare colour values:

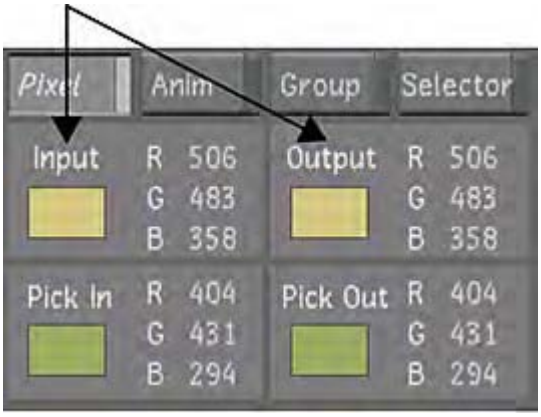
1. Click a pixel in the image.

   Both the original (Pick In) and modified (Pick Out) colours are displayed simultaneously in the lower section of the Pixel Analyser. The RGB values for the selected pixel are also displayed.



2. Compare the colours in your shot to a set of reference values. View the reference image, and then place the mouse cursor over a pixel in the image without clicking it.

   The original (Input) and modified (Output) colours in the reference image are displayed simultaneously in the upper section of the Pixel Analyser. If you click the reference image, the reference values will overwrite the previously sampled values.



Modifying Default LUT Values

When you convert logarithmic data to linear data with the Log to Lin conversion LUT, you can modify several settings from the Input LUT menu:

- The reference white and reference black levels
- The gamma of the incoming film footage
- The softclip value—the degree to which the shoulder of the upper end of the conversion curves is softened

Modifying Reference White and Reference Black Levels

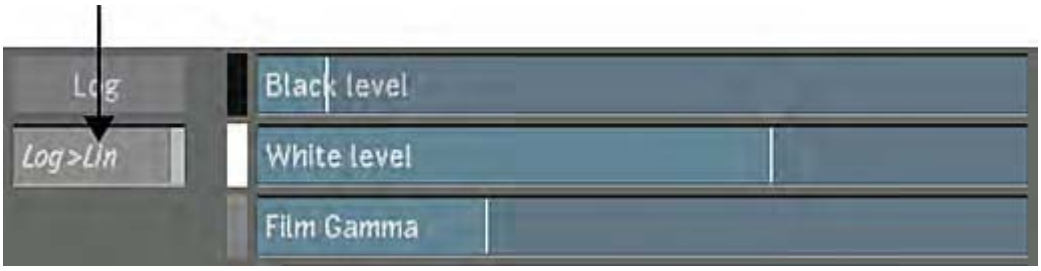
Reference white and reference black levels define the source white and black range. By default, reference white is set to 685 and reference black is set to 95. Using these values (which correspond with typical Kodak film stock results):

- All incoming pixels with values between 685 and 1023 produce white pixels.
- All incoming pixels with values between 1 and 95 produce black pixels.

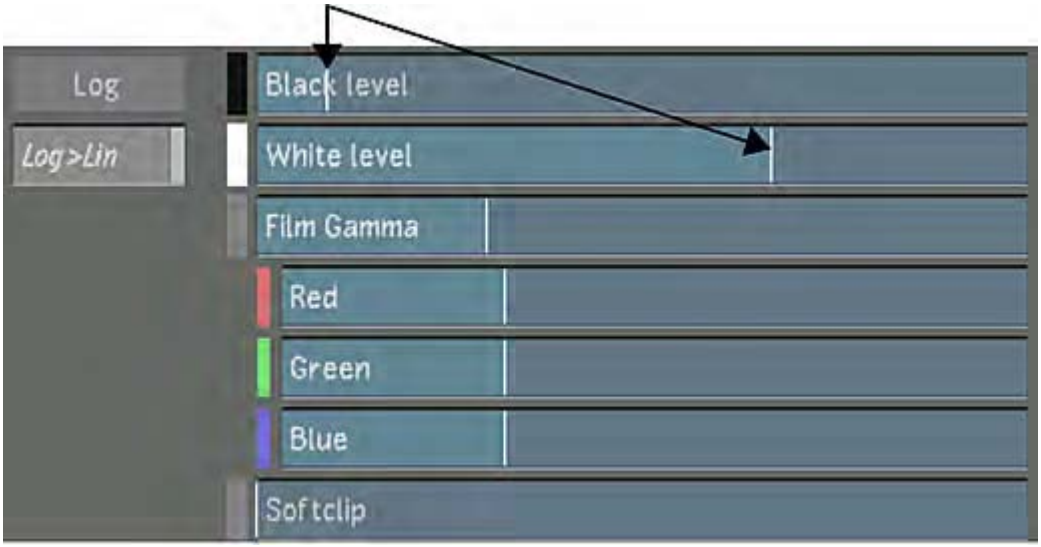
You can modify the reference white and reference black levels for the red, green, and blue channels proportionally or independently.

To modify reference white or black levels:

1. In the Input LUT menu, enable the Log>Lin button.



2. To modify the red, green, and blue channels in the reference white or black levels proportionally, drag the White Level or Black Level slider.



While dragging, the reference white or black level is displayed in the slider.

3 To modify the red, green, and blue channels for the reference white or black levels independently, click the White Level tab or Black Level tab to expand the parameter, and then drag the Red, Green, or Blue slider.



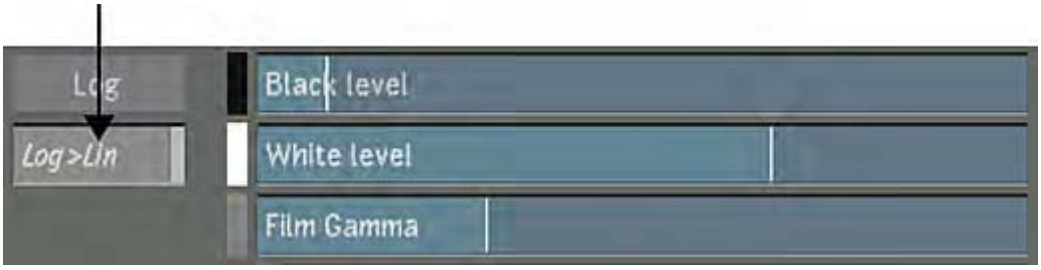
While dragging, the reference white or black level of the selected channel is displayed in the slider.

Modifying the Gamma of the Incoming Film Footage

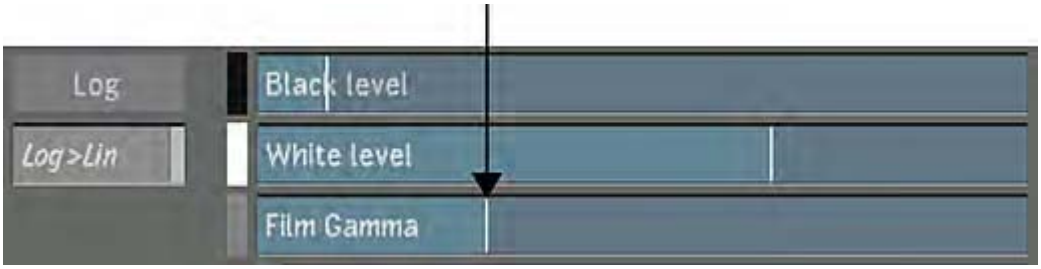
Typically, the gamma of a film negative is 0.6. Only adjust this value if your film negative is of a non-standard gamma. You should not have to modify this value by very much. The Film Gamma values affect the gamma correction curve inversely.

To modify the gamma correction curve:

1 In the Input LUT menu, enable the Log>Lin button.

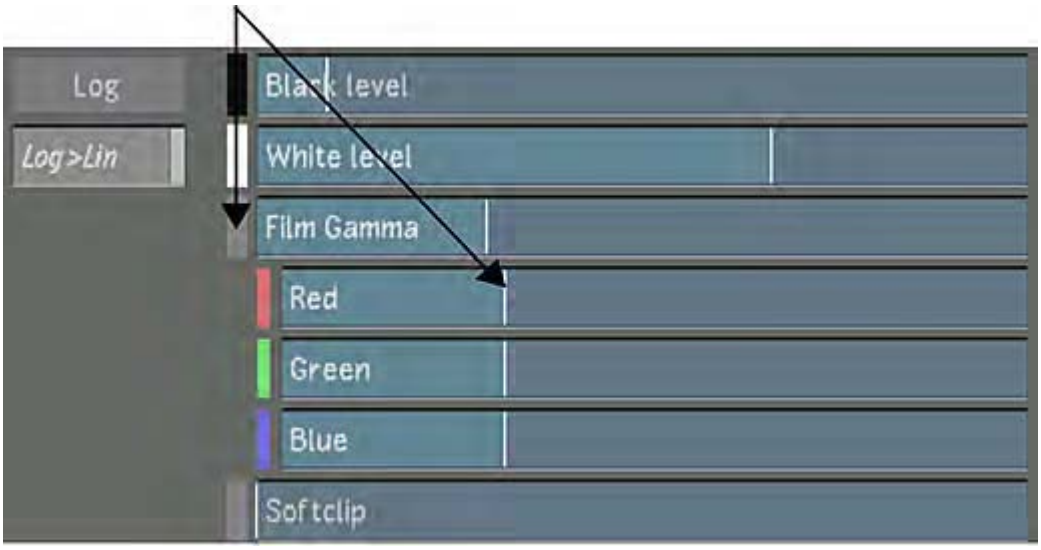


2 To modify the red, green, and blue channels for the film gamma levels proportionally, drag the Film Gamma slider.



While dragging, the film gamma level is displayed in the slider.

3 To modify the red, green, and blue channels for the film gamma independently, click the Film Gamma tab and then drag the Red, Green, or Blue slider.



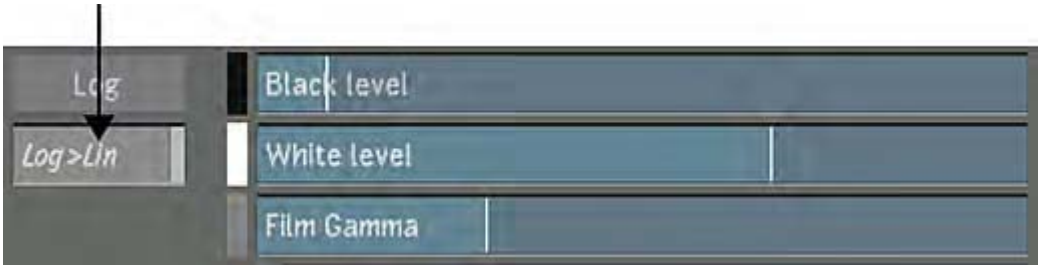
While dragging, the film gamma level for the selected channel is displayed in the slider.

Adjusting Softclip Values

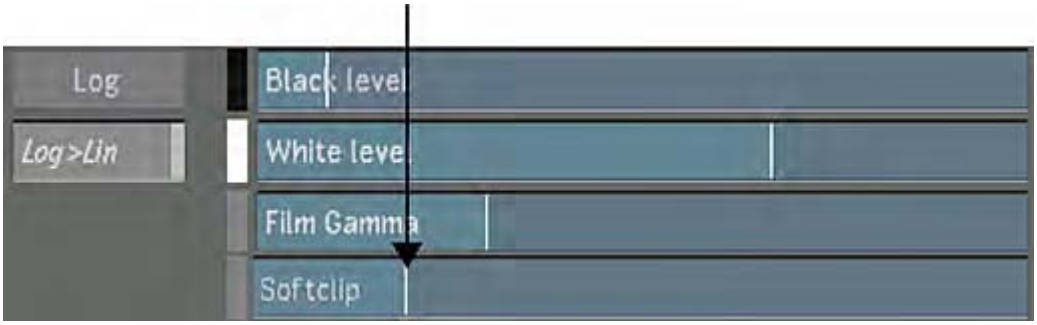
You can soften the shoulder of the gamma correction curve by adjusting the softclip levels. When you soften the shoulder of the conversion curve, you soften the transition of colours toward the reference white value. This results in softer highlights in your shot.

To soften the shoulder of the gamma correction curve:

1 In the Input LUT menu, enable the Log>Lin button.



2 To modify the red, green, and blue channels for the softclip levels proportionally, drag the Softclip slider.



While dragging, the softclip level is displayed in the slider.

3 To modify the red, green, and blue channels for the softclip levels independently, click the Softclip tab and then drag the Red, Green, or Blue slider.



While dragging, the softclip level for the selected channel is displayed in the slider.

Working with Half Float Media

OpenEXR is a high-dynamic range (HDR) media format supporting multi-resolutions and an arbitrary number of channels and channel types, such as specular, diffuse, alpha, RGB, normals, in a single file. Lustre supports OpenEXR 16-bit RGB only (files with other channels cannot be read). This format is sometimes referred to as "16-bit half float".

Lustre can obtain 16-bit floating-point media in four distinct ways. First, it can acquire the media by way of a Wiretap server. That is to say, by browsing the clip libraries and associated framestores of Visual Effects and Editing applications. Second, it can acquire the media directly, by importing OpenEXR media using a Wiretap Gateway server. Third, OpenEXR files can be read directly without using the Wiretap Gateway server. Fouth, media shot with a RED camera in HDRx mode and where "High Dynamic Range" is selected in the HDRx settings section of the Transcode tab also enter Lustre as 16-bit floating-point. In all cases, you need to tell Lustre how to work with the half float media.

As indicated, OpenEXR represents pixels as floating-point numbers. Internally, Lustre uses a 16-bit integer buffer format (although processing is done in floating-point). Thus, before you can grade OpenEXR media, it must be converted from floating-point to Lustre's special extended-range integer format. This is done by setting a floating-point conversion LUT for the project. This setting is located in the calibration tab of the project settings. Several floating point conversion LUTs are provided as examples.

LUT	Description
lin_default	A float-conversion LUT which converts the scene-linear floating-point media into an extended range video space suitable for grading using Lustre's linear grading controls.
log_default	A float-conversion LUT which converts the scene-linear floating-point media into an extended range logarithmic space

LUT	Description
	suitable for grading using Lustre's logarithmic grading controls. This is the recommended way of grading HDR media.
lin_black_limit1	Similar to lin_default but accomodates media from some digital cameras which may have spurious negative or very low values.
log_black_limit1	Similar to log_default but accomodates media from some digital cameras which may have spurious negative or very low values.
ACESproxy_CDL_workflow	Similar to log_default but the range of scene exposures presented to the ASC CDL match the ACESproxy range. Values outside this range are not clamped.

Whenever loading half float media, Post Input LUT should be selected in the Image/CDL menu.

NOTE The floating-point conversion LUT causes linear images to appear similar to log images in Lustre, instead of the way they appear in Flame by default.

Similar to logarithmic media, HDR media requires a viewing LUT in order to look correct on a display or to render to a video color space. This is sometimes referred to as a tone-mapping LUT. This LUT should be selected in the Output tab of the Render settings. See Rendering Shots (page 2216).

You can use a tone-mapping Colour Transform such as the ones in the RRT+ODT directory. Alternatively, you can use one of the legacy 1d-LUT tone maps:

LUT	Description
lin_default_tonemap	An example tone-map output LUT to be used with the lin_default floating-point conversion LUT.
log_default_tonemap	An example tone-map output LUT to be used with the log_default floating-point conversion LUT.
identityExtended	An output LUT which avoids clipping of HDR information when rendering back to floating-point.

When your goal is to output a floating-point image, either directly to an OpenEXR file, or via rendering to Wiretap as part of an interoperability workflow with an Autodesk Visual Effects and Finishing application, use identityExtended as an output LUT to preserve the extended-range floating-point information. For example, if you are pre-grading OpenEXR media preparing for VFX work, use the identityExtended output LUT during the render.

Colour Grading Basics

Lustre provides interactive colour grading tools for the creation of continuity grades and artistic colour effects. You can perform advanced colour grading in either logarithmic or linear colour space as well as save colour grading settings for future sessions and ongoing use. While you manipulate the colour content of a

shot, you can monitor reference images, as well as changes you make to the colour distribution of the shot to ensure that you achieve the result you want.

Because grades are saved as metadata rather than as part of the shot, the original image is always preserved—you can adjust colours without the risk of permanently losing the original colour of your image. Grade metadata is stored on a shot-by-shot basis, ensuring that your colour grades are applied to the correct shot regardless of any changes you make to a cut.

Colour Grading Workflow

The procedures required to digitally colour grade footage depend on your goal, the shots used in your scene, and the architecture—logarithmic or linear—that you are working in. Scenes consisting of footage shot on the same camera equipment and under consistent conditions may be colour graded quickly and easily. Scenes created from footage shot on multiple cameras across several months and under varying lighting and weather conditions require more work.

Because you often work with a wide variety of shots, there are a variety of workflows you can use when grading a scene. However, it is important to keep in mind the order in which colour grading tasks are processed in Lustre.

The following table outlines processing order and a typical colour grading workflow from balancing a shot to performing final adjustments. You may not complete all the procedures. You may also revisit procedures as you create continuity grades and artistic effects.

Step:	**Refer to:**
1. Select the project colour space.	Setting Logarithmic or Linear Mode (page 1786).
2. With primary colour grading, balance colours in the shot and develop a continuity grade to create a consistent colour look.	Primary Colour Grading (page 2058).
3. Further develop the colour look using the RGB and Hue curves.	Colour Grading RGB and Hue Curves (page 2074).
4. Perform secondary colour grading to colour grade objects and colour ranges in the shot.	Secondary Colour Grading (page 2083).
5. Use Sparks plugins to apply image processing effects such as grain reduction.	Creating Lustre Sparks Effects (page 2156).
6. Adjust the colour look using the RGB curves.	Colour Grading RGB and Hue Curves (page 2074).
7. Perform any necessary primary colour grading to adjust the colour look and complete the colour grade.	Primary Colour Grading (page 2058).

Colour Grading Concepts

Altering the colour content of your footage to create a colour look across a series of shots, scenes, or the entire project is the primary objective of colour grading (also referred to as colour timing). The following concepts are used throughout the colour grading chapters.

Log and Linear Modes

In Lustre, you can work in Log or Linear mode. The mode you select defines the colour space, your work environment, and the availability of some hotkeys. In Log mode, you colour grade shots using a film-based toolset and can grade in printer lights. In Linear mode, you colour grade shots using a video-based toolset. For example, if you are familiar with printer light grading, or you are grading logarithmic images, work in Log mode (Log mode is a better grading tool, mathematically, for logarithmic images). If you have more experience working with video and are familiar with the linear toolset, you can use Linear mode. Also, if you are working with linear images and are outputting to linear, it is simpler to work in Linear mode than to convert images to and from Log mode.

Primary Colour Grading

Primary colour grading is applied to the entire image and is used to obtain an overall colour look for each shot used in a series of shots, scene, or entire project. When you perform primary colour grading, you modify the brightness and contrast of the red, green, and blue channels independently. You can modify the red, green, and blue channels together (with the Brightness and Contrast sliders); however, the channels are not codependent—a change made to the red channel does not depend on a change made to the green or blue channel.

You perform primary colour grading at the beginning and end of the colour grading process. In Log mode, you modify brightness, contrast, and saturation across the entire image, as well as brightness in the shadows, midtones, and highlights. In Linear mode, you modify lift, gain, gamma, saturation, and contrast for the red, green, and blue channels across the entire image or in the shadows, midtones, and highlights. You can also clamp minimum and maximum luminance values.

NOTE The intensity of specific colour channels (red, green, and blue) determines whether the pixel is part of the image shadows, midtones, or highlights.

Curves Colour Grading

Use curves colour grading to further modify the RGB and Hue curves. Curves colour grading is well suited, for example, for colour grading a specific range of colours without having to pull a key. Alter the RGB curves to remap red, green and blue values either simultaneously or individually. Use the Hue curves to perform hue shifts, lighten or darken colour ranges, and saturate or suppress colour or luminance ranges. You can also plot colours and add vertices for increased precision.

RGB curves are processed after the initial primary colour grade, and after the application of Sparks effects (if applicable). Hue curves are processed after the initial primary colour grade only.

Secondary Colour Grading

Use secondary colour grading to colour grade specific hues and areas in an image. Create secondaries by generating keys and geometries. You can combine keys and geometries to define the area for modification—keys to define a colour or range of colour and geometries to define an area. After areas have been defined, you can colour grade them using the Grading and Curves menus. You can also track objects in the image to animate a geometry used for a secondary colour grade. You can create up to 48 secondary layers. Secondary colour grading is applied after the initial primary colour grade.

Continuity Grade

One of the first steps in the colour grading process is the creation of a continuity grade—a consistent colour look across a series of shots and over time. Continuity grades convey the time frame and should be invisible to the viewer. Generally, continuity grades are created when you balance the colours in the shots—during the initial primary colour grade.

Printer Lights

Printer lights are measurements or settings used by colour labs to operate the optical printer that produces the film copy. Because you may want to grade your shots according to printer lights and fstops, Lustre displays the modifications you make to the red, green, and blue content in printer lights when you perform primary and secondary colour grading in Log mode. You set printer light steps that calibrate Lustre with the equipment in the film lab in the Display & Interface tab of the user configuration. See Display & Interface Settings (page 1759).

Accessing Colour Grading Menus

Use the Colour menu to access the Grading, Curves, Secondaries, Degrain, and Timeline menus.

To access the Colour menu:

1 In the Main menu, click Colour.
The Colour menu appears.



(a) Colour grading menus

You can access the following colour grading menus from the Colour menu.

Grading Create the primary colour grade to colour balance your shots, as well as develop the continuity grade. In Log mode, adjust the brightness, contrast, and saturation across the entire image, or adjust the brightness in the shadows, midtones, and highlights. In Linear mode, adjust lift, gain, gamma, saturation, and contrast across the entire image or in the shadows, midtones, and highlights. You can also clamp minimum and maximum luminance values. Enable the Input button for initial primary colour grading, and the Output button to create the final look.

Curves Refine the primary or secondary colour grade. Remap colour values for the red, green, and blue channels individually or all together using the RGB curves. Adjust the hue, saturation, lightness, and luminance using the Hue curves. Enable the Input button to refine the initial primary grade, and the Output button to refine the final colour grade.

Secondaries Define up to 48 secondary layers that can then be graded using the Grading, Curves, Degrain, and Sparks Effects menus. Generate keys that define your secondaries by isolating colour ranges in the image. Create and track geometries to include with, or exclude from, the secondary colour grade.

Degrain Analyse the grain profile of a frame, and use spatial degrain operations to remove grain and noise with secondary layers. See Removing Grain (page 2008).

NOTE To analyse the source image before you begin primary grading, access the Degrain menu from the Image menu instead.

Timeline The colourist's timeline is a simple user interface that allows the colourist to work in a timeline interface without all the timeline editing tools. In this interface you can work in Normal or Solo mode, add cuts and dissolves, and create a new version of a shot. You do not have access to the trimming tools.

Displaying Colour Menus in the Player

You can display the Grading, Curves and Secondaries colour grading menus in the Player next to the image in the currently active playhead. This allows you to:

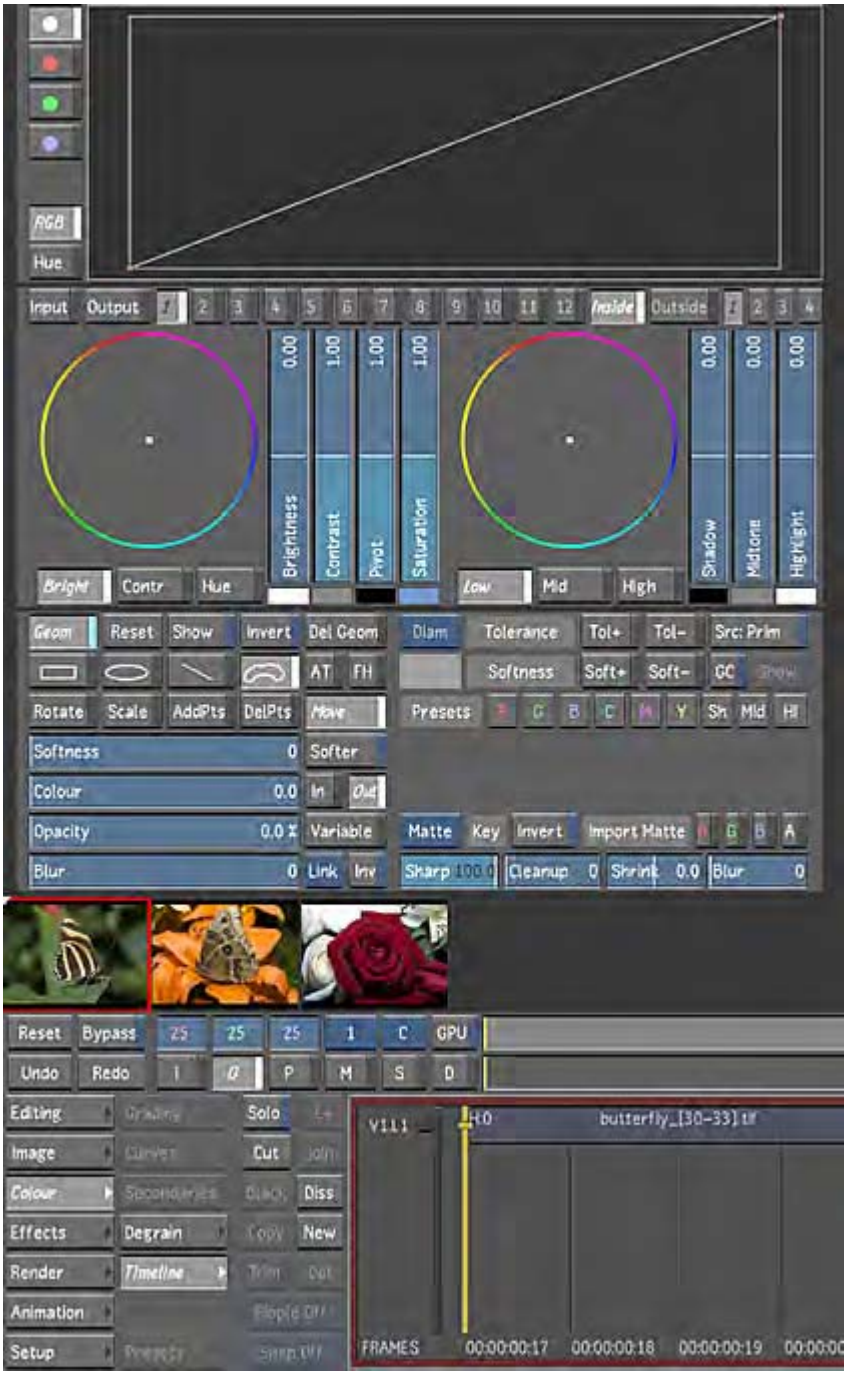
- Keep the Colour menus available even if you are accessing another menu.
- Display viewing options such as split-screen view on the second head of the graphics card.
- Use the mouse cursor to access the HDSDI head and draw shapes that will appear on the broadcast monitor.

For details on assigning shots to playheads, see Viewing Reference Images (page 1970).

To display Colour menus in the Player:

1. Press F7.
   The Colour menus appear alongside the image, and are disabled in the main user interface.

   NOTE The Hue cube of the Diamond Keyer does not appear in this view mode.



GPU Acceleration

Processing with the Graphics Processing Unit (GPU) is available for certain features and is faster than processing with the Central Processing Unit (CPU).

Support for GPU acceleration depends on the version of the NVIDIA graphic card that is installed on your system. The following configurations are supported for GPU acceleration.

Configuration	Feature available?
FX5800	Yes
FX5600	Yes
FX5500	Yes (except Noise plugin 3.1)

WARNING

Due to the enhancements in the GPU, we do not recommend using Lustre to render grades created in Lustre 2011 or earlier if you are looking for pixel accuracy rendering.

GPU acceleration is available for the following features:

- Playback of cached degrained frames
- 1D or 3D on calibration LUTs
- Animated or still repositions (including rotations)
- Animated or still input primary grading
- Input and output primary grading with RGB, hue and light saturation (LS) curves
- Secondary grading with RGB curves
- Secondary grading with hue curves
- Secondary black clip/reference and white clip/reference in Linear mode
- Secondary key cleanup and shrink
- Imported mattes for secondaries
- Gamma and contrast adjustments to secondaries in Linear mode
- Low, mid, and high adjustments to secondaries
- Input and render/output LUT
- Lustre Sparks plugins:
  - Blur mix
  - Glow
  - Noise plugin 3.1
  - Printbleach

  NOTE A performance hit can occur when using the Lustre Sparks plugins.

For more information about these features, see:

- Primary Colour Grading (page 2058).
- Colour Grading RGB and Hue Curves (page 2074).
- Creating Lustre Sparks Effects (page 2156).

GPU acceleration is also available for certain secondary grading features with up to 48 secondary layers enabled. You can apply, and optionally animate, each of the following secondary grading features and continue to use GPU acceleration:

- Primary grading inside and outside geometries, including adjustments to overall brightness, contrast, hue, and saturation
- Softness, Colour, and Opacity slider values for geometries, with Softer and Variable optionally enabled
- Key-in shapes based on hue, luminance, and saturation, and refined by tolerance and softness range definition
- Key and geometry blurs

  NOTE A performance hit can occur when using the geometry blur.

- Multiple point changes for geometries

For more information about these features, see Secondary Colour Grading (page 2083).

When GPU acceleration is enabled, the histogram, vectorscope, and waveform monitors do not dynamically update. They retain the colour distribution of the image before GPU was enabled.

The following parameters cannot be accelerated by the GPU. These parameters are processed by the CPU instead.

Lustre Sparks plugins:

- Add noise
- Blur
- Defocus
- Directional blur
- Field zoom
- Gold
- Noise plugin 2
- Noise plugin 3
- Silver

NOTE If GPU acceleration is enabled and Lustre encounters a shot with features that cannot be processed by the GPU, the CPU is used for the shot. The GPU button remains enabled, but is greyed out until you navigate to the next shot in the timeline that contains features available for GPU acceleration.

To enable GPU acceleration:

1. Do one of the following:
   - Click GPU.
   - Press Y.



A GPU flag appears in the upper-right section of the Player.



NOTE GPU acceleration only works in progressive scan mode. Before you render a project, you need to switch the scan type to interlaced (located in the Setup > Grade menu).

To disable GPU acceleration:

1 Do one of the following:
- Click GPU.
- Press Y.

GPU acceleration is disabled.

GPU Auto Switch

By default, when you use a feature that is not supported by GPU (while in GPU acceleration mode) it automatically switches to CPU processing mode. The GPU button is greyed out in the user interface to show you that this feature is not GPU compatible. You can prevent the switch to CPU mode by disabling the GPU Auto Switch feature, see Display & Interface Settings (page 1759). When you disable the GPU Auto Switch button, you can only see features which are supported by GPU (when GPU acceleration is enabled). To see all of the features, you have to disable the GPU button.

Colour Grading With the Control Surfaces

The control surfaces facilitate colour grading work by accelerating interactivity with the shots. You can use control surfaces to perform many colour grading tasks. The dials and trackballs give you precise control over the colour effects you create.

Viewing Multiple Shots

You can view multiple shots in the Player while you work. These views are useful when you want to compare shots in a cut to ensure continuity, match colours, or view a group of shots in which the colour grading parameters you are modifying are ganged. See Using Dual View (page 1986) and Using Multi View (page 1988).

Reference Images

You can view a reference image while you work. For example, use a reference image to visually compare the current shot to another when you develop a continuity grade. Reference images are stored in the frame buffer and are loaded from your cut or from the Grade bin. You can only have one reference image stored in the frame buffer at a time. You can switch between different reference images to ensure continuity and consistency throughout the project. See Viewing Reference Images (page 1970).

Viewing Proxies

After you generate proxies, you can view proxies while you colour grade your shots. It is useful to view proxies if interaction slows down. This can occur when you perform secondary colour grading on high-resolution images because a large amount of processing is required to display the result at full resolution.

Proxy view is set on a shot-by-shot basis. To view proxies, you must first generate them. See Generating and Viewing Proxies (page 2227).

To view proxies:

1. Do one of the following:
   - On the function control surface for the Autodesk control surface, press the Proxy button.
   - On the Tangent CP100 control surface, press the Resolution button.
   - Press `F9`.
   - Click the Resolution button.



   The full-resolution image in the Player is replaced by the proxy (half-resolution) image. Use the same procedure to toggle between the full-resolution image and the proxy.

   NOTE If you have not generated a proxy for the shot, an X will appear in the Player.

Analysing the Image

While you colour grade, you can use different tools to view and analyse colours. Use the Analyse tools to view the colour distribution in the current frame. Use the Pixel Analyser to sample the image to compare colour values used in the input (original) and output (result) images.

Sampling Colour Values

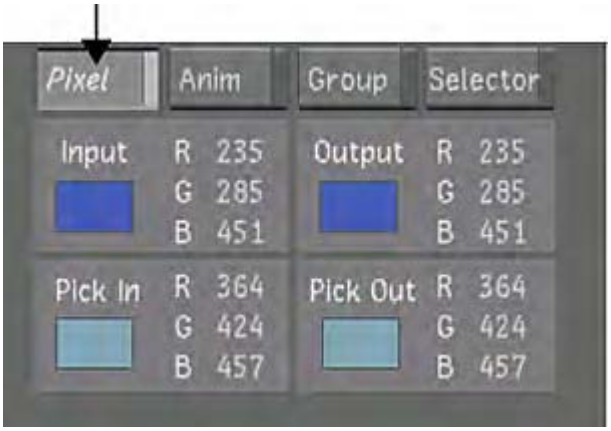
When you generate mattes, modify shots, or match colours between shots, you can sample the current image in the Player. When you sample the image, the input and output colour values are displayed in the Pixel Analyser.

The sample will appear in the Curves menu so you can view input colour values when you refine the image. R, G, and B colour values are plotted separately on the RGB curves while the combined RGB colour value is plotted on the Hue, Saturation, Lightness, and Luminance curves.

When you create a key, you must sample the image to set tolerance and softness values. See Creating a Secondary by Keying a Range of Colours (page 2116).

To sample colour values:

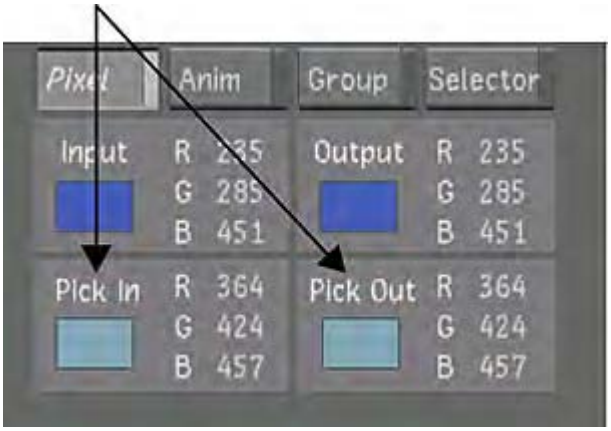
1. Select the frame in the shot you want to sample.
2. Click Pixel to display the Pixel Analyser.



3 In the Player, do one of the following:

- Drag the image to sample a single pixel in the image. You can drag through the image until you locate the pixel you want to sample.
- `Alt`-click to sample an average taken from a range of colours in the image.

The sample appears in the Pixel Analyser.



4 (Optional) Press `E` to change the display value. You can display the colour in three different formats: colour, F-stop/float value, or high dynamic range (HDR).



Display Value	Description
Colour	Displays the colour value for all types of media.
F-stop/Float value	F-stop is only applicable for10-bit media (whereas the 8 and 16-bit media retains the colour value). Float value is only applicable for the Input value of OpenEXR media.
HDR	Displays the colour input value from -1 to 3 for OpenEXR media. For the other media, the colour value is from 0 to 1.

NOTE For OpenEXR media the F-stop value is replaced by the media's float value and the HDR range is only applicable to the Input value. The Input HDR range is from -1 to 3 for OpenEXR media, but for all other media the range is from 0 to 1.

Viewing Colour Distribution

There are a variety of Analyse tools that you can use to view the colour distribution in the current frame. Analyse tools are useful when you want to determine the original structure of the image and see how a grade transforms the colour space.

You can display up to three Analyse tools at one time. Use these tools to help match colours, view the colour distribution, adjust shadows and highlights, and ensure that broadcast and other standards are met. You can display the following Analyse tools while you colour grade.

Histogram Displays the distribution of image pixels across the luma range for the R, G, and B channels.

Waveform monitor Displays the R, G, B channels separately or together as a waveform.

Vectorscope Displays the distribution of image pixels across luma and hue ranges.

The Analyse tools show the changes you make to the colour content in a shot. Each tool is dynamically updated to reflect your changes as you modify the image.

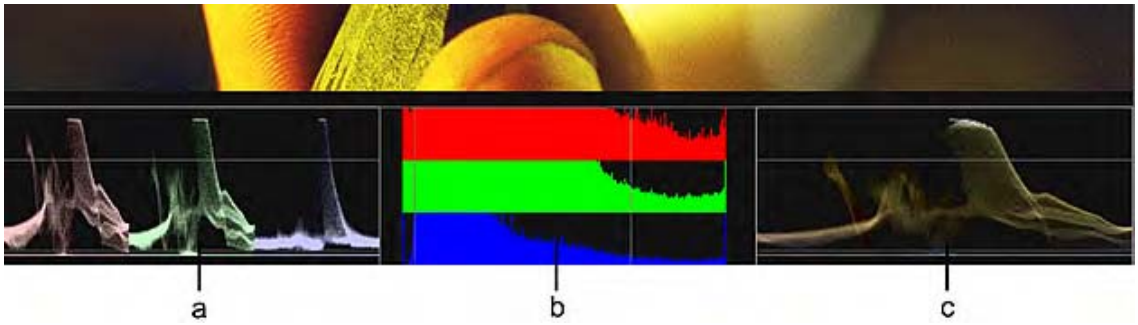
NOTE The histogram, vectorscope, and waveform monitor do not dynamically update if GPU acceleration is enabled. They retain the colour distribution of the image prior to GPU being enabled.

To view colour distributions:

1 Display up to three Analyse tools to view the colour distribution of the image.

Press:	To display:
Alt+1	The RGB waveform monitor.
Alt+2	The RGB histogram.
Alt+3	The luminance waveform monitor.
Alt+4	The vectorscope.

TIP You may need to move the shot in the Player up in order to see the Analyse tools below it.



(a) Waveform monitor (with RGB channels displayed separately) (b) Histogram (c) Waveform monitor (with RGB channels displayed together)

NOTE Right-clicking over any of these monitoring tools either zooms you in, or adjusts the tool's luminance value.

To view the RGB histogram for two images simultaneously:

1 Press F5.

The last used view is displayed—either Dual View or Multi View. Dual View displays two frames at the same time while Multi View displays the entire frame of one or more shots.

2 If Multi View is displayed, press F4 to activate Dual View.

The current frames on Playheads A and B are displayed either horizontally or vertically, depending on the last displayed orientation.

3 Press F1 or select A from the Playhead button and then select the shot in the Storyboard.

The shot you are working on is assigned to Playhead A.

4 Press Alt+2 to display the histogram for the shot assigned to Playhead A.

5 Press F2 or select B from the Playhead button.

The shot is assigned to Playhead A.

6 Press Alt+2 to display the histogram for the shot assigned to Playhead B.

Saving and Loading Presets Using the Presets Lists

To increase the efficiency and speed of the colour grading process, you can save a limited set of the colour grading parameters that you plan to reuse on a regular basis. These predefined settings, or presets, are saved on a menu-by-menu basis and are stored in the Presets lists.

Saving Presets

Save settings from the current Colour menu to a preset. You can save settings to the following Presets lists:

- Grade presets
- Curves presets
- Geometry presets
- Gmask raw import
- Tracker presets
- Key presets
- Degrain presets

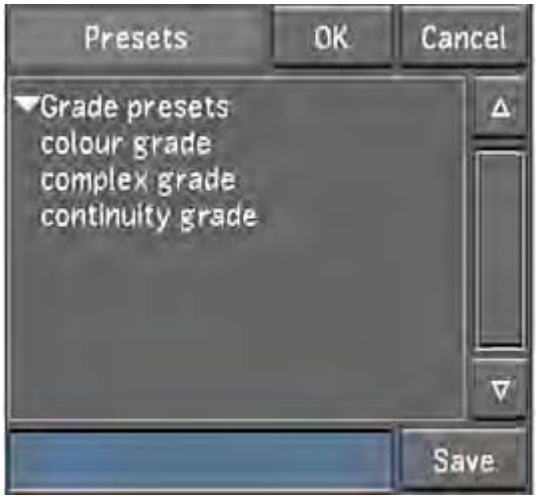
NOTE Saving to a preset saves the settings currently displayed in the Player. For example, if you save to a preset in the middle of a colour dissolve or animation, this timeline position is saved to the preset file. Saving to a preset is a good way to save and load animation keyframes.

To save a preset:

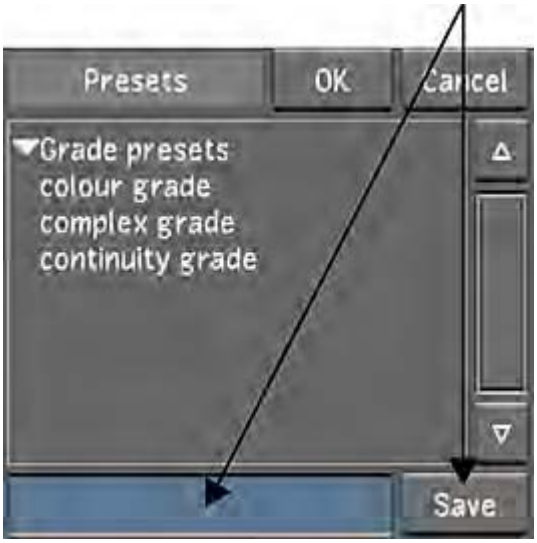
1. Click the Presets button in the Colour menu or right-click Grading, Curves, or Secondaries.

   NOTE You can also right-click the Hue, Luminance, or Saturation button in the HLS Keyer as well the Presets colour buttons in the Diamond Keyer.

   The affiliated Presets list appears. For example, if you right-clicked the Grading menu, the Grade presets list appears.



2. In the Colour menu, make any necessary modifications to the settings to create the colour look you want.
3. In the Presets Name field, enter a name for the preset, and then click Save.



NOTE Spaces are not permitted in naming conventions. It is recommended that underscores be used instead of spaces when working on either Linux or Windows.

The settings in the current menu are saved to the preset. The preset appears in the Presets list.

TIP To save the settings to an existing preset, select the preset in the Presets list.

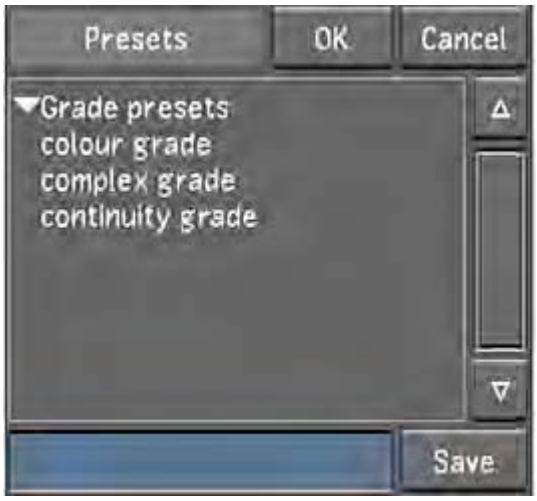
Loading Presets

Increase efficiency by loading presets to the current shot when you want to reuse Colour menu settings. Preview the effect a preset will have on a shot before you apply the settings.

To load a preset from a Presets list:

1 Click the Presets button in the Colour menu or right-click Grading, Curves, Secondaries, Degrain, or the Keyer.

The affiliated Presets list appears. For example, if you right-clicked the Grading menu, the Grade presets list appears.



2 Select a preset in the list.



The image in the Player is updated with the settings from the preset so that you can preview the effect.

3 Do one of the following:

- Click OK to apply the settings.

  The settings are applied to the image and the Colour menu. The other Colour menus are unaffected.

- Click Cancel to revert to the previous settings in the Colour menu.



Colour Grade Animation

Use the Animation controls to animate colour grades. You can animate any parameter from the Colour menus except those in the Curves menu. For example, animate a colour grade to convey a change in the ambient light. To access the Animation controls, click the Anim button. See Animation (page 2171).

Grade Bin Storage

During the colour grading process, use the Global Grade bin or Scene Grade bin as a framestore for reference images and as a scratch pad for grades. Grade settings are stored in the Grade bins on a shot-by-shot basis. See Using Grade Bins (page 1821) and Using the Expanded Grade Bin (page 1828).

Colourist Multi-Layer Timeline

The colourist multi-layer timeline allows you to perform limited editing functions within the Colour menu. Grading information (a selection of a shot's grading information or all of the grading information) can be copied from one shot to another or to several others.

NOTE The functions in the colourist multi-layer timeline are not accessible if your timeline is in sort mode. See Timeline Sort Mode (page 1990).

Accessing the Colourist Timeline Menu

To edit shots in the colourist timeline view, you need the Timeline menu.

To access the colourist timeline menu:

1. In the Main menu, click Colour.
2. Click Timeline.
   The colourist timeline menu appears.



The timeline is made up of the following buttons.



Solo Within a multi-layer timeline, Solo mode displays the layer which the Focus point is positioned on. The Storyboard is updated to only display the shots in the soloed layer. When Solo mode is disabled, the top layer, by default, is displayed in the Storyboard and Player.

NOTE If you are using the solo function as a method to create editorial or grade version, please keep in mind that the status is not part of the grade file data structure. Therefore, this state will not be saved upon restarting Lustre.

Cut Adds a splice at the positioner location on the focused layer.

Diss Adds a dissolve at the positioner location on the focused layer.

New Copies the current element to the next available top layer with grade data.

For more information about the multi-layer timeline, see Multi-Layer Timeline (page 1932).

Copying Grading Information in the Colourist Timeline

There are several procedures for copying grading information using the colourist timeline. You can easily drag and drop all or a selection of grading information from shot to shot (or to multiple shots). You can also drag and drop grading information from multiple shots to an equal number ot shots.

To copy all grading information from one shot to another:

1. In the colourist multi-layer timeline, hold down the `Ctrl` key and drag from the graded shot to the destination shot.

   When you move your mouse over a shot, a green indicator displays the graded shot's name. A red indicator displays the destination shot name.



2. Drop the selection when the cursor is over the desired destination shot.

   The entire shot's grading information is copied to the destination shot.

To copy all grading information from one shot to multiple shots:

1. In the colourist multi-layer timeline, create a timeline selection of shots to which you wish to copy grading information. See Selecting Elements (page 1937).
2. Hold down `Ctrl` and `Shift` and drag from the graded shot to any of the selected shots.

   When you move your mouse over a shot, a red indicator displays the destination shot name.



3 Drop the selection when the cursor is over any destination shot that is in your selection.

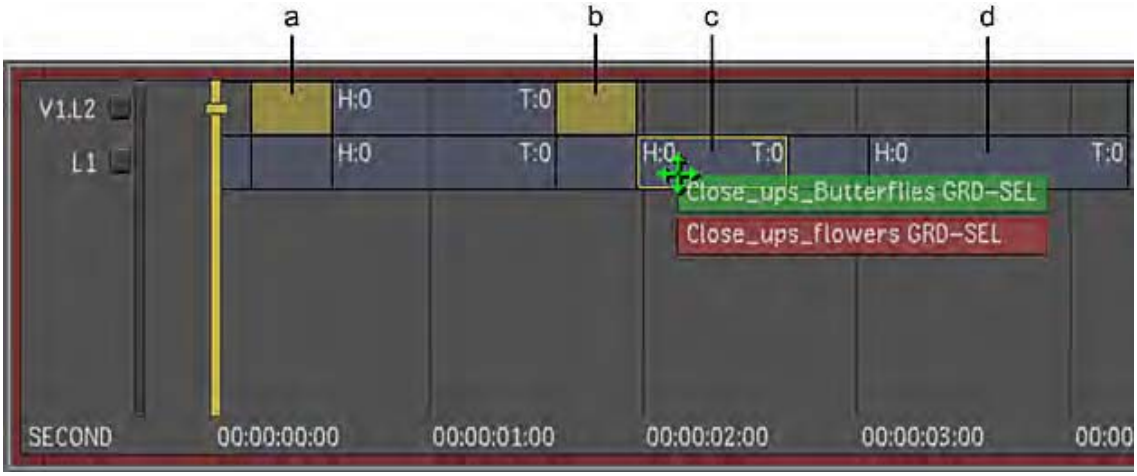
The entire shot's grading information is copied to the selected shots.

To copy all grading information from multiple shots to a different set of shots:

1 In the colourist multi-layer timeline, select two or more shots with the grades you wish to copy. See Selecting Elements (page 1937).

2 Hold down the `Ctrl` and `Shift` keys and drag the first shot in the selection to the first destination shot to be graded.

When you move your mouse over a shot, a green indicator displays the shot name and a red indicator displays the destination shot name.



(a) First graded shot selection (b) Second graded shot selection (c) First destination shot (d) Second destination shot

3 Drop the selection when the cursor is over the first destination shot.

The grading of each shot is applied to a corresponding destination shot. In the example above, if you drag a broken selection of the first and third shot and drop it on the fifth shot in another layer, then the fifth and seventh shot in that layer will be graded, respectively.

To copy a defined selection of grading information from one shot to another:

1 Using the various grading tools, grade a shot to your satisfaction.

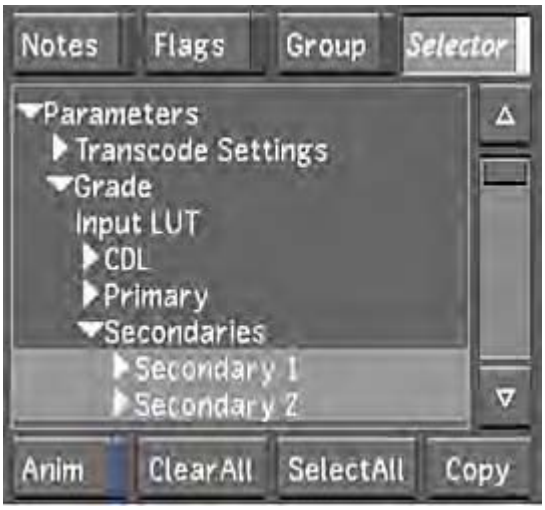
2 In the Timeline menu, enable the Selector button.



3 Specify what parameters to copy:

- To select all the channels in a directory, select the directory.
- To select specific channels in a directory, expand the directory and select any of its channels.

The channels within selected directories become bold white.



4 Hold down the `Ctrl` and `Alt` keys and left-click+drag from the graded shot to the desired destination shot.

A green indicator displays the graded shot's name. A red indicator displays the destination shot's name when you move your mouse over a shot.

5 Release the `Ctrl` and `Alt` keys and mouse button when the cursor is over the destination shot.

The grade is copied to the destination shot.

To copy a defined selection of grading information from one shot to multiple shots:

1 Using the various grading tools, grade a shot to your satisfaction.

2 In the colourist multi-layer timeline, create a timeline selection of shots to which you wish to copy grading information. See Selecting Elements (page 1937).

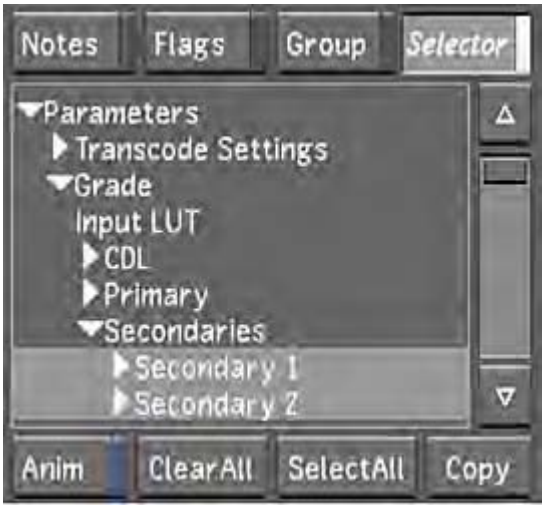
3 In the Timeline menu, enable the Selector button.



4 Specify what parameters to copy:

- To select all the channels in a directory, select the directory.
- To select specific channels in a directory, expand the directory and select any of its channels.

The channels within selected directories become bold white.



5 Hold down `Ctrl`, `Shift`, and `Alt` and left-click+drag from the graded shot to any of the selected shots.

A green indicator displays the graded shot's name. A red indicator displays the destination shot's name when you move your mouse over a shot.

6 Release the `Ctrl`, `Shift`, and `Alt` keys and mouse button when the cursor is over any destination shot in your selection.

The grade is copied to the destination shot selection.

Primary Colour Grading

Primary colour grading is applied to the entire image and is used to obtain an overall colour look. With primary colour grading, you modify the primary colour channels (red, green, and blue) independently. Although you can manipulate these colour channels simultaneously, the modifications you make to one colour channel will not depend on the pixel intensity of the other colour channels.

When you perform primary colour grading, you can develop a continuity grade to create a consistent colour look for shots at the beginning of the colour grading process, and then perform any final adjustments to readjust the colour look and complete the colour grade at the end of the process.

You perform primary colour grading in either Log mode or Linear mode. In Log mode, you can modify brightness, contrast, pivot point, and saturation. In Linear mode, you can modify lift, gain, gamma, saturation, and contrast values, as well as clamp minimum and maximum luminance values.

TIP For faster playback on shots with input primary grading applied, enable GPU processing by clicking the GPU button. See GPU Acceleration (page 2043).

WARNING The Histogram, Waveform, and Vectorscope functions do not update when GPU processing is enabled.

Accessing the Grading Menu

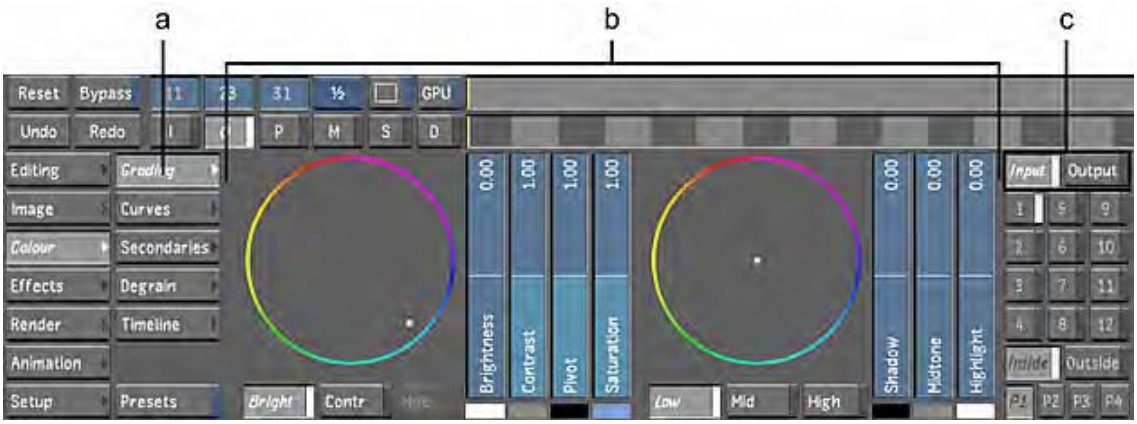
Primary colour grading is done from the Grading menu.

To access the Grading menu:

1 Do one of the following:
- In the Main menu, click Colour and then Grading.

■ Press F7. See Displaying Colour Menus in the Player (page 2042).

The Grading menu appears.



(a) Grading button (b) Grading menu (Log mode) (c) Input and Output buttons

2 Enable one of the following buttons.

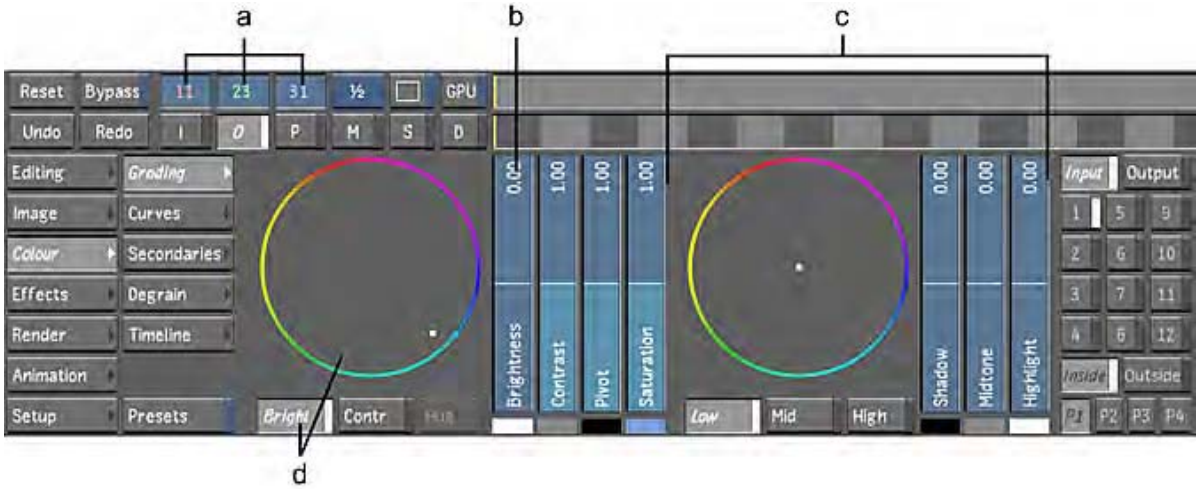
Input Enables input primary grading. Use it to perform primary colour grading at the beginning of the colour grading process.

Output Enables output primary grading. Use it to perform primary colour grading at the end of the colour grading process, that is, after RGB and Hue modifications, as well as selective grading have been performed.

NOTE The Hue setting is only enabled for selective colour grading.

Logarithmic Mode: Modifying the Brightness

When you modify brightness, colours appear to emit more or less light. You can modify brightness across the entire image or just in the shadows, midtones, and highlights. You can also view and make printer light adjustments. Reset the brightness settings to their original values at any time.



(a) Printer light settings (b) Brightness slider (c) Brightness controls for shadows, midtones, and highlights (d) Brightness Balance wheel and button

If you want to modify R, G, B curves individually and within a particular range, use the RGB curves. See Modifying Red, Green, and Blue Channels (page 2076).

Modifying Brightness in the Entire Image

You can increase or decrease colour brightness across the entire image. Use the Brightness slider to modify the brightness of the red, green, and blue levels uniformly. Use the Brightness Balance wheel to balance colours, remove or add a colour cast, create a sense of time, or create ambience. When you use the Brightness Balance wheel, you modify RGB levels proportionally without changing the overall brightness. For example, if you drag the Brightness Balance wheel toward blue, blue will appear to emit more light while red and green will appear to emit less; the image may therefore develop a blue cast.

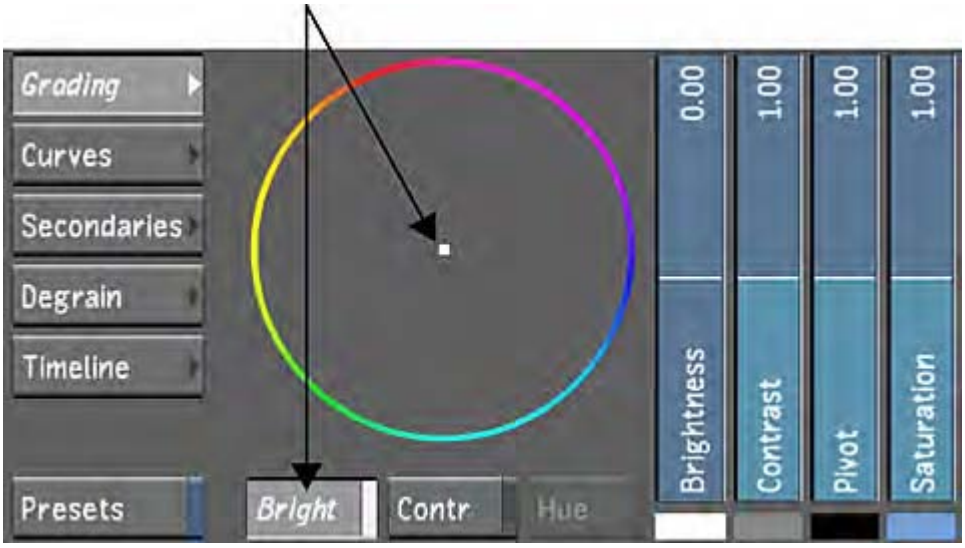
NOTE When you use a slider or colour balance wheel to colour grade a shot, the colour range is limited by the control (the edge of the colour wheel, for example). Use either the Autodesk control surface or the Tangent CP100 control surface to reach the full spectrum of colour.

To modify the brightness in the entire image:

1. Indicate whether this is the initial or final primary grade:
   - To work on the initial primary colour grade, click Input.
   - To make final adjustments to the primary colour grade, click Output.
2. Modify the overall brightness:
   - To increase the overall brightness in the image, drag the Brightness slider up.

     The colours in the image appear to emit more light. The printer light settings are also updated to reflect your changes.
   - To decrease the overall brightness in the image, drag the Brightness slider down.

     The colours in the image appear to emit less light.
3. To balance the colours in the image, create ambience, or develop a colour cast, enable Brightness and then drag the Brightness Balance wheel towards the colour you want to increase in the image. For example, to create a cooler image, drag toward blue; to create a warmer image, drag toward red.



The colours are balanced in the image without affecting the overall brightness. The printer light settings are also updated to reflect your changes.

TIP When you click on the printer lights, the printer light values are replaced by values based on the center point of the colour wheel.

4 Make adjustments to the image using the Brightness slider and Brightness Balance wheel until you are satisfied with the results.

TIP When working with any of the vertical sliders in the Grading menu, refer to their numerical values in order to make more precise adjustments. The default value for each of the sliders reflects the mid-point in the range of available values. For example, the Brightness slider has a range of -25 to 25, and a default mid-point value of 0. The Contrast slider, on the other hand, has a range of 0 to 2, making for a mid-point default value of 1.

Adjusting Printer Lights for Primary Grading

You can make printer light adjustments to the entire image when you want to grade your shots according to printer lights and fstops. You can adjust the level of the overall colour brightness or the level of the RGBCMY values independently. You can then send the printer light settings to the film lab for processing. Printer light steps are set in the Display & Interface page of the user configuration. See Display & Interface Settings (page 1759).

NOTE Hotkeys for printer light adjustments only work in Log mode.

To modify brightness by printer lights:

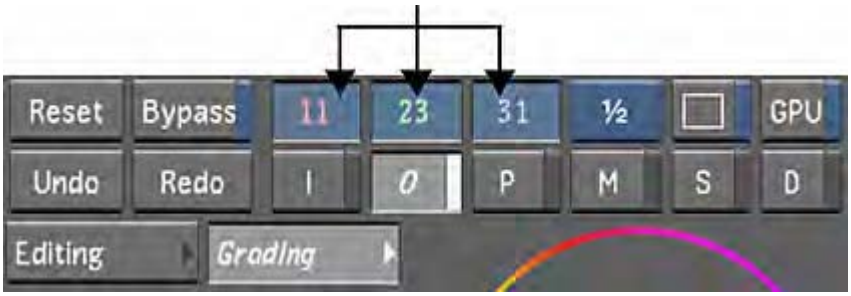
1 Indicate whether this is the initial or final primary grade:
- To work on the initial primary colour grade, click Input.
- To make final adjustments to the primary colour grade, click Output.

2 Use the following keypad hotkeys to make printer light adjustments to the image.

Press:	To:
+ (plus sign)	Increase overall brightness by one step. Press `Shift+(+)` to increase overall brightness by a half step.
`Enter`	Decrease overall brightness by one step. Press `Shift+Enter` to decrease overall brightness by a half step.
`NumLock`	Increase red brightness by one step. Press `Shift+NumLock` to increase red brightness by a half step.
`7`	Decrease red brightness by one step. Press `Shift+7` to decrease red brightness by a half step.
/ (divide sign)	Increase green brightness by one step. Press `Shift+/` to increase green brightness by a half step.
`8`	Decrease green brightness by one step. Press `Shift+8` to decrease green brightness by a half step.
* (multiply sign)	Increase blue brightness by one step. Press `Shift+*` to increase blue brightness by a half step.

Press:	To:
9	Decrease blue brightness by one step. Press Shift+9 to decrease blue brightness by a half step.
4	Increase cyan brightness by one step. Press Shift+4 to increase cyan brightness by a half step.
1	Decrease cyan brightness by one step. Press Shift+1 to decrease cyan brightness by a half step.
5	Increase magenta brightness by one step. Press Shift+5 to increase magenta brightness by a half step.
2	Decrease magenta brightness by one step. Press Shift+2 to decrease magenta brightness by a half step.
6	Increase yellow brightness by one step. Press Shift+6 to increase yellow brightness by a half step.
3	Decrease yellow brightness by one step. Press Shift+3 to decrease yellow brightness by a half step.
– (minus sign)	Reset overall brightness.

The printer light settings for the R, G, and B values are updated to reflect your changes.



NOTE Printer light settings do not include any modifications you make to brightness in the shadows, midtones, or highlights. See Modifying Brightness in the Shadows, Midtones, and Highlights (page 2062).

TIP When you click on the printer lights, the printer light values are replaced by values based on the center point of the colour wheel. If you then right-click one of the colour wheel values, the calculator appears in the UI, allowing you to enter a value.

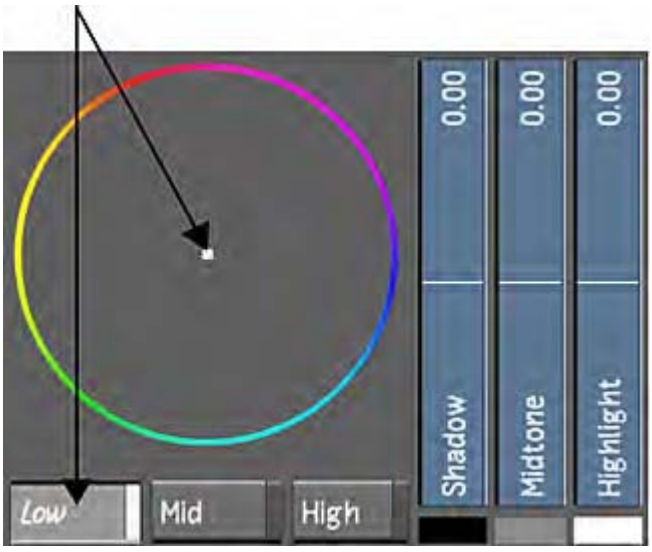
Modifying Brightness in the Shadows, Midtones, and Highlights

You can increase or decrease the brightness in the shadows, midtones, or highlights. Use the Shadow, Midtone, or Highlight slider to modify the brightness of the red, green, and blue levels uniformly. Use the Low, Mid, or High Balance wheel to balance colours by modifying RGB levels proportionally.

NOTE When you modify brightness in the shadows, midtones, or highlights, your changes are not reflected in the printer light settings.

To modify the brightness in the shadows, midtones, and highlights:

1. Indicate whether this is the initial or final primary grade:
   - To work on the initial primary colour grade, click Input.
   - To make final adjustments to the primary colour grade, click Output.
2. Adjust the overall brightness:
   - To adjust the overall brightness in the shadows, drag the Shadow slider. Drag up to increase brightness. Drag down to decrease brightness.
   - To adjust the overall brightness in the midtones, drag the Midtone slider. Drag up to increase brightness. Drag down to decrease brightness.
   - To adjust the overall brightness in the highlights, drag the Highlight slider. Drag up to increase brightness. Drag down to decrease brightness.
3. Balance the brightness of colours:
   - To balance the brightness of the colours in the highlights, enable High and then drag the High Balance wheel toward the colour you want to increase in the image.
   - To balance the brightness of the colours in the midtones, enable Mid and then drag the Mid Balance wheel toward the colour you want to increase in the image.
   - To balance the brightness of the colours in the shadows, enable Low and then drag the Low Balance wheel toward the colour you want to increase in the image.



Resetting Overall Brightness

You can reset the image brightness using the keypad hotkey.

To reset Brightness settings:

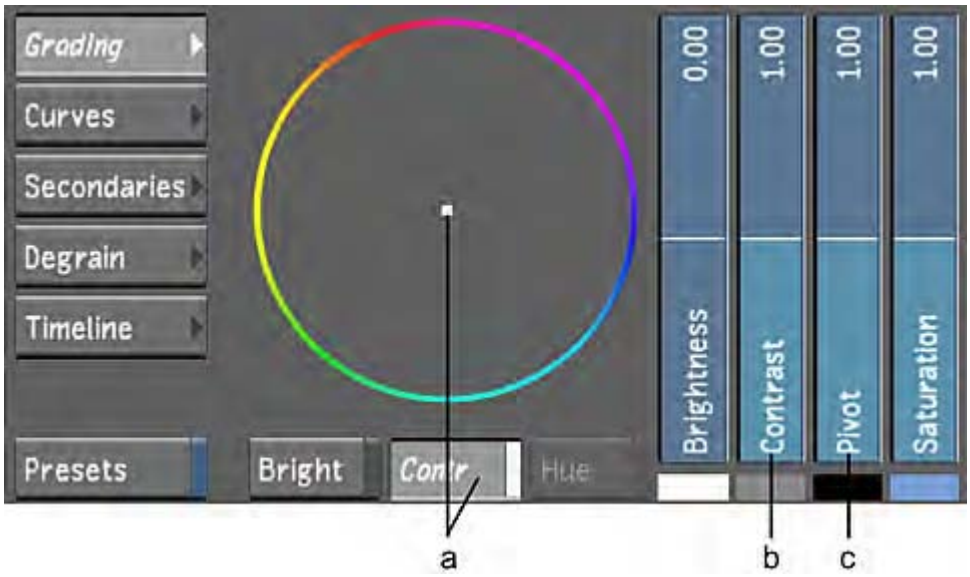
1. Press – (minus sign) on the keypad.

The Brightness slider and Brightness Balance wheel are reset to their default settings. Printer light settings are also reset and the image reverts to its original printer light settings.

NOTE Modifications you made to the image using the Brightness controls for shadows, midtones, and highlights are not reset to their default values. See Resetting Parameters (page 1804).

Logarithmic Mode: Improving Contrast

Poor image contrast can be caused by various factors. For example, when you remove a colour cast from a shot, the shot may become washed out. Washed out images usually occur when image contrast and saturation levels are too low. To complete the task of balancing the colours in the shot, use the Contrast controls to correct poor contrast. When you correct contrast, you can also set the pivot point and saturation levels. See Setting the Pivot Point (page 2065) and Modifying Saturation (page 2066).



(a) Contrast Balance wheel and button (b) Contrast slider (c) Pivot slider

NOTE When you use a slider or colour balance wheel to colour grade a shot, the colour range is limited by the control (the edge of the colour wheel, for example). Use the Autodesk control surface or the Tangent CP100 control surface to reach the full spectrum of colour. You can also use the standard Lustre calculator to extend beyond the UI colour range. To access the calculator, click on the printer lights and then right-click one of the colour wheel values.

Modifying Image Contrast

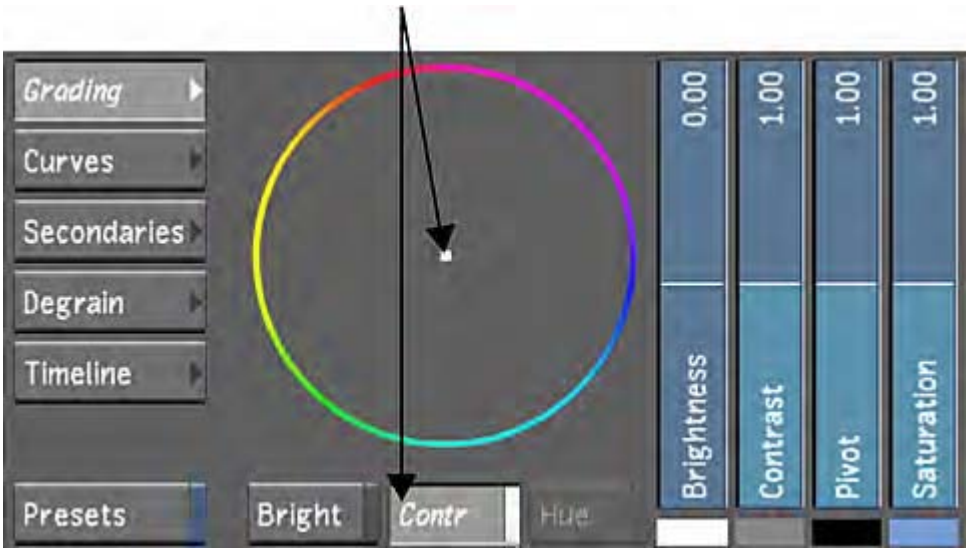
Improve image contrast by modifying the film gamma with the Contrast slider. Use the Contrast Balance wheel to increase the contrast of any channel while decreasing the contrast of other channels. These changes in contrast are relative to the pivot point.

NOTE To see how balancing the contrast works, view the histogram. Press `Alt+2` while modifying the contrast balance and the pivot point.

WARNING The Histogram, Waveform, and Vectorscope functions do not update when GPU processing is enabled.

To improve image contrast:

1. Indicate whether this is the initial or final primary grade:
   - To work on the initial primary colour grade, click Input.
   - To make final adjustments to the primary colour grade, click Output.
2. Modify the contrast:
   - To increase the contrast in the image, drag the Contrast slider up.
   - To decrease the contrast in the image, drag the Contrast slider down.
3. To increase contrast in a range of colour in the image, enable Contrast and then drag the Contrast Balance wheel towards the colour.



The colour you drag toward increases in contrast while the opposing colours decrease in contrast. The overall image contrast remains the same.

NOTE The pivot point is the point around which the contrast balanced. See Setting the Pivot Point (page 2065).

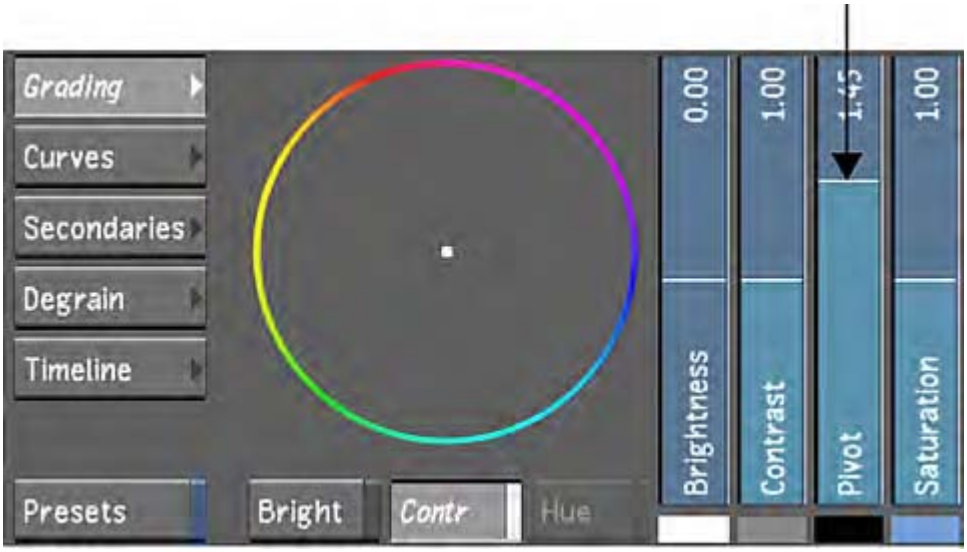
4. Make adjustments to the image using the Contrast slider and Contrast Balance wheel until you are satisfied with the results.

Setting the Pivot Point

You can set the pivot point for contrast. The pivot point is the anchor around which contrast is scaled. By default, the pivot point is set to the midpoint in the image. You can move the pivot point to scale the contrast around the image highlights or shadows.

To set the pivot point:

1. Indicate whether this is the initial or final primary grade:
   - To work on the initial primary colour grade, click Input.
   - To make final adjustments to the primary colour grade, click Output.
2. Drag the Pivot slider to move the anchor around which the contrast is modified. Drag up to modify contrast around highlights. Drag down to modify contrast around shadows.

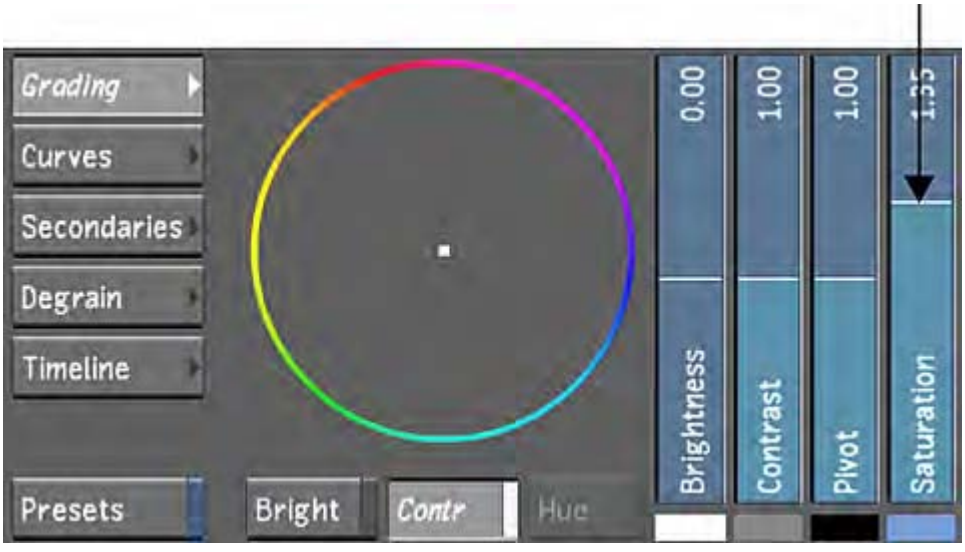


Modifying Saturation

Control the image's colour intensity by modifying saturation levels.

To modify the saturation level:

1. Indicate whether this is the initial or final primary grade:
   - To work on the initial primary colour grade, click Input.
   - To make final adjustments to the primary colour grade, click Output.
2. Modify the saturation:
   - To increase the saturation in the image, drag the Saturation slider up.
   - To decrease the saturation in the image, drag the Saturation slider down.



Linear Mode: Selecting Levels of the Image

You can modify lift, gain, gamma, and saturation in the low, mid, and high levels of the image, or across the entire image. Select the range you want to modify while performing primary colour grading. You can also define values to clamp minimum and maximum luminance values.

NOTE You can use the Tangent CP100 control surface when you want to colour grade the entire image, not an individual level. You can use the Autodesk control surface for either the entire image or an individual level.

To select a level of the image for colour grading:

1. Indicate whether this is the initial or final primary grade:
   - To work on the initial primary colour grade, click Input.
   - To make final adjustments to the primary colour grade, click Output.
2. Enable the appropriate button to select the entire image or a level of the image for primary colour grading.



Enable:	To perform primary colour grading on:
Master	The entire image.
Low	The low levels of the image.
Mid	The mid levels of the image.
High	The high levels of the image.

Linear Mode: Modifying Lift

You can lift colour values in a selected range or in the entire image, and then offset ranges of colour. Changes in lift mostly affect black level. Although lift affects the entire image or image level, visually, it mostly affects the shadows.

NOTE When you use a slider or colour balance wheel to colour grade a shot, the colour range is limited by the control (the edge of the colour wheel, for example). Use the Autodesk control surface or the Tangent CP100 control surface to reach the full spectrum of colour.

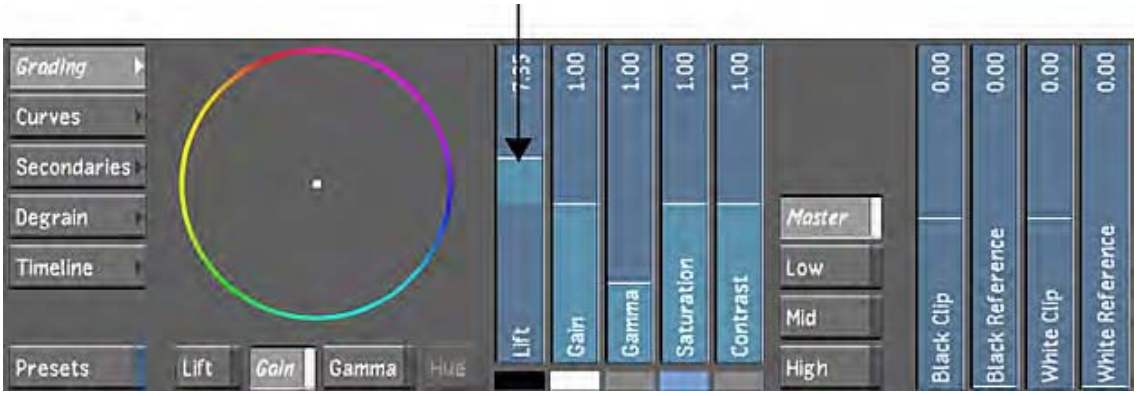
To lift colour values:

1 Indicate whether this is the initial or final primary grade:
- To work on the initial primary colour grade, click Input.
- To make final adjustments to the primary colour grade, click Output.

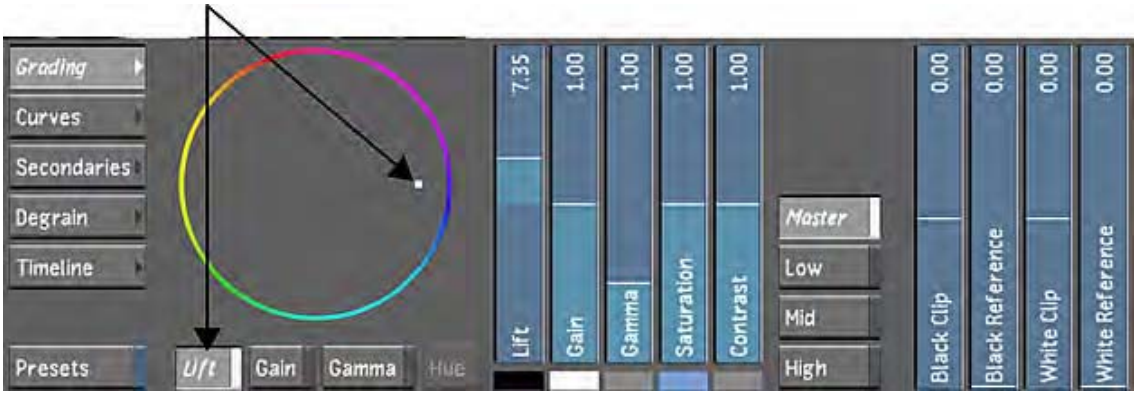
2 Select the image level you want to modify.



3 Modify the black levels:
- To increase the black level, drag the Lift slider up.
- To decrease the black level, drag the Lift slider down.



4 To offset the colours while maintaining the overall luminance, enable Lift and then drag the Lift Balance wheel towards a colour.



The selected colour is lifted while other colours in the image are offset to compensate for the change in luminance.

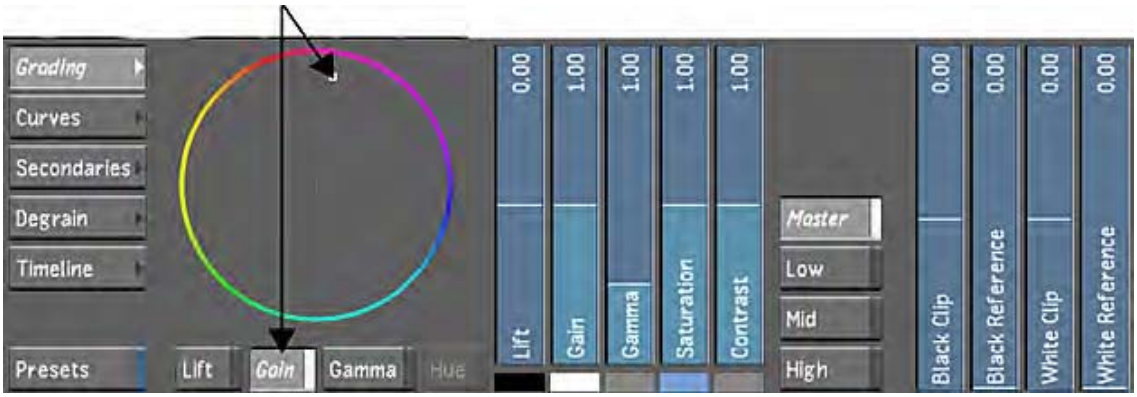
Linear Mode: Modifying Gain

You can boost image colours in the highlights of the selected range or entire image. Changes in gain mostly affect the highlights and do not change the black point of the image. Gain is similar to brightness in Log mode.

NOTE When you use a slider or colour balance wheel to colour grade a shot, the colour range is limited by the control (the edge of the colour wheel, for example). Use the Autodesk control surface or the Tangent CP100 control surface to reach the full spectrum of colour.

To modify gain:

1. Indicate whether this is the initial or final primary grade:
   - To work on the initial primary colour grade, click Input.
   - To make final adjustments to the primary colour grade, click Output.
2. Select the image level you want to modify.
3. Modify the image colours:
   - To increase image colours uniformly, drag the Gain slider up.
   - To decrease image colours uniformly, drag the Gain slider down.
4. To offset the colours while maintaining the overall gain, enable Gain and then drag the Gain Balance wheel towards a colour you want to boost.



The selected colour is boosted while other colours in the image are offset to compensate for the change in gain.

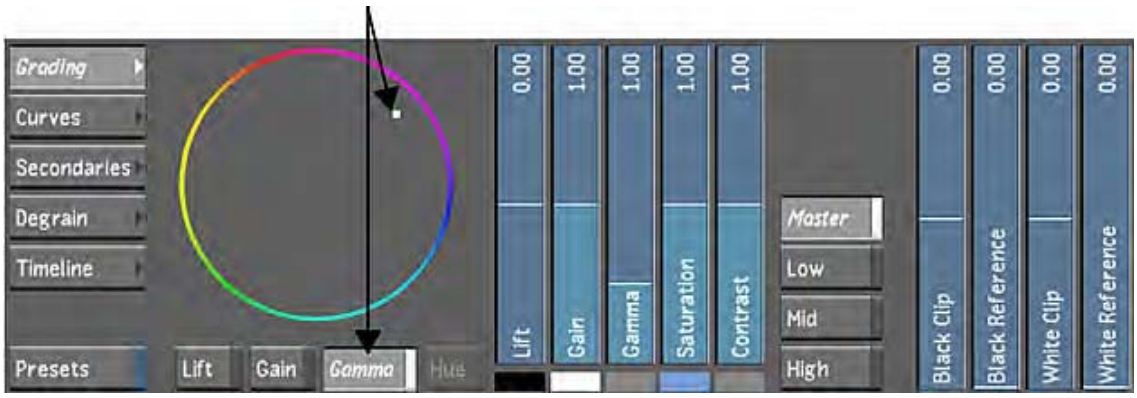
Linear Mode: Modifying Gamma

You can adjust the grey values of an image by adjusting gamma. This allows you to brighten or darken the image and modify the gamma level without greatly affecting the shadows and highlights.

NOTE When you use a slider or colour balance wheel to colour grade a shot, the colour range is limited by the control (the edge of the colour wheel, for example). Use the Autodesk control surface or the Tangent CP100 control surface to reach the full spectrum of colour.

To modify gamma:

1. Indicate whether this is the initial or final primary grade:
   - To work on the initial primary colour grade, click Input.
   - To make final adjustments to the primary colour grade, click Output.
2. Select the range you want to modify.
3. Do one of the following:
   - To decrease contrast, drag the Gamma slider up.
   - To increase contrast, drag the Gamma slider down.
4. To balance the colours while maintaining the overall gamma level, enable Gamma and then drag the Gamma Balance wheel towards a colour.



The amount of the selected colour is increased in the image while the overall gamma level is maintained. The middle of the image curve is the most affected.

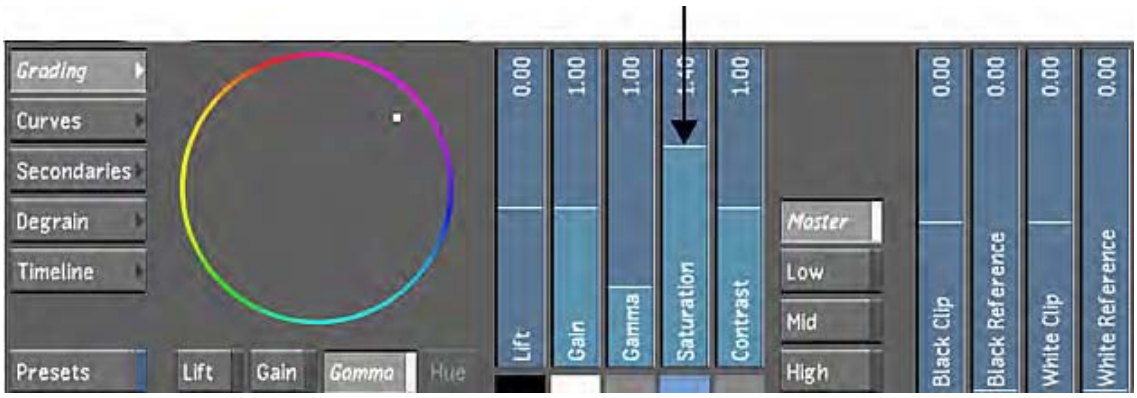
NOTE Select Low, Mid, or High to further differentiate between the areas of the image that are affected.

Linear Mode: Modifying Saturation

Control the image's colour intensity by modifying the saturation levels.

To modify image saturation:

1. Indicate whether this is the initial or final primary grade:
   - To work on the initial primary colour grade, click Input.
   - To make final adjustments to the primary colour grade, click Output.
2. Modify the image saturation:
   - To increase image saturation, drag the Saturation slider up.
   - To decrease image saturation, drag the Saturation slider down.

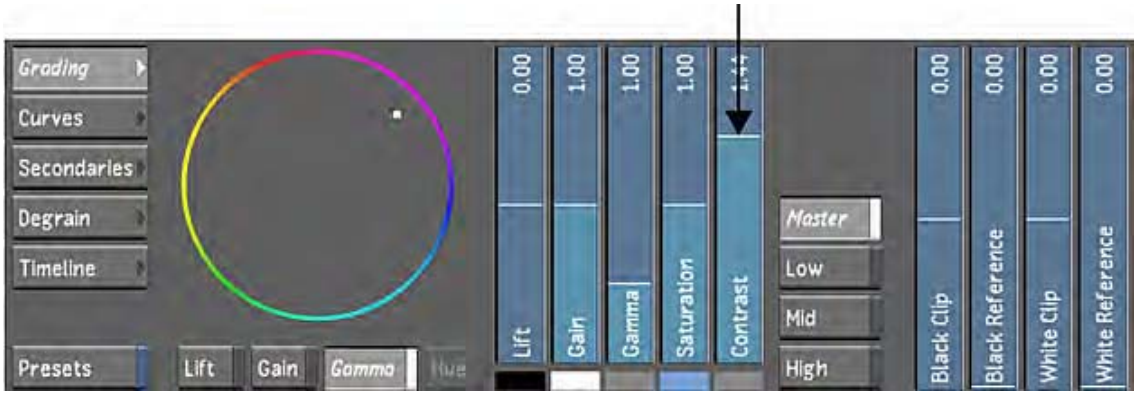


Linear Mode: Modifying Contrast

As in Log mode, you can complete the task of balancing the shot's colours by modifying the contrast. With the Contrast slider, you can change the image's Linear luminance curve to a film-contrast-like S curve.

To modify image contrast:

1. Indicate whether this is the initial or final primary grade:
   - To work on the initial primary colour grade, click Input.
   - To make final adjustments to the primary colour grade, click Output.
2. Modify the image contrast:
   - To increase image contrast, drag the Contrast slider up.
   - To decrease image contrast, drag the Contrast slider down.



Linear Mode: Setting Black Clip and White Clip

You can define values to clamp minimum and maximum luminance values using the clip and reference sliders.

To clamp minimum and maximum luminance values:

1. Indicate whether this is the initial or final primary grade:
   - To work on the initial primary colour grade, click Input.

■ To make final adjustments to the primary colour grade, click Output.

2 Modify the clip and reference values using the following sliders.

Black Clip Determines the extent to which values will be smoothed out in the shadows.

Black Reference Defines the range of values in the shadows where the Black Clip will be effective.

White Clip Determines the extent to which values will be smoothed out in the highlights.

White Reference Defines the range of values in the highlights where the White Clip will be effective.



Matching Colours

Use the Match feature to match a sample from one shot to other shots to improve continuity between shots in a scene. You can match highlights, shadows, or overall saturation. You can also use Match when you want to match colours such as the colour of the sky or skin tone.

To match colours:

1 In the Grading menu, click Input.

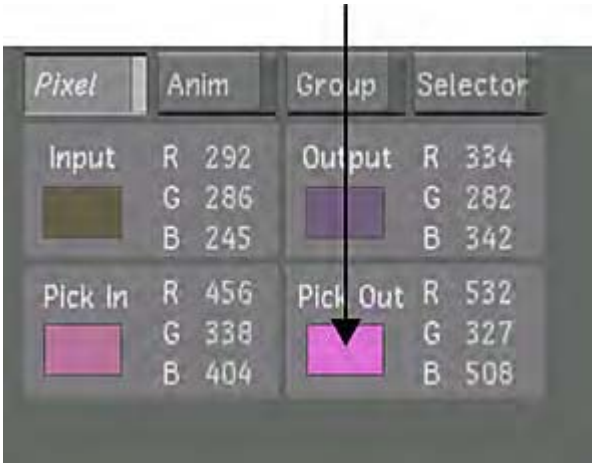
2 Show the clips you want to match in the Player. Press `F3` to display Multi View and then set the number of viewers. See Setting the Viewing Options (page 1984).

3 Sample the image you want to use as reference. This is the shot to which other clips are matched. Sample an area that is predominantly black, white, or contains a wide spectrum of colours.

You can:	To:
Click	Sample a single pixel in the image. You can drag through the image until you locate the pixel you want to sample, then click.
`Shift`-drag	Sample an average taken from a range of colours in the image.

The sample appears in the Pixel Analyser.

NOTE You can only sample the original image in the viewer. You cannot sample images that are loaded to the frame buffer. In order for sampling to function, geometries cannot be displayed. Be sure to disable the Show button in the Secondaries menu.



4 Sample the colours in the shot you want to match. Try to match whites with whites, blacks with blacks, or select a broad range from both images to match saturation.

You can:	To:
`Alt`-click	Sample an individual pixel and set the Brightness Balance wheel to match the reference sample. You can drag through the image until you locate the pixel you want to sample, then `Alt`-click.
`Shift+Alt`-drag	Sample an average taken from a range of colours in the image and set the Brightness Balance wheel to match the reference sample.
`Ctrl`-click	Sample an individual pixel and set the Brightness Balance wheel and Brightness slider to match the reference sample. You can drag through the image until you locate the pixel you want to sample, then `Ctrl`-click.
`Shift+Ctrl`-drag	Sample an average taken from a range of colours in the image and set the Brightness Balance wheel and Brightness slider to match the reference sample.

The sampled colours in the second shot are matched to the reference image.

5 Continue matching other shots to the reference image.

6 To refine the result, use other controls in the Grading menu and then reuse the Match feature.

Autograding Shots

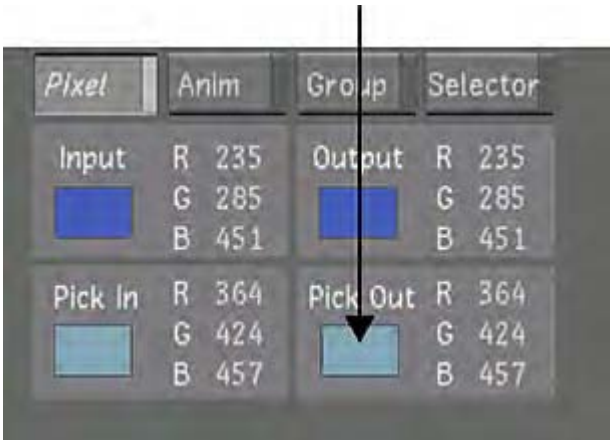
From the Grading menu, you can automatically colour grade a shot based on sampled pixels. Autograding balances the colours according to the Brightness Balance wheel—the sampled pixels are changed to a more neutral colour. For example, if you sample an area that contains more blue than any other colour, yellow (the colour that is opposite on the colour wheel) is added to the image. To properly autograde a shot, sample an area of the image that should be neutral or have no hue cast. Sample blacks, whites, or greys.

To autograde a shot:

1 Sample an area of the image that should be neutral.

You can:	To:
Click	Sample a single pixel in the image. You can drag through the image until you locate the pixel you want to sample, then click.
`Shift`-drag	Sample an average taken from a range of colours in the image.

The sample appears in the Pixel Analyser.



2 Press / (forward slash).

The entire image is autograded according to the sample. The colours are rebalanced as shown on the Brightness Balance wheel.

Colour Grading RGB and Hue Curves

When you want to colour grade a specific range of colour without generating a key, you can modify the RGB and Hue curves. The RGB curves are inserted in the processing pipeline in two places—after the initial primary colour grade and after the addition of any Sparks effects. Sparks are inserted after secondary colour grading. The Hue curves are inserted in the processing pipeline in one place—after the initial primary colour grade. RGB and Hue curves can also be applied to secondary colour grades.

The RGB curves consist of red, green, blue, and RGB curves. Modify these curves when you want to remap colour values for the red, green, and blue channels individually or together. The Hue curves consist of Hue, Lightness, Saturation, and Luminance curves. Modify these curves when you want to perform hue shifts, lighten colours in the image, and suppress or saturate a particular range of colour or luminance area.

You can also plot colours and add vertices to the curves so that you can remap any value in the colour range precisely.

For information on secondary colour grading, see Secondary Colour Grading (page 2083). The Luminance curve can only be modified on the Curves menu.

TIP For faster playback on shots with input RGB and Hue curve modifications, enable GPU processing by clicking the GPU button. See GPU Acceleration (page 2043).

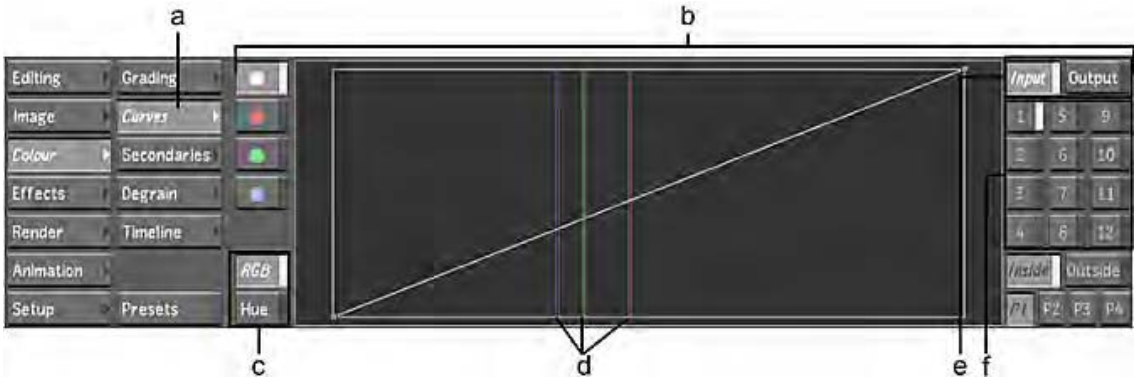
Accessing the Curves Menu

Use the Curves menu to modify RGB and Hue curves.

To access the Curves menu:

1 Do one of the following:
- In the Main menu, click Colour and then Curves.
- Press F7. See Displaying Colour Menus in the Player (page 2042).

The Curves menu appears.



(a) Curves button (b) Curves menu (c) RGB and Hue curve buttons (d) Sampled colour values (e) Input and Output buttons (f) Secondary layers

2 Enable one of the following buttons.

Input Enables RGB and Hue curve modification for primary colour grading performed prior to secondary colour grading.

Output Enables RGB curve modification for primary colour grading performed after secondary colour grading and the creation of Sparks effects.

Secondary layer Enables a layer for RGB and Hue curve modification for secondary colour grading.

NOTE Hue curves are not functional when the Output button is enabled.

Plotting Colour Values when Adjusting Curves

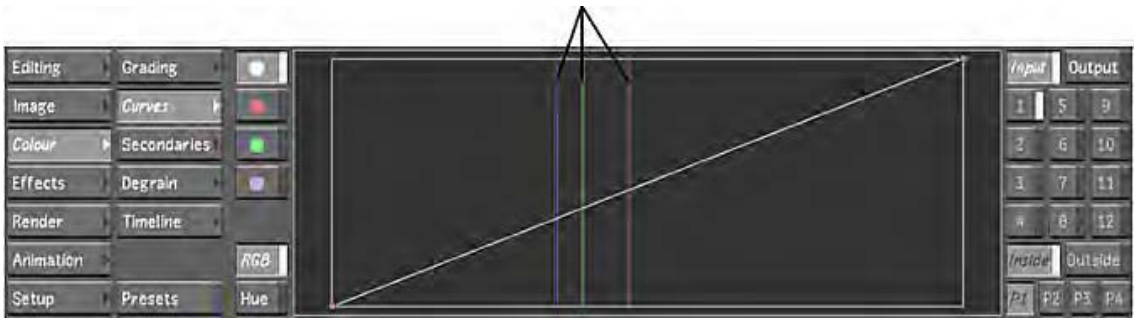
Plot the colour value of any sample on RGB and Hue curves. While you make adjustments, you can plot colours as often as necessary to isolate the colours you want to modify.

To plot colour values when adjusting curves:

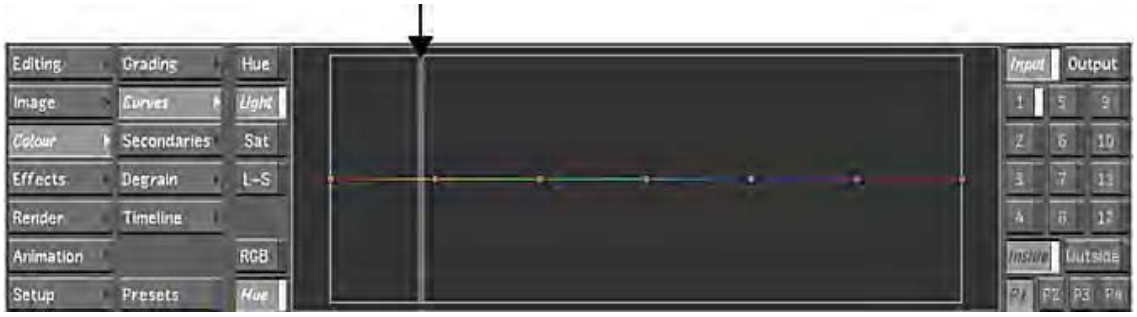
1 Indicate the colour grading stage at which the curve modification is being applied:
- To work on the initial primary grade, click Input.
- To make final adjustments to the primary grade, click Output.
- To make curve modifications for secondary colour grading, enable a secondary layer.

2 In the Player, sample the colour you want to use as reference. Click the image to sample a single pixel. You can drag through the image until you locate the pixel you want to sample, and then click.

On the RGB curve, red, green, and blue vertical lines appear, representing the red, green, and blue colour channels.



On the Hue curves, a single vertical line appears, representing the colour on the hue spectrum.



3 To zoom in on a curve using rectangle zoom, drag a selection box around the desired area while pressing `Alt`, and then release the mouse.

   The area of the curves contained by the selection box now occupies the entire Curve canvas.

4 To zoom in on a curve using proportional zoom, press `Alt` while dragging right or left with the right mouse button.

   The aspect ratio of the Curve canvas is maintained while zooming in and out.

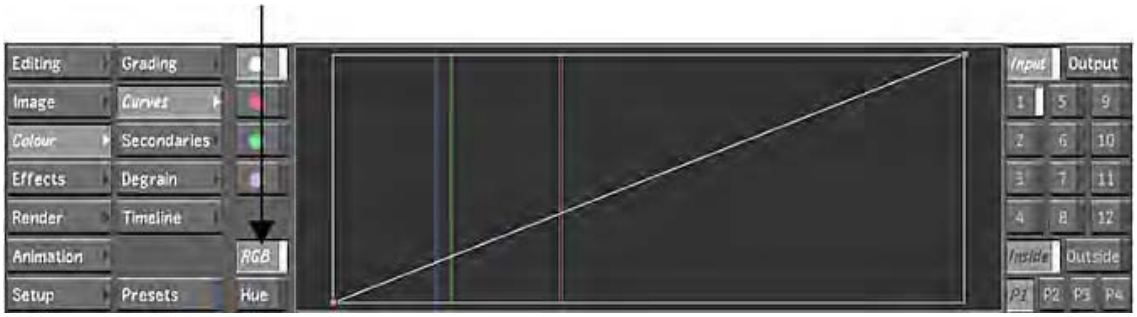
Modifying Red, Green, and Blue Channels

Modify the RGB, red, green, or blue curves to fine-tune the colour grade. When you modify the RGB curve, the red, green, and blue channels are adjusted equally. When you modify the red, green, or blue curve, only the selected colour channel is adjusted. You can add vertices to a curve to refine your adjustments.

To modify the red, green, blue, or RGB curve:

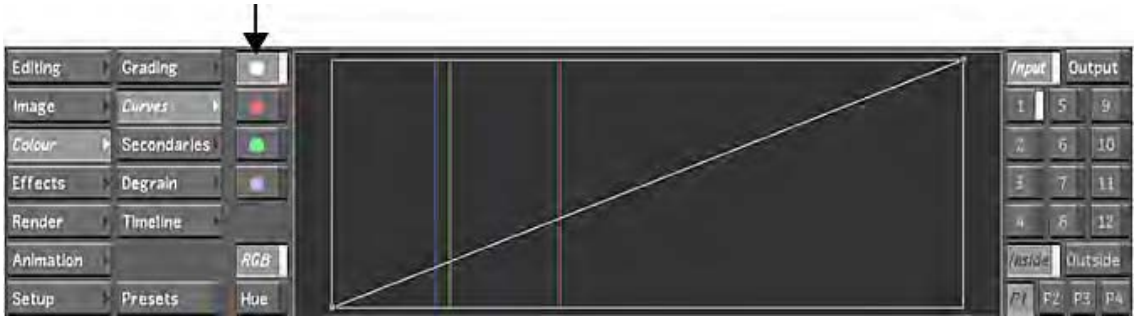
1 Indicate the colour grading stage at which the curve modification is being applied:
   - To work on the initial primary grade, click Input.
   - To make final adjustments to the primary grade, click Output.
   - To make curve modifications for secondary colour grading, enable a secondary layer.

2 Click RGB to display the RGB curves.



3 Show the curve you want to modify by clicking the button corresponding to the colour of the curve. The selected curve is highlighted.

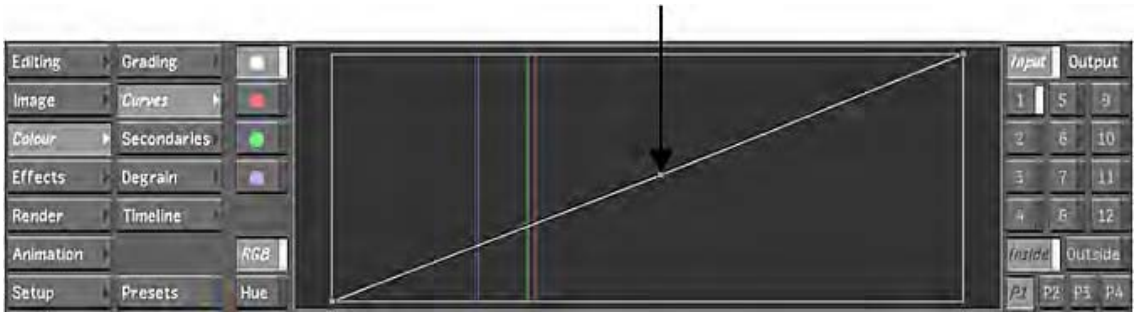
NOTE To display the RGB curve, click the white button.



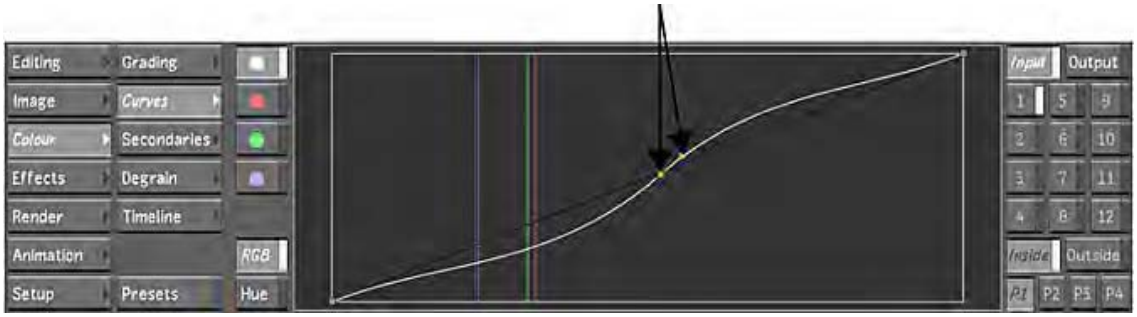
4 Sample the colour you want to use as a reference for your adjustments. See Plotting Colour Values when Adjusting Curves (page 2075).

Red, green, and blue vertical lines representing the colour channels are plotted on the curve.

5 To add a vertex to the curve, place the mouse cursor over the area on the curve where you want to add the vertex and press `A`. To lock the curve in place while adding a vertex, press `Shift+A` on the curve.



6 Click a vertex to display its tangent handle and then drag the handle to adjust the curve. You can also adjust the curve by dragging the vertex.



7 Modify the curve until you are satisfied with the results:
   - To select a vertex, click it. To select several vertices, draw a selection box around them.
   - To move one or several selected vertices, drag a selected vertex. To restrict vertex movement to the Y axis, press `Shift` while moving the mouse. To restrict vertex movement to the X axis, press `Shift+Alt` while moving the mouse.
   - To delete vertices, select them and then press `D`.

Modifying Hue, Saturation, Luminance, and Lightness

Modify curves to adjust the hue, saturation, luminance, and lightness of an image. You can:

- Perform hue shifts.
- Suppress or saturate colour ranges.
- Suppress or saturate luminance ranges.
- Lighten or darken colour ranges.
- Add vertices to the curves to refine your adjustments.

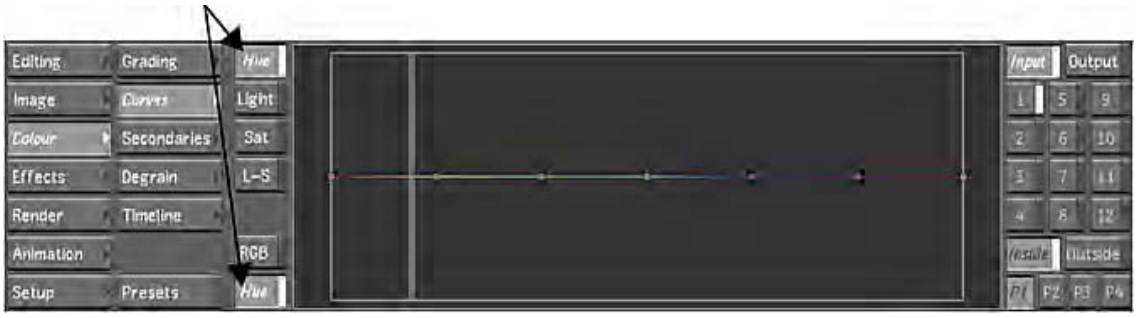
Shifting Hue

Shift the hue of the entire image or a range of colour in the image.

To perform a hue shift:

1 Indicate the colour grading stage at which the curve modification is being applied:
   - To work on the initial primary grade, click Input.
   - To make curve modifications for secondary colour grading, enable a secondary layer.

2 Click the Hue button located below the RGB button, and then click the upper Hue button to display the Hue curve.



The Hue curve is plotted to the Hue spectrum.

3 Sample the colour you want to use as a reference for your adjustments. See Plotting Colour Values when Adjusting Curves (page 2075).

A vertical line representing the reference colour is plotted on the Hue curve.

4 Drag vertices up or down to shift the hue of the colour displayed in the hue spectrum.



The hue of the selected colours moves through the hue spectrum as you drag.

TIP You can also select a vertex to use tangent handles to adjust the curve.

5 Modify the curve until you are satisfied with the results:

- To select a vertex, click it. To select several vertices, draw a selection box around them.
- To move several selected vertices, select them and then drag one. To restrict vertex movement to the Y axis, press `Shift` while moving the mouse. To restrict vertex movement to the X axis, press `Shift+Alt`.
- To add a vertex to the curve, place the mouse cursor over the area on the curve where you want to add the vertex and then press `A`.
- To lock the curve in place while adding a vertex, press `Shift+A` on the curve.
- To delete vertices, select them and then press `D`.

Saturating and Suppressing Colours

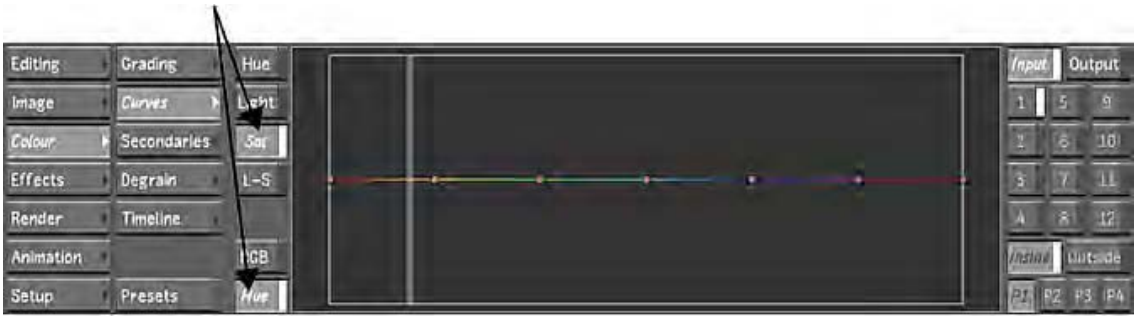
You can adjust the purity of colours in the image by modifying the saturation. You can suppress or saturate colours in a range or across the entire image.

To saturate and suppress colours in the image:

1 Indicate the colour grading stage at which the curve modification is being applied:

- To work on the initial primary grade, click Input.
- To make curve modifications for secondary colour grading, enable a secondary layer.

2 Click Hue and then click Sat to display the Saturation curve.



The Saturation curve is plotted to the hue spectrum.

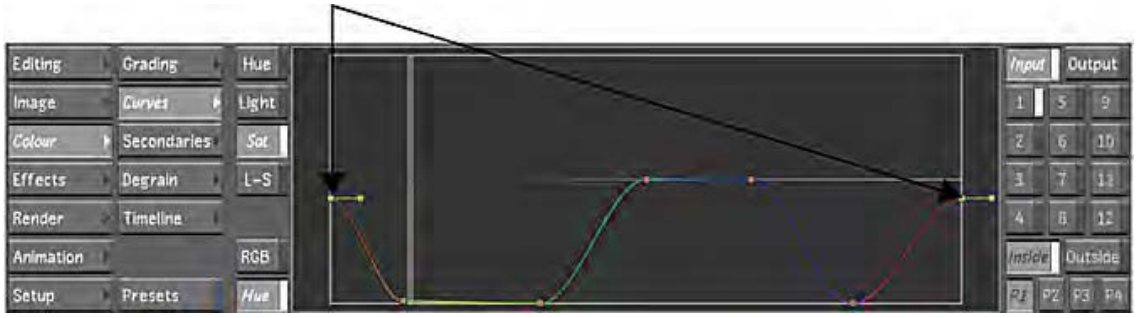
3 Sample the colours you want to use as a reference for your adjustments. See Plotting Colour Values when Adjusting Curves (page 2075).

A vertical line representing the reference colour is plotted on the Saturation curve.

4 To select a vertex, click it. To select several vertices, draw a selection box around them.

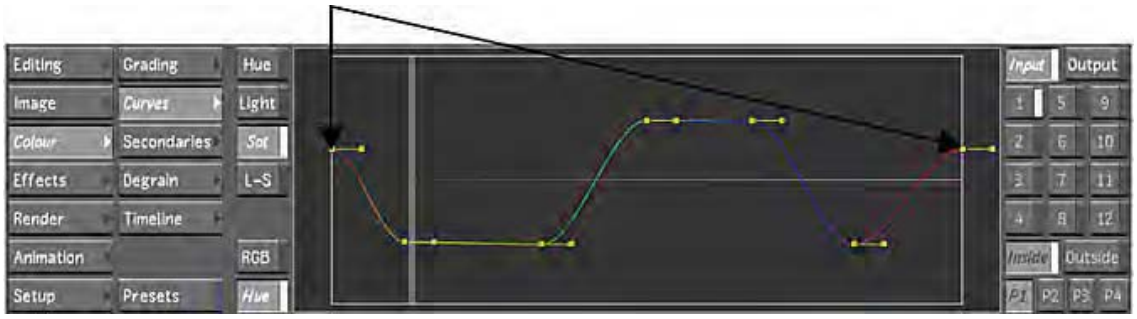
5 Saturate or suppress the colour:

- To saturate the colour displayed in the hue spectrum, drag selected vertices up.
- To suppress the colour displayed in the hue spectrum, drag selected vertices down.



TIP You can also select a vertex to use tangent handles to adjust the curve.

6 To move several selected vertices, select them and then drag one. To restrict vertex movement to the Y axis, press `Shift` while moving the mouse. To restrict vertex movement to the X axis, press `Shift+Alt`.



TIP To create a greyscale image, select all the vertices and then drag them all the way down. You can then drag vertices up to saturate isolated ranges of colour.

7 To add a vertex to the curve, place the mouse cursor over the area on the curve where you want to add the vertex and then press `A`. To lock the curve in place while adding a vertex, press `Shift+A` on the curve.

8 To delete vertices, select them and then press D.

9 Edit the curve until you are satisfied with the results.

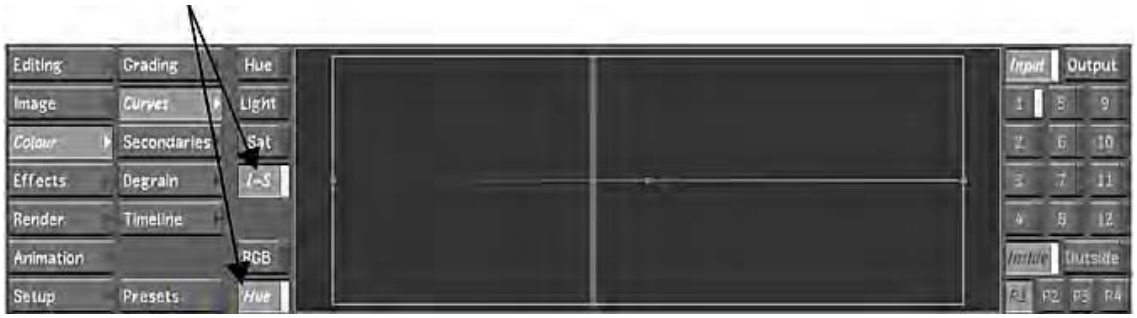
Modifying Image Luminance

The Luminance curve represents the gamut of light from black to white. Use the Luminance curve to saturate or desaturate different luminance areas by adding or removing hue from black and white points. Like any other curve, you can add points to refine ranges of an effect.

To suppress or saturate luminance areas in the image:

1 Indicate the colour grading stage at which the curve modification is being applied:
- To work on the initial primary grade, click Input.
- To make curve modifications for secondary colour grading, enable a secondary layer.

2 Click Hue and then click L-S to display the Luminance curve.



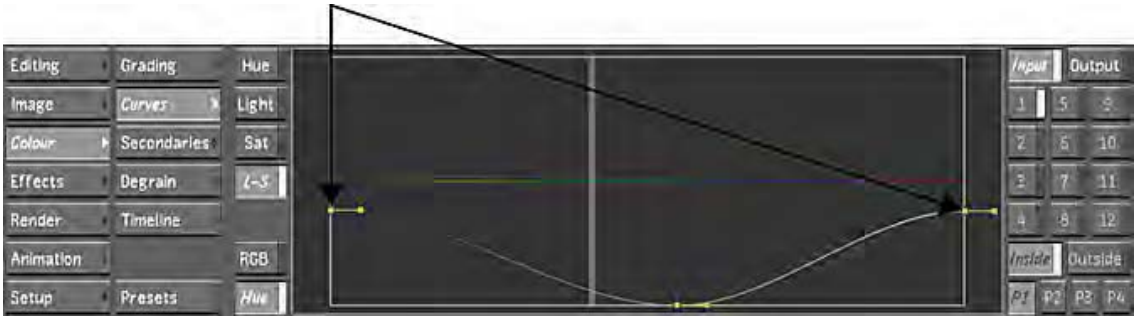
The Luminance curve is plotted to the hue spectrum.

3 Sample the luminance areas you want to use as a reference for your adjustments. See Plotting Colour Values when Adjusting Curves (page 2075).

A vertical line representing the reference colour is plotted on the Luminance curve.

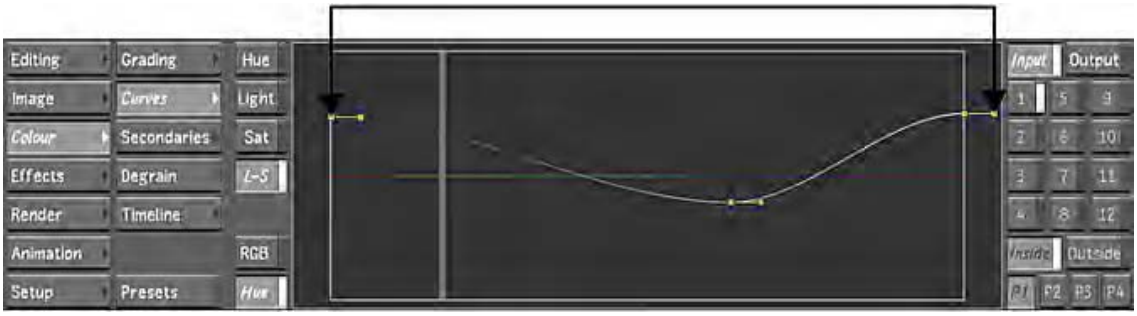
4 To select a vertex, click it. To select several vertices, draw a selection box around them.

5 Saturate or suppress the luminance area:
- To saturate the luminance area displayed in the hue spectrum, drag selected vertices up.
- To suppress the luminance area displayed in the hue spectrum, drag selected vertices down.



TIP You can also select a vertex to use tangent handles to adjust the curve.

6 To move several selected vertices, select them and then drag one. To restrict vertex movement to the Y axis, press Shift while moving the mouse. To restrict vertex movement to the X axis, press Shift+Alt.



7 To add a vertex to the curve, place the cursor over the area on the curve where you want to add the vertex and then press `A`. To lock the curve in place while adding a vertex, press `Shift+A` on the curve.

8 To delete vertices, select them and then press `D`.

9 Edit the curve until you are satisfied with the results.

Modifying Image Lightness

Increase or decrease the lightness in specific colour ranges or across the image.

To modify image lightness:

1 Indicate the colour grading stage at which the curve modification is being applied:
- To work on the initial primary grade, click Input.
- To make curve modifications for secondary colour grading, enable a secondary layer.

2 Click Hue and then click Light to display the Lightness curve.



The Lightness curve is plotted to the hue spectrum.

3 Sample the colours you want to use as a reference for your adjustments.
A vertical line representing the reference colour is plotted on the Lightness curve.

4 To select a vertex, click it. To select several vertices, draw a selection box around them.

5 Modify the lightness:
- To increase the lightness of the colour displayed in the hue spectrum, drag selected vertices up.
- To decrease the lightness of the colour displayed in the hue spectrum, drag selected vertices down.



TIP You can also select a vertex to use tangent handles to adjust the curve.

6. To move several selected vertices, select them and then drag one. To restrict vertex movement to the Y axis, press `Shift` while moving the mouse. To restrict vertex movement to the X axis, press `Shift+Alt`.
7. To add a vertex to the curve, place the cursor over the area on the curve where you want to add the vertex and then press `A`. To lock the curve in place while adding a vertex, press `Shift+A` on the curve.
8. To delete vertices, select them and then press `D`.
9. Edit the curve until you are satisfied with the results.

Secondary Colour Grading

In Lustre, you perform secondary colour grading on shots. With secondary colour grading, you isolate and then colour grade colour ranges and objects in your shots. Secondaries are defined by a combination of keys and geometries—keys to isolate colours and geometries to isolate objects or areas in the image. To colour grade an object that moves through time, you can animate the associated geometry using trackers.

NOTE Trackers are used for precise secondary animation; however, when a tracker is not required, keyframes are predominantly used to animate secondaries.

Secondary colour grading is done in the middle of the colour grading process—after the first modifications you make to RGB and Hue curves and before applying Sparks. In Log mode, you can modify brightness, contrast, and hue. In Linear mode, you can modify lift, gain, gamma, and hue.

NOTE If you create effects for your image using Sparks, they are processed after secondary colour grading. See Creating Lustre Sparks Effects (page 2156).

You can create and colour grade up to 48 unique secondary layers for each shot. However, you can toggle the secondary layers on and off and view the results in the Player. You can save key presets, geometry presets, and degrain presets. You can also import garbage mask setups, tracker data, and stabilizer data from other Autodesk products.

Secondaries are included in the processing pipeline. With the Geom button activated, you can use the colour grading controls in the Grading menu to perform secondary colour grading to the area defined by the key and geometry.

TIP GPU acceleration makes faster playback available for certain secondary grading features. See GPU Acceleration (page 2043).

NOTE The histogram, waveform, and vectorscope monitors do not dynamically update when GPU acceleration is enabled. They retain the colour distribution of the image prior to GPU acceleration being enabled. See Viewing Colour Distribution (page 2049).

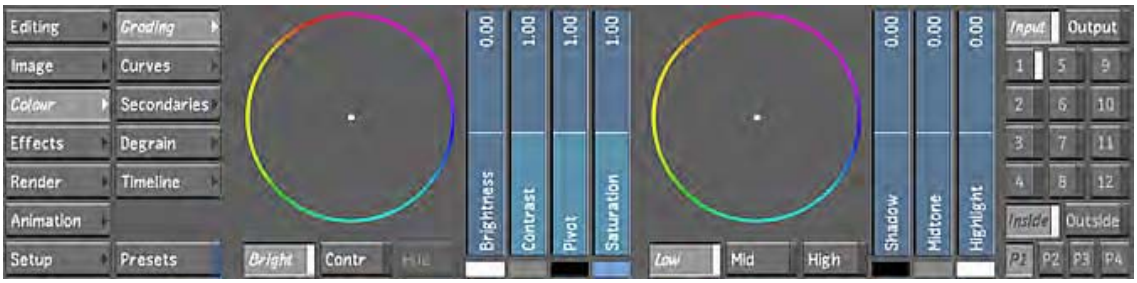
Accessing Secondary Grading Menus

You perform secondary colour grading with the Secondaries and Grading menus.

To access the menus for secondary colour grading:

1 Do one of the following:
- In the Main menu, click Colour.
- Press F7. See Displaying Colour Menus in the Player (page 2042).

The Colour menu appears.



2 Click one of the following buttons.

Secondaries Displays the Secondaries menu, where you have access to the geometry, keyer, presets, secondary layers, and tracker panels. See The Secondaries Menu (page 2084).

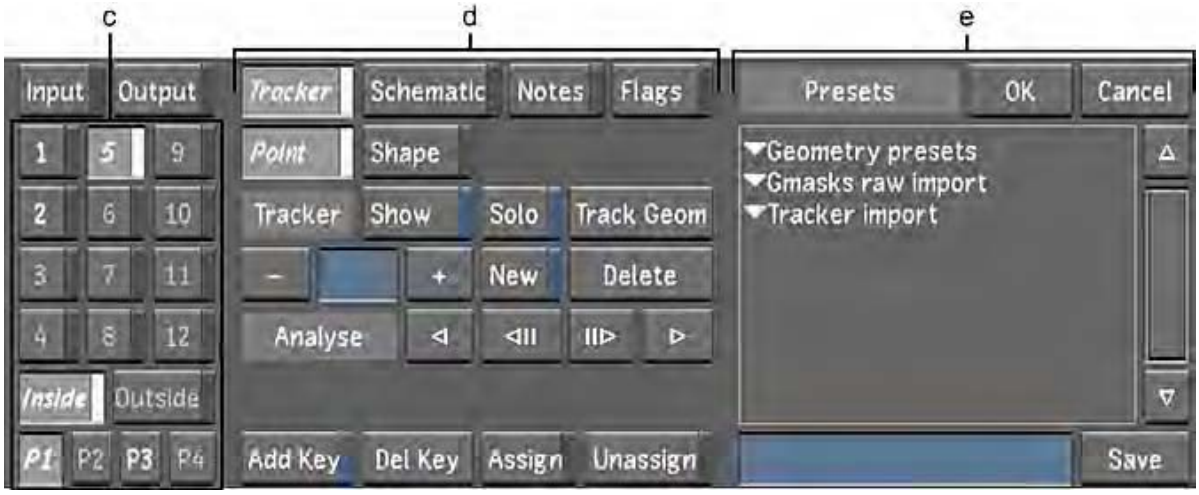
Grading Displays the Grading menu. Once geometries have been created, use the Grading menu for secondary colour grading. In Log mode, make adjustments to brightness, contrast, saturation, and hue. In Linear mode, make adjustments to lift, gain, hue, gamma, and saturation. Use logical operations to link geometries across secondaries. See Primary Colour Grading (page 2058).

NOTE You can also modify RGB and Hue curves for secondaries using the Curves menu. For information, see Colour Grading RGB and Hue Curves (page 2074).

The Secondaries Menu

The Secondaries menu allows you to create and track geometries to isolate objects or areas within the shot, and create keys to isolate ranges of colour. You are then able to use logical operations to link geometries across secondaries. The following is a breakdown of the Secondaries menu.





(a) Geometry panel (b) HLS and Diamond Keyer panel (c) Secondary Layers panel (d) Shape and Point Tracker panel (e) Presets panel

Geometry panel Creates geometries of various shapes in order to isolate objects or areas within a shot. Once a geometry has been graded, you can modify the softness, colour, opacity, and blur of the geometry to acquire a certain effect. See About Geometries (page 2091).

HLS and Diamond Keyer panel Extracts a key to isolate a colour within your shot. See Creating a Secondary by Keying a Range of Colours (page 2116).

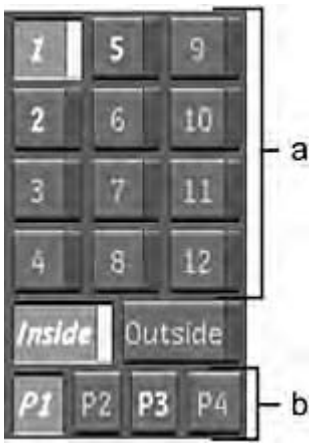
Secondary Layers panel Creates up to 48 layers of secondary grading for each shot. See Secondary Layers (page 2085).

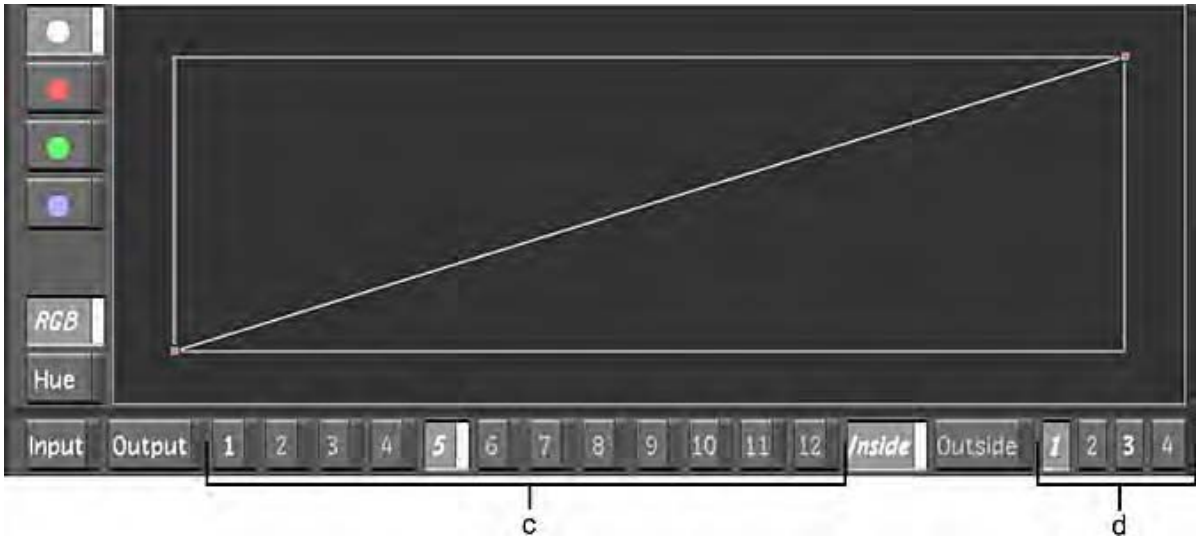
Shape and Point Tracker panel Tracks either objects or points throughout a shot. See Shape Tracker (page 2139) and Animating the Point Tracker (page 2144).

Presets panel Lists the previously saved geometry, gmask raw, tracker, and key presets that allows you to increase the efficiency and speed of the colour grading process. See Saving and Loading Presets Using the Presets Lists (page 2051).

Secondary Layers

The secondary layers panel allows you to create up to 48 secondaries to be applied to each shot in your project. Once a secondary layer is selected, you can choose whether or not to apply the grading changes to the shot.





(a) Secondary layers (b) Secondary layer pages (c) Secondary layers within the F7 mode (d) Secondary layer pages within the F7 mode

Secondary Layer buttons There are 48 secondary layers that can be used to contain your secondary grading information. When a layer is selected, you are able to modify the grading applied to that layer. When a layer is activated, the grading is applied to the shot and is visible within the Player. Only one layer can be selected for editing at one time, but you can activate up to 48 layers at one time. There are 12 secondary layers on each page and there are four pages of secondary layers.

Inside button When selected, the secondary grading is applied inside the geometry. This is the default. In order to view the grading, make sure the button is activated (i.e., the text is white). If not, right-click the button to activate it. See Colour Grading Inside and Outside Secondaries (page 2090).

Outside button When selected, the secondary grading is applied outside of the geometry. In order to view the grading, make sure the button is activated (i.e., the text is white). If not, right-click the button to activate it. See Colour Grading Inside and Outside Secondaries (page 2090).

Secondary Layer Page buttons When a page is selected, it displays the 12 secondary layers associated with that page (e.g. P1 displays layers 1-12, P2 displays layers 13-24, etc.). If a page number is highlighted but not selected, it lets you know that there are active secondary layers on that page.

Secondary Layer Hotkeys

The following is a list of the secondary layer hotkeys. Refer to the Autodesk Control Surface User GuideAdding Secondary Layers (page 2465) for a list of the control panel hotkeys. These hotkeys can be used wherever the secondary layer panel is visible on the user interface.

NOTE These hotkeys cannot be used with the secondary option within the Effects menu.

Press:	To:
`1 to 0, -, =`	Select a secondary layer (1-12, 13-24, 25-36, or 37-48).
`\`	Activate/deactivate the selected secondary layer.
`Ctrl+(1, 2, 3, or 4)`	Select the secondary layers page 1, 2, 3, or 4.

Press:	To:
Scroll Lock	Cycle through all the activated secondary layers.
Ctrl-click secondary layer button	Copy the grading information from the current menu (e.g., Grading, Curves, or Secondaries) to the selected secondary layer.
Shift-click secondary layer button	Copy all the colour grading information to the selected secondary layer.

Selecting and Activating Secondary Layers

When a secondary layer is selected, you can edit that layer. You can then toggle the activation of the secondary layer and view the results in the Player. Only one layer can be selected at a time for editing, but up to 48 layers can be activated and displayed within the Player.

To select and activate a secondary layer:

1. Click the layer number button. The grey bar on the side changes colour to show that a layer has been selected.

   NOTE Click on another layer number to select another layer to edit.

2. Right-click the selected layer number to activate it. The number on the secondary button becomes white and all the changes applied to that layer are now visible within the Player.

   To deactivate the layer, right-click the layer number button again. The number on the secondary button goes from white to grey to indicate that it is no longer activated. The layer is then bypassed in the processing pipeline.

   NOTE If you do not activate the layer, you can manipulate the geometry while keeping the colour correction hidden.



(a) Selected and activated secondary layer (b) Activated and unselected secondary layer (c) Current secondary layer page with activated layer(s) (d) Secondary layer page with activated layer(s)

Colour Grading Secondaries

Use the colour grading controls in the Grading menu to perform secondary colour grading on your shots. Once the layer has been selected in the Secondaries menu, switch to the Grading menu and perform secondary grading, either in Logarithmic or Linear mode, just as you would for a primary grade.



To perform secondary colour grading:

1 In the Secondaries menu, right-click a secondary layer to activate it.



2 Define the area to be selectively colour graded by creating a key, adding one or more geometries, or combining a key with one or more geometries. See About Geometries (page 2091) and Creating a Secondary by Keying a Range of Colours (page 2116).

If you are colour grading geometries, enable Geom. This ensures that the colour correction will be applied selectively, rather than to the whole image.



3 Click Grading to display the Grading menu.

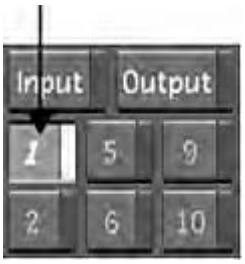
4 Perform colour grading as you would for a primary grade. See Primary Colour Grading (page 2058).

Shifting Hue

You can shift the hue of the area defined by a secondary using the Hue Rotator.

To shift the hue:

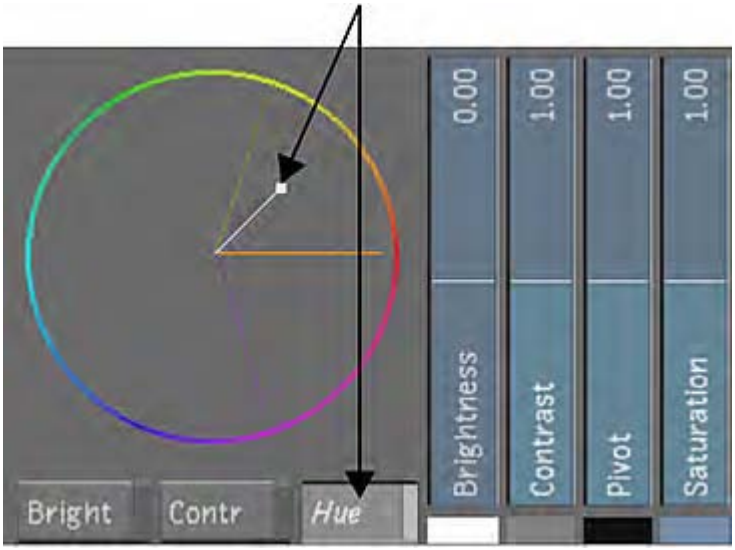
1. In the Secondaries menu, right-click a secondary layer to activate it.



2. Define the area to be selectively colour graded by creating a key, adding one or more geometries, or combining a key with one or more geometries.
3. If you are colour grading geometries, enable Geom. This ensures that the colour correction will be applied selectively, rather than to the whole image.



4. Click Grading to display the Grading menu.
5. Click Hue and then drag the Hue Rotator to shift the hue. To shift the hue clockwise through the hue spectrum, drag right. To shift the hue counter-clockwise, drag left.



Adjusting Printer Lights for Secondary Grading

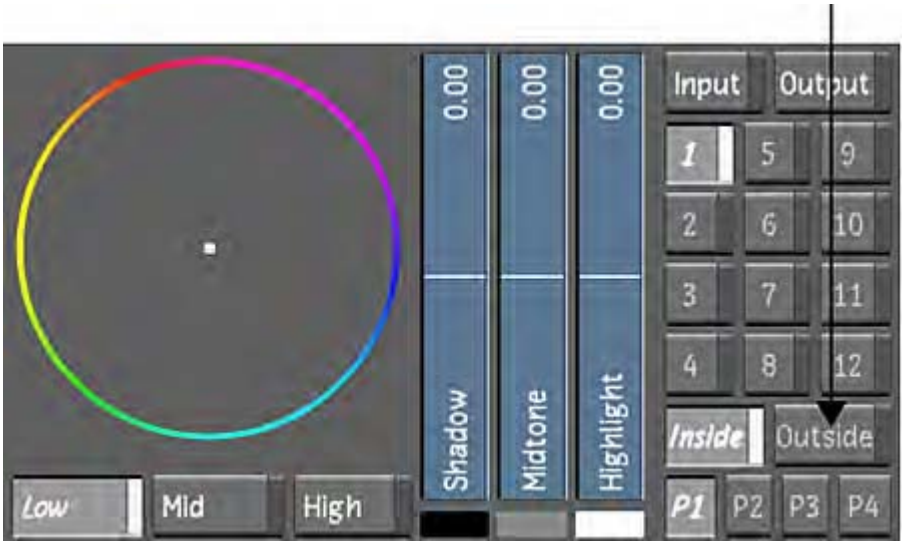
You can make printer light adjustments to secondaries just as you would to primary colour grades. See Adjusting Printer Lights for Primary Grading (page 2061).

Colour Grading Inside and Outside Secondaries

Secondaries can have unique colour grades applied both inside and outside a defined shape. The secondary's default state is the inside.

To grade a secondary and its surrounding area:

1. Colour grade the secondary.
2. In the secondary layers panel, click Outside to switch out of inside colour grading mode.



3. Right-click Outside to activate the outside colour grading mode.

   NOTE Selecting Inside or Outside allows you to colour grade the inside or outside area of the secondary. If you do not activate either button, its colour grade will be hidden in the Player.

   If gradients are used to define the shape, the appropriate blend will be applied between the shapes.

Copying Shapes and Grades

You can speed up the secondary colour grading process by copying a shape, or a shape with its parameters, from one secondary to another. This functionality is useful when colour grading the same shape(s) across several secondaries.

To copy a shape from one secondary to another:

1. Click Secondaries to display the Secondaries menu.
2. In the Player, select the shape's axis.
3. Hold down the `Ctrl` key while clicking a new secondary.
   The new secondary is enabled and the shape appears in the same position as on the previous secondary.

To copy a shape and selected parameters from one secondary to another:

1. Click Secondaries to display the Secondaries menu.
2. In the Player, select the shape's axis.
3. Click Selector and choose the parameters to be copied.

4 `Shift`-click the new secondary.

The new secondary is enabled and the shape with its parameters appears in the same position as on the previous secondary.

Applying a Secondary Colour Grade to the Entire Image

Once a secondary has been graded, you can test how the secondary grade would look like as a primary grade.

To view a secondary's colour grading as a primary colour grade:

1 In the Secondaries menu, deactivate Geom and the keyer.



NOTE To reapply the secondary colour grade, enable the Geom button again.

About Geometries

Geometries are spline-based objects you draw directly on a shot or an image. When performing secondary colour grading, you can create geometries to isolate particular objects or areas in the image to include with, or exclude from, the secondary colour grade.

You can include multiple geometries with each secondary.

Once geometries have been added, you can:

- Resize, reposition, and rotate them.
- Modify the softness, colour, opacity, and blur values of areas isolated by geometries.
- Parent multiple shapes to an axis.
- Change the priority order.
- Show or hide geometries in the Player.
- Invert geometries.
- Delete or reset geometries.
- Link geometries using logical operations from the previous secondary layer to the next one.
- Animate and track geometries.

Geometries are useful in many colour grading situations, from selectively colour grading large objects like the sky to small objects like a person's eyes. When you want to colour grade an object that moves through the shot, you can use the tracker to make a geometry follow the moving element. See Animating the Point Tracker (page 2144).

You can work with tangents and vertices to modify, scale, and rotate a geometry around its axis. You can also use the axis of the geometry to move it around the image. You can add and delete axes, and use one axis to move multiple geometries.

NOTE Field-based projects viewed on the data monitor will demonstrate a half-pixel downshift. Geometries added to the image will also display this downshift.

Accessing the Geometry Controls

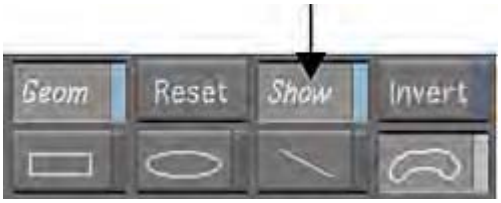
To draw geometries, you work with the Secondaries menu.

To access the Geometry controls:

1 In the Secondaries menu, right-click a secondary layer button to activate it.

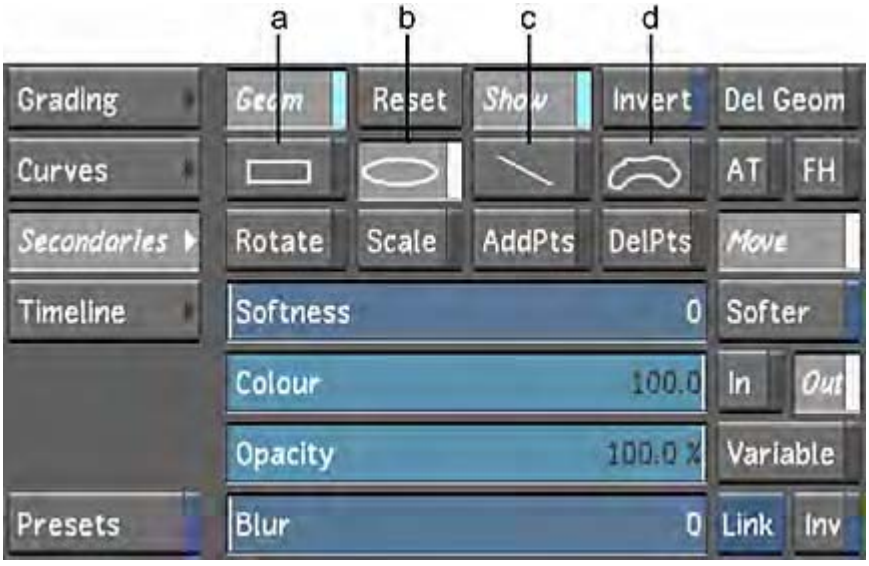


2 Enable Show.



If a geometry exists, it is displayed in the Player.

3 Enable a geometry.



(a) Rectangle geometry (b) Circle geometry (c) Split screen geometry (d) Free-form (bezier) geometry

NOTE The Keyer picker is automatically deactivated when a geometry is being created.

Drawing Basic Geometries

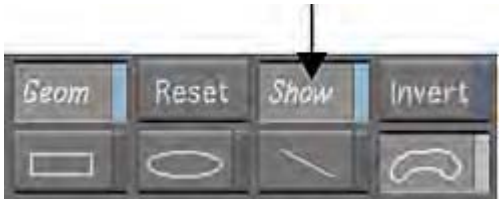
For each secondary, you can draw multiple rectangles or circles to isolate parts of the shot or image for secondary colour grading. Basic geometries can also be converted into free-form (Bezier) shapes.

To draw a rectangle or a circle:

1. In the Secondaries menu, right-click a secondary layer button to activate it.

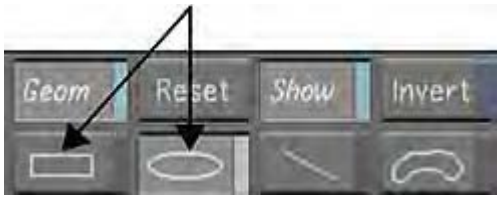


2. Enable Show.



   If a geometry exists, it is displayed in the Player.

3. Enable either the rectangle or the circle shape.



4. Draw the shape on the image. In the Player, drag to set the initial dimensions for the geometry. The geometry appears in the Player.

   NOTE To add more than one of the same basic shape to a secondary, you must enable the shape each time.

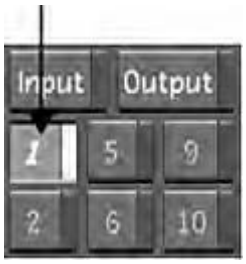
5. (Optional) To resize the geometry, drag vertices or a vertex point. To scale it, drag the Z-axis.
6. (Optional) To convert a basic shape to a free-form (Bezier) shape, right-click the shape in the Player.

Drawing Free-Form Geometries

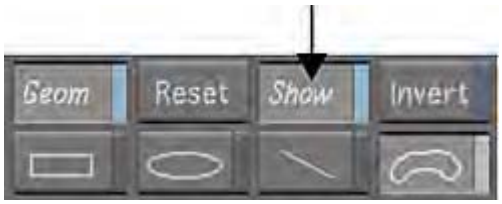
For each secondary layer, you can draw multiple free-form (Bezier) geometries to isolate parts of the shot or image for secondary colour grading. When you draw a free-form geometry, you can draw point-to-point using the Free-Form (Bezier) button, use Auto-Tangent mode or Freehand mode, or manually add vertices and tangents.

To manually draw a free-form geometry:

1 In the Secondaries menu, right-click a secondary layer button to activate it.



2 Enable Show.



If a geometry exists, it is displayed in the Player.

3 Enable the Free-Form (Bezier) button.



NOTE To add more than one of the same free-form shape to a secondary, you must enable the Free-Form shape button each time.

4 Make sure that the FH (freehand) button is disabled.



5 In the Player, click the image to set the first vertex.

6 Click again to add the second vertex.

7 To extend the tangents of a vertex and create a smooth curve, drag when you add a vertex.

8 Continue adding vertices and create the geometry.

9 To close the free-form geometry, click the first vertex.

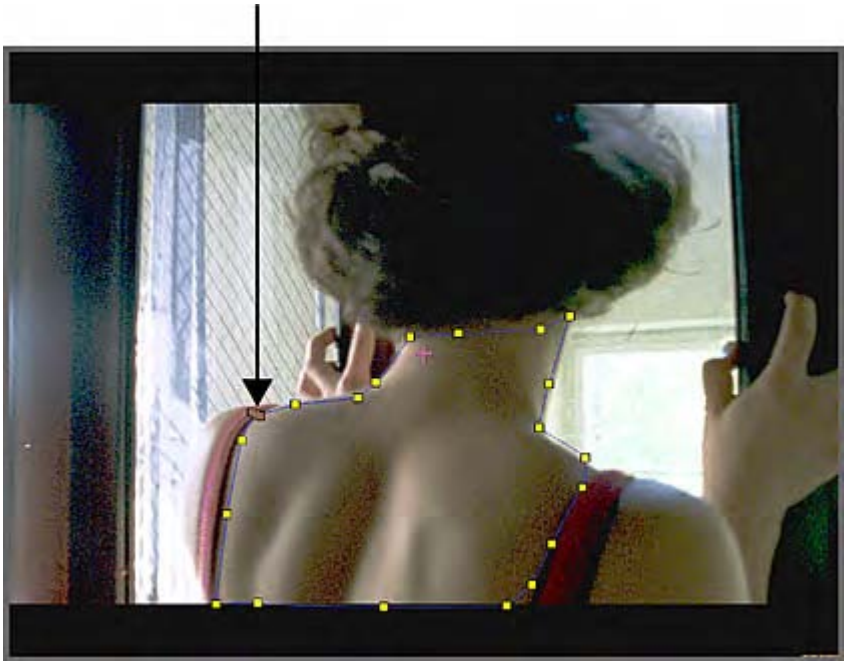


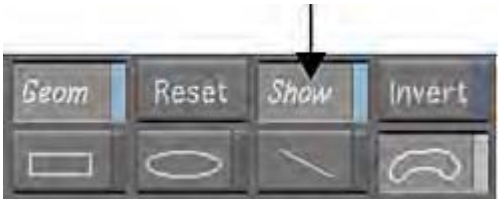
Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

To draw a free-form geometry using AutoTangent mode:

1 In the Secondaries menu, right-click a secondary layer button to activate it.



2 Enable Show.



Any pre-existing geometries will be displayed in the Player.

3 Enable the Free-Form (Bezier) button.



NOTE To add more than one of the same free-form shape to a secondary, you must enable the Free-Form shape button each time.

4 Enable AT (auto tangent).



5 In the Player, click the image to set the first vertex.

6 Click again to add the second vertex.

As you move from one vertex to the next, the tangents are automatically interpolated to create a smooth curve between vertices.

7 Continue clicking the image to add vertices and create the geometry.

8 To close the free-form geometry, click the first vertex.

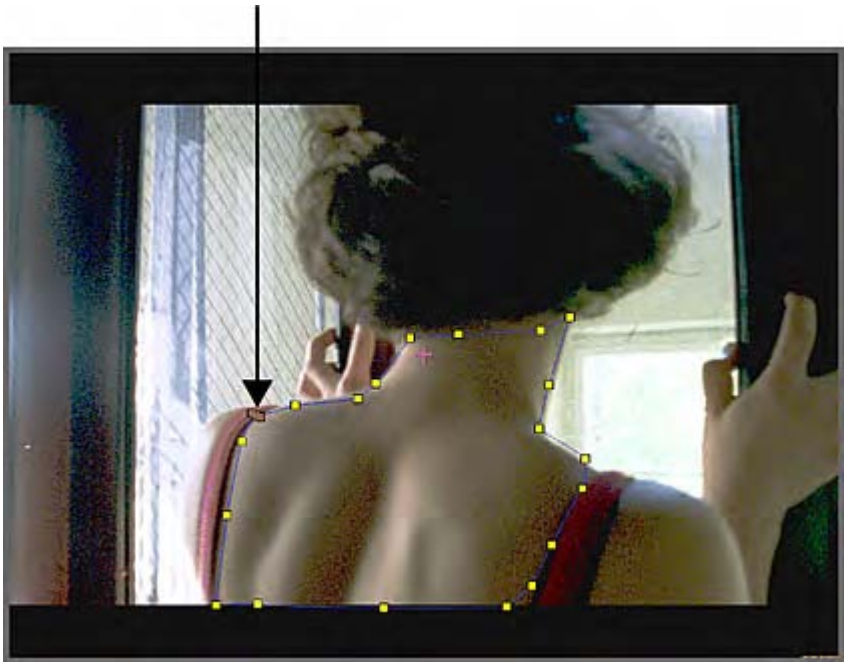


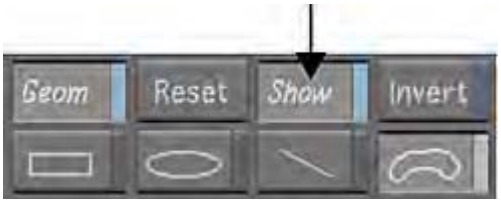
Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

To draw a free-form geometry using Freehand mode:

1 In the Secondaries menu, right-click a secondary layer button to activate it.



2 Enable Show.



Any pre-existing geometries will be displayed in the Player.

3 Enable the Free-Form (Bezier) button.



NOTE To add more than one of the same free-form shape to a secondary, you must enable the Free-Form shape button each time.

4 Enable FH (freehand).



5 In the Player, click and drag on the image to draw the geometry.

The vertices and tangents of the geometry are automatically added as you drag the mouse. The optimal number of vertices is used to create the smoothest curve.

6 To close the free-form geometry, release the mouse.

Modifying Geometries

Once you finish drawing a basic or free-form geometry, you can modify its shape, position, and appearance. You can:

- Reposition axes.
- Adjust tangents and vertices.

- Apply transformations.
- Apply constant or variable softness.
- Apply a blur filter.
- Adjust the colour transparency.

TIP To modify the geometry's parameters without the colour correction being visible, right-click the geometry's layer number button. To re-display the colour correction, right-click the button again.

Moving Axes

An axis can be moved from its default central position within the geometry. This affects how a geometry will behave when you scale or rotate it.

To move an axis:

1 Press `Alt`, right-click the axis, and drag it to another position.

Adjusting Tangents and Vertices

Adjust the shape of a geometry by manipulating its tangents and vertices. You can:

- Modify the position of vertices and tangents.
- Add and remove vertices and tangents.
- Break tangents.

Moving Vertices and Tangents

In the Player, you can move vertices and tangents to adjust the shape of a geometry.

To move vertices and tangents:

1 If you are working with a basic shape, right-click it in the Player to convert it to a free-form geometry.

2 In the Secondaries menu, enable Move.



3 Do any of the following:

- To move a vertex, drag it in the Player.
- To move multiple vertices, drag from anywhere on the screen without vertices. Then draw a selection box around the vertices you want to move and drag them in the Player.

  TIP To deselect multiple vertices, click away from the selected ones.

- To modify the shape or curve of a side, move the adjacent tangents. Drag a tangent handle to change its position.

Adding and Removing Vertices

You can add and remove vertices from a geometry. When you add or remove a vertex, the curve is redrawn according to the change of shape.

To add vertices:

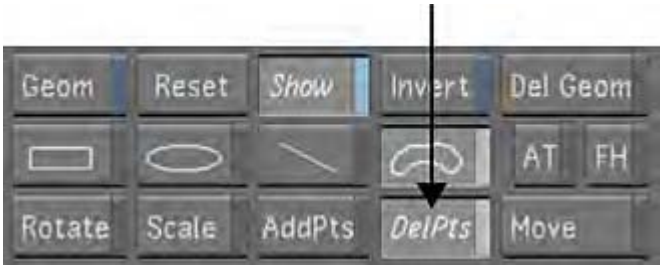
1. If you are working with a basic shape, right-click it in the Player to convert it to a free-form geometry.
2. In the Secondaries menu, enable Add Pts.



3. In the Player, move the mouse over the mask border of the geometry you want to modify.
   The current position on the geometry is highlighted.
4. Click the geometry where you want to add a vertex.
   The vertex is added to the geometry.

To remove vertices:

1. In the Secondaries menu, enable Del Pts.



2. In the Player, move the mouse over the mask border of the geometry you want to modify.
   The current vertex on the geometry is highlighted.
3. Click the highlighted vertex.
   The vertex is deleted from the geometry. The shape of the geometry is adjusted according to the change.

Breaking and Repairing Tangents

You can separate two tangent handles (*break* the tangent) and move them separately. You can also repair broken tangents.

To break and repair tangents:

1. To break and move a tangent handle, `Ctrl`-click the tangent handle and drag it.

2 To repair a broken tangent handle, Ctrl-click it.

Transforming Geometries

You can apply transformations to change the position and size of a geometry. You can:

- Move geometries.
- Rotate geometries.
- Scale geometries.

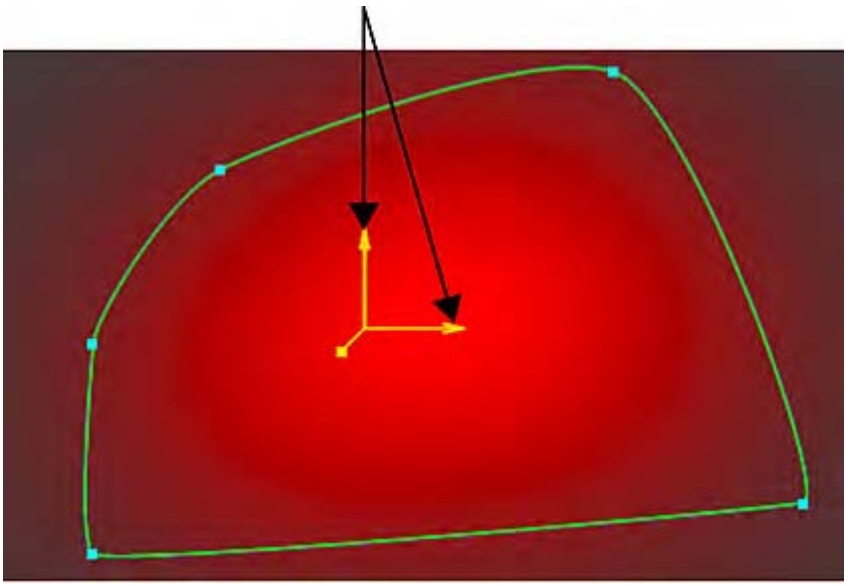
To move a geometry:

1 In the Secondaries menu, enable Move.



2 In the Player, drag the axis of the geometry you want to move.

3 To lock the geometry to the X- or Y-axis when you move it, you can drag from the arrowheads of the X- or Y-axis.



To rotate a geometry:

1 Do one of the following:
   - In the Player, right-click the axis and hold down the right mouse button while dragging.
   - In the Secondaries menu, enable Rotate. Then, in the Player, drag the axis of the geometry you want to rotate.



To scale a geometry:

1 In the Secondaries menu, enable Scale.



2 In the Player, drag along the axis of the geometry in the direction you want to scale.

Drag the:	To scale the geometry:
X-axis	Along the X-axis.
Y-axis	Along the Y-axis.
Global Scale axis	Along the Z-axis.

TIP You can drag the Z-axis to scale a geometry in any transformation mode—in Rotate, Move, or Scale mode.

Applying Softness to Geometries

You can increase and decrease the softness range to soften the edges of geometries. You can set the softness of the mask edge using:

- Constant softness.
- Variable softness to vary the softness range for different parts of the mask.

Applying Constant Softness

You can apply a constant range of softness to the edge of a geometry on a secondary. You can apply constant softness to the inside or outside of the mask border. You do not need to convert a basic shape to a free-form geometry in order to apply constant softness.

To soften the edges of the geometry:

1 Do one of the following:
- In the Player, select a geometry.
- From the Axes list, select a geometry or its axis.



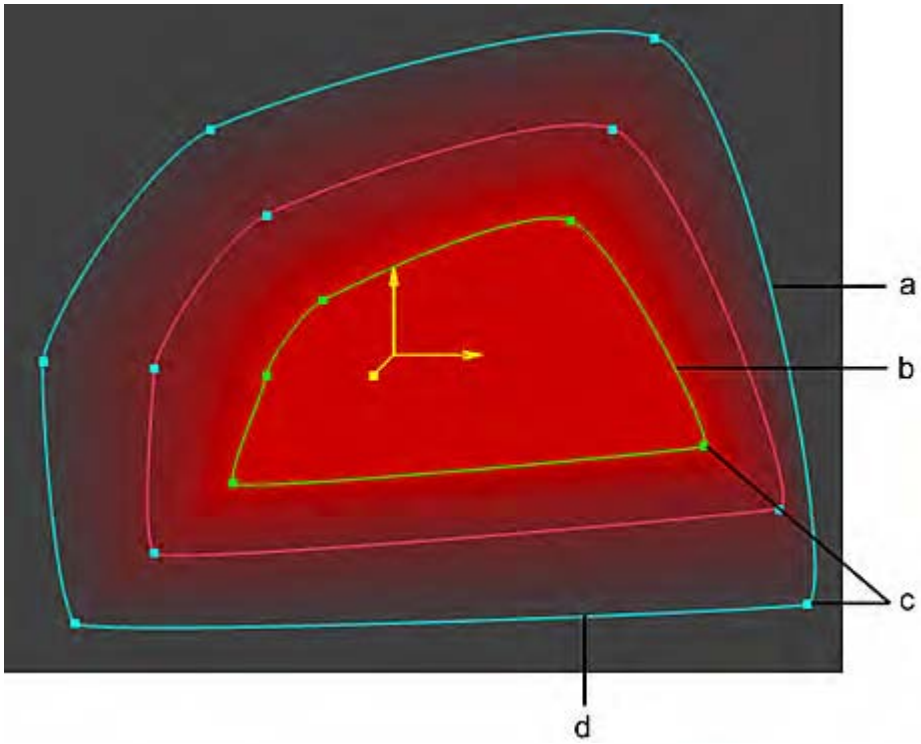
2 In the Secondaries menu, do one of the following:
- To apply softness to the inside of the mask border, enable In, and then drag the Softness slider. Drag right to increase softness. Drag left to decrease softness.
- To apply softness to the outside of the mask border, enable Out, and then drag the Softness slider. Drag right to increase softness. Drag left to decrease softness.



The softness level set by the Softness slider is applied to the geometry.

Applying Variable Softness

You can apply variable softness to the edges of one or more geometries to customize the softness at different parts of the mask. Variable softness has two softness borders, one inside and one outside the mask border. It also includes inner and outer softness vertices for each regular mask vertex.



(a) Outer softness border (b) Inner softness border (c) Softness vertices (d) Mask border

You can customize the softness at different parts of the mask by adjusting the distance of each softness vertex from the mask border—you set a softness value based on each vertex in the shape.

Variable softness can only be applied to free-hand geometries, or to basic geometries that have been converted to a free-form geometry.

To apply variable softness to the edges of the geometry:

1. Do one of the following:
   - In the Player, select a geometry.
   - From the Axes list, select a geometry or its axis.



2. In the Secondaries menu, enable Variable.



Softness is applied inside and outside the mask border.

3 Drag the Softness slider. Drag right to increase softness. Drag left to decrease softness.



The softness level set by the Softness slider is applied to the geometry.

4 Drag the vertices and tangents to adjust the shape and vary the softness at the edges of the geometry. See Adjusting Tangents and Vertices (page 2098).

Blurring Geometries

You can apply a Gaussian blur to soften all geometries on the current secondary layer.

To apply a Gaussian blur:

1 In the Secondaries menu, right-click a secondary layer button to activate it.



2 Drag the Blur slider. Drag right to increase the blur. Drag left to decrease the blur.



Setting the Colour and Opacity of Geometries

You can adjust the colour and opacity of a geometry to affect the output image.

If you are applying only one geometry or several non-overlapping geometries to a secondary, modify either the colour or opacity to adjust the matte—in this case, colour and opacity will have the same effect on the image. If you are applying several overlapping geometries to a secondary, modify both colour and opacity to adjust the matte—colour will set the intensity of the colour of the geometry, whereas opacity will set the transparency, creating a blend between the overlapping geometries.

To set the colour and opacity of a geometry:

1 Do one of the following:
- In the Player, select a geometry.
- From the Axis list, select a geometry or its axis.



2 In the Secondaries menu, drag the Opacity slider to adjust the transparency of the mask.



The Opacity slider defines the blend between overlapping geometries. A value of 50% is a 50/50 blend. A value of 100% shows the exact colour of the geometry. A value of 0% shows what is behind the selected geometry.

3 To set the intensity of the colour of the geometry, drag the Colour slider to set a colour value between 0% and 100%. A value of 50% means the area is 50% grey.

Adjusting the Colour and Opacity of a Matte Overlay

You can change the way a matte is displayed to help you easily view the isolated colour. The colour and opacity of the unselected region in a matte can be easily modified by adjusting the Matte Overlay parameters.

To adjust the colour and opacity of a matte overlay:

NOTE This procedure will not affect the output image.

1 Select and then right-click a secondary on which a matte is present.

2 Depending on your user settings, press `F11` once or twice to enable the Matte view. See Display & Interface Settings (page 1759).

3 In the Setup menu, select Interface to display the Matte Overlay panel.

NOTE The default values are displayed.



4 Drag the Opacity slider as well as the R, G, and B sliders to adjust the opacity and colour of the unselected region. See Matte Overlay Settings (page 1790).



Secondary view (default)



Secondary view (adjusted)

TIP Click Reset twice if you wish to reset the sliders to their default values.

Adding a Split Screen

The split screen feature lets you divide the screen in two in order to apply separate colour gradings to each half of the image.

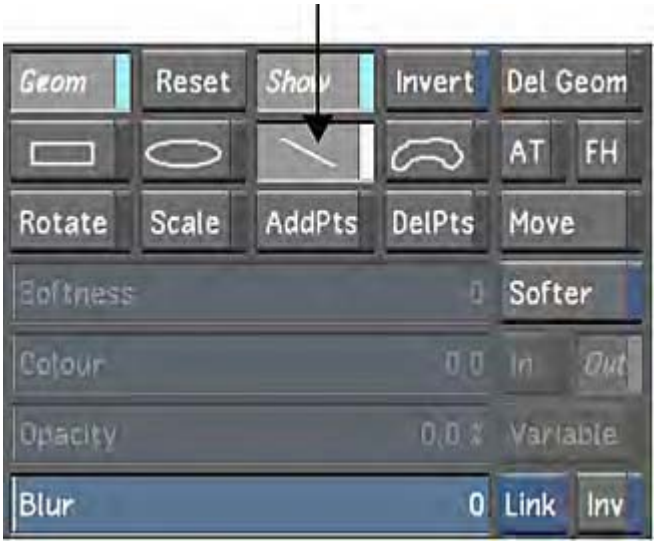
Split screens can be moved and rotated.

To add a split screen:

1 In the Secondaries menu, right-click a secondary layer button to activate it.



2 Enable the Split Screen button.



3 In the Player, click where you would like the split screen applied.

The split screen is applied horizontally in the spot where you clicked. You can now colour grade this part of the image.

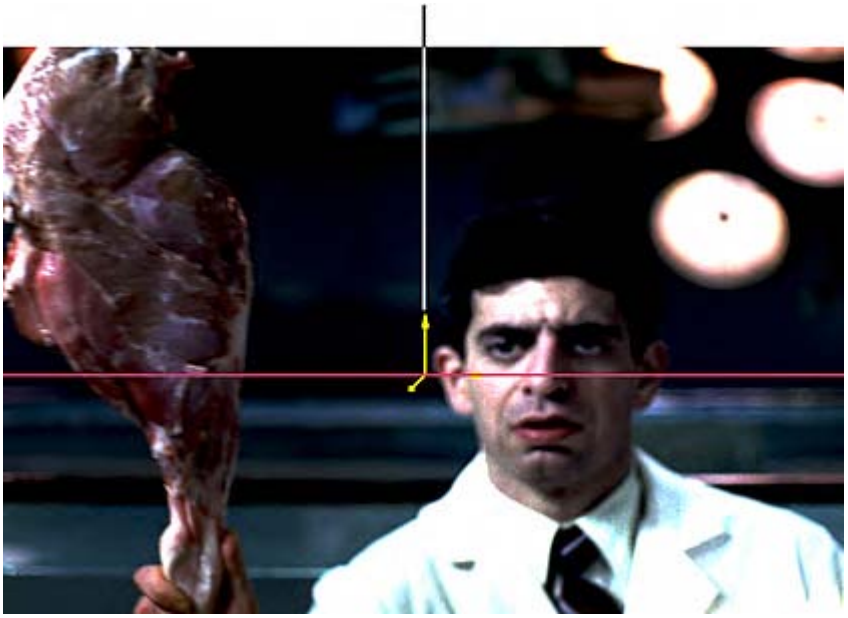


Image courtesy of Technicolor (formerly Toybox)

4 To grade the other half of the split screen, enable Outside (see Colour Grading Inside and Outside Secondaries (page 2090)).

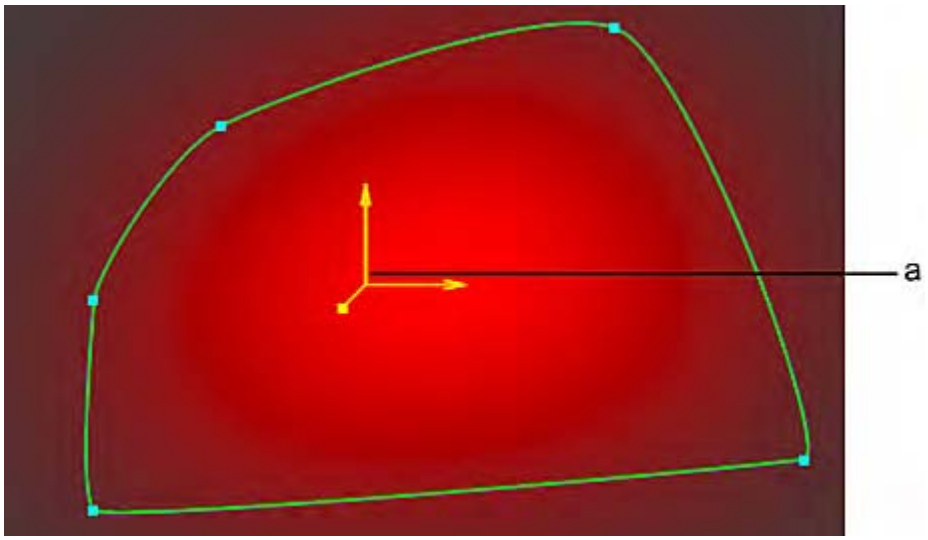
5 To move the split screen up or down, click and drag the split screen's axis.

6 To rotate the split screen, do one of the following:

- Enable Rotate and then drag the axis.
- Right-click the axis and drag.

Adding and Deleting Axes

By default, every geometry you draw has an axis. Each geometry and its axis are displayed in the Player.



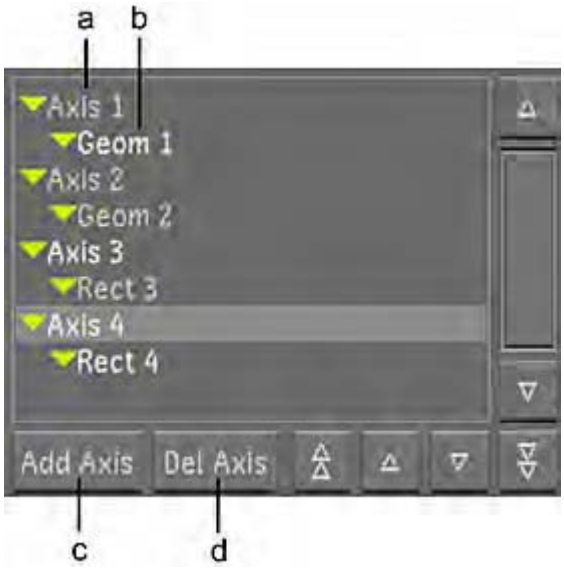
(a) Geometry axis

You use the Axes list to add and delete axes, as well as parent multiple axes to a master axis. You use the schematic view to create parent/child connections among shapes and axes.

A parent axis allows you to move, scale, rotate, and track multiple geometries.

The Axes List

The Axes list appears in almost all Lustre menus and provides a list view of geometries and their axes.



(a) Axis (b) Geometry (c) Add Axis button (d) Del Axis button

When you select the axis or the geometry in the Axes list, it is also selected in the Player. The inverse is also true.

To create a parent axis:

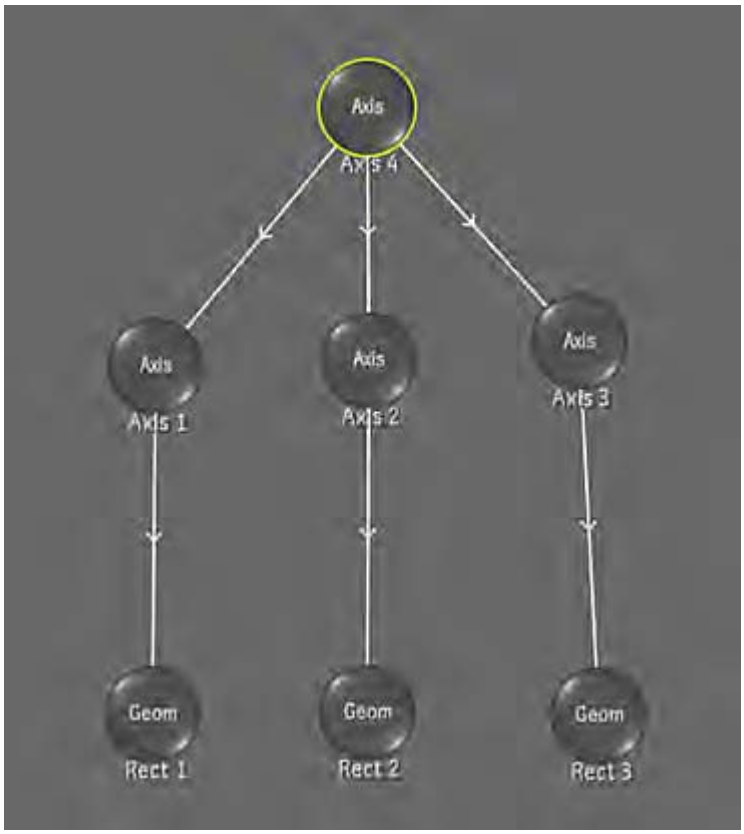
1. In the Axes list, select the axis you want to parent. To select multiple axes, `Shift`-click each axis.
2. Click Add Axis.
   The axes you selected are now parented to a new master axis.

To delete an axis:

1. In the Axes list, select the axes you want to delete. To select multiple axes, `Shift`-click each axis.
2. Click Del. Axis.
   The axis is deleted from the Player and the Axes list.

Schematic View

As an alternative to using the Axes list to view the axis hierarchy and create parent axes, you can use the schematic.



The schematic appears in the Player and allows you to parent shapes and axes in any combination you like. For example, a shape could parent an axis, or vice versa.

Creating a master axis and using it to parent multiple axes simultaneously is performed the same way as when using the Axes list. However, in the schematic, you can also make connections one at a time.

NOTE Axes cannot be deleted in the schematic.

To create parent/child connections using the schematic:

1 Press the tilde key (~) to switch to a schematic view of the axis hierarchy.

NOTE To pan within the schematic, hold down the middle mouse button and drag.

2 To connect one item to another, click just inside the border of the parent object and drag to the intended child object. The connection is represented by a white line (yellow when selected), with the hierarchy indicated by an arrow that points away from the parent object and towards the child.

NOTE To move an item, click in its centre and drag. To break a link, drag a red line across the connection line.

3 Switch back to default view by pressing ~ again.

The changed hierarchy is displayed in the schematic.

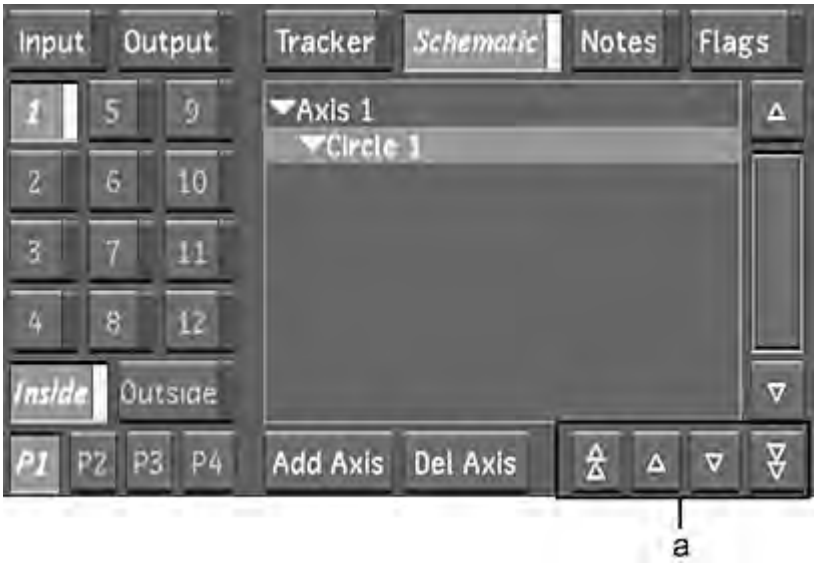
Changing the Priority Order of Geometries

When you create several free-form geometries on a secondary layer, you can change the order in which they are drawn, or layered in the scene. This affects the resulting image because a geometry with a higher priority has precedence over one with a lower priority. You can use the priority order to create complex colour effects involving multiple geometries.

To set the priority of a mask:

1 Select the geometry.

2 Enable Schematic to display the priority order buttons.



(a) Priority order buttons

3 Do any of the following.

Click:	To move the geometry:
	To the top layer.

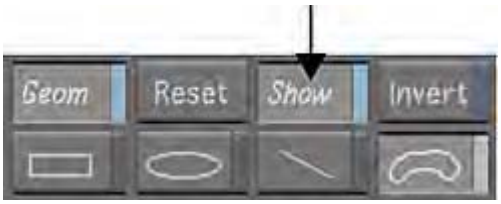
Click:	To move the geometry:
	To the bottom layer.
	Down one layer.
	Up one layer.

Showing and Hiding Geometries

You can show or hide geometries after you create them. Geometries that are hidden do not appear in the Player, but are applied to the secondary—depending on which logical operation you use.

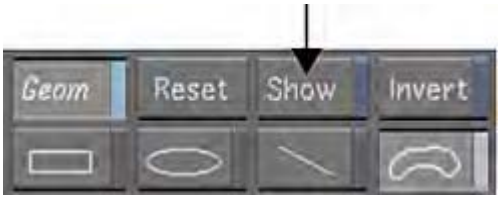
To show a geometry:

1 In the Secondaries menu, enable Show.



To hide a geometry:

1 Disable Show.

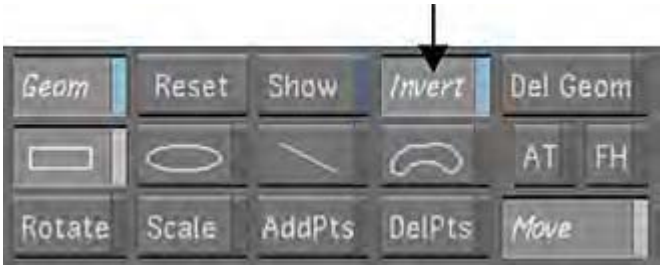


Inverting Geometries

You can invert geometries on the current secondary layer.

To invert geometries:

1 In the Secondaries menu, enable Invert.



Deleting Geometries and Resetting Secondaries

Delete geometries one at a time from a secondary, or reset the secondary to delete all geometries simultaneously.

To delete a geometry:

1 In the Secondaries menu, enable Del Geom.



2 In the Player, select the geometry.

The selected geometry is deleted.

3 Select the geometry's axis.

The selected axis is deleted. Once both the axis and geometry have been deleted, the secondary colour grade is applied to the entire image.

NOTE You can also delete geometries and their axes from the Axes list. To delete a geometry or axis, select it in the Axis list (or `Shift`-click to select several), and then click Del. Axis.

To reset a secondary:

1 In the Secondaries menu, enable a Secondary button.

2 Do one of the following:

- To clear all geometries from the layer and apply the secondary colour grade to the entire image, click the Reset button in the Secondaries menu and then click again to confirm.



- To clear all geometries from the layer without applying the secondary colour grade to the entire image, click the Reset button in the main menu.



Linking Geometries across Secondary Layers

You can link geometries from the previous secondary layer to the current secondary layer by selecting an option from the Link option box, located in the Secondaries menu. Use the Link option box when you have already applied a key, a geometry, or both to the current secondary layer.

NOTE You can use the Link option box without applying any key or geometry to the current secondary layer.

When you use the Link option box, a logical operation is applied to determine how the geometries from the previous layer are combined with the key and geometries on the current secondary layer, and what areas of the image are colour graded. Depending on what you want to accomplish with the secondary, you can use a logical operation to add or exclude the inside of the geometries from the previous layer to the key and geometries on the current secondary layer, or intersect the geometries with the current secondary layer. You can also invert the geometries from the previous layer using the Invert button.

To combine geometries from the previous secondary layer with the current secondary layer:

1. Make sure you have multiple secondary layers containing keys and geometries, and then in the Secondaries menu, enable the secondary you want to work on.

   NOTE The previous secondary layer must have some sort of mask applied—it can be a key, a geometry, or both.

2. Select a logical operation from the Link option box. The logical operation you select determines what is colour graded by the current secondary.



Select:	To:
	Add the current secondary to the previous one.
	Subtract the current secondary from the previous one.
	Intersect the current secondary with the previous one.

NOTE If you do not want to apply a logical operation, select the Link option.

3 To invert the previous selection, enable Inv.



When you invert geometries from the previous secondary layer, the logical operation you choose will combine the invert of the geometries from the previous secondary layer with the current one.

Creating a Secondary by Keying a Range of Colours

You can create secondaries by defining transparent regions in a shot, based on a specific range of colour. This process is known as *keying*.

When you extract a key to isolate a colour, you can view it in the Player. Keys are based on the chrominance and luminance of a selected colour from the original image. Once you have selected a colour, you then set the tolerance and softness ranges by sampling the original image. Tolerance defines how similar luminance and chrominance values will be keyed (made transparent) in the secondary. A higher tolerance setting extracts a broader range of luminance and chrominance values, while a lower setting extracts a narrower range, leaving the rest of the image untouched. Softness defines how similar luminance and chrominance values are softened (made partially transparent) for the secondary. A higher softness setting extracts a broader range of luminance and chrominance values, while a lower setting extracts a narrower range, leaving the rest of the image untouched. You can soften the edge of a key to blend the secondary colour grading with the rest of the image.

Keys are, by default, based on original scans, bypassing any input primary colour grading. This allows you to modify an image upstream of any secondary without changing the key. For example, after you key a secondary, you can safely perform a hue shift from the Curves menu. The key, based on the original image colour, is unaffected. To base a key on colour corrected sources, enable the Src: Prim (Source Primary) button.

For extracting keys to isolate colours, Lustre provides two keyers that work independently of one another:

- The Hue, Luminance, and Saturation (HLS) Keyer, which is used to extract a key based on the softness and tolerance of the hue, luminance, and saturation channels of the source image. See Using the Hue, Luminance, and Saturation (HLS) Keyer to Extract a Key (page 2116).
- The Diamond Keyer, which is used to extract a key based on the chrominance range and luminance levels of the source image. See Using the Diamond Keyer to Extract a Key (page 2124).

Using the Hue, Luminance, and Saturation (HLS) Keyer to Extract a Key

With the HLS Keyer, you can extract a key based on hue, luminance, and saturation channels. You can use one or more of these channels to create a key.

You can also use previously saved keyer presets. See Saving and Loading Presets Using the Presets Lists (page 2051).

Once you have extracted a key, you can modify a key by adjusting the Master Tolerance and Master Softness sliders or by using the Add/Remove Softness and Tolerance buttons. You can further modify a key using the Cleanup, Shrink, and Blur parameters.

Before extracting a key, you can modify the source image by using the Input Transform Log to Lin and Soften options. See Tools Settings (page 1765).

NOTE To disable the HLS Keyer at any time, right-click on the Keyer option box.

Use the following workflow to extract a key for colour isolation.

Step:	Refer to:
1. Determine which channels you want to use in a key.	See Setting Keying Parameters in the HLS Keyer (page 2117).
2. Select the colour that you want to key.	See Sampling a Colour in the HLS Keyer (page 2118).
3. View your secondary to determine the range of colour you have keyed.	See Viewing Secondaries in the HLS Keyer (page 2120).

Step:	Refer to:
4. Set the tolerance to widen or narrow the range of luminance and chrominance values to extract in a key.	See Setting the Tolerance Range in the HLS Keyer (page 2121).
5. Set the softness to widen or narrow the range of luminance and chrominance values to extract at the edge of a key.	See Setting the Softness Range in the HLS Keyer (page 2123).
6. Blend the light and dark edges of a key by shrinking, eroding, or blurring the edges of a key.	See Modifying the Edges of the Key (page 2134).
7. Clean up a key for colour grading.	See Removing Stray Pixels from a Key (page 2135).
8. Increase or decrease the softness of pixels that are in the tolerance range.	See Sharpening a Key Source Image (page 2135).

The following procedures are not essential for extracting a key. You can perform these procedures at any time after extracting a key.

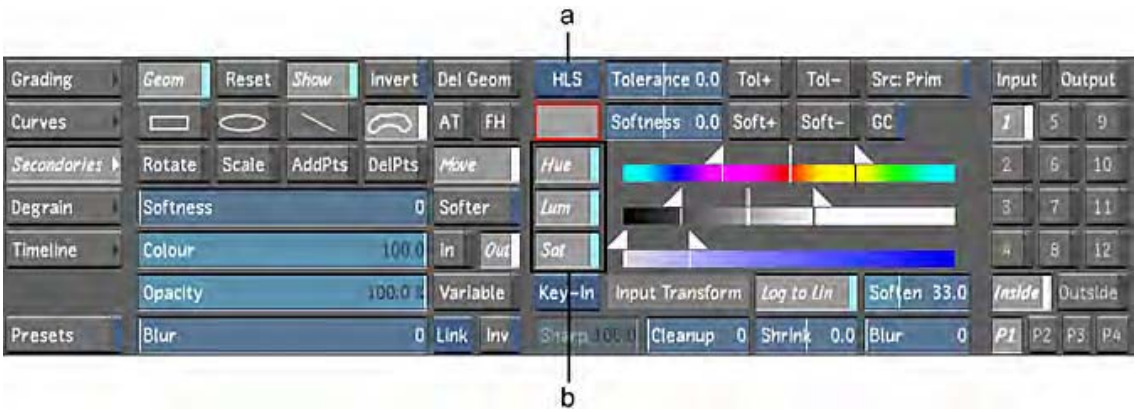
Step:	Refer to:
1. Remove colours that you do not want to include in a key.	See Inverting Keys (page 2136).
2. Exclude a key that you do not want to appear in a secondary.	See Excluding Keys from Secondaries (page 2136).

Setting Keying Parameters in the HLS Keyer

You need to determine which channels (hue, luminance, and saturation) you will use to extract your key.

To set keying parameters:

1. In the Colour menu, click Secondaries.
   The Secondaries menu appears.
2. Click the Keyer option box to toggle to the HLS Keyer.
   The HLS Keyer appears in the Secondaries menu.



(a) Keyer option box (b) Hue, luminance, and saturation buttons

3 To select the basis of a key, enable one, two, or all three of the channels. Channel selection depends what you want to base your key on: chrominance, luminance, or a combination of both.

- To extract a key based on the hue of the selected colour range, enable Hue. To exclude hue from a key, disable Hue.
- To extract a key based on the luminance of the selected colour range, enable Lum. To exclude luminance from a key, disable Lum.
- To extract a key based on the saturation of the selected colour range, enable Sat. To exclude saturation from a key, disable Sat.

NOTE Chrominance is comprised of both the Hue and Saturation channels.

4 Enable the Source Primary button to use primary grading results as the input to the keyer.

NOTE When Src: Prim is disabled, the raw scan is used as the input to the keyer, effectively bypassing any input LUTs and primary grading that has been applied to your shot.



(a) Source Primary button (b) Log to Lin button (c) Soften slider

5 Enable the Log to Lin button. See Tools Settings (page 1765).

6 Adjust the Soften slider. See Tools Settings (page 1765).

Sampling a Colour in the HLS Keyer

The typical way to extract a key is to sample a pixel or an average colour in the image using the colour picker. You can also use predefined colour grading parameters, called presets, by right-clicking the Hue, Luminance, or Saturation button. See Saving and Loading Presets Using the Presets Lists (page 2051).

To sample a colour:

1 Set the view mode to Output. You can click the O button.

NOTE Pressing `F12` to set the view mode to Output only works from Matte view.



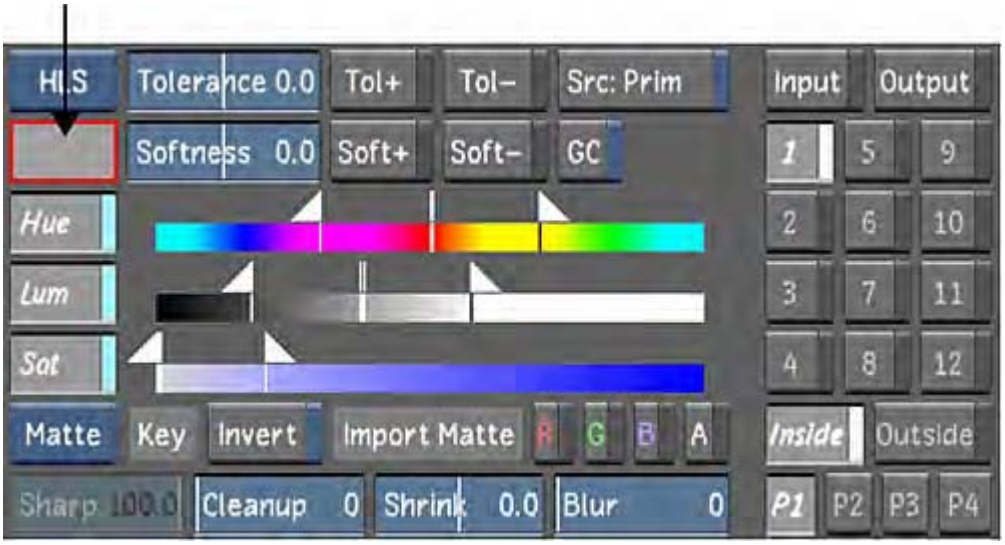
2 In the Secondary layers panel, activate a secondary layer.



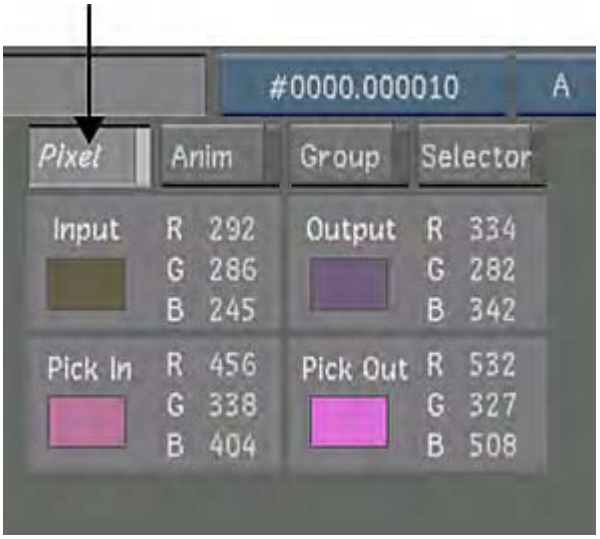
NOTE If you do not activate the secondary layer, the colour pot will not be updated with your colour sample.

3 Right-click the Keyer option box to enable the HLS Keyer.
The letters in the Keyer option box turn bright white indicating that the HLS Keyer is enabled.

4 Click the colour pot.
The HLS Keyer is enabled and the colour pot is outlined in red, indicating that it is in use.



5 Click Pixel to enable the Pixel Analyser.



6 Set the Log to Lin parameter. See Tools Settings (page 1765).

7 Adjust the Soften slider. See Tools Settings (page 1765).

8 Sample the image:

- To sample a single pixel, click and drag in the image until you locate the pixel you want to sample. As you drag, the colour of the current pixel appears in the Pixel Analyser. When you locate the pixel you want to sample, release the mouse.
- To sample an average taken from a range of colours in the image, `Alt`-drag the image in the Player.

The cursor changes to a colour picker when you move it over the image in the Player. When you release the mouse, the centre tolerance and the initial softness and tolerance ranges are set for the channels that are enabled. The colour pot displays the sampled colour.



9 Click the colour pot again to finish defining the colour.

Viewing Secondaries in the HLS Keyer

Once you have sampled a colour for a key, you can view the resulting secondary in the Player. You can view a key in Secondary view or Matte view by toggling the `F11` key. Secondary view displays the keyed out colour as is; unsampled colours are overlayed with a uniform colour by default. Matte view displays the alpha channel, a black and white template indicating which parts of the image are transparent (black), and which are opaque and selected (white).

To view a secondary:

1. Select the secondary you want to view.
2. Display the image in Secondary view by pressing `F11`.

   In Secondary view, the colour that is keyed out in the current secondary is unchanged; unsampled colours are overlayed with a uniform colour by default. See Matte Overlay Settings (page 1790).



3. Press `F11` to display Matte view. See Display & Interface Settings (page 1759).

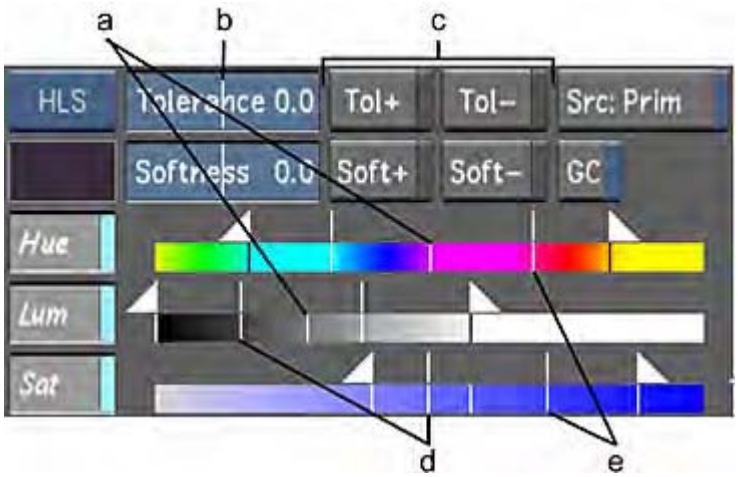
   In Matte view, the white parts of the matte represent the selected colours which can be colour graded. The black parts represent the unselected colours in the secondary. Greys indicate the zone of softness.



Setting the Tolerance Range in the HLS Keyer

You can set the tolerance range of colours to be fully keyed (transparent) and to remove unwanted greys from a key. The maximum and minimum tolerance values define the range of colours included in the secondary. The closer a pixel matches the key colour, the greater its transparency. Conversely, the less a pixel matches the key colour, the greater its opacity.

To set the tolerance, you can use the colour picker, the Master Tolerance slider, or the Tolerance Range indicators. With the colour picker and the Master Tolerance slider, you set the tolerance range for hue, luminance, and saturation together. With the Tolerance Range indicators, you set the tolerance range for hue, luminance, and saturation individually.



(a) Centre Tolerance indicators (b) Master Tolerance slider (c) Add Tolerance/Remove Tolerance buttons (d) Minimum Tolerance Range indicators (e) Maximum Tolerance Range indicators

To set the tolerance range using the colour picker:

1. View the secondary in the Player. Press `F11` to toggle between Matte view and Secondary view.
2. In the Secondaries menu, enable the Add Tolerance or Remove Tolerance button beside the Master Tolerance slider to increase or decrease tolerance.
3. Click and drag in the image to include it in the tolerance range. `Alt`-drag in the image to select a larger area.

   The tolerance is modified.

To set the tolerance range using the Master Tolerance slider:

NOTE The Master Tolerance slider controls the position of all of the sliders in all of the channels (even when they are not enabled).

1. View the secondary in the Player. Press `F11` to toggle between Matte view and Secondary view.
2. Set the tolerance range:
   - To increase tolerance, drag the Master Tolerance slider to the right.
   - To decrease tolerance, drag the Master Tolerance slider to the left.

   When you release the slider, the tolerance is modified and the slider returns to its original position.

To set the tolerance range for the hue, luminance, or saturation channel:

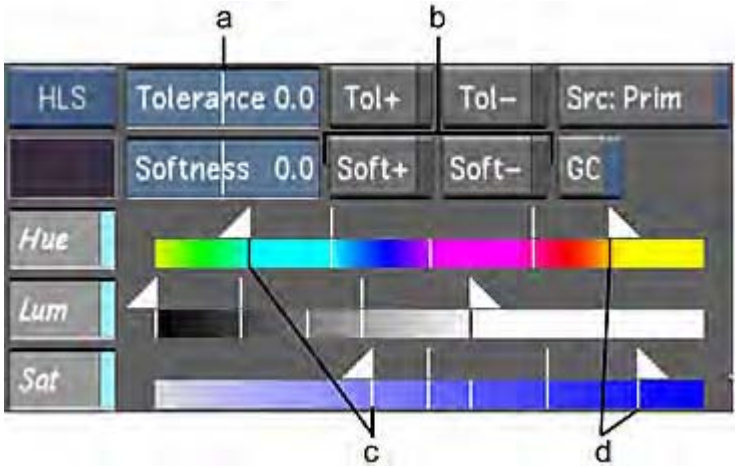
1. View the secondary in the Player. Press `F11` to toggle between Matte view and Secondary view.
2. For the hue, luminance, or saturation channel, set the centre tolerance value using the Centre Tolerance indicator. The Centre Tolerance indicator acts as a reference point for the tolerance range. Make sure to place your cursor directly over the indicator and then drag it to the left or right.

   When you move the Centre Tolerance indicator, the other indicators move as well—the softness and tolerance ranges are shifted.
3. For the hue, luminance, or saturation channel, set the minimum value for the tolerance range using the Minimum Tolerance Range indicator. Drag the indicator to the left to increase tolerance. Drag the indicator to the right to decrease tolerance.
4. For the hue, luminance, or saturation channel, set the maximum value for the tolerance range using the Maximum Tolerance Range indicator. Drag the indicator to the left to decrease tolerance. Drag the indicator to the right to increase tolerance.

Setting the Softness Range in the HLS Keyer

Once you have set the tolerance range, you can then set the softness range. Softness sets the range of colours to make semi-transparent and can create a softer key. Use softness to create a more natural and softer look when applying secondary colour correction.

To set the softness of a key, you can use the colour picker, the Master Softness slider, or the Softness Range indicators. With the colour picker and the Master Softness slider, you set the softness range for hue, luminance, and saturation together. With the Softness Range indicators, you set the softness range for hue, luminance, and saturation individually.



(a) Master Softness slider (b) Add Softness/Remove Softness buttons (c) Minimum Softness Range indicators (d) Maximum Softness Range indicators

To set the softness range using the colour picker:

1. View the secondary in the Player. Press `F11` to toggle between Matte view and Secondary view.

   In Matte view, you can see the modifications you make to the softness. You can toggle between Matte view and Secondary view while you fine-tune a key.
2. In the Secondaries menu, enable the Add Softness or Remove Softness button to increase or decrease the softness.
3. Click and drag an area of the image. `Alt`-drag in the image to select a larger area.

   The softness is modified.

To set the softness range using the Master Softness slider:

NOTE The Master Softness slider controls the position of all of the sliders in all of the channels (even when they are not enabled).

1. View the secondary in the Player. Press `F11` to toggle between Matte view and Secondary view.

   In Matte view, you can see the modifications you make to the softness. You can toggle between Matte view and Secondary view while you fine-tune a key.
2. Set the softness range:
   - To increase softness, drag the Master Softness slider to the right.
   - To decrease softness, drag the Master Softness slider to the left.

   When you release the slider, the softness is modified and the slider returns to its original position.

To set the softness range for the hue, luminance, or saturation channel:

1. View the secondary in the Player. Press `F11` to toggle between Matte view and Secondary view.

   In Matte view, you can see the modifications you make to the softness. You can toggle between Matte view and Secondary view while you fine-tune a key.
2. For the hue, luminance, or saturation channel, set the minimum value for the softness range using the Minimum Softness Range indicator. This controls the minimum softness of edges between the image and the key colour. Drag the indicator to the left to increase the softness range. Drag the indicator to the right to decrease the softness range.
3. For the hue, luminance, or saturation channel, set the maximum value for the softness range using the Maximum Softness Range indicator. This controls the maximum softness of edges between the image and the key colour. Drag the indicator to the left to decrease the softness range. Drag the indicator to the right to increase the softness range.

Using the Diamond Keyer to Extract a Key

The Diamond Keyer provides comprehensive controls to set the key colour, as well as the chrominance range and luminance levels for softness and tolerance on the hue cube. You can set these by using sliders and numeric fields, or by sampling colours in the image.

With the Diamond Keyer, you can extract a key based on a preset red, green, blue, cyan, magenta, or yellow colour, or on a sampled colour. You can also use previously-saved keyer presets. See Saving and Loading Presets Using the Presets Lists (page 2051). By manipulating the chrominance Softness and Tolerance Diamonds, which define the boundaries of the chrominance range for softness and tolerance on the hue cube, you can quickly and easily isolate the colour for which you want to extract a key.

You can modify a key by adjusting the luminance levels using the Luminance gradient controls and by adjusting the chrominance range and luminance levels using the Add/Remove Softness and Tolerance buttons. You can further modify a key using the Sharpness, Cleanup, Shrink, and Blur parameters.

NOTE To disable the Diamond Keyer at any time, right-click on the Keyer option box.

Before extracting a key, you can modify the source image by using the Input Transform Log to Lin and Soften options. See Tools Settings (page 1765).

Use the following workflow to extract a key for colour isolation.

Step:	Refer to:
1. Access the Diamond Keyer.	See Accessing the Diamond Keyer (page 2125).
2. Select the colour that you want to key.	See Sampling a Colour in the Diamond Keyer (page 2126).
3. View your secondary to determine the range of colour to key.	See Viewing Secondaries in the Diamond Keyer (page 2128).
4. Set the luminance levels and chrominance range for tolerance to extract in the key.	See Setting the Tolerance Range in the Diamond Keyer (page 2129).
5. Set the luminance levels and chrominance range for softness to extract in the key.	See Setting the Softness Range in the Diamond Keyer (page 2131).

Step:	Refer to:
6. Blend the light and dark edges of a key by shrinking, eroding, or blurring the edges of a key.	See Modifying the Edges of the Key (page 2134).
7. Remove stray pixels from a key to clean it up for colour grading.	See Removing Stray Pixels from a Key (page 2135).
8. Increase or decrease the softness of pixels that are in the tolerance range.	See Sharpening a Key Source Image (page 2135).

The following procedures are not essential for extracting a key. You can perform these procedures any time after extracting a key.

Step:	Refer to:
1. Remove colours that you do not want to include in the key.	See Inverting Keys (page 2136).
2. Exclude a key that you do not want to appear in a secondary.	See Excluding Keys from Secondaries (page 2136).

Accessing the Diamond Keyer

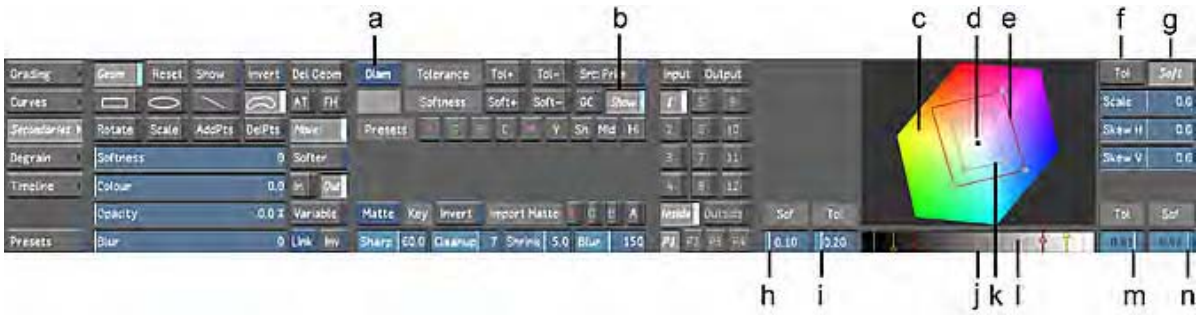
Use the comprehensive controls in the Diamond Keyer to extract a key.

To access the Diamond Keyer:

1 In the Secondaries menu, toggle the Keyer option box to access the Diamond Keyer.

The Diamond Keyer interface appears.

NOTE Disable the Show button to display the Shape Tracker, Schematic, Notes, Flags, Pixel, Animation, Group, and Selector panels.



(a) Keyer option box (b) Show Diamond Keyer (c) Hue cube (d) Sampled pixel (e) Chrominance Softness Diamond (f) Chrominance Tolerance Diamond selector (g) Chrominance Softness Diamond selector (h) Minimum Luminance slider for softness (i) Minimum Luminance slider for tolerance (j) Luminance level of sampled pixel (k) Chrominance Tolerance Diamond (l) Luminance gradient (m) Maximum Luminance slider for tolerance (n) Maximum Luminance slider for softness

The Diamond Keyer interface is made up of the following elements.

Keyer option box Toggles to display the HLS Keyer or the Diamond Keyer. The default keyer is configured in the user settings. See Tools Settings (page 1765).

Show Displays the Diamond Keyer. Disable to display the Shape Tracker, Schematic, Notes, Flags, Pixel, Animation, Group, and Selector menus.

Hue cube Representation of the colour space in which you perform colour isolation for the extraction of a key.

Sampled pixel Black dot represents the sampled colour on the hue cube.

Chrominance Softness Diamond Used to set the chrominance range for softness.

Chrominance Tolerance Diamond selector Selects the chrominance Tolerance Diamond for scaling and skewing.

Chrominance Softness Diamond selector Selects the chrominance Softness Diamond for scaling and skewing.

Minimum Luminance slider for softness Used to set the minimum luminance levels for softness.

Minimum Luminance slider for tolerance Used to set the minimum luminance levels for tolerance.

Luminance level of sampled pixel Orange bar represents the luminance level of the sampled pixel in the Luminance gradient.

Chrominance Tolerance Diamond Used to set the chrominance range for tolerance.

Luminance gradient Displays the level of luminance for the sampled pixel and the Minimum and Maximum luminance levels for tolerance and softness.

Maximum Luminance slider for tolerance Used to set the maximum luminance level for tolerance.

Maximum Luminance slider for softness Used to set the maximum luminance level for softness.

Sampling a Colour in the Diamond Keyer

The typical way to extract a key is to sample a pixel or an average colour in the image using the colour picker. Lustre also provides colour presets as a starting point for the key. You can also use previously saved keyer presets. See Saving and Loading Presets Using the Presets Lists (page 2051).

To sample a colour:

1 Set the view mode to Output. You can click the O button.

NOTE Pressing `F12` to set the view mode to Output only works from Matte view.



2 In the secondary layers panel, activate a layer.



NOTE If you do not activate the secondary layer, the colour pot will not be updated with your colour sample.

3 Right-click the Keyer option box to enable the Diamond Keyer.

The letters in the Keyer option box turn bright white indicating that the Diamond Keyer is enabled.

4 Enable the Source Primary button to use primary grading results as the input to a key.

NOTE When Src: Prim is disabled, the raw scan is used as the input to the keyer, effectively bypassing any input LUTs and primary grading that have been applied to your shot.



(a) Source Primary button (b) Log to Lin button (c) Soften slider

5 Enable the Log to Lin button. See Tools Settings (page 1765).

6 Adjust the Soften slider. See Tools Settings (page 1765).

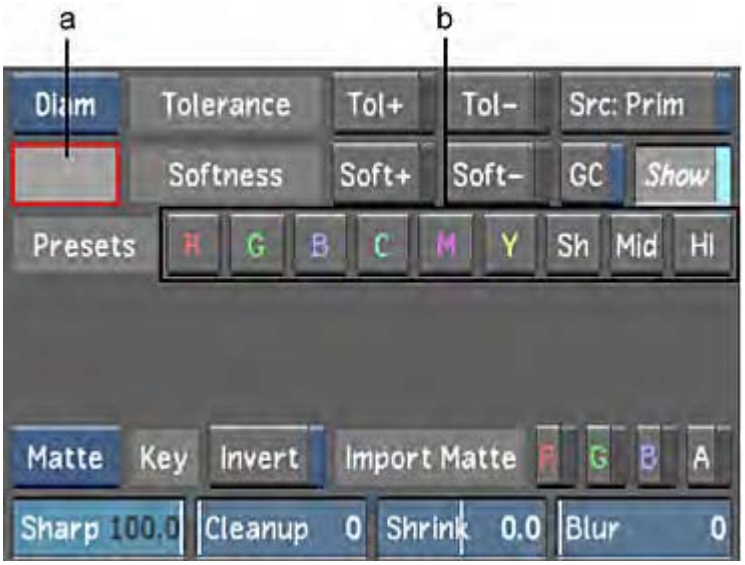
7 To sample a colour, do one of the following:

- Click the colour pot.

  The colour pot is outlined in red.

- Click a Presets option.

  The colour presets allow you to quickly select a starting point for your key. The R, G, B, C, M, and Y colour presets position the chrominance Tolerance and Softness Diamonds in the corresponding regions of the hue cube. The Sh (shadow), Mid (midtones), and Hi (highlights) presets select luminance ranges along the luminance axis.



(a) Colour pot (b) Presets options

8 Sample the image:

- To sample a single pixel, click and drag in the image until you locate the pixel you want to sample. As you drag, the colour of the current pixel appears in the colour pot. When you locate the pixel you want to sample, release the mouse.
- To sample an average taken from a range of colours in the image, `Alt`-drag the image in the Player.

The cursor changes to a colour picker when you move it over the image in the Player. When you release the mouse, the initial chrominance range is set for softness and tolerance on the hue cube, and the colour pot displays the sampled colour.

NOTE The chrominance of the sampled colour is represented by a black dot on the hue cube and the luminance of the sampled colour is represented by an orange bar in the Luminance gradient.

9 If the colour pot is enabled, click it again to finish defining the colour.

Viewing Secondaries in the Diamond Keyer

Once you have sampled a colour for the key, you can view the resulting secondary in the Player. You can view the key in Secondary view or Matte view by toggling the `F11` key. Secondary view displays the keyed out colour as is; unsampled colours are overlayed with a uniform colour by default. Matte view displays the alpha channel, a black and white template indicating which parts of the image are transparent (black), and which are opaque and selected (white). The white parts of the image can be colour graded.

To view a secondary:

1 Select the secondary you want to view.

2 Display the image in Secondary view by pressing `F11`.

In Secondary view, the colour that is keyed out in the current secondary is unchanged; unsampled colours are overlayed with a uniform colour by default. See Matte Overlay Settings (page 1790).



3 Press F11 to display Matte view.

In Matte view, the white parts of the matte represent the selected colours that can be colour graded. The black parts represent the unselected colours in the secondary. Greys indicate the zone of softness.



Setting the Tolerance Range in the Diamond Keyer

You can set the tolerance for a range of colours to be fully keyed (transparent) and to remove unwanted greys from the key. You can modify the chrominance and luminance levels for tolerance across the entire image or just in the shadows, midtones, and highlights. The Tolerance Diamond defines the range of colours included in the secondary.

To set the chrominance range for tolerance, you can manipulate the Tolerance Diamond or sample the chrominance and luminance of the image using the Add Tolerance and Remove Tolerance buttons. You can also use the Minimum and Maximum Luminance sliders to set the luminance levels for tolerance.

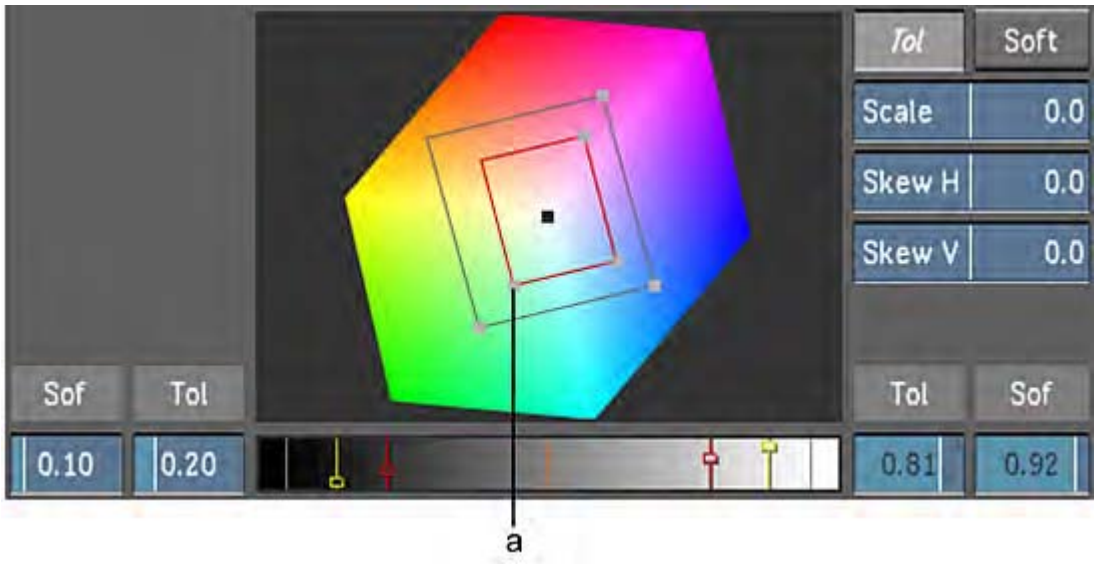
To set the chrominance range for tolerance by moving and modifying the Tolerance Diamond:

1 View the secondary in the Player. Press F11 to toggle to Secondary view.

In Secondary view, you can see the effect of the modifications you make to the chrominance range for tolerance while you fine-tune the key.

2 Click and drag the Tolerance Diamond or click and drag a vertex of the Tolerance Diamond.

NOTE If the Tolerance Diamond vertex is too small to drag with the mouse, you can use the scale and skew controls to expand the diamond. See Using the Scale and Skew Controls in the Diamond Keyer (page 2133). You can also zoom in and out on the hue cube. Right-click and drag to the right to zoom in or drag to the left to zoom out. The Tolerance Diamond is red when selected and grey when unselected.

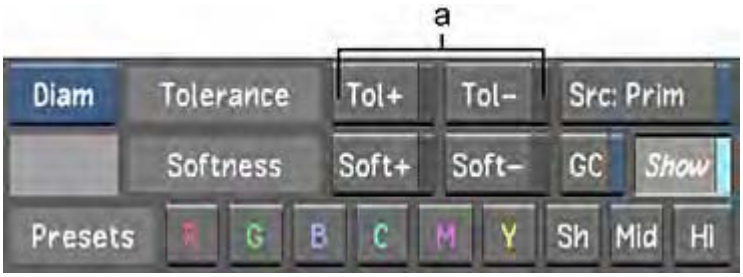


(a) Tolerance Diamond vertex

The chrominance range for tolerance is modified.

To set the luminance levels and chrominance range for tolerance using the Add/Remove Tolerance buttons:

1. View the secondary in the Player. Press `F11` to toggle between Secondary view and Matte view.
2. In the Secondaries menu, enable the Add Tolerance or Remove Tolerance button to increase or decrease the chrominance and luminance levels for tolerance.



(a) Add Tolerance and Remove Tolerance buttons

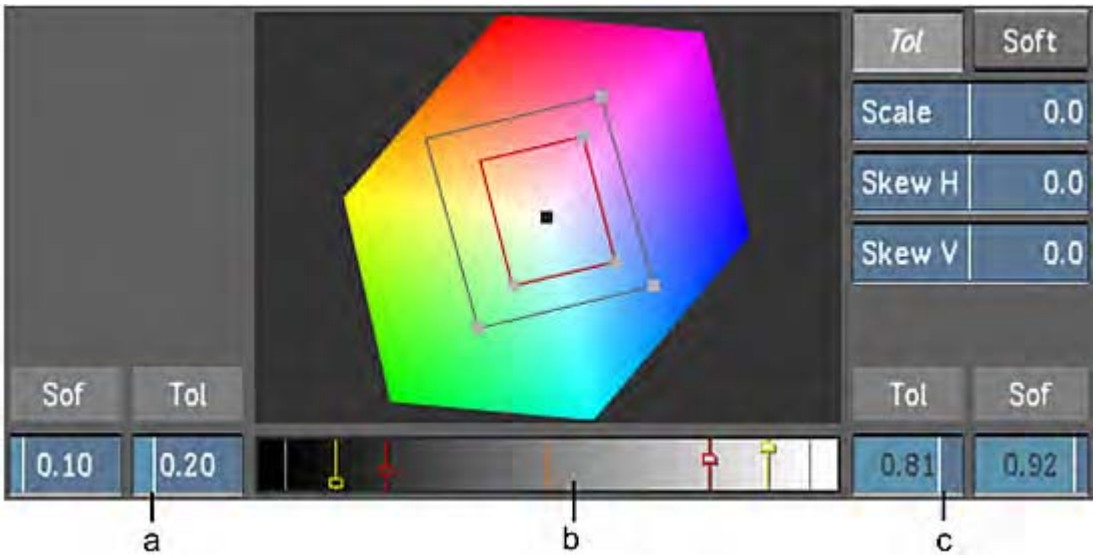
3. Click or drag in the image. `Alt`-drag in the image to select a larger area.

   The chrominance range and luminance levels for tolerance are modified.

To set the luminance levels for tolerance using the Minimum and Maximum Luminance sliders:

1. View the secondary in the Player. Press `F11` to toggle to Matte view.

   In Matte view, you can see the effect of the modifications you make to the luminance levels for tolerance while you fine-tune the key.
2. Drag the Minimum and Maximum Luminance sliders (next to the Luminance gradient) to set the luminance levels for tolerance.



(a) Minimum Luminance slider for tolerance (b) Luminance gradient (c) Maximum Luminance slider for tolerance

NOTE As you move the Minimum and Maximum Luminance sliders for tolerance, the luminance levels are represented in the Luminance gradient by corresponding red markers.

The luminance levels for tolerance are modified.

Setting the Softness Range in the Diamond Keyer

Once you have set the tolerance range, you can then set the softness range. Softness sets the range of colours to make semi-transparent and creates a softer key. Use softness to create a more natural and softer look when applying secondary colour correction.

To set the chrominance range for softness, you can manipulate the chrominance Softness Diamond. You can use the Add Softness and Remove Softness buttons to set the chrominance range and luminance levels for softness. You can also use the Minimum and Maximum Luminance sliders to set the luminance levels for softness.

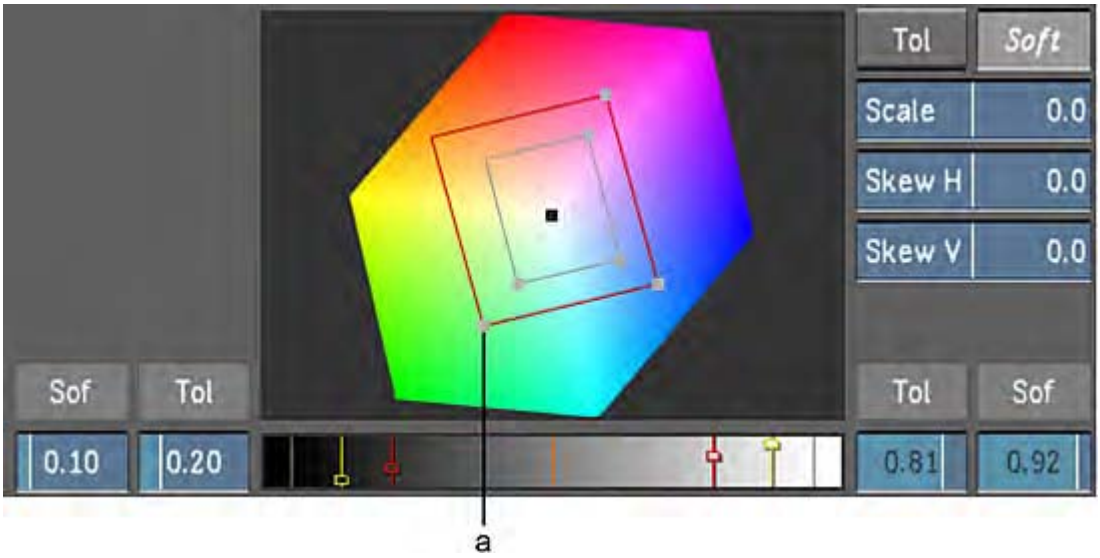
To set the chrominance range for softness by moving and modifying the Softness Diamond:

1. View the secondary in the Player. Press `F11` to toggle to Secondary view.

   In Secondary view, you can see the effect of the modifications you make while you fine-tune the key.

2. Click and drag the Softness Diamond or click and drag a vertex of the Softness Diamond.

   NOTE If the Softness Diamond vertex is too small to drag with the mouse, you can use the scale and skew controls to expand the diamond. See Using the Scale and Skew Controls in the Diamond Keyer (page 2133). You can also zoom in or out on the hue cube. Right-click and drag to the right to zoom in or drag to the left to zoom out. The Softness Diamond is red when selected and grey when unselected.

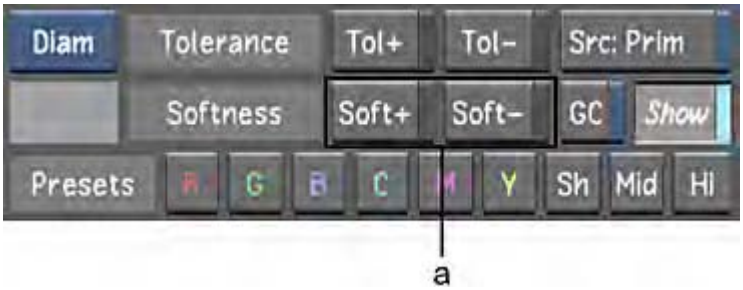


(a) Softness Diamond vertex

The chrominance range for softness is modified.

To set the chrominance range and luminance levels for softness using the Add Softness and Remove Softness buttons:

1. View the secondary in the Player. Press `F11` to toggle between Secondary view and Matte view.
2. In the Secondaries menu, enable the Add Softness or Remove Softness button to increase or decrease the chrominance range and luminance levels for softness.



(a) Add Softness and Remove Softness buttons

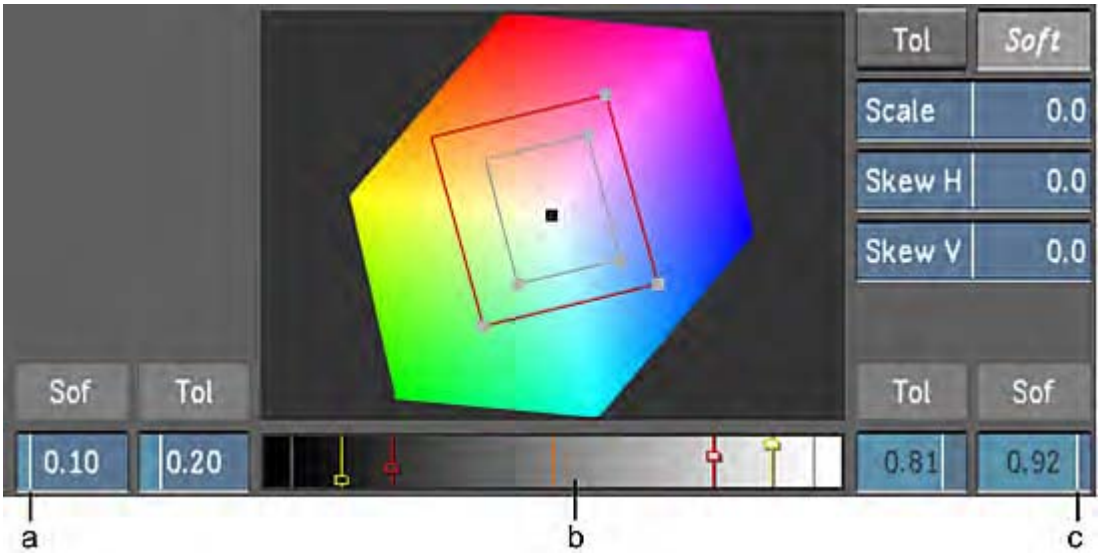
3. Click or drag in the image. `Alt`-drag in the image to select a larger area.

   The chrominance and luminance levels for softness are modified.

To set the luminance levels for softness using the Minimum and Maximum Luminance sliders:

1. View the secondary in the Player. Press `F11` to toggle to Matte view.

   In Matte view, you can see the effect of the modifications you make while you fine-tune the key.
2. Drag the Minimum and Maximum Luminance sliders (next to the Luminance gradient) to adjust the luminance levels for softness.



(a) Minimum Luminance slider for softness (b) Luminance gradient (c) Maximum Luminance slider for softness

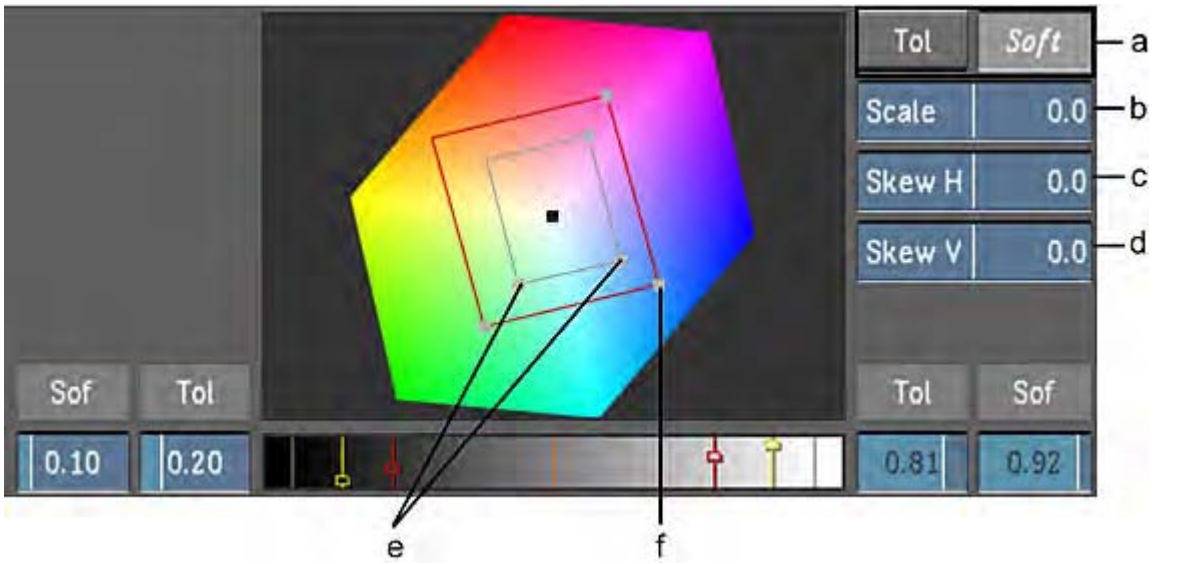
NOTE As you move the Minimum and Maximum Luminance sliders for softness, the luminance levels are represented in the Luminance gradient by corresponding yellow markers.

The luminance levels for softness are modified.

Using the Scale and Skew Controls in the Diamond Keyer

With the Scale control, you can scale the chrominance Softness and Tolerance Diamonds to increase or decrease the range of colour to isolate. With the Skew controls, you can skew the chrominance Softness and Tolerance Diamonds horizontally and vertically relative to the orientation of the selected diamond to increase or decrease the range of colour to isolate.

NOTE To zoom in or out on the hue cube, right-click and drag to the right to zoom in or drag to the left to zoom out.



(a) Tolerance and Softness selectors (b) Scale slider (c) Skew Horizontal slider (d) Skew Vertical slider (e) Horizontal control vertices (f) Vertical control vertex

To scale the range of colour:

1. View the secondary in the Player. Press F11 to toggle to Secondary view.

   In Secondary view, you can see the effect of the modifications you make while you fine-tune the key.
2. Click the Tolerance button or Softness button to select the Tolerance Diamond or Softness Diamond.
3. Drag the Scale slider to the left or to the right to decrease or increase the range of colour to isolate.

To skew the range of colour horizontally:

1. View the secondary in the Player. Press F11 to toggle to Secondary view.

   In Secondary view, you can see the modifications you make to the chrominance range while you fine-tune the key.
2. Click the Tolerance button or Softness button to select the Tolerance Diamond or Softness Diamond.
3. Depending on the orientation of the diamond, drag the Skew slider to the left to horizontally decrease the range of colour to isolate or to the right to horizontally increase the range of colour to isolate.

To skew the range of colour vertically:

1. View the secondary in the Player. Press F11 to toggle to Secondary view.

   In Secondary view, you can see the modifications you make to the chrominance range while you fine-tune the key.
2. Click the Tolerance button or Softness button to select the Tolerance Diamond or Softness Diamond.
3. Depending on the orientation of the diamond, drag the Skew slider to the left to vertically decrease the range of colour to isolate or to the right to vertically increase the range of colour to isolate.

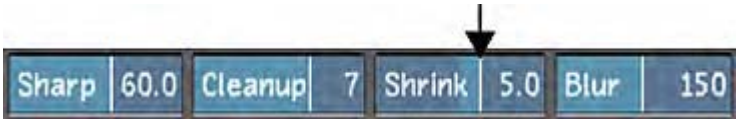
Modifying the Edges of the Key

You can shrink, erode, or blur the edges of a key. Shrink a key to remove pixels from the edge of the key. Erode a key to blend the light and dark edges. Blur a key to apply a softening filter to its edge.

NOTE The following steps apply to both the HLS and Diamond Keyers.

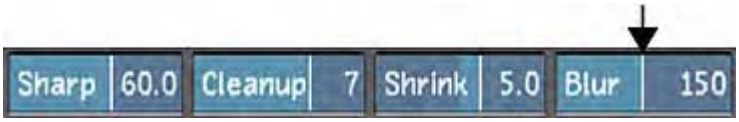
To shrink or erode the edge of the key:

1. View the secondary in the Player. Press F11 to toggle between Matte view and Secondary view.
2. Fine-tune the edge of the key:
   - To shrink the edge of the key, drag the Shrink slider to the left.
   - To erode the edge of the key, drag the Shrink slider to the right.



To blur the edge of the key:

1. View the secondary in the Player. Press F11 to toggle between Matte view and Secondary view.
2. Drag the Blur slider to the right.



The edge of the key is softened.



Before



After

Images courtesy of Hungarian Academy of Film & Theatre, 3rd year

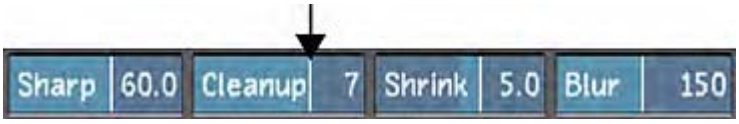
Removing Stray Pixels from a Key

When you extract a key, pixels in other parts of the image may be included. You can remove these stray pixels from the key to clean it up for colour grading.

NOTE The following steps apply to both the HLS and Diamond Keyers.

To remove stray pixels:

1. View the secondary in the Player. Press `F11` to toggle between Matte view and Secondary view.
2. Drag the Cleanup slider to the right to apply a median filter to remove stray pixels.



Sharpening a Key Source Image

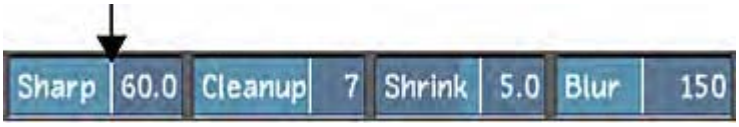
Sharpen a key source image to increase or decrease the softness of only the pixels that are within the chrominance and luminance range for tolerance.

NOTE This procedure applies to the Diamond Keyer only.

To sharpen a key:

1. View the secondary in the Player. Press `F11` to toggle between Matte view and Secondary view.

2 Drag the Sharpness slider to adjust softness and reduce noise if the matte appears grainy.



Inverting Keys

Invert a key when it is easier to exclude rather than include colours in a secondary.

NOTE The following steps apply to both the HLS and the Diamond Keyers.

To invert a key:

1 In the Secondaries menu, toggle the Matte/Key-in option box to display the Key Invert panel, and then enable Invert.



The area of the key that was opaque becomes transparent and the area that was transparent becomes opaque.



Before



After

Images courtesy of Hungarian Academy of Film & Theatre, 3rd year

Excluding Keys from Secondaries

You can exclude keys from secondaries. When you exclude a key from a secondary that includes a geometry, only the geometry is colour graded.

NOTE The following steps apply to both the HLS and the Diamond Keyers.

To exclude a key from a secondary:

1 In the Secondaries menu, select and then right-click the Secondary button that contains the key you want to exclude.

If a key is included with the secondary, the keyer is enabled.



TIP Click M in the View Mode buttons and then toggle the `F11` key to display the Matte view.

2 Right-click the Keyer option box to disable the current Keyer.

NOTE The HLS and Diamond Keyer letters turn from bright white to faded white, indicating that the keyer is disabled.



The key is excluded from the colour grade. The whole image or the areas defined by a geometry will be colour graded by the secondary.

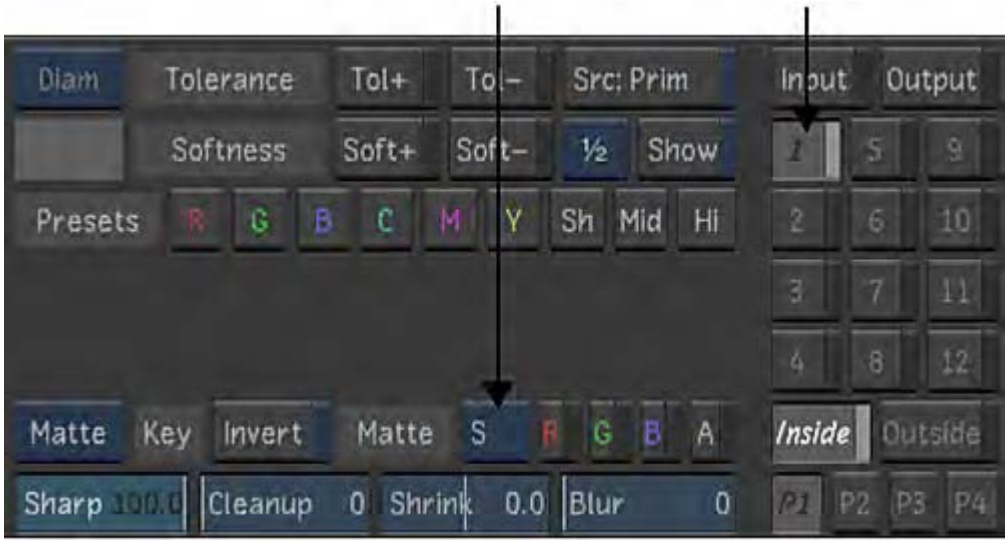
Saving and Loading Key Presets

You can save user-defined keys, or previously saved keyer presets. You can then quickly load them to the current shot for comparison. See Saving and Loading Presets Using the Presets Lists (page 2051).

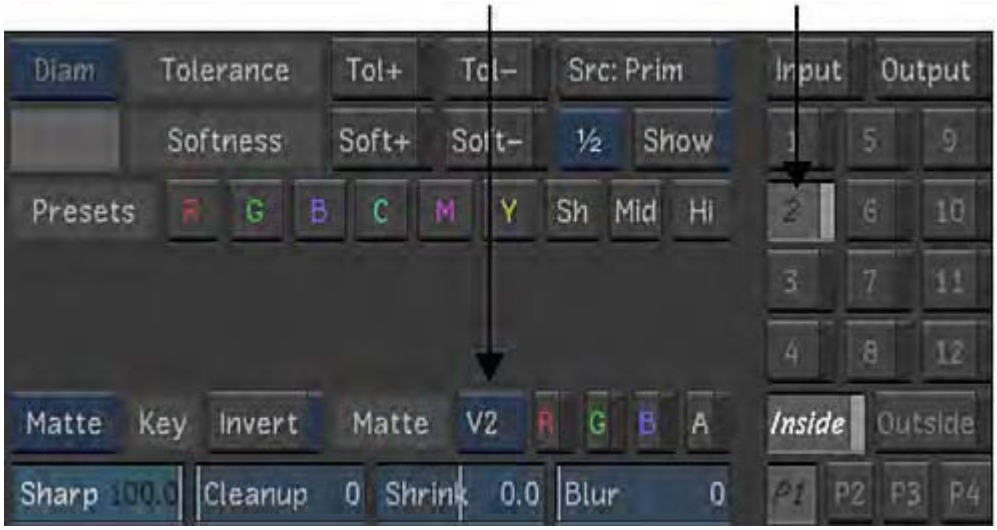
Working with Mattes



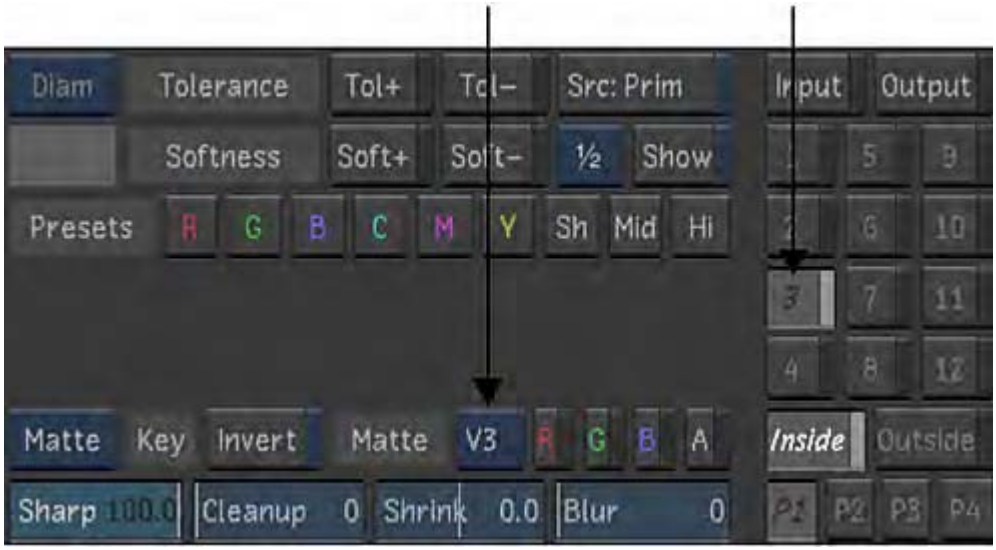
In the following screenshot, the matte on the secondary track is applied to the first secondary colour grading layer.



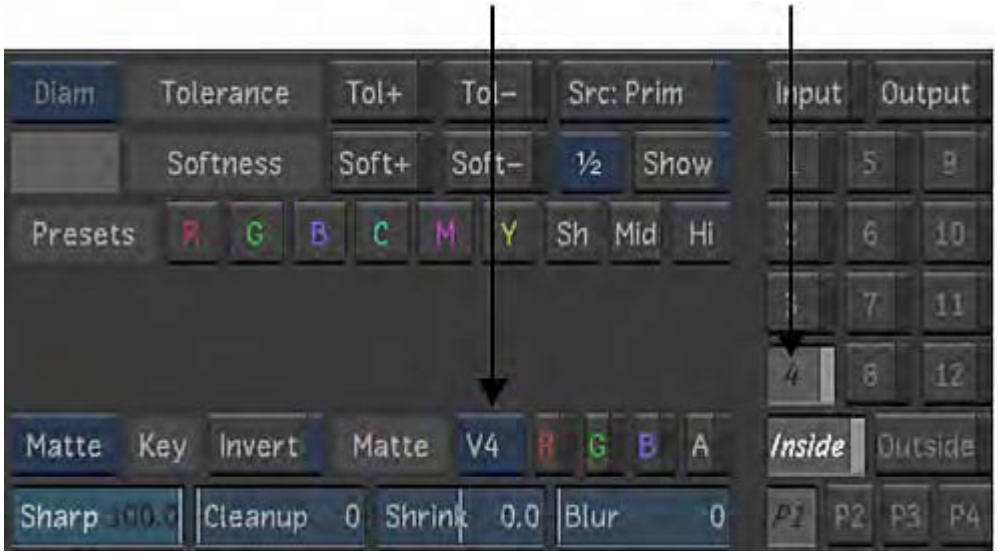
In the following screenshot, the matte on the V2 track is applied to the second secondary colour grading layer.



In the following screenshot, the matte on the V3 track is applied to the third secondary colour grading layer.



In the following screenshot, the matte on the V4 track is applied to the fourth secondary colour grading layer.



NOTE If the matte you are using does not contain an alpha channel, the alpha button is not available.

NOTE Playback performance may vary based on the number of active mattes, file formats, resolution, storage bandwidth and type of project. For example, if you have two matte tracks in a matte container that are both being used on secondaries, three playback streams of media are required. You may not achieve real-time playback based on storage bandwidth, graphics card processing, etc. Performance may also be compromised if you are working with dissolves or stereoscopic timelines.

Shape Tracker

The Shape Tracker tool is a 2-dimensional tracking device that allows you to track objects within your shot. It allows you to create a geometry around the object you want to track, analyse the object through the shot, and then grade the geometry.

To use the Shape Tracker you need to define a region of interest (ROI) by drawing a geometry around the object you want to track. The area inside this geometry is where Lustre searches for trackable points. This group of trackable points is called a point cloud. By tracking a cloud instead of a single point, Lustre can track shapes through translation, rotation, and scaling.

In order to track a shape properly, the ROI needs to be large enough so there are enough trackable points in the point cloud. If the ROI is large enough, the object in the shot will be tracked properly. If the object you want to track is small and does not contain enough trackable points, you need to link the smaller graded geometry to a larger tracked geometry.

The Shape Tracker Panel

To track 2D shapes within a shot, you need to enable the Shape Tracker tool and create an ROI.

To access the Shape Tracker panel and create an ROI:

1. In the Main menu click Colour.
2. Click Secondaries.

   The Secondaries menu with the Shape Tracker panel appears.

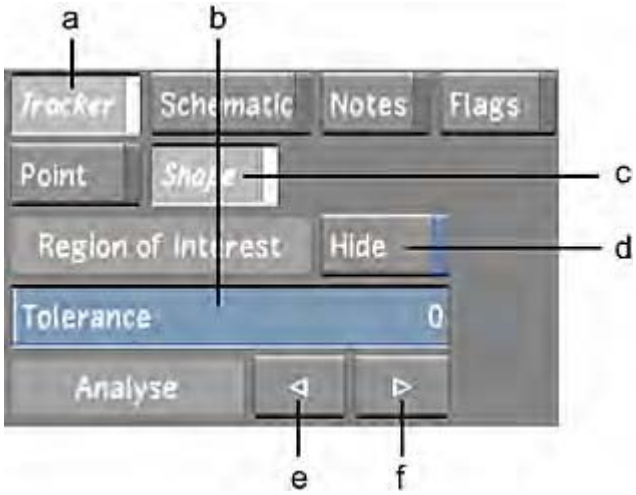


NOTE When the Diamond Keyer is selected, disable the Show button to display the Shape Tracker panel (as well as the Schematic, Notes, Flags, Pixel, Animation, Group, and Selector panels).



Parts of the Shape Tracker

The Shape Tracker is made up of the following components.



(a) Tracker button (b) Tolerance slider (c) Shape button (d) Hide button (e) Analyse backward button (f) Analyse forward button

Tracker button Enables the tracking option (Point Tracker or Shape Tracker).

Tolerance slider Defines the tolerance level of the Shape Tracker analysis. The range is from 0 to 10.

A tolerance level of 0 (low tolerance) means the object that is being tracked does not change in shape throughout the shot. If an external object passes in front of the tracked object, a low tolerance ensures that the motion of the external object does not get tracked through the analysis.

A tolerance level of 10 (high tolerance) means the object which is being tracked has a slight change in shape throughout the shot. For example, there is slight 3D movement from the shape. If you have set the tolerance level to high, external objects that pass in front of the tracked object may be included in the analysis.

Shape button Enables the Shape Tracker tool.

Hide button When enabled, it hides the selected geometry and grading (if applicable). You can also use the hotkey, left `Ctrl+H`, to hide and show the geometry.

Analyse backward button When selected, the Shape Tracker analyses the object within the selected ROI from your insertion point to the beginning of the shot.

Analyse forward button When selected, the shape tracker analyses the object within the selected ROI from your insertion point to the end of the shot.

Tracking an Object using Shape Tracker

There are two ways to use the Shape Tracker. If you can create a large enough ROI around the object you want to track, you can use the Shape Tracker to track the object and then grade the same geometry. If the object you want to track and grade is small and it does not have enough trackable points to use the Shape Tracker effectively, you need to track a larger ROI, create a graded geometry of the smaller object, and link the ROI geometry to the graded geometry.

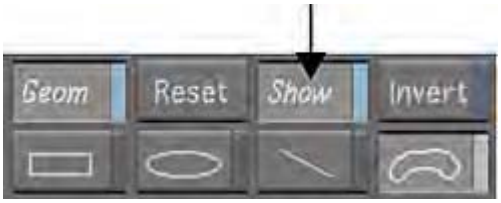
The following procedure is used when the object has a large enough ROI to track.

To track an object within a shot:

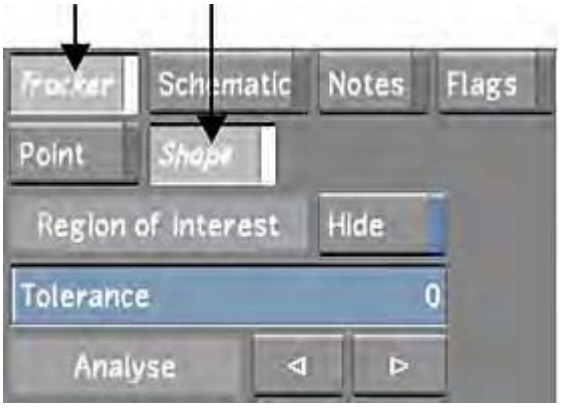
1. Make sure you are on the first frame of where you want the tracking to start.
2. In the Secondaries menu, right-click a secondary layer to activate it.



3 Click Show



4 Enable the Shape Tracker tool by clicking on Tracker and Shape.



5 Create an ROI by drawing either a basic or free-form geometry around the object you want to track. See Drawing Basic Geometries (page 2093) and Drawing Free-Form Geometries (page 2093).

NOTE Make sure your ROI is as large as possible, without including too many non-rigid shapes, in order to give Shape Tracker a sufficient area to analyse.

6 Make sure the geometry you want to track is selected.

The axis of the geometry should be yellow.

7 Set the tolerance slider between 0-10.

8 Click the analyse backward, or analyse forward button, to analyse the shot segment.

When the analysis is complete, you can grade the geometry.

NOTE If the analysis stops before the end of the shot or it does not begin at all, the ROI area may be too small to track. Refer to the following procedure below.

The following procedure is used if the object you want to track is small and does not contain enough trackable points.

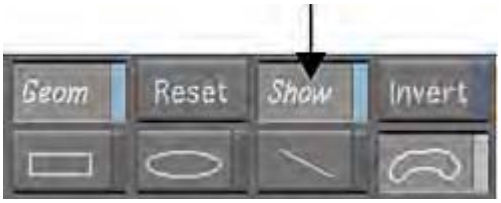
To track a small object that does not contain enough trackable points:

1 Make sure you are on the first frame of where you want the tracking to start.

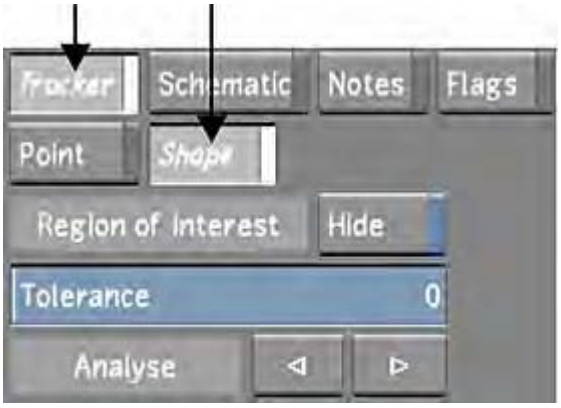
2 In the Secondaries menu, right-click a secondary layer button to activate it.



3 Click Show.



4 Enable the Shape Tracker tool by clicking on Tracker and Shape.



5 Create an ROI by drawing a basic geometry around a large area in the shot. See Drawing Basic Geometries (page 2093).

NOTE Depending upon your shot, the large ROI area may or may not contain the smaller object you want to track.

6 Make sure the geometry you want to track is selected.

The axis of the geometry should be yellow.

7 Set the tolerance slider between 0-10.

8 Click the analyse backward, or analyse forward button, to analyse the shot segment.

The analysis is now complete.

9 Click Hide.

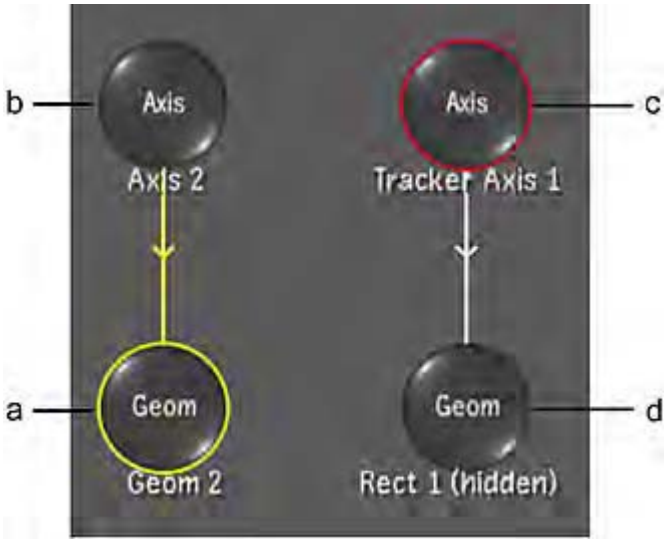
The tracked geometry is now hidden.

10 Create a geometry around the object you want to grade.

11 Grade this new geometry.

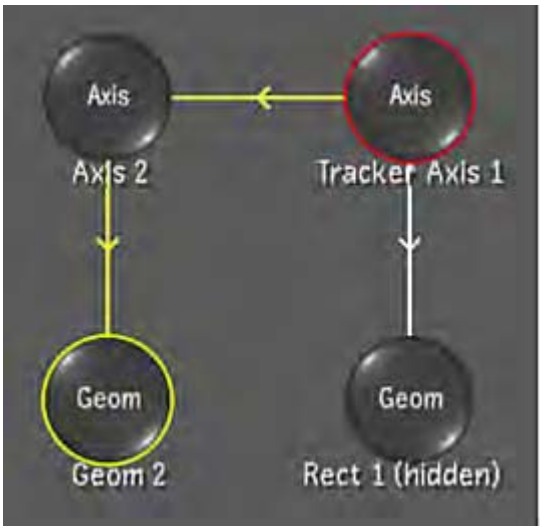
12 Press ~ (the tilde key) to enter the Schematic view.

You should see the axis and geometry of the tracked shape and the graded shape.



(a) Geometry of graded shape (b) Axis of graded shape (c) Axis of ROI (d) Geometry of ROI

13 Position your cursor over the edge of the circle representing the ROI axis and left-click and drag a line to the edge of the circle representing the graded shape axis.



The graded shape is now linked to the ROI and therefore follows its tracked path.

14 Press ~ to return to the Player view.

Animating the Point Tracker

Point Tracker is a useful tool for tracking points through a shot. In situations where tracking a point cloud is impossible, Point Tracker provides an excellent alternative. For example, if Shape Tracker's ROI provides no rigid shapes to track, Point Tracker's ability to track a point remains the best choice. With Point Tracker, you can animate a geometry by applying tracking data to it. Users can assign one or several trackers to a geometry. This is useful when you want a geometry to follow a moving element in a shot. With Point Tracker, you can:

- Animate the position of an entire geometry without changing its shape by assigning a tracker to the geometry's axis.

- Animate the shape of a geometry by assigning a tracker to each vertex. If you are animating a basic geometry, convert it to a free-form geometry first.

When you apply tracking data to a geometry, only the axis or the position of the vertices is animated. Because you can assign multiple trackers to a geometry, you can assign trackers to both its axis and vertices.

Tracking data appears in the Channel Editor. Tracking data for each tracker is contained in a Trackers directory of the channel hierarchy. Use the channel hierarchy to select the tracker and then modify the tracking data. In the channel hierarchy, the channels for each tracker appear.

Tracker Channel	Description
Ref X	The point being tracked on the X-axis
Ref Y	The point being tracked on the Y-axis
Track X	Last tracked (or current) tracker position along the X-axis
Track Y	Last tracked (or current) tracker position along the Y-axis

You can also animate the softness of geometries and the position of their individual vertices using the Animation controls. See Animation (page 2171).

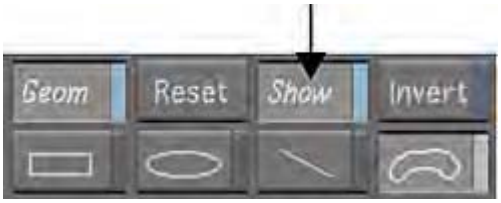
The Point Tracker is further improved with sub-pixel accuracy. Subsequently, any subtle movement (including movements less than a single pixel in dimension) can now be analysed and written to the Lustre Animation Channel.

Animating the Position of Geometries

You can animate the position of geometries by defining a Point Tracker and assigning it to the axis of the geometry.

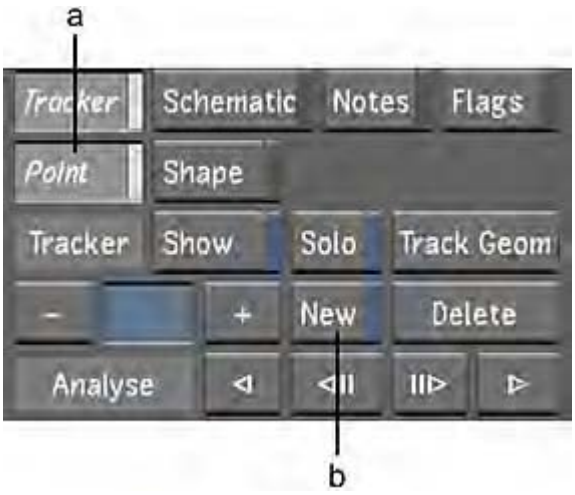
To animate the position of a geometry with the Point Tracker:

1. In the Secondaries menu, activate the secondary layer button that contains the geometry you want to track.
2. Click Show.



3. The Point Tracker option should already be enabled by default.

   NOTE When the Diamond Keyer is selected, disable the Show button (beneath the Source Primary button) to display the Tracker panel.

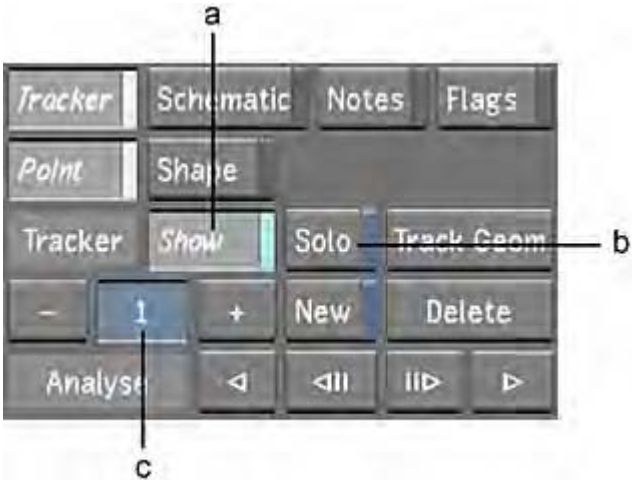


(a) Point Tracker button (b) New button

4 Enable a new tracker by clicking New.

5 In the Player, go to the frame where you want to start tracking, and then click a tracking point in the image.

In the Player, the tracker is assigned to, and positioned over, the tracking point. A tracker number is assigned to the tracker and the tracker is set to On.



(a) Tracker status (b) Solo/Gang button (c) Tracker list

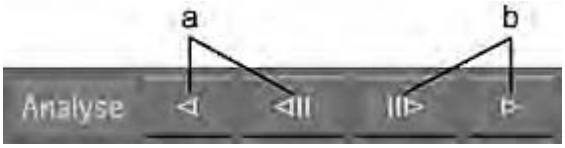
When a tracker is on, it is displayed in the Player and can be analysed.

6 Disable the Solo/Gang button to select Solo mode.

In Solo mode, only the current tracker is analysed.

7 Use the Analyse buttons to analyse the shot:

- To track the tracking point forward in time, click the Analyse Forward buttons.
- To track the tracking point backward in time, click the Analyse Backward buttons.



(a) Analyse Backward buttons (b) Analyse Forward buttons

The system analyses the image and generates tracking data. For each frame in the shot, the position of the tracking point is displayed in the Player. The current position of the tracking point is highlighted in magenta.

8 Enable Assign.



9 In the Player, click the geometry's axis.

The tracker is assigned to the geometry's axis or vertex.

10 If the tracker loses the tracking point, click the image to stop the process, and then use the Step Backward or Step Forward button in the Player controls to return to the last frame that was correctly tracked.

11 In the Player, do any of the following:

- To resize the tracker, modify the Match area (magenta rectangle) and Search area (yellow rectangle) by dragging the vertices of the rectangles.
- To reposition the tracker, drag the centre point of the tracker.
- To reposition the tracker and reference point, middle-click and drag the centre point of the tracker.

TIP You can reposition the tracker and reference point when the reference point moves off-screen.

12 Continue using the Analyse buttons to analyse the shot.

13 To manually add or reposition a tracking point, go to the frame where you want to modify the tracking data, enable Add Key, and then click a tracking point in the image.



The position of the tracking point is displayed in the image.

14 To manually delete the position of a tracking point, go to the frame where you want to remove the tracking data, and then click Del Key.



The tracking data is deleted.

NOTE Linear interpolation is used to set the tracking position for frames without tracking data. For these frames, the previous defined tracking position and the next defined position are used to interpolate the tracking position.

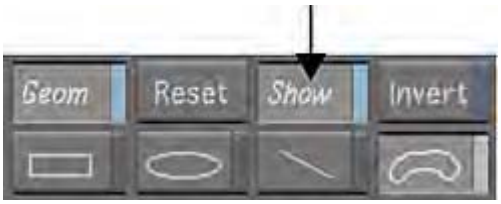
Animating the Shape of Geometries with the Point Tracker

You can use the Track Geom button to animate the shape of a geometry when using Point Tracker. When you animate the shape of a geometry, the vertices of the geometry change position as the moving elements in the shot are tracked.

When you use Track Geom, a Point Tracker is simultaneously assigned to each vertex in the current geometry. You then analyse the shot to generate the tracking data.

To animate the shape of a geometry:

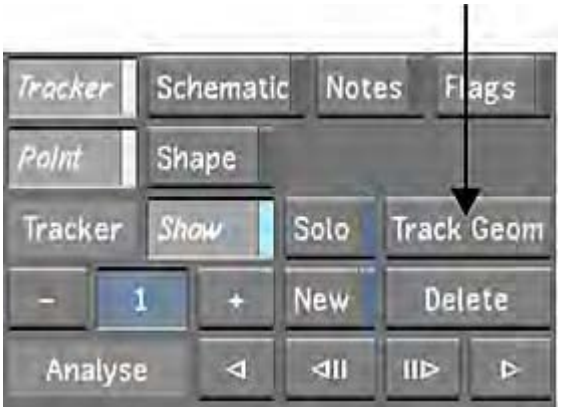
1. If working with a basic shape, right-click it in the Player to convert it to a free-form geometry.
2. In the Secondaries menu, activate the secondary layer button that contains the geometry you want to track.
3. Enable Show.



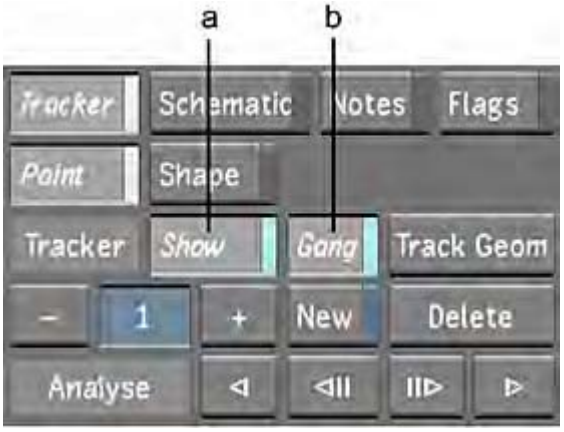
The geometries are displayed in the Player.

4. Go to the frame where you want to start tracking.
5. Select the geometry you want to track. In the Player, draw a selection box around the geometry to select all vertices on the shape. Alternatively, click a vertex to select it.
6. Click Track Geom.

NOTE When the Diamond Keyer is selected, disable the Show button (beneath the Source Primary button) to display the Tracker panel.



In the Player, a Point Tracker is assigned to, and positioned over, each vertex on the shape. A tracker number is assigned to each tracker. Gang mode is selected and the Point Trackers are turned on.



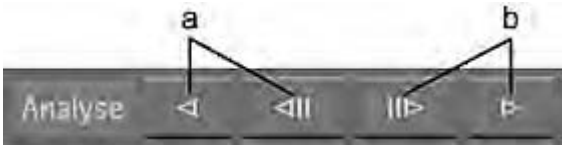
(a) Tracker status (b) Gang mode

When a Point Tracker is on, it is displayed in the Player and can be analysed. When Gang mode is selected, all Point Trackers assigned to the geometry (and any others included in the gang) will be analysed simultaneously.

NOTE If other Point Trackers exist, they will be turned off and hidden so as not to be included in the gang. However, if they are assigned to another geometry, their tracking data will still be applied.

7 Use the Analyse buttons to analyse the shot:

- To track the tracking point forward in time, click the Analyse Forward button.
- To track the tracking point backward in time, click the Analyse Backward button.



(a) Analyse Backward buttons (b) Analyse Forward buttons

The system analyses the shot and generates the tracking data.

8 If the Point Tracker loses the tracking point, click the image to stop the process, and then use the Step Backward or Step Forward button in the Player controls to return to the last frame that was correctly tracked.

9 In the Player, do any of the following:

- To resize the tracker, modify the Match area (magenta rectangle) and Search area (yellow rectangle) by dragging the vertices of the rectangles.
- To reposition the Point Tracker, drag the centre point of the tracker.
- To reposition the Point Tracker and reference point, middle-click and drag the centre point of the tracker.

TIP You can reposition the Point Tracker and reference point when the reference point moves off-screen.

10 Continue using the Analyse buttons to analyse the shot.

11 To manually add or reposition a tracking point, go to the frame where you want to modify the tracking data, enable Add Key, and then click a tracking point in the image.



The position of the tracking point is displayed in the image.

12 To manually delete the position of a tracking point, go to the frame where you want to remove the tracking data, and then click Del Key.



The tracking data is deleted.

NOTE Linear interpolation is used to set the tracking position for frames without tracking data. For these frames, the previous defined tracking position and the next defined position are used to interpolate the tracking position.

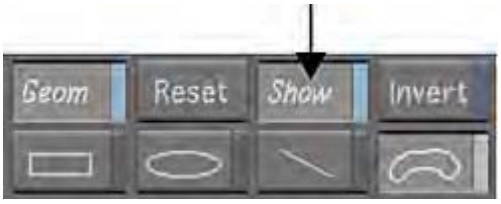
Assigning and Unassigning Point Trackers to Geometries

You can assign Point Trackers to, or unassign Point Trackers from, geometries at any time after you create them.

You can assign a Point Tracker to the axis of any geometry when you want to animate the entire geometry with or without changing its shape. You can also assign a Point Tracker to a vertex of a geometry when you want part of the edge of the geometry to change as it tracks a moving element in the shot. However, if you want to assign Point Trackers to all the vertices in a geometry, use Track Shape. See Animating the Shape of Geometries with the Point Tracker (page 2147).

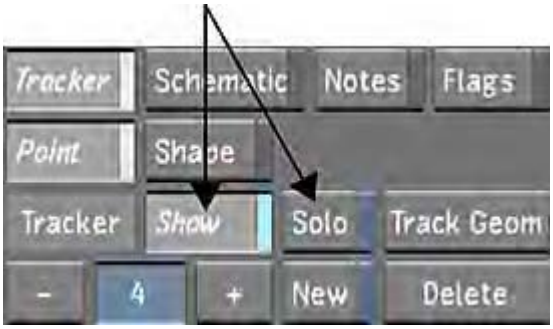
To assign a Point Tracker to a geometry:

1. In the Secondaries menu, activate the secondary layer button that contains the geometry you want to track.
2. Enable Show.



The geometry is displayed in the Player.

3. Do one of the following:
   - Add a new Point Tracker.
   - Click the + or - button to select a Point Tracker from the Tracker list. Make sure the tracker is set to On and is in Solo mode.



The current Point Tracker is displayed in the Player.

4. Enable Assign.



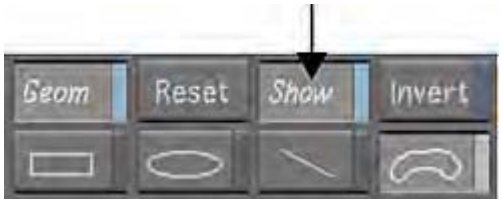
5. In the Player, do one of the following:
   - To animate the entire geometry without changing its shape, click the geometry's axis.
   - To animate a single vertex in a geometry, click the vertex.

   The Point Tracker is assigned to the geometry's axis or vertex.

NOTE You cannot assign a Point Tracker to more than one point on a geometry. If the tracker is already assigned to a vertex or axis on the geometry, you must unassign it.

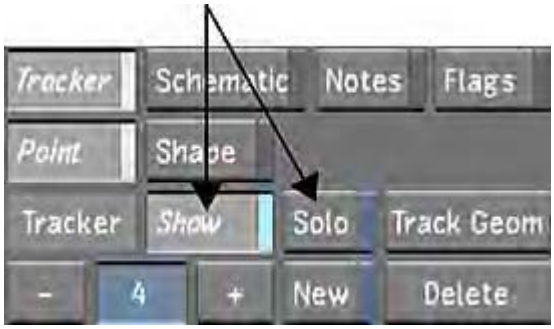
To unassign a Point Tracker from a geometry:

1. Select the shot that contains the tracking data.
2. In the Secondaries menu, enable Show.



The geometry is displayed in the Player.

3. Click the + or - button to select a Point Tracker from the Tracker list. Make sure the tracker is set to On and is in Solo mode.



The current tracker is displayed in the Player.

4. Enable Unassign and then click the vertex or axis to which you assigned the tracker.



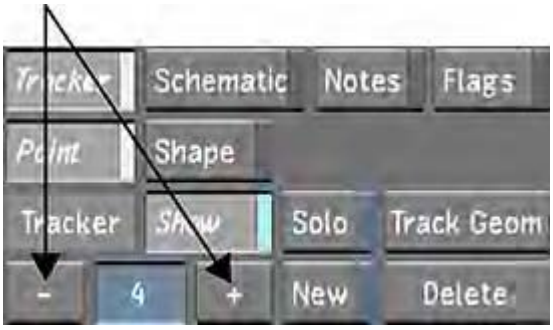
The Point Tracker is unassigned to the vertex but remains in the Tracker list.

Soloing and Ganging Point Trackers

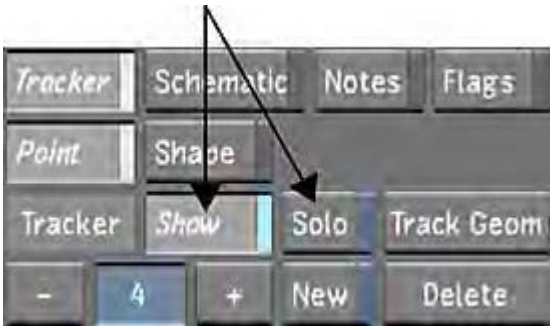
You can solo or gang Point Trackers. Solo a Point Tracker when you want to view and generate tracking data for the current tracker only. Gang Point Trackers when you want to view and generate tracking data for multiple Point Trackers at the same time.

To solo a Point Tracker:

1. Click the + or - button to select a Point Tracker from the Tracker list.



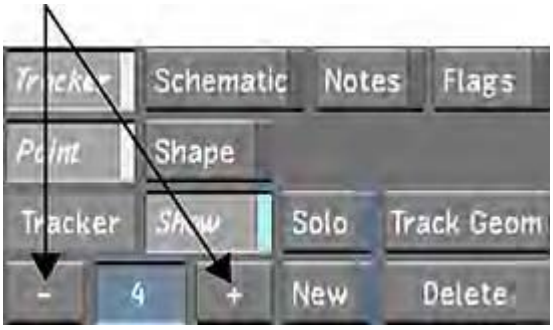
2 Make sure the Point Tracker is set to On and is in Solo mode.



Only the current Point Tracker is displayed in the Player.

To gang Point Trackers:

1 Click the + or - button to select a Point Tracker from the Tracker list.



2 Make sure the tracker is set to On and is in Gang mode.



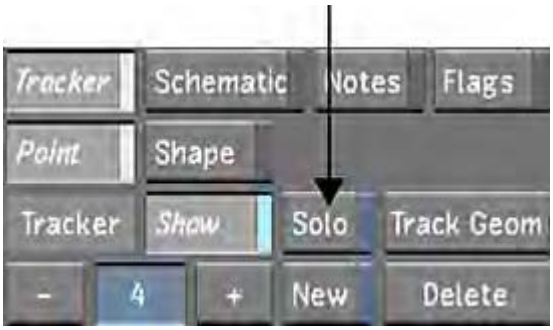
3 Repeat steps 1 and 2 for all the Point Trackers you want to add to the gang.
The ganged trackers that are set to On are displayed in the Player and can be analysed as a group.

Deleting Point Trackers

You can delete Point Trackers from a shot.

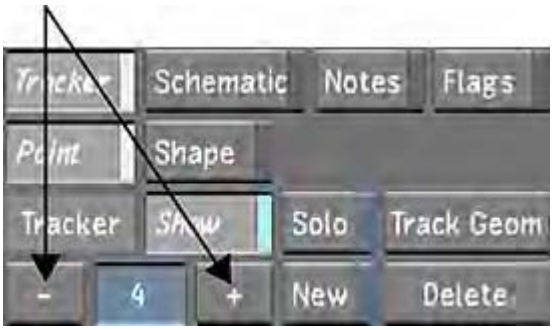
To delete a Point Tracker:

1 Select the shot that contains the tracking data.
2 In the Secondaries menu, select Solo mode.



When you use Solo mode, only the current Point Tracker is displayed in the Player. If the tracker does not appear, set the tracker to On.

3 Click the + or - button to scroll to the Point Tracker you want to delete.



If the current Point Tracker is on, it is displayed in the Player.

4 Click Delete.



The Point Tracker is deleted.

Loading Tracker and Stabilizer Data from Autodesk Applications

You can load tracker and stabilizer data created with other Autodesk products (Flint®, Flame®, Inferno®, Fire®, Smoke®, and Autodesk Combustion®) into Lustre. To import tracker and stabilizer data, you must export it from the applicable Autodesk product as a raw setup file, and then save it in the *C:\Program Files\Autodesk\<version number>\settings\geompresets* folder (Windows), or in the */usr/autodesk/<version number>/settings/geompresets* directory (Linux).

NOTE You must create the *geompresets* directory manually to store your raw setup files. However, if you have created a preset, the *geompresets* directory is automatically generated.

To load tracker or stabilizer data:

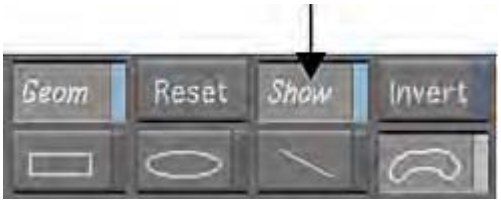
1 Save a tracker or stabilizer raw setup file from the applicable Autodesk product.

2 Do one of the following:

- If using the Windows version of Lustre, store the raw setup file in the *C:\Program Files\Autodesk\<version number>\settings\geompresets* folder.
- If using the Linux version of Lustre, store the raw setup file in the */usr/autodesk/<version number>/settings/geompresets* directory.

3 In the Secondaries menu, enable the secondary layer button that contains the geometry you want to track.

4 Enable Show.



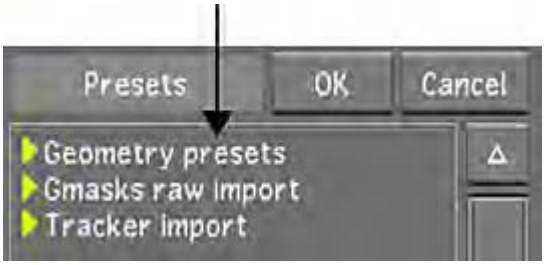
The geometries are displayed in the Player.

5 Click Tracker.



6 Click Point.

7 Click Presets to display the Presets list.



8 Expand Tracker import, and then select the tracker or stabilizer file.

9 Click OK.

The tracker appears in the Player and can be assigned to the points or axes of geometries.

Loading Garbage Mask Setups from Autodesk Applications

You can load garbage mask setups created with other Autodesk products (Flint, Flame, Inferno, Fire, and Smoke) to use as geometries in Lustre. To import a garbage mask setup, you must export it from the applicable Autodesk product as a raw setup file, and then save it to the Lustre program directory. You can then load the garbage mask setup to the current secondary for use as a geometry.

NOTE You must create the *geompresets* directory manually to store your raw setup files. However, if you have created a preset, the *geompresets* directory is automatically generated.

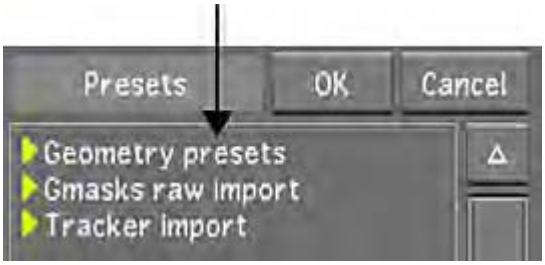
To load a garbage mask setup:

1 Export a raw setup file from the applicable Autodesk product.

2 Do one of the following:

- If using the Windows version of Lustre, store the raw setup file in the *C:\Program Files\Autodesk\<version number>\settings\geompresets* folder.
- If using the Linux version of Lustre, store the raw setup file in the */usr/autodesk/<version number>/settings/geompresets* directory.

3 Click Presets to display the Presets list.



4 Expand Gmasks raw import, and then select the raw setup file.

5 Click OK to load the garbage mask setup.

The geometry appears in the Player.

Creating Lustre Sparks Effects

You can apply effects to your shots using Lustre Sparks plugins created by Autodesk or third-party developers. These plugins are image processing effects that you can add to individual shots. You can apply up to six effects to the same shot.

Lustre Sparks plugins occupy a specific, fixed spot in the processing pipeline in Lustre— between the secondary and output colour grading. You can therefore create them anytime and the effect will be the same. It is suggested to add them after the secondary grading process so that you can see the effect on the graded shot.

Lustre Sparks plugins are integrated into the application and function in the same way as any effect in Lustre. For example, as with colour grading effects, you can apply a plugin created on one shot to another shot. When you save the grade, plugin data is saved with colour grading data in the grade file. Likewise, Lustre Sparks plugins are rendered with all other effects in the processing pipeline and are reset in the same way as other effects.

Installing Lustre Sparks Plugins

Lustre Sparks plugins are installed automatically during Lustre installation. They are included with all Lustre configurations, as well as with all render components, including the Linux render farm package for background rendering.

NOTE You can modify plugin parameters on the Master Station or the Lustre HD Station. On the Lustre Station, you can only view the plugins.

All Lustre Sparks plugins reside in the plugins directory. If developing your own plugins, you must place them in the plugins directory and they must have a **.dll* extension. If running the Windows version of Lustre, save your plugins in the *C:\Program Files\Autodesk\<version number>\plugins* folder. If running the Linux version of Lustre, save your plugins in the */usr/autodesk/<version number>/plugins* directory.

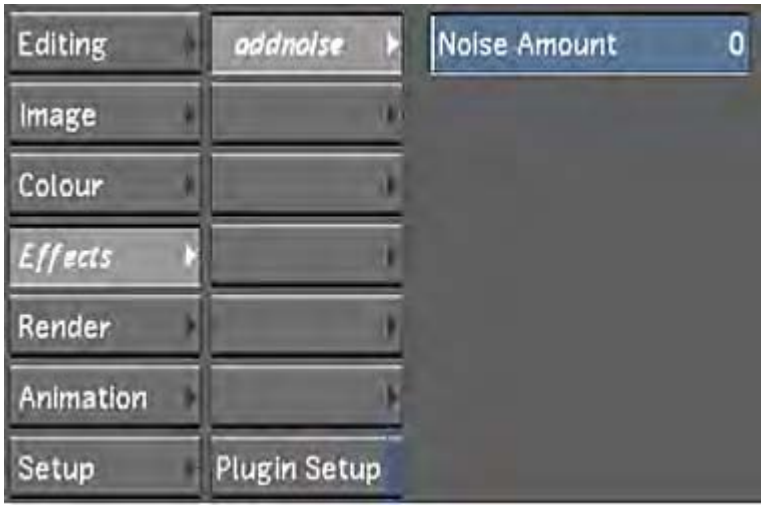
Removing a Lustre Sparks plugin consists of deleting it from the plugins directory.

For details on developing Lustre Sparks plugins, see Developing Lustre Sparks Plugins (page 2171).

Lustre Sparks Plugin Descriptions

There are several Lustre Sparks plugins that are included with Lustre. These plugins, and their parameters, are described in the following sections.

Add Noise Grain-type plugin that you can use for adding random noise to an image. Use the Noise Amount parameter to adjust the amount of noise added to the shot.



Blur Mix Blur-type plugin that performs a multiply mix between the original image with the blurred image. You can also use this plugin to sharpen or tint an image.



Use:	To:
Defocus	Set the amount of blur.
Aspect	Adjust the orientation of the blur. A negative value produces a horizontal blur and a positive value produces a vertical blur.
Mix	Adjust the amount of mix between the original image and the unblurred image. By default, this value is set to 1. If you set the value to zero, no mix occurs and you see only the original image. Setting the mix to a negative value sharpens the image.
Boost	Multiply key values when using the plugin on a secondary key. This allows you to visually grow your key by increasing the range of the key value.

Use:	To:
Red Mix	Adjust the red gain of the blurred image.
Green Mix	Adjust the green gain of the blurred image.
Blue Mix	Adjust the blue gain of the blurred image.

Blur Simple blur plugin using a high-speed algorithm.



Use:	To:
Kernel Size	Adjust the amount of blur.
Aspect	Adjust the orientation of the blur. A negative value produces a horizontal blur and a positive value produces a vertical blur.

Defocus Blur-type plugin that simulates a camera out-of-focus effect. Along with the blur parameters, this plugin includes variable iris types, additional highlight controls, as well as a control for softening the edge of a secondary.



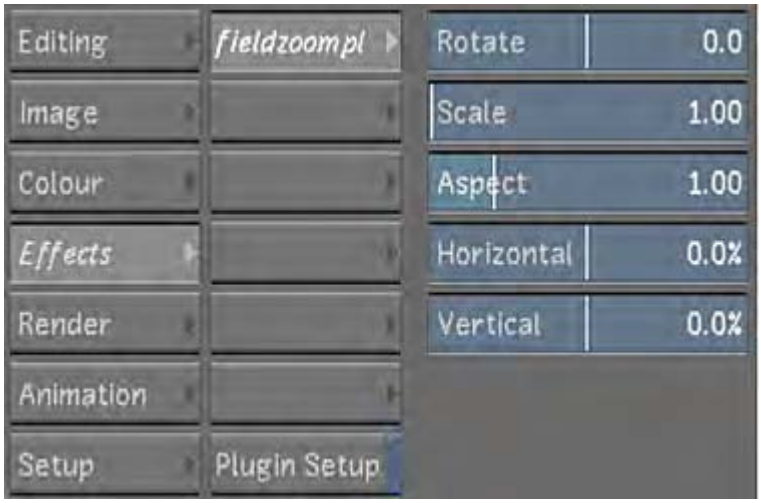
Use:	To:
Kernel Size	Set the amount of the camera defocus.
Highlight Offset	Offset the highlight range.
Highlight Boost	Increase the highlight range.
Edge	Adjust the edge softness of a matte when using this plugin with a secondary.
Opacity	Change the opacity of the image. By adding transparency, you see through to the original image.
Aspect	Adjust the orientation of the blur. A negative value produces a horizontal blur and a positive value produces a vertical blur.
Circle, Hexagon, Octagon	Specify the shape of the plugin matrix (or camera iris) used to calculate how the defocus is applied.
Rotation	Rotate the plugin matrix (or iris).

Directional Blur Simple directional blur along a specified direction.



Use:	To:
Size	Adjust the amount of blur.
Orientation	Change the direction of the blur.

Field Zoom Pan, scan, scale, and rotate interlaced footage. The parameters in this plugin work in the same way as the tools in the Image > Reposition menu, except that they are applied to individual fields instead of the entire frame. This yields a smooth image from interlaced footage even after repositioning.



Use:	To:
Rotate	Rotate the fields.
Scale	Resize the fields.
Aspect	Adjust the aspect ratio of the fields.
Horizontal	Adjust the horizontal position of the fields.

Use:	To:
Vertical	Adjust the vertical position of the fields.

Glow Blur-type plugin that can be used by itself or combined with a secondary key.



Use:	To:
Defocus	Adjust the amount of blur.
Aspect	Adjust the orientation of the blur. A negative value produces a horizontal blur and a positive value produces a vertical blur.
Mix	Adjust the amount of mix between the original image and the blurred image.
Boost	Multiply key values when using the plugin on a secondary key. This allows you to visually grow your key by increasing the range of the key value.
Red Mix	Adjust the red gain of the blurred image.
Green Mix	Adjust the green gain of the blurred image.
Blue Mix	Adjust the blue gain of the blurred image.

Gold Blur-type effect that allows you to soften and tint an image. The defocused and coloured image is multiplicatively combined with the original image to create a coloured soft-filter look.



Use:	To:
Kernel Size	Adjust the amount of blur.
Aspect	Adjust the orientation of the blur. A negative value produces a horizontal blur and a positive value produces a vertical blur.
Hue Value	Set the hue that is used to tint the image.
Amount	Adjust the amount of the selected hue.
Mix	Adjust the amount of mix between the original image and the blurred, tinted image. The other parameters have no effect until you set the Mix value.
Brightness	Adjust the brightness of the tinted image.

Noise Plugin 1 This grain-type plugin is a first generation advanced noise reducer that uses a spatial filter with detail preservation control. It is useful for grain, noise, or detail reduction. During playback, this plugin updates dynamically as you adjust the values, so you can see how the grain removal works in context of the shot.

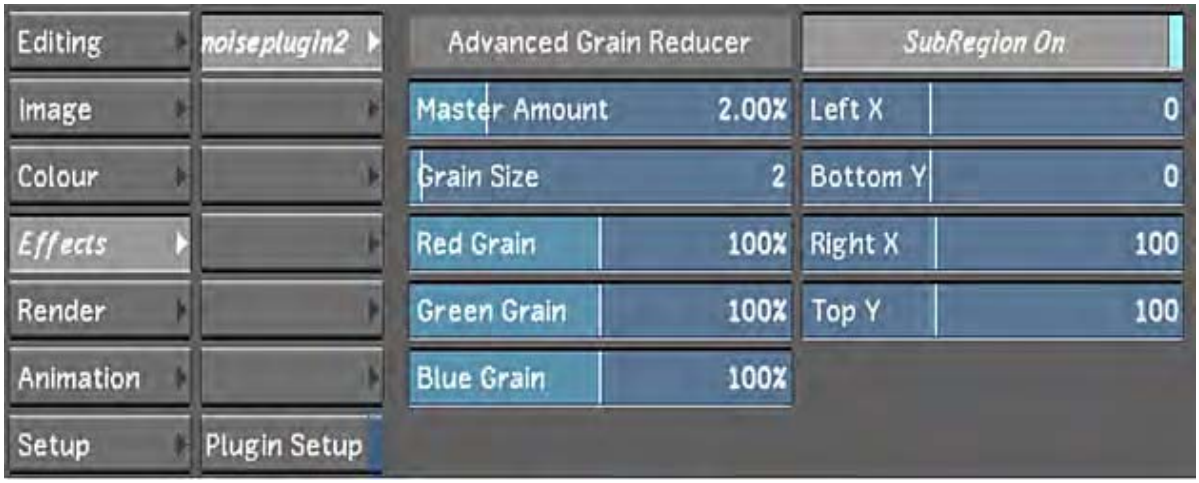
If the plugin processing slows down image playback, you can restrict grain reduction to a subregion of the frame.



Use:	To:
Master Amount	Set how much the grain is reduced. The default value (or less) generally produces a good result for grain removal.
Grain Size	Set the size of grain that is reduced.
Red Grain	Independently adjust grain removal on the red colour channel.
Green Grain	Independently adjust grain removal on the green colour channel.
Blue Grain	Independently adjust grain removal on the blue colour channel.
SubRegion On	Restrict the grain setting to the subregion defined by the bounding box. Right-click in the subregion to drag it to another position. Adjust the size of the subregion by clicking and dragging a corner of the subregion bounding box.
Left X, Bottom Y, Right X, Top Y	You can adjust the size of the subregion bounding box by adjusting these parameters.

Noise Plugin 2 This grain-type plugin is a second generation advanced noise reducer that uses a spatial filter with detail preservation control. It is useful for grain, noise, or detail reduction. During playback, this plugin updates dynamically as you adjust the values, so you can see how the grain removal works in context of the shot.

If the plugin processing slows down image playback, you can restrict grain reduction to a subregion of the frame.



Use:	To:
Master Amount	Set how much the grain is reduced. The default value (or less) generally produces a good result for grain removal.
Grain Size	Set the size of grain that is reduced.
Red Grain	Independently adjust grain removal on the red colour channel.
Green Grain	Independently adjust grain removal on the green colour channel.
Blue Grain	Independently adjust grain removal on the blue colour channel.
SubRegion On	Restrict the grain setting to the subregion defined by the bounding box. Right-click in the subregion to drag it to another position. Adjust the size of the subregion by clicking and dragging a corner of the subregion bounding box.
Left X, Bottom Y, Right X, Top Y	You can adjust the size of the subregion bounding box by adjusting these parameters.

Noise Plugin 3 This grain-type plugin is a third generation advanced noise reducer that uses a spatial filter with detail preservation control. You can control luminance and colour noise processing separately.



Use:	To:
Master Amount	Set how much the grain is reduced. The default value (or less) generally produces a good result for grain removal.
Grain Size	Set the size of grain that is reduced.
Colour Grain	Adjust grain reduction on colour grain only.
Luminance Grain	Adjust grain reduction on luminance grain only.

Noise Plugin 3.1 This grain-type plugin is a third generation advanced noise reducer that uses a spatial filter with detail preservation control. It is useful for colour noise processing, and allows you to specify whether processing should occur in the shadows, midtones, or highlights. During playback, this plugin updates dynamically as you adjust the values, so you can see how the grain removal works in context of the shot. It is the fastest of the four noise plugins available in Lustre.

If the plugin processing slows down image playback, you can restrict grain reduction to a subregion of the frame.



Use:	To:
Master Amount	Set how much the grain is reduced. The default value (or less) generally produces a good result for grain removal.
Grain Size	Set the size of grain that is reduced.
Colour Grain	Adjust grain reduction on colour grain only.
Shadow Grain	Adjust grain reduction in the shadows only.
Midtone Grain	Adjust grain reduction in the midtones only.
Highlight Grain	Adjust grain reduction in the highlights only.
SubRegion On	Restrict the grain setting to the subregion defined by the bounding box. Right-click in the subregion to drag it to another position. Adjust the size of the subregion by clicking and dragging a corner of the subregion bounding box.
Left X, Bottom Y, Right X, Top Y	You can adjust the size of the subregion bounding box by adjusting these parameters.

Print Bleach Blur-type plugin that creates an effect similar to the chemical print bleach process. This plugin increases contrast for the shadows while desaturating the image. When you use this plugin, you first set a mix value so that you mix the image onto itself, and then you blur and brighten the underlying layer.



Use:	To:
Blur	Adjust the amount of blur. The blur parameter has no effect until you set the mix parameter.
Aspect	Adjust the orientation of the blur. A negative value produces a horizontal blur and a positive value produces a vertical blur.
Mix	Change the amount of the mix. By increasing the mix value, you increase the contrast and decrease the saturation of the image.
Brightness	Adjust the amount of brightness of the blurred image. The brightness parameter has no effect until you set the mix parameter.

Silver Blur-type effect that mixes a black-and-white (desaturated) image with the original image creating a soft-filtered look.



Use:	To:
Blur	Adjust the amount of blur. The blur parameter has no effect until you set the mix parameter.

Use:	To:
Aspect	Adjust the orientation of the blur. A negative value produces a horizontal blur and a positive value produces a vertical blur.
Mix	Adjust how much the black and white image is mixed with the original. Use a negative value to give your image a silver look.

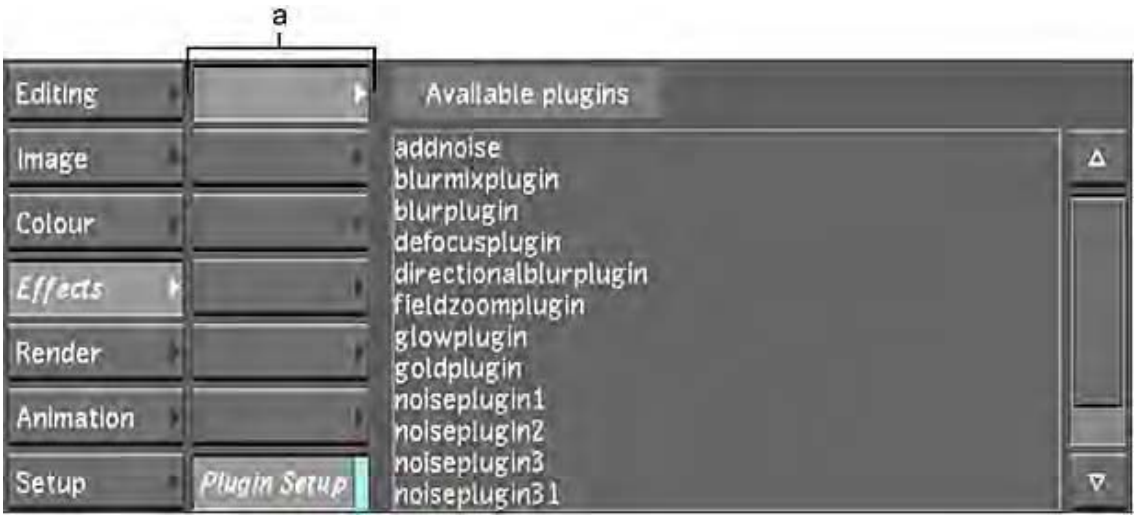
Creating an Effect with a Lustre Sparks Plugins

Load and use Lustre Sparks plugins from the Effects menu. You can load up to six plugins at a time.

Each plugin has various parameters that you set to create the desired effect. Apply the effect to the whole shot, or an area of a shot defined by a secondary. You can animate the effect to make it change over time. You can also create an effect using a single or multiple plugins on the same shot. The order of the plugin in the menu will affect the final result, as they are processed sequentially starting with the plugin at the top of the menu and continuing down.

To load a plugin and create an effect:

1. In the Storyboard, select the shot to which you want to apply the effect.
2. From the Main menu, click Effects to display the Effects menu.
3. Enable Plugin Setup.
   A list of the available plugins is displayed.



(a) Blank plugin buttons

4. Click the blank button in which to load the plugin.
5. Select a plugin from the Available plugins list.
   The plugin is loaded and its name appears on the selected button. The plugin parameters appear to the right of the button.



TIP You can also replace a loaded plugin with another by loading the second plugin into the button used by the first. In this case, the settings of the replaced plugin are deleted.

6 Set the parameters as needed.

The parameters are applied to the entire shot and the image is dynamically updated to reflect the changes.

7 If you wish, modify the effect by repeating the previous steps on the shot with other plugins.

To apply an effect to an area defined by a secondary:

1 In the Storyboard, navigate to the shot you are applying the effect on.

2 Create one or more secondaries for the shot to which you want to add an effect.

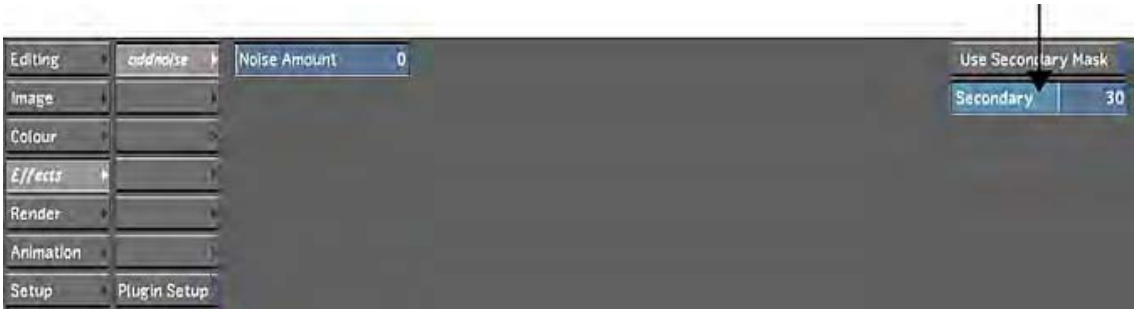
3 In the Main menu, click Effects to display the Effects menu.

4 Select a blank plugin button and choose a plugin from the Available plugins list.

5 Select a secondary layer that corresponds to the secondary you want to use:

- Right-click the Secondary slider to display the calculator and enter the number of the secondary layer.
- Click and drag within the Secondary slider to select the secondary layer.

NOTE The Field Zoom plugin does not allow you to select a secondary layer.



6 Enable the Use Secondary Mask button.

The plugin is applied to the area defined in the secondary layer.

7 Set the parameters as needed.

To animate an effect:

1. Place the positioner on the frame where you want the effect to start.
2. Enable AutoK.

   The AutoK button turns yellow.



3. In the Main menu, click Effects to display the Effects menu.
4. Make sure the plugin you want to animate is selected.

   The plugin name on the button is highlighted and italicized and its parameters appear to the right of the buttons.



5. Set one or more of the parameters.

   As soon as you change a parameter, a blue line in the Shot timebar appears indicating a keyframe.
6. Move the positioner to the next frame where you want to set another keyframe and adjust one or more of the parameters. Continue to add keyframes as needed.

Disabling and Re-enabling Lustre Sparks Plugins

You can disable and re-enable loaded Lustre Sparks plugins. Disabling a plugin allows you to temporarily view the shot without the effect while retaining the settings.

To disable and re-enable a plugin:

1. Right-click the plugin button to toggle between disabling and re-enabling the plugin.

   When the plugin is disabled, the text on the button is grey. When the plugin is enabled, the text on the button is white.



(a) Enabled plugin (b) Disabled plugin (c) Selected and disabled plugin

Developing Lustre Sparks Plugins

An application programming interface (API) is available to third-party developers for creating Lustre Sparks. The API is provided on the Lustre CD. If you are interested in developing commercial Lustre Sparks, you must apply to the system Sparks program. To request an application, send an e-mail to and indicate your interest in developing specifically for Lustre. For more information, see the *Plugin Reference Guide*.

Animation

In Lustre, you create animations with the Channel Editor by making channel values change over time. When you set values for a channel, they are plotted on a curve in the Channel Editor. This curve maps frame numbers (time) on the horizontal axis and channel values on the vertical axis.

When you explicitly set a channel value at one frame, you are creating a keyframe. If you set more than one keyframe, the values between the keyframes are automatically calculated, creating an animation curve. This is called interpolation. You can customize the shape of the animation curve to control how quickly and by how much the parameters change.

By controlling interpolation and keyframes, you can create complex animations involving numerous variable parameters. For example, you can animate primary and secondary colour as well as Sparks effects (such as a blur). You can also make a dull stationary shot dynamic and therefore more visually interesting using frame positioning and scaling.

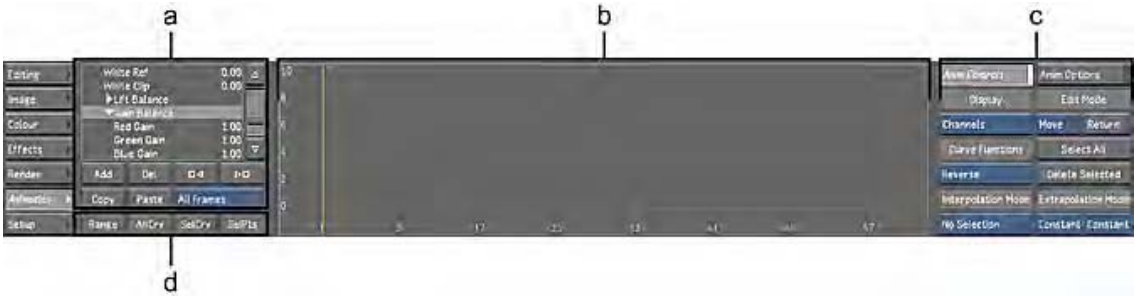
Once you create an animation, you can use the Track Editor to adjust animation timing or duration without changing the keyframe values.

Accessing the Channel Editor

Use the animation curves in the Channel Editor to create animations.

To access the Channel Editor:

1 In the Main menu, click Animation.
The Channel Editor appears.



(a) Channel hierarchy (b) Animation Curves window (c) Animation controls (d) Channel Display buttons

The Channel Editor is made up of the following elements.

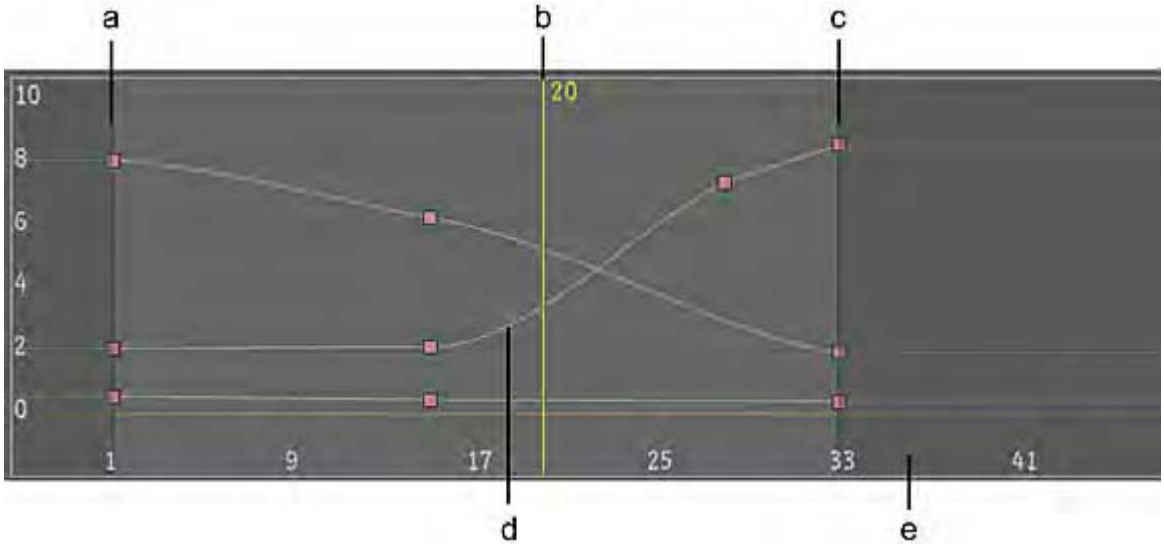
Channel hierarchy Use to select the folders and channels to animate. You can also refer to the Channel hierarchy to quickly see what channels contain keyframes.

Animation Curves window Use to create and view animation curves, adjust interpolation, and add, edit, or delete keyframes. The Track Editor (page 2201), used to adjust animation timing, appears in place of the Animation Curves window when you select the Tracks option in the Display option box.

Animation Controls and Animation Options Use to switch between animation modes, select and view channels, and control interpolation.

Animation Curves Window

You perform most animation operations in the Animation Curves window.



(a) Start of current shot (b) Positioner (c) End of current shot (d) Animation curve (e) Trimmed frames

The Animation Curves window is made up of the following elements.

Current Frame positioner A vertical yellow line represents the current frame position.

Start and end of current shot Vertical thick grey lines represent the start and end of the current shot. If the shot is trimmed, the trimmed handles are greyed out.

Trimmed frames If you trimmed frames from the current shot, these frames are greyed out before or after the shot.

Animation curves Selected curves are white. Deselected curves are black.

Navigating in the Animation Curves Window

Using the Animation Curves window, you can go directly to any frame for the current shot, or you can go directly to previous or next keyframes. The Current Frame positioner always moves to reflect your new frame position in the current shot.

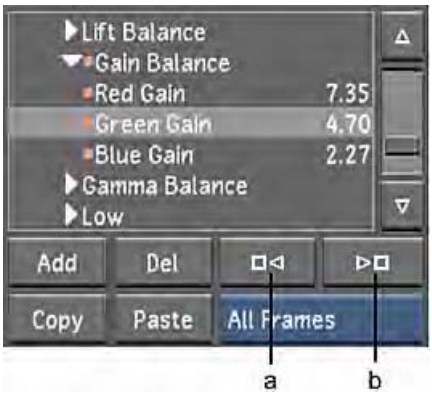
To move to any frame in the Animation Curves window:

1. Drag in the Shot timebar to scrub between frames in the current shot.
   The Current Frame positioner moves in sync with the timebar positioners.

   TIP You can also use the playback controls to navigate through the Animation Curves window.

To move between keyframes:

1. Do one of the following:
   - Click Next to move to the next keyframe.
   - Click Previous to move to the previous keyframe.



(a) Previous button (b) Next button

Displaying Curves in the Animation Curves Window

Use the buttons to the left of the Animation Curves window to change its display.

To view the entire channel range between keyframes in the Channel Editor:

1. Click Range.
   All keyframes are loaded into the Animation Curves window.

To view all animated curves in the Channel Editor:

1. Click AllCrv.
   All animated channels appear in the Animation Curves window.

To view the selected curve in the Channel Editor:

1. In the Channel hierarchy, select the animated channels.
2. Click SelCrv.
   The selected curves are loaded into the Animation Curves window.

To view the selected keyframes in the Channel Editor:

1. In the Animation Curves window, select the keyframes. See Selecting Keyframes (page 2184).
2. Click SelPts.

   The animation curves are scaled so that the selected keyframes fill the Animation Curves window.

Panning and Zooming the Animation Curves Window

You can pan and zoom the Animation Curves window using the mouse. Do this when you need to see a part of the window that is not visible.

To pan the Animation Curves window:

1. Middle-click and drag to pan the Animation Curves window view horizontally or vertically.

To zoom the Animation Curves window view:

1. Do one of the following:
   - To zoom horizontally, hold down the right mouse button and drag to the right to increase the zoom factor and to the left to decrease it.
   - To zoom vertically, hold down the right mouse button and drag up to increase the zoom factor and down to decrease it.

   TIP Hold down `Alt` while dragging to centre the zoom at the point where you began dragging.

Channel Hierarchy

Use the Channel hierarchy to select the channels to animate. In the following example, Green Gain is selected. You can then set keyframes for this channel and create an animation. See Animating with Keyframes (page 2176).

Channels are grouped together in folders. When you select a folder, all its channels are selected and all its curves appear in the Animation Curves window. You can also expand a folder and select a single channel.



(a) Collapsed folder (b) Expanded folder (c) Selected channel (d) Deselected folder

Animation Controls

Use the Animation controls to modify channels. The settings are displayed when the Anim Controls toggle button is enabled in the Animation editor or the Dissolve editor (some options are unavailable in the Dissolve editor).



Display box Toggle between displaying the Channel Editor view (Curve in previous versions of Lustre) or in Track Editor view (Timeline in previous versions).

NOTE

In the Dissolve editor, only the Channels display is available.

Edit Mode box Select how you want to modify keyframes:

Select:	**To:**
Add (A)	Add a keyframe.
Move (M)	Move a keyframe.
Delete (D)	Delete a keyframe.
Break	Break a tangent.
Repair	Repair a broken tangent.

Return button Modify a keyframe and return to Move mode.

Curve Functions box Modify the current channel by reversing the order of keyframes in time (Reverse), or inverting the value of the keyframes (Invert).

Select All button Selects all keyframes on the current channel.

Delete Selected button Deletes selected keyframes from the current channel.

Interpolation Mode box Select how the curve between keyframes is shaped: as a Constant, Linear, Hermite or Bezier curve; Mixed is displayed only when a selection contains keyframes with different interpolations; Natural is available only when importing timelines with dissolves from Wiretap server.

Extrapolation Mode box Select the shape of the curve before and after the first and last keyframes.

Animation Options

Use the Animation Options to determine the way in which the animation channels function. The settings are displayed when the Anim Options toggle button is enabled. The default settings for these options are set in the Animation panel of the Tools Settings page (page 1765) (some options are unavailable in the Dissolve editor).



Animating with Keyframes

To create an animation, you select channels with animatable parameters, and then you add keyframes at points in time where the parameters change. The following procedure outlines the basic steps required to create an animation.

To create an animation:

1. In the Channel hierarchy, select the channel or parent folder you want to animate. See Selecting Channels (page 2177).
2. Move the positioner to the frame you wish to mark as a keyframe and then add a keyframe. See Adding Keyframes (page 2177).
3. Set the value by dragging the keyframe or by using the individual controls in the appropriate tool.
4. Set the interpolation and extrapolation (if needed) for the keyframe. See Setting Interpolation (page 2179) and Setting Extrapolation (page 2181).
5. Move to the next frame and add another keyframe.
6. Set the keyframe value at the new position.
7. Make any additional modifications to the animation curve.
8. Move to the start of the shot and play the animation.
9. If necessary, use the Track Editor to adjust the animation timing. See The Track Editor (page 2201).

Selecting Channels

To create an animation, you select channels with animatable parameters, and then you add keyframes at points in time for which the parameters change. You select animation channels in the Channel hierarchy. Once you select a channel, its curve appears in the Animation Curves window.

To select a channel:

1. In the Main menu, click Animation.
2. In the Channel hierarchy, expand a folder and click a channel to select it. To select all channels in the folder, select the folder.

   NOTE You can also select a channel in the Animation Curves window by clicking the inactive curve. You can see inactive curves in the Animation Curves window only when the Hide Inactive Curves button in the Animation Options panel is disabled.



(a) Select folder (b) Selected channel (c) Deselected channel

   Selected channels are highlighted.
3. Click a selected channel to deselect it.

   NOTE Deselecting a channel does not change the selection state of that channel's keyframes.

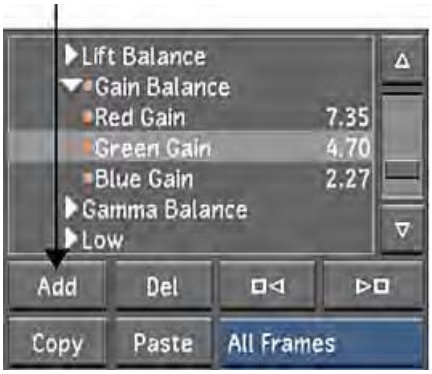
Adding Keyframes

You can add keyframes at any point in your animation curves. There are three methods of adding keyframes:

- Add a single keyframe to the selected channel(s) at the current position.
- Interactively add keyframes to any point on the selected curves. There must be at least one keyframe on a channel before you can interactively add keyframes.
- Set keyframes automatically when you adjust a parameter in any menu. For example, set up an animation directly in the Colour menu and then fine-tune the animation curve in the Animation Curves window.

To add a keyframe:

1. In the Channel hierarchy, select the channel to which you want to add a keyframe.
2. Move the positioner to the frame you wish to mark as a keyframe.
3. Click Add.



When you add a keyframe to a channel, an orange indicator appears to the left of the channel in the Channel hierarchy.

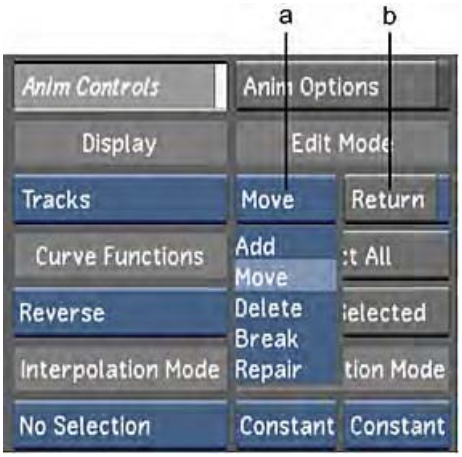
4 Set the channel value as needed by dragging the keyframe.

Once you add keyframes to a shot, blue markers appears in the Shot timebar indicating the location of the keyframes.

To add multiple keyframes interactively:

1 You must explicitly add one keyframe using the Add button or Autokey before you can add keyframes interactively in Add mode.

2 Display the Animation controls.



(a) Edit Mode option box (b) Return toggle button

NOTE If Return mode is enabled, you are returned to Edit mode after adding one keyframe.

3 In the Channel hierarchy, select the channel to which you want to interactively add keyframes.

4 In the Animation controls, select Add in the Edit Mode option box.

5 Move the mouse cursor over the animation curve.

Keyframes appear as you approach each frame.

6 Click the animation curve to add a keyframe at that frame.

NOTE The new keyframe is automatically set to Bezier interpolation.

Once you add keyframes to a shot, markers appears in the Shot timebar indicating the location of the keyframes.

To add keyframes in Autokey mode:

1 Enable AutoK.



NOTE Autokey is available from all menus in Lustre.

2 Access the tool you want to use to create the animation. For example, in the Main menu, click Colour, and then click Grading.

3 Move the Shot timebar to the frame where you want to add a keyframe.

4 Adjust the colour parameters.

A keyframe is added automatically to the channels you modified.

5 Move to another frame and adjust the colour parameters and set another keyframe.

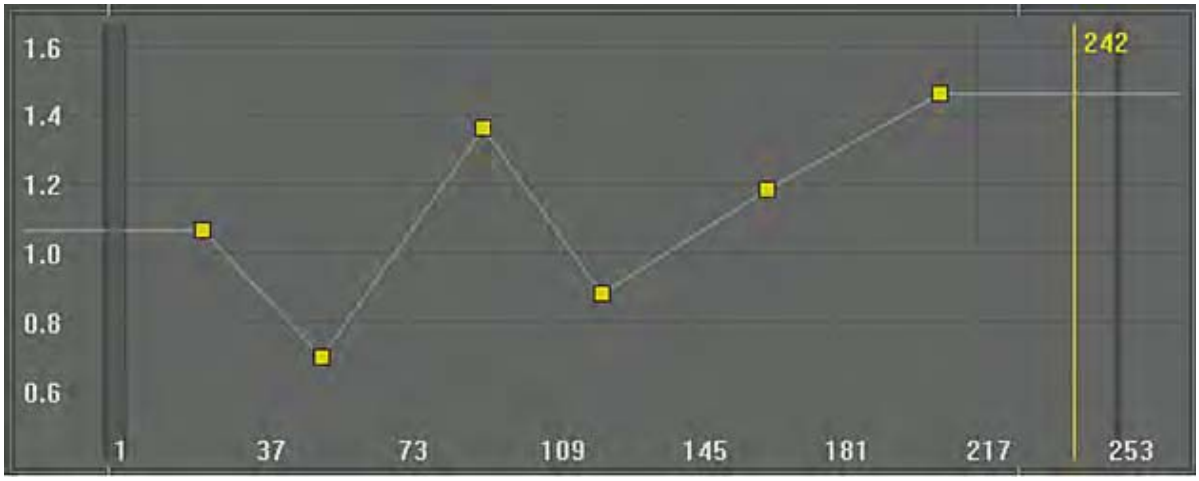
6 Keep adjusting colour as needed.

Once you have added keyframes to a shot, blue markers appears in the Shot timebar indicating the location of the keyframes.

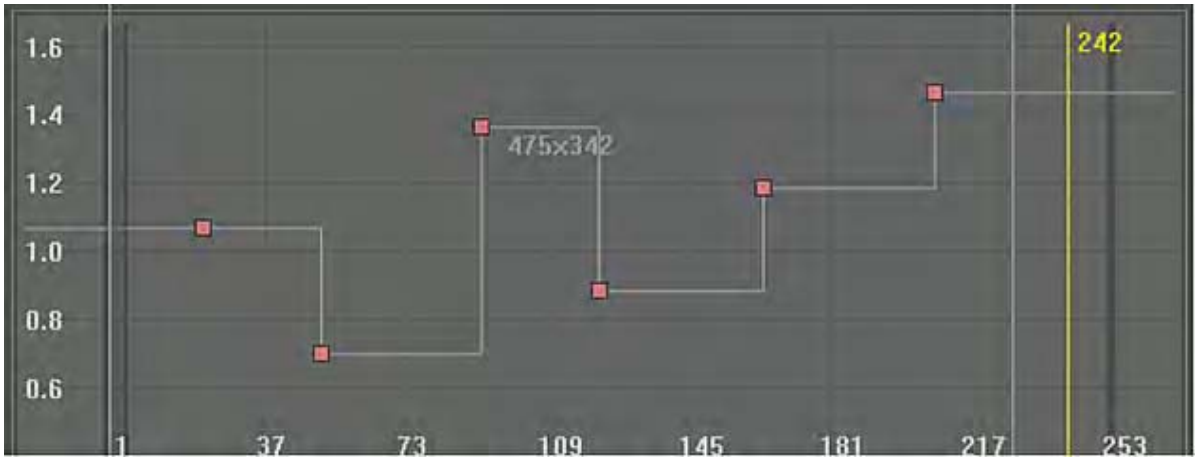
Setting Interpolation

Interpolation defines the shape of an animation curve between keyframes. You can choose from the following interpolation settings depending on how you want the channel values to change over time.

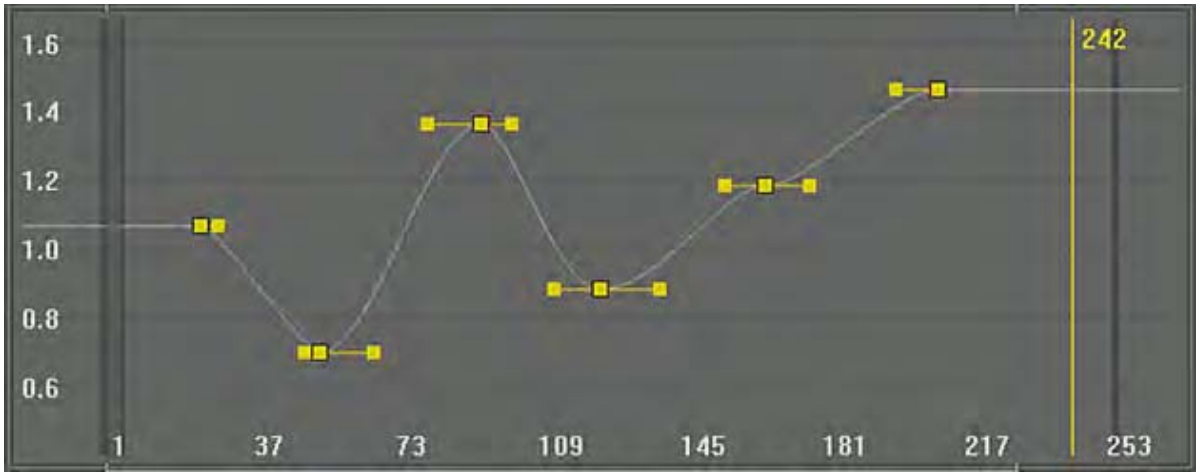
Linear Joins keyframes with a straight line.



Constant Produces a square curve. The value of one keyframe is held at a constant value until the next keyframe. This setting can produce abrupt changes between keyframes.



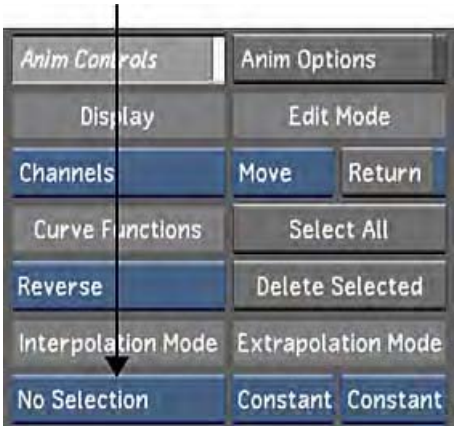
Hermite and Bezier Produces a smooth curve with a smooth transition between keyframes. Each keyframe on the animation curve has an associated tangent handle. You can change the shape of the animation curve by dragging the tangent handle. Bezier requires fewer keyframes than Hermite to produce a similar curve. This is because there are two tangents at each keyframe: one incoming, one outgoing.



NOTE You can create animation curves with mixed interpolation types.

To set the interpolation type for a keyframe:

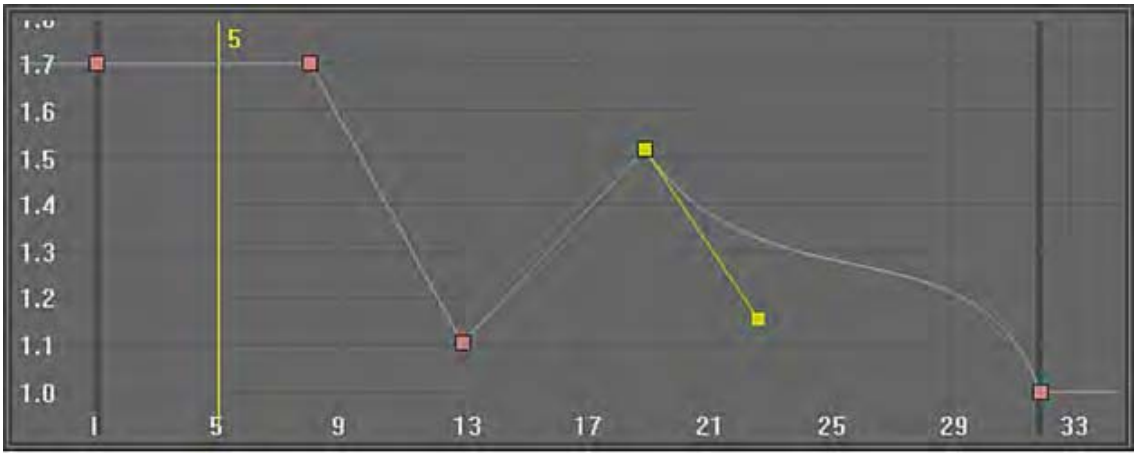
1. Display the Channel Editor (page 2171).
2. Select the keyframe for which you want to set the interpolation.
3. In the Animation controls, select the type of curve from the Interpolation Mode option box.



To create an animation curve with mixed interpolation:

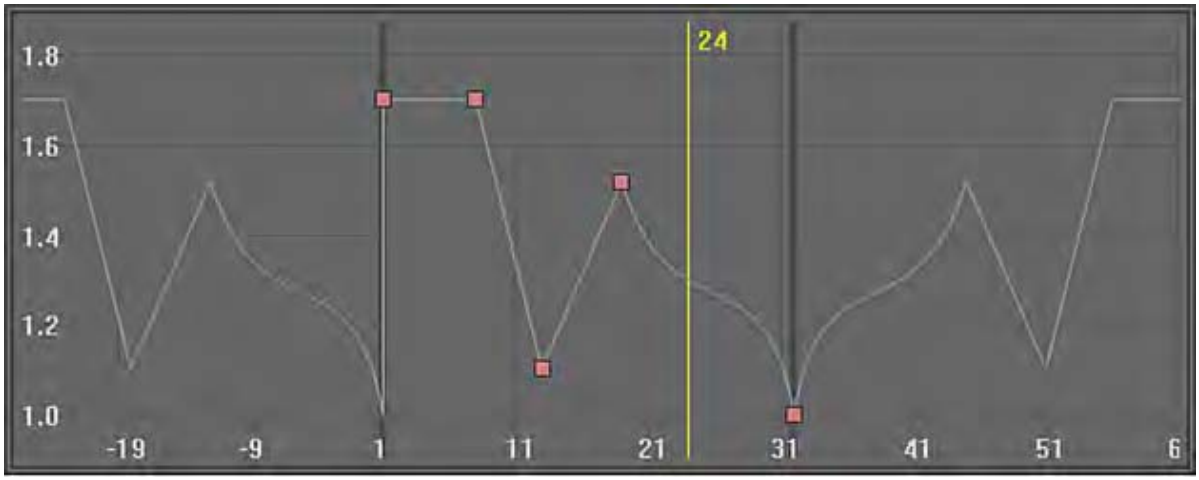
1. Create an animation with at least four keyframes. Do not worry about the interpolation setting at this point.
2. Display the Channel Editor (page 2171).
3. Make sure you are in Move mode and then select the first keyframe. See Editing Keyframes (page 2184).
4. Select Constant to set the interpolation for the first keyframe to Constant.
5. Select the second keyframe.
6. Select a different interpolation mode, such as Linear.
7. Select the third keyframe, and then select a different mode, such as Bezier.

   Your animation curve may resemble the following.



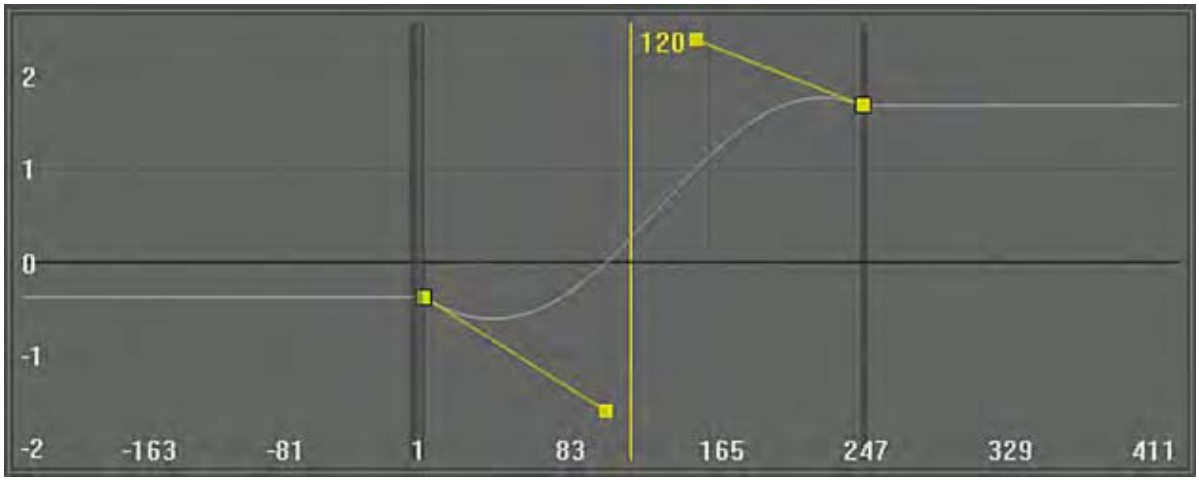
Setting Extrapolation

Extrapolation defines the shape of the animation curve outside the keyframes. You can set up a cycle in which channel values change repeatedly in the same manner over time. You can set the extrapolation before the first keyframe separately from the extrapolation after the last keyframe.

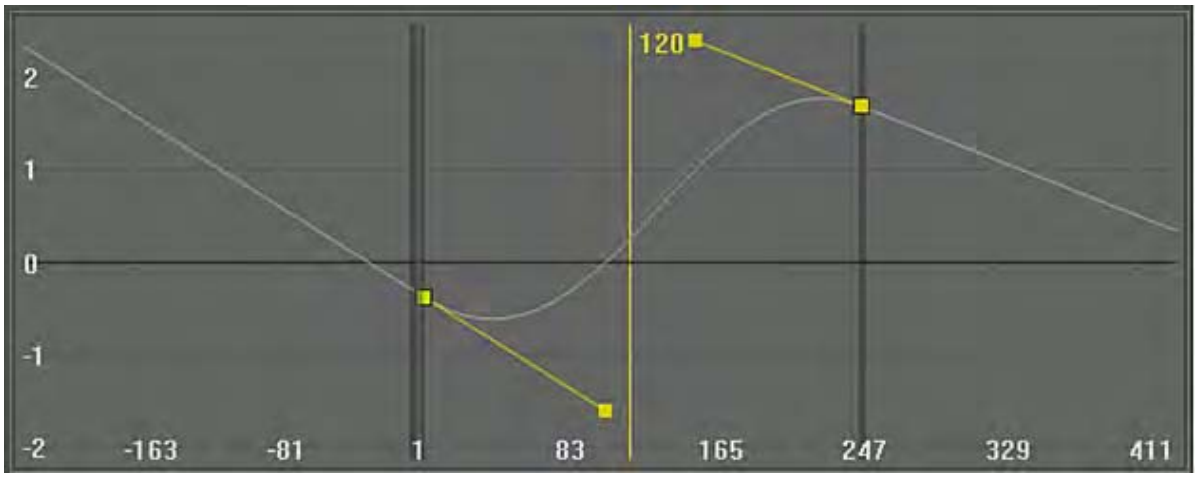


You can specify how keyframes are extrapolated using the following settings.

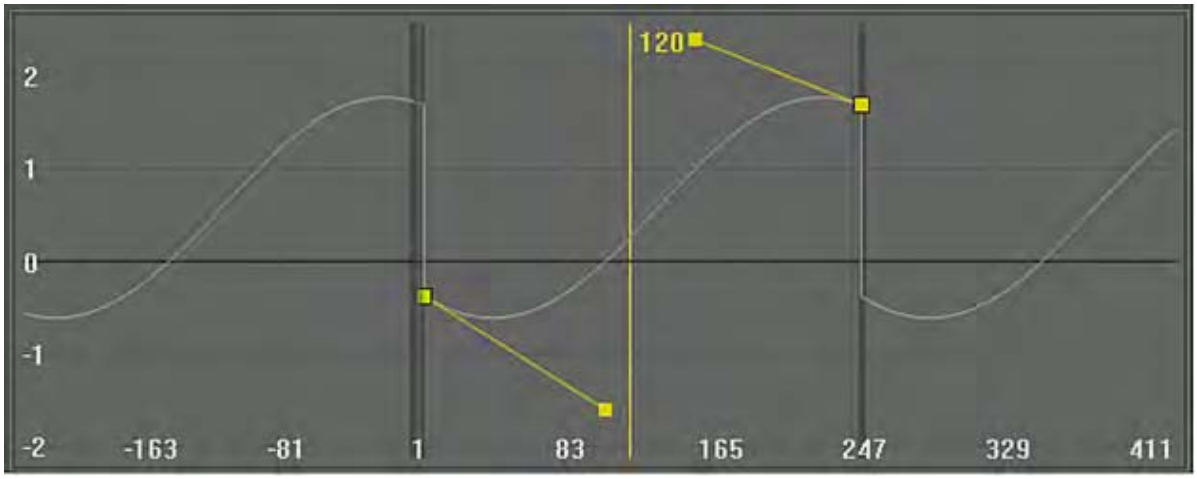
Constant Creates an extrapolated animation curve that stays at the same value as the first or last keyframe.



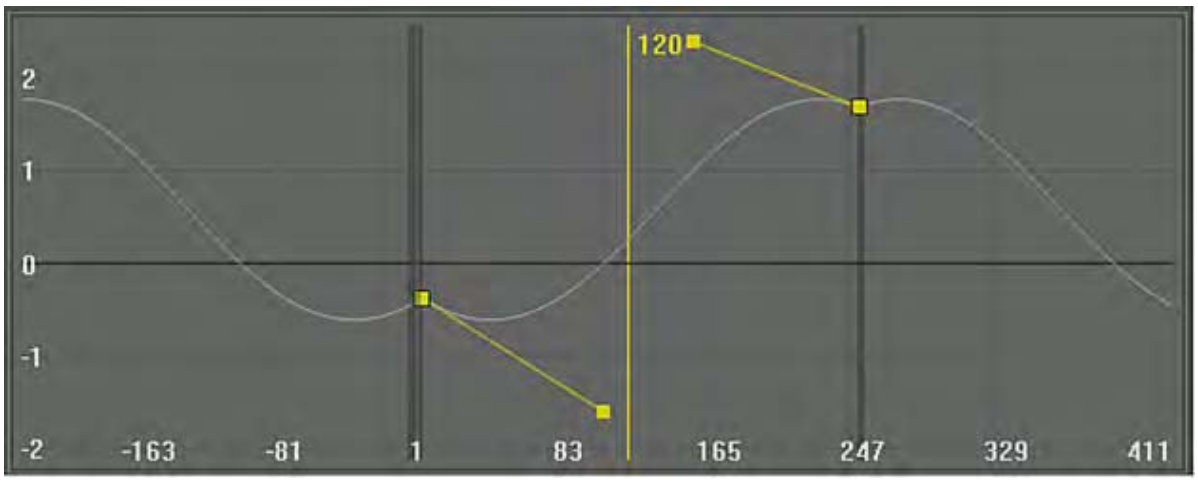
Linear Creates an animation curve that changes in a linear fashion before the first or after the last keyframe.



Repeat Creates a mirror image of the animation curve before the first or after the last keyframe.

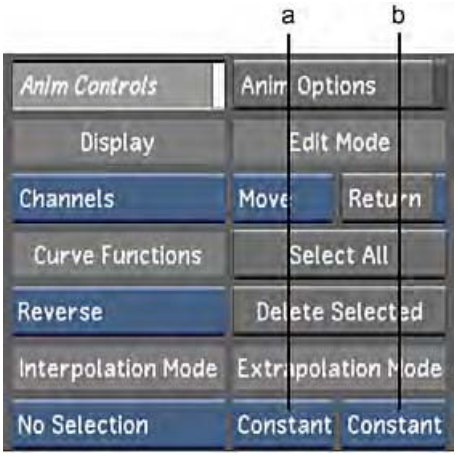


Cycle Creates a pattern of repeating keyframes before the first of after the last keyframe.



To set the extrapolation type for a channel:

1. Select the keyframe for which you want to set the extrapolation.
2. Click Animation.
3. Enable the Animation Controls panel.
4. In the Extrapolation Mode panel of the Animation controls, do any of the following:
   - In the PreType controls, click one extrapolation type button to set the extrapolation before the first keyframe.
   - In the PostType controls, click one extrapolation type button to set the extrapolation after the last keyframe.



(a) PreType Extrapolation option box (b) PostType Extrapolation option box

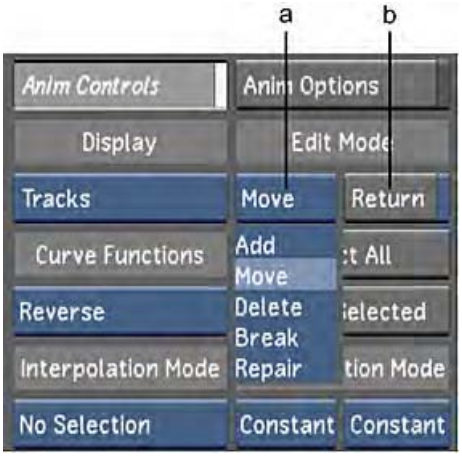
Editing Keyframes

As you create animations, you may need to edit keyframe values, modify animation curves, and manipulate keyframes. Make sure that you are in the appropriate Edit mode when selecting and editing keyframes.

NOTE You cannot edit keyframes in Add mode or Delete mode.

To enable Edit mode:

1. Click Animation.
2. In the Animation controls, select Add, Move, Break or Repair from the Edit Mode option box.



(a) Edit Mode option box (b) Return toggle button

NOTE To return to Move mode after editing a keyframe, enable the Return toggle button.

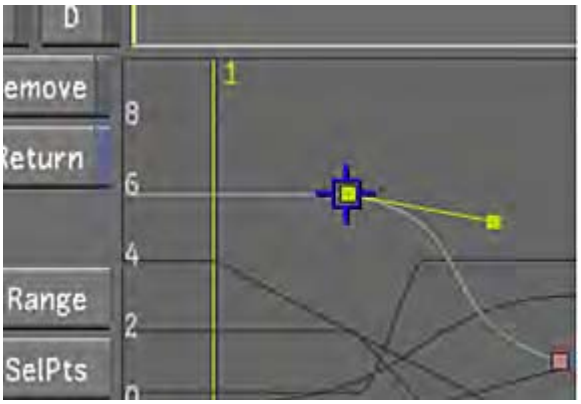
Selecting Keyframes

You need to select keyframes in order to edit them. You can select one keyframe at a time, marquee select a group of keyframes, or select all keyframes in a given channel.

Selected keyframes are yellow. Unselected keyframes are red.

To select a keyframe:

1 In the Animation Curves window, position the cursor over the keyframe and click.

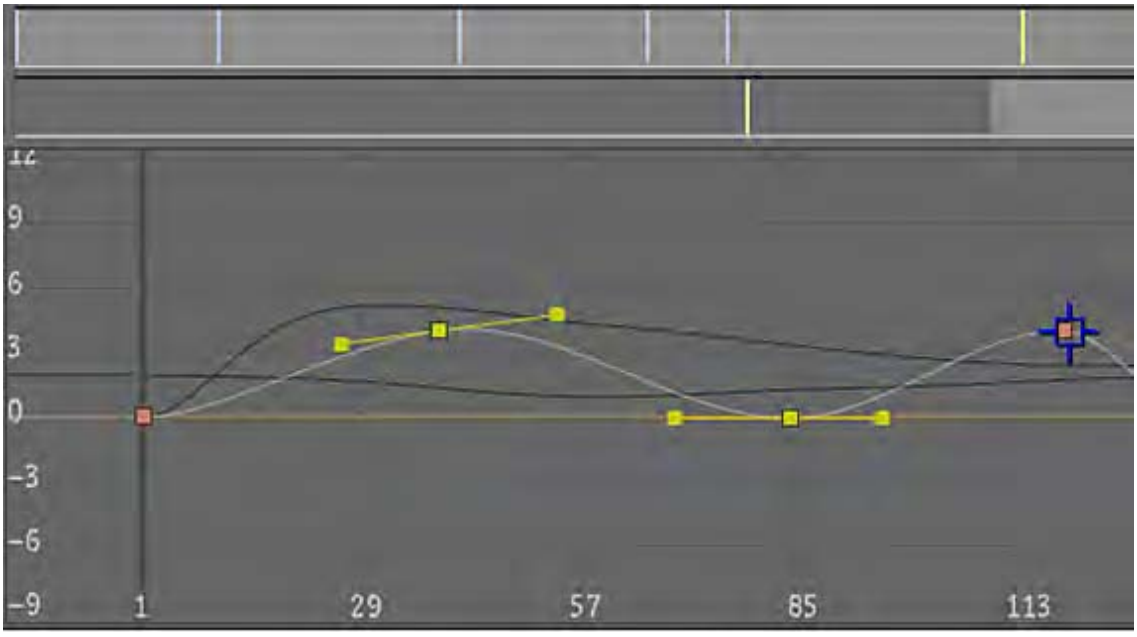


The keyframe is selected.

To select multiple keyframes:

1 In the Animation Curves window, click a keyframe to select it and then hold down `Ctrl` and click additional keyframes to add them to the selection.

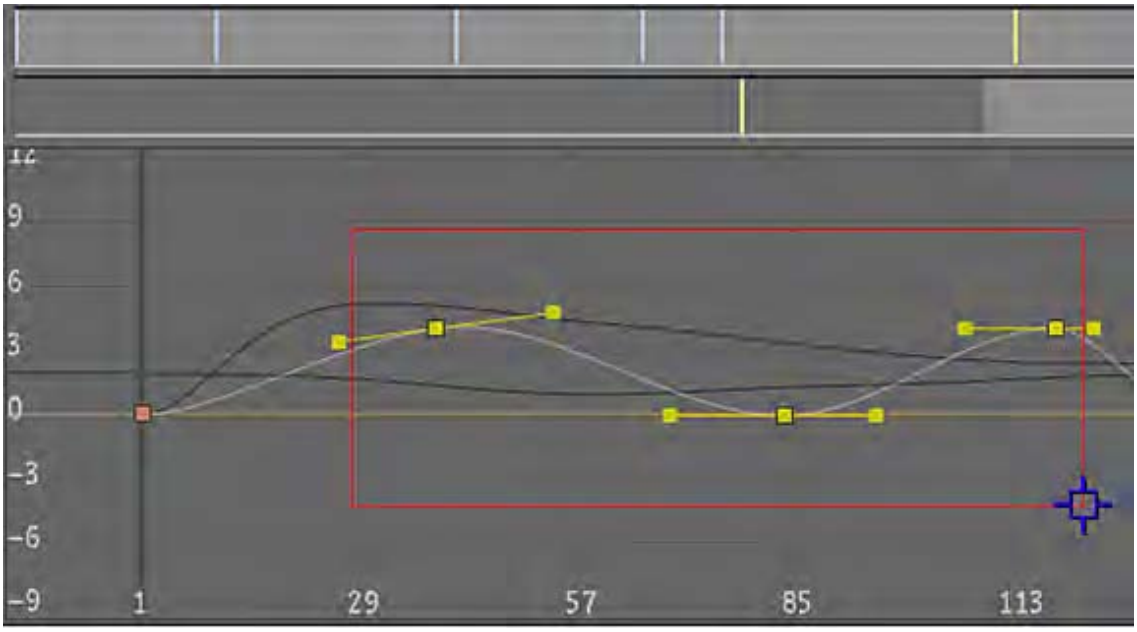
NOTE To deselect a selected keyframe, hold down `Ctrl` and click the keyframe. This is useful when you make a marquee selection but you would like not to include one or more keyframes in the selection.



If you release the `Ctrl` key and then click a keyframe, this keyframe is selected and the previous selection is cleared.

To marquee select multiple keyframes:

1 In the Animation Curves window, drag to draw a marquee selection rectangle around the keyframes you want to select.

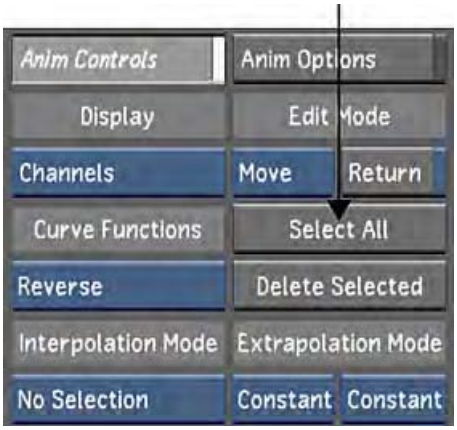


Keyframes inside the marquee selection are selected.

2 To zoom in on the selected area, hold down the `Alt` key when drawing the marquee selection.

To select all keyframes in the selected curve:

1 In the Animation Controls panel, click Select All.



Modifying Your Animation

The rate at which your animation changes depends on the slope of the animation curve. A steep slope results in a more rapidly changing animation. A gradual slope represents a slower change in channel values over time. By changing the shape of the animation curve, you can modify the speed at which the animation occurs between keyframes. Although you can use Constant or Linear interpolation to create animations,

with Bezier and Hermite interpolation you have much more control over the shape of the animation curve, and therefore, the speed of the animation.

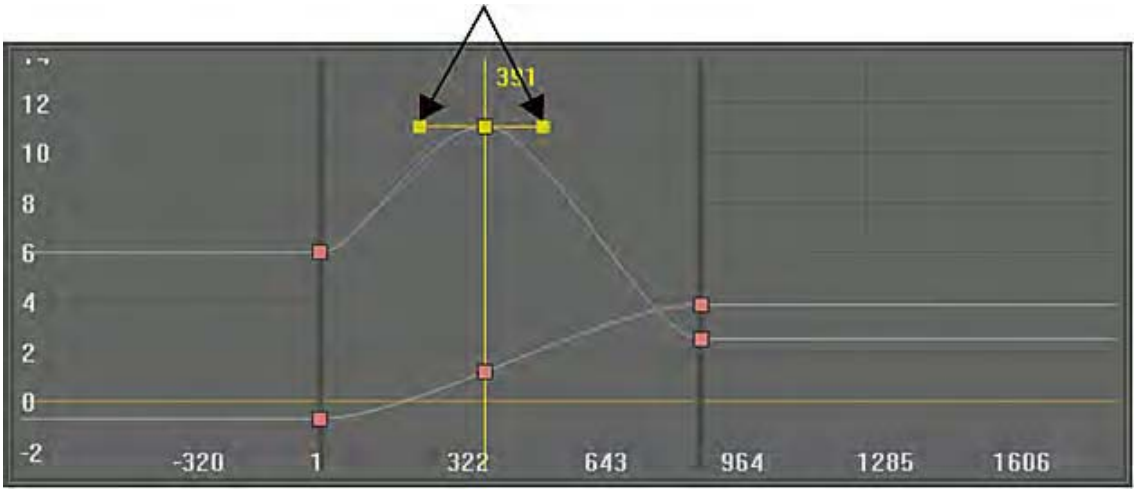
You can hide curves that are not selected. This can make it easier to manage your keyframes and modify animations.

To modify the shape of an animation curve:

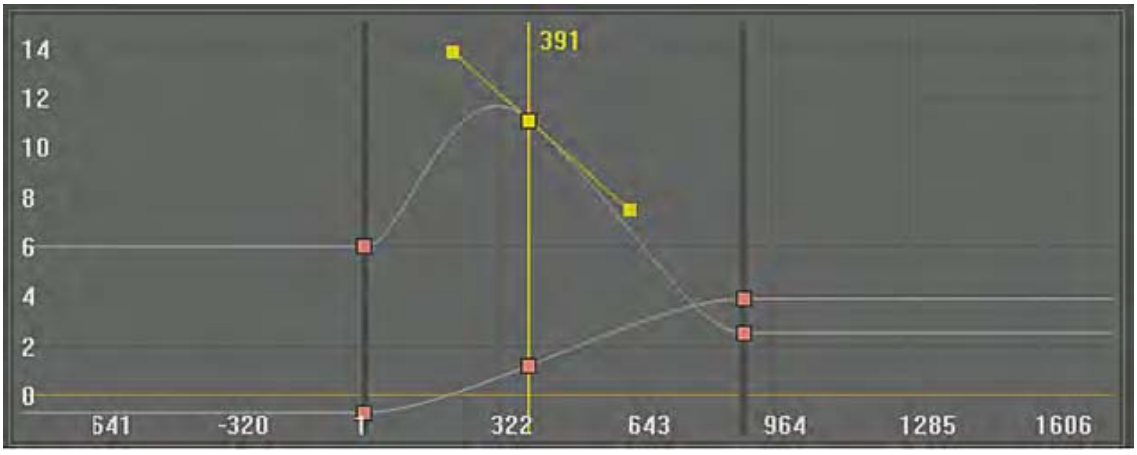
1. In the Channel hierarchy, select the animated channel you want to modify.
2. Click Range to view the entire curve between the first and last keyframes.



3. Select the first keyframe and set the interpolation to Bezier.
   A pair of Bezier tangent handles appear for the selected keyframe.

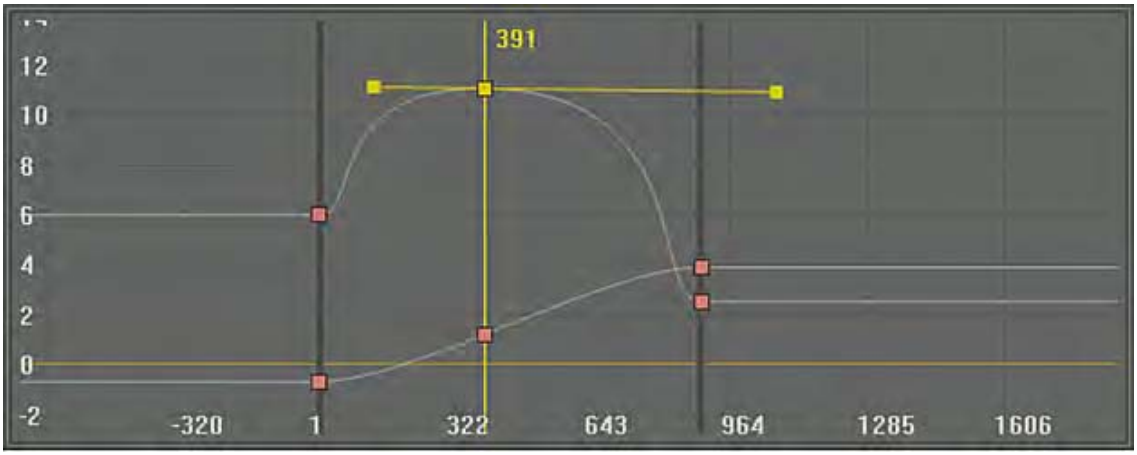


4. Drag one of the tangent handles.
   The shape of the curve changes as you move the tangent handle.



NOTE When you drag a tangent handle, its adjacent tangent handle moves in the opposite direction. If you want to move the tangent handles independently, you need to first break them. See Breaking Tangent Handles (page 2188).

5 You can lengthen a tangent handle independently without breaking the pair.



To hide curves that are not in use:

1 Enable the Animation Options (page 2176) panel.
2 Enable the Hide Inactive Curves toggle button.

Synchronizing Keyframe Functionality

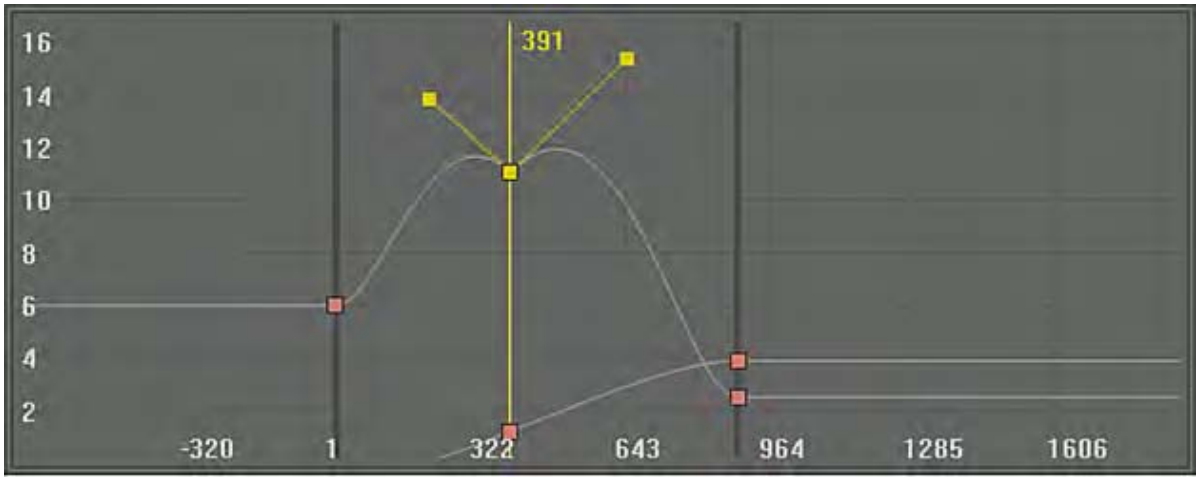
When you assemble with the Record base of assembly, dissolved shots become one shot on the timeline and the dissolve start and end points are automatically keyframed. The ACS Animation System option ensures consistent keyframe animation functionality between the control surface and the Lustre user interface. This means that if you modify colour settings that occur before the dissolve, the colour settings will fade into the next shot correctly whether you are using the control surface or the user interface.

To ensure consistent keyframe functionality between the control surface and the user interface:

1 Click Animation.
2 Enable the Animation Options (page 2176).
3 Enable the ACS Animation System button.

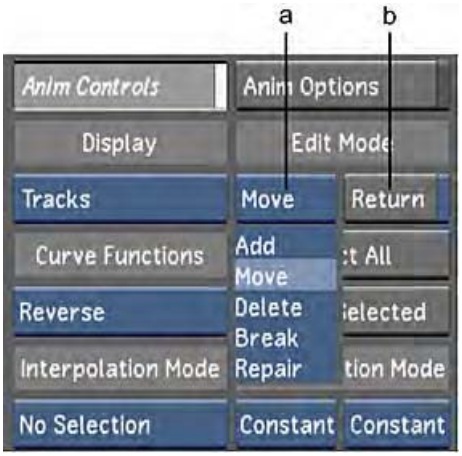
Breaking Tangent Handles

Normally when you move one Bezier or Hermite tangent handle, the other moves in the opposite direction. You can break tangent handles so that you can adjust each tangent independently and create abrupt changes in your animation.



To break a tangent handle:

1 In the Edit Mode option box, select Move to select the keyframes you want to break. You can also click Select All to select all keyframes.



(a) Edit Mode option box (b) Return toggle button

2 In the Edit Mode option box, select Break.

NOTE If you want to break the tangents for more than one keyframe, disable Return. If Return mode is enabled, you are returned to Edit mode after performing one break operation.

3 In the Animation Curves window, click and drag the selected tangent handle you want to break.

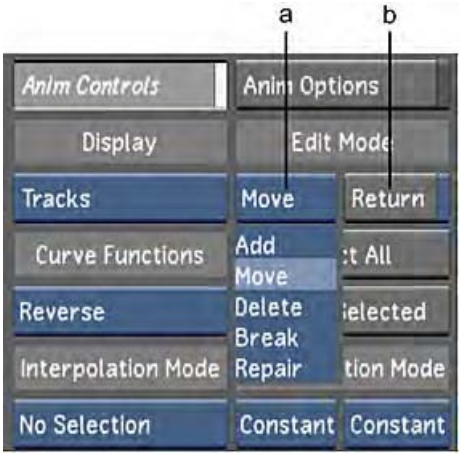
4 If Return mode is disabled, click additional keyframes to break the tangent handles and modify them as needed.

Resetting Tangent Handles

Use Repair mode to reset broken tangent handles to an unbroken state.

To reset broken tangent handles:

1 In the Edit Mode option box, select Move to select the broken keyframes you want to repair. You can click the Select All button to select all keyframes.



(a) Edit Mode option box(b) Return toggle button

2 In the Edit Mode option box, select Repair.

All broken tangent handles for the selected keyframes are restored to an unbroken state.

Hiding Tangent Handles

You can hide tangent handles for keyframes that are not selected.

To hide inactive tangent handles:

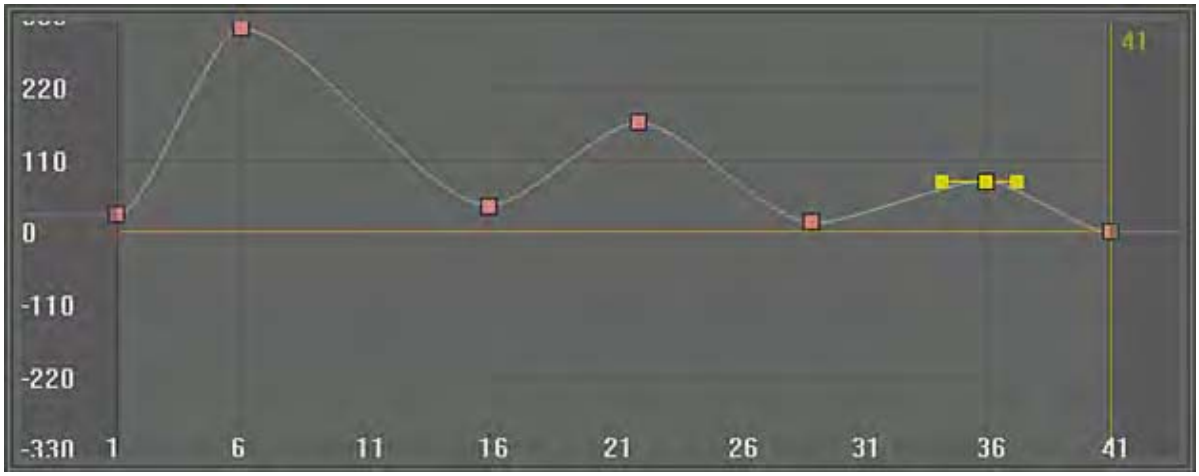
1 Click Animation.

2 Enable the Animation Options (page 2176).

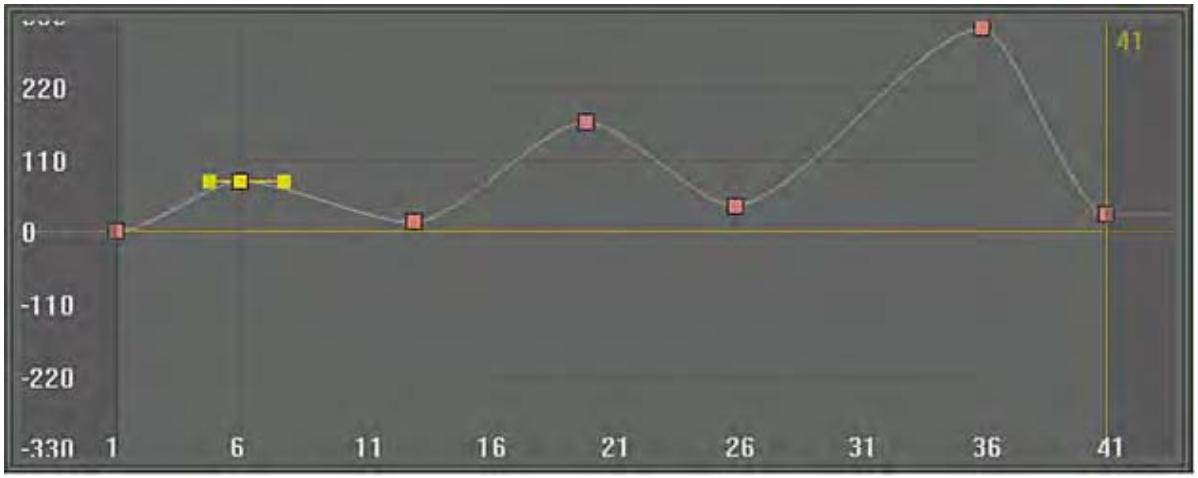
3 Enable the Hide Inactive Tangents toggle button.

Reversing Keyframes

You can reverse the order of keyframes in time. When you reverse keyframes, the first keyframe takes the place of the last, and the last takes the place of the first.



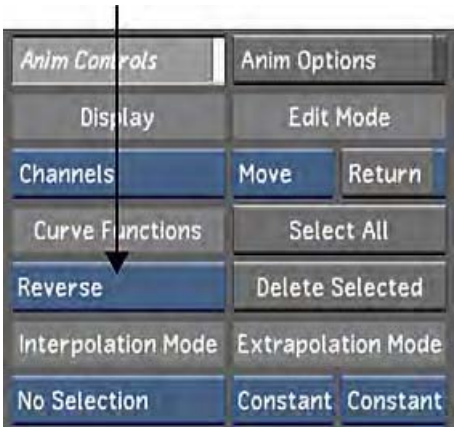
Before reversing the animation curve



After reversing the animation curve

To reverse keyframes for a channel:

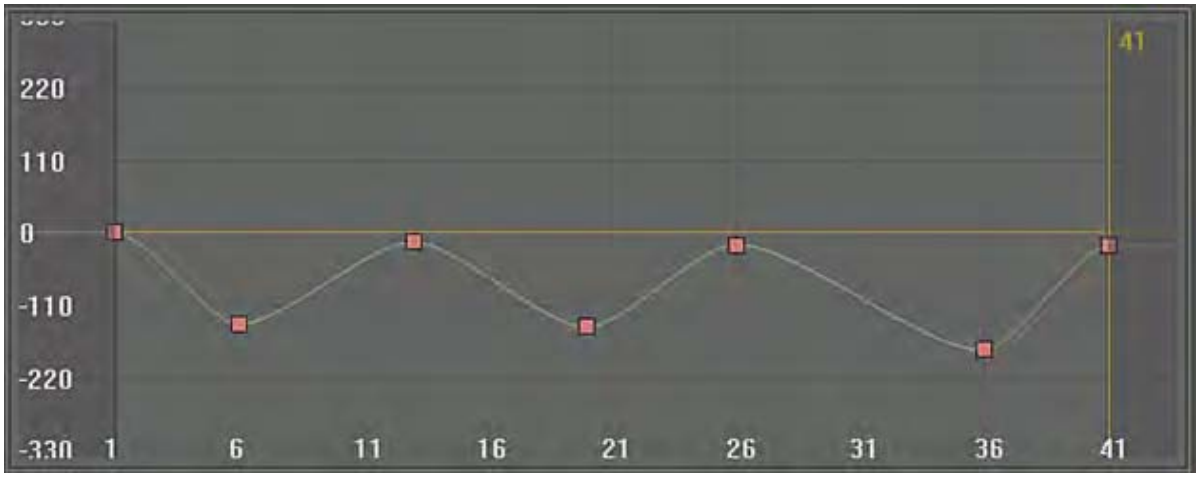
1. Click Animation.
2. In the Channel hierarchy, select the channel you want to reverse.
3. Enable the Animation Controls panel.
4. In the Curve Functions option box, select Reverse.



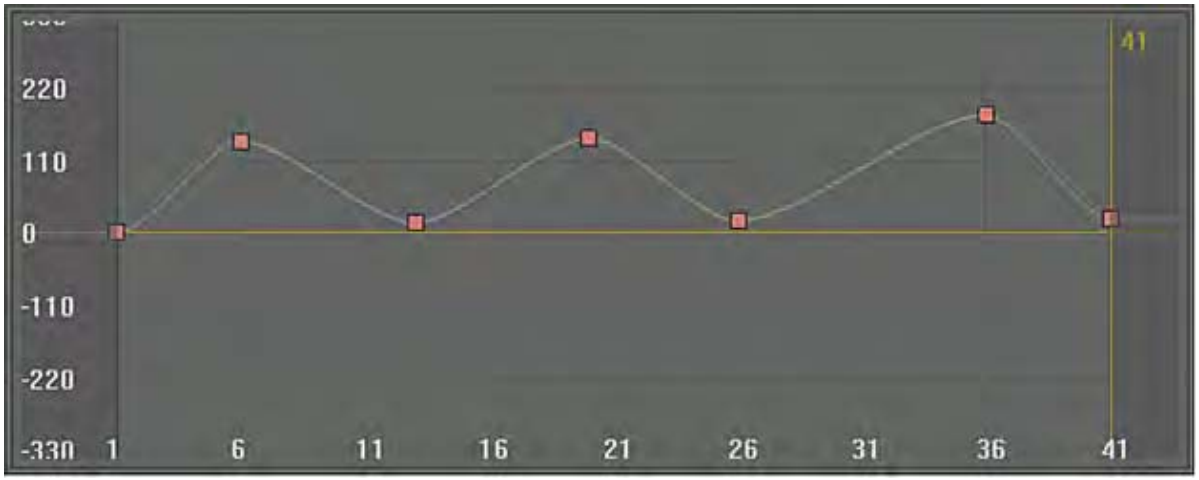
The selected channel's keyframes are reversed.

Inverting Keyframes

You can invert the value of keyframes. When you invert keyframes, a mirror of the value is produced about the horizontal time axis.



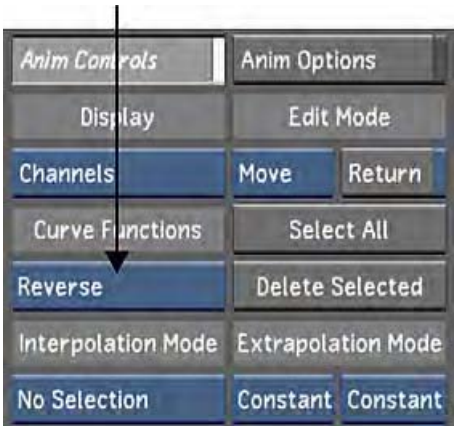
Before inverting the animation curve



After inverting the animation curve

To invert keyframes for a channel:

1. Click Animation.
2. In the Channel hierarchy, select the channel you want to invert.
3. Enable the Animation Controls panel.
4. In the Curve Functions option box, select Invert.



The selected channel's keyframes are inverted.

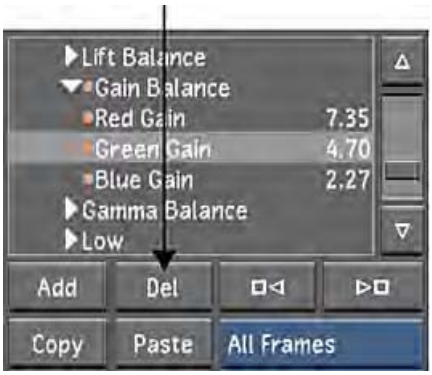
Deleting Keyframes

You can delete keyframes when they are no longer needed. You can:

- Delete a single keyframe.
- Delete a selection of keyframes.
- Delete keyframes interactively, enabling you to click directly on keyframes in the Animation Curves window to delete them.

To delete one keyframe:

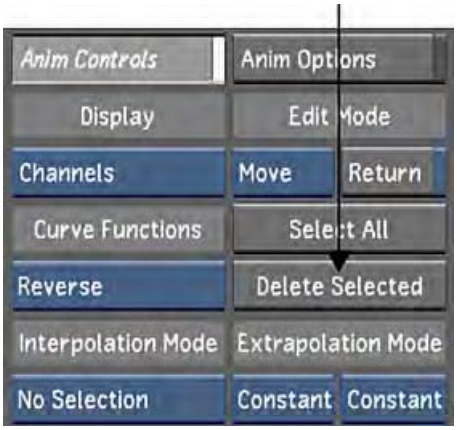
1. Scrub the positioner to the frame with the keyframe you want to delete.
2. Click the Delete button.



The selected keyframe is deleted.

To delete a selection of keyframes:

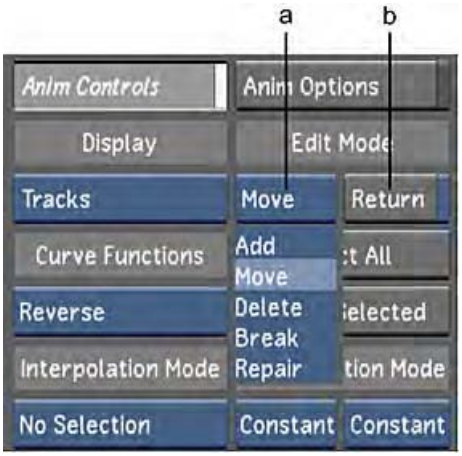
1. In the Edit Mode box, select Move.
2. In the Animation Curves window, select the keyframes you want to delete.
3. Enable the Animation Controls.
4. Click Delete Selected and then confirm the deletion. You can also press `Delete`.



The selected keyframes are deleted.

To delete multiple keyframes interactively:

1. Enable the Animation Controls.
2. In the Edit Mode box, select Delete.



(a) Edit Mode option box (b) Return toggle button.

NOTE Disable Return if you want to delete multiple keyframes. If Return mode is enabled, you are returned to Edit mode after deleting one keyframe.

3. In the Animation Curves window, click a keyframe to delete it.
4. You can click to delete as many keyframes as needed.

Copying and Pasting Keyframes

Copying and pasting keyframes eliminates the need for manually duplicating your creative accomplishments. With copying and pasting, you only need to set up a keyframe once and then you can reuse it as many times as your project demands. With Lustre, you can easily copy a single keyframe or multiple keyframes from a source channel or folder (group of channels) and paste them in a target channel or folder in the Channel hierarchy.

Keyframes for grading, geometries, point trackers, and Effects plugins are just some of the types of keyframes that you can copy and paste. When copying and pasting keyframes, keep in mind that you can only copy and paste the same channels. For example, when you copy a keyframe for a grading channel, you can only paste it to another grading channel. For copying and pasting keyframes for geometries, point trackers, and Effects plugins, see Guidelines for Geometries, Point Trackers, and Effects Plugins (page 2200).

NOTE A red dot next to a channel or folder indicates keyframes have been applied.

There are three different tools you can use to perform a copy and paste procedure:

- The user interface. See About the Copy/Paste Keyframe Controls (page 2195).
- Hotkeys. See About Copy/Paste Hotkeys (page 2195).
- The Autodesk Control Surface (ACS). See the *Autodesk Control Surface User Guide*.

There are four main ways to copy and paste keyframes:

- On the current frame. See Copying and Pasting Keyframes on the Current Frame (page 2196).
- All of the keyframes in the shot. See Copying and Pasting All Keyframes in a Shot (page 2197).
- A manual selection of keyframes. See Copying and Pasting a Selection of Keyframes (page 2198).

■ Over a selection of keyframes. See Copying and Pasting Over a Selection of Keyframes (page 2198).

In the case where you want to create a dissolve using keyframes, you can paste a sequence of keyframes that exceeds the target shot boundary.

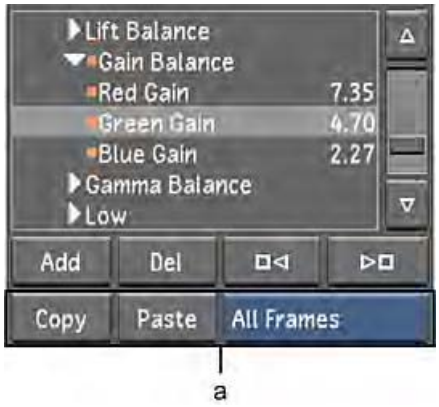
See Pasting a Keyframe Sequence That Exceeds a Shot Boundary (page 2199).

About the Copy/Paste Keyframe Controls

The Copy/Paste Keyframe controls are located in the Channel hierarchy, which is displayed in the user interface in all but the following menus:

- Editing > Timeline
- Editing > Capture
- Editing > Playout
- Colour > Timeline

The Channel hierarchy and Copy/Paste Keyframe controls do, however, remain operational, even though they are not displayed in these menus. Any Copy Keyframe actions and channel/folder selections you perform remain active, even after you leave the Channel hierarchy.



(a) Copy/Paste Keyframe controls

Copy Click to copy one or more keyframes to memory. The Copy function is dependent on what is selected in the Channel hierarchy, the location of the positioner, and whether One Fr or All Fr is enabled.

Paste Click to paste the copied keyframes held in memory. The Paste function is dependent on what is selected in the Channel hierarchy, the location of the positioner, and whether or not keyframes are selected in the target channel or folder.

Frame Selection option box Select whether to copy the current keyframe present at the location of the positioner (One Frame) or all keyframes (All Frames) for a selected channel or folder.

NOTE If you manually select keyframes in the Channel Editor, you do not need to use the Frame Selection option box. Lustre automatically copies and pastes the selected keyframes.

About Copy/Paste Hotkeys

You can use the copy/paste hotkeys in any of Lustre's menus to copy and paste keyframes. For the procedures below, the following hotkeys can be used:

- `CTRL(left)+C` to copy keyframes

- CTRL(left)+V to paste keyframes

See Animation (page 2360) for a complete list of Animation hotkeys.

Copying and Pasting Keyframes on the Current Frame

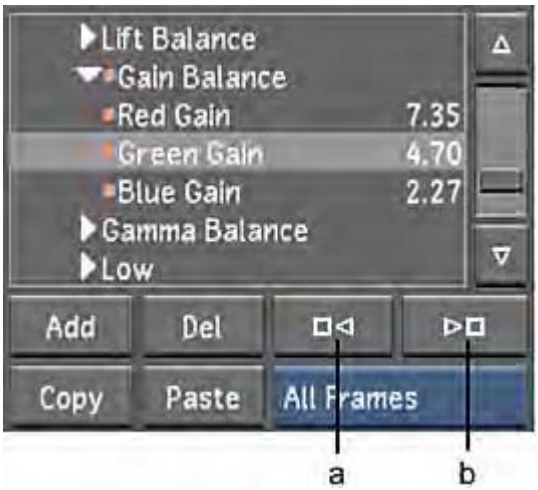
This procedure enables you to copy from the source only the keyframes on the current frame and paste them in the target where the positioner is located.

To copy and paste only the keyframes on the current frame:

1. Navigate to the shot from which you want to copy keyframes.
2. In the Channel hierarchy, select the source channel or folder whose keyframes you want to copy.

   TIP If you want to view the entire range of keyframes for the selected channel or folder, click Range in the Animation > Curves menu.

3. Set the copy mode. Do one of the following:
   - In the Frame Selection option box, select One Frame.
   - Use the ACS.
4. Click the Next or Previous buttons to navigate to the keyframes that you want to copy.



(a) Previous button (b) Next button

   The positioner moves to the next keyframe in the sequence.
5. When you have located the appropriate keyframes, do one of the following:
   - Click Copy.
   - Press CTRL(left)+C.
   - Use the ACS.
6. Navigate to the target shot and then place the positioner on the frame where you want to paste the keyframes.
7. In the Channel hierarchy, select the target channel or folder into which you want to paste the copied keyframes.

   NOTE If there are keyframes present in a channel or folder at the location of the positioner, they will be overwritten.

8 Paste the keyframes held in memory. Do one of the following:
- Click Paste.
- Press CTRL(left)+V.
- Use the ACS.

The shot is updated in the Player with the pasted keyframe data.

Copying and Pasting All Keyframes in a Shot

Copying a sequence of keyframes and pasting it in a target that has no keyframes selected, overwrites any keyframe data that are present, beginning at the location of the positioner (for the length of the source sequence). For example, if the source sequence of keyframes is ten frames long, then only ten frames of keyframes are overwritten in the target, beginning at the location of the positioner.

NOTE Should the pasted keyframes extend beyond the target shot boundary, see Pasting a Keyframe Sequence That Exceeds a Shot Boundary (page 2199).

This procedure enables you to copy all of the keyframes in a shot for the selected source channel or folder, regardless of the location of the positioner, and paste them in a target channel or folder, starting from where the positioner is located.

To copy and paste all of the keyframes in the shot for the selected channel or folder:

1 Navigate to the shot from which you want to copy keyframes.
2 In the Channel hierarchy, select the source channel or folder whose keyframes you want to copy.
3 Set the copy mode. Do one of the following:
- In the Frame Selection option box, select All Frames.
- Use the ACS.

4 Copy all the keyframes. Do one of the following:
- Click Copy.
- Press CTRL(left)+C.
- Use the ACS.

5 Navigate to the target shot and then place the positioner on the frame where you want to paste the keyframes.
6 Select the target channel or folder into which you want to paste the copied keyframes.

NOTE If there are keyframes present in a channel or folder at the location of the positioner, they will be overwritten.

7 Paste the keyframes held in memory. Do one of the following:
- Click Paste.
- Press CTRL(left)+V.
- Use the ACS.

The shot is updated in the Player with the pasted keyframe data.

Copying and Pasting a Selection of Keyframes

This procedure enables you to copy a manual selection of keyframes in a shot for the selected channel or folder.

To copy and paste a selection of the keyframes in the shot for the selected channel or folder:

1. Navigate to the shot from which you want to copy keyframes.
2. In the Channel hierarchy, select the source channel or folder whose keyframes you want to copy.
3. Click Range to view the entire range of keyframes for the selected channel or folder.
4. Select the keyframes you wish to copy. See Selecting Keyframes (page 2184).
5. Copy the selected keyframes. Do one of the following:
   - Click Copy.
   - Press `CTRL(left)+C`.
   - Use the ACS.
6. Navigate to the target shot and then place the positioner on the frame where you want to paste the keyframes.
7. Select the target channel or folder into which you want to paste the copied keyframes.

   NOTE If there are keyframes present in a channel or folder at the location of the positioner, they will be overwritten.

8. Paste the keyframes held in memory. Do one of the following:
   - Click Paste.
   - Press `CTRL(left)+V`.
   - Use the ACS.

The shot is updated in the Player with the pasted keyframe data.

Copying and Pasting Over a Selection of Keyframes

This procedure enables you to copy and paste keyframes over a selection of existing target keyframes while preserving unselected target keyframe(s).

To copy and paste keyframes over a selection of keyframes:

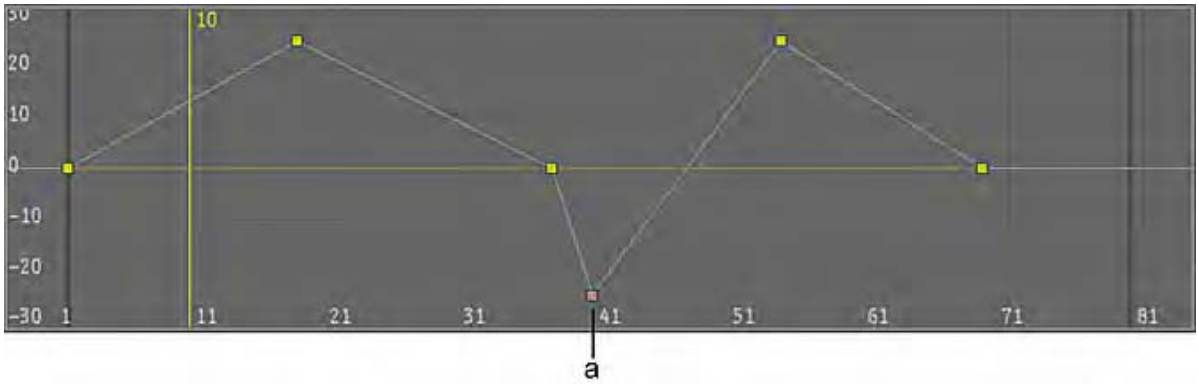
1. Navigate to the shot from which you want to copy keyframes.
2. In the Channel hierarchy, select the source channel or folder whose keyframes you want to copy.
3. Click Range to view the entire range of keyframes for the selected channel or folder.
4. Select the keyframes you wish to copy. See Selecting Keyframes (page 2184).
5. Copy the selected keyframes. Do one of the following:
   - Click Copy.
   - Press `CTRL(left)+C`.
   - Use the ACS.
6. Navigate to the target shot and then place the positioner on the frame where you want to paste the keyframes.
7. Select the target channel or folder into which you want to paste the copied keyframes.

8 In the Animation Curves window, manually select the keyframes that you want to overwrite with the copied keyframe data.

9 Paste the keyframes held in memory. Do one of the following:

- Click Paste.
- Press `CTRL(left)+V`.
- Use the ACS.

In the Animation Curves window, unselected keyframe(s) are preserved, as seen in the following example.

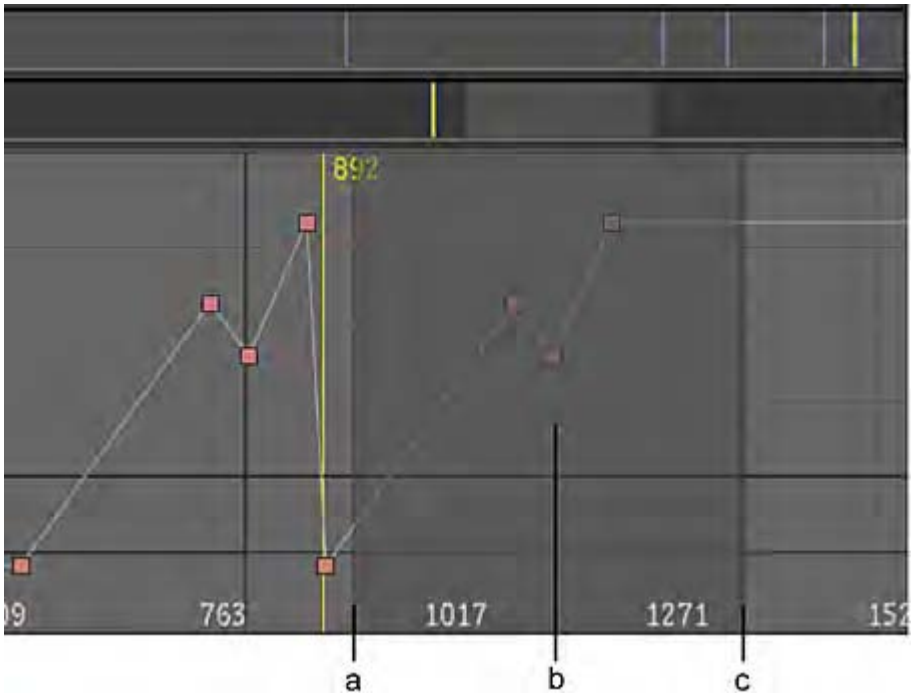


(a) Keyframe preserved

Pasting a Keyframe Sequence That Exceeds a Shot Boundary

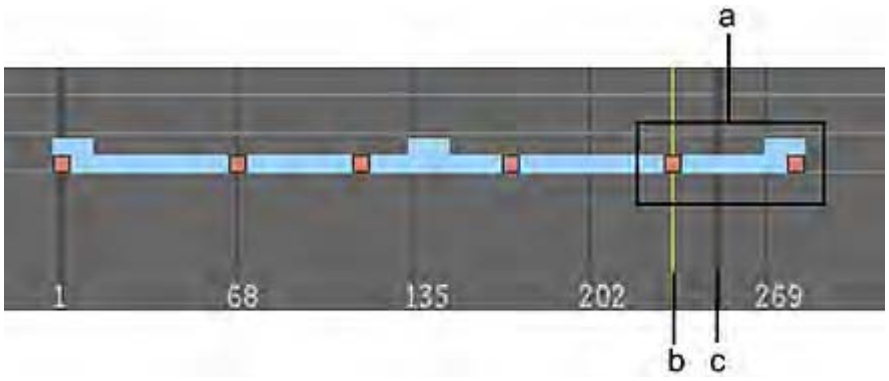
Pasting a keyframe sequence that extends beyond the boundary of the target shot can be really useful when creating a dissolve. See Creating Dissolves (page 1927).

The target shot can be shorter in length than the keyframe sequence due to its actual length or its having been trimmed. When you copy and paste a keyframe sequence that exceeds the boundary of the target shot that has been trimmed, the excess keyframes in the sequence are still pasted, but they appear in a greyed out area. The greyed out area represents the available handles of a trimmed shot and contains the “inactive” keyframes, as seen in the following example. This means the keyframes are available for creating a dissolve.



(a) Shot boundary (b) Excess keyframe data (c) Beginning of next shot

You can also paste a keyframe sequence to a target shot that is of equal or greater length and then manually trim the shot. The keyframes will still exist in the handles (greyed out area). The excess keyframes can also be viewed in the Track Editor (page 2201).



(a) Pasted keyframe sequence (b) Positioner (where first of copied keyframes is pasted) (c) Shot boundary

Guidelines for Geometries, Point Trackers, and Effects Plugins

When copying and pasting keyframes for geometries, point trackers, and Effects plugins, you need to be aware of the following guidelines:

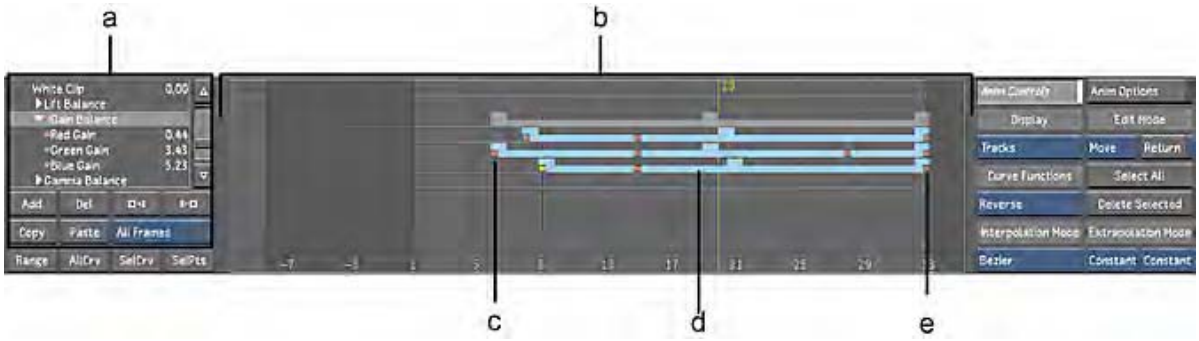
- You can only copy from and paste to corresponding existing channels. For example, an Effects plugin channel can only be copied from and pasted to a matching Effects plugin channel. Non-matching channels will not be pasted.
- When there are more source channels in the copy buffer than there are corresponding channels in the target, Lustre will match channels (in top-to-bottom order) with their corresponding target channels. Non-matching channels will not be pasted.

- You can only copy and paste the keyframes for common channels from one geometry to another. For example, when copying a rectangle geometry and pasting it in a circle geometry, all but the control points are copied.
- Axes are not saved with absolute values; therefore, it is not possible to copy and paste an axis while maintaining its original position. If you perform such an action, you will have to manually modify the appropriate axis channel values.

The Track Editor

Use the Track Editor to adjust animation timing after you create an animation. With the Track Editor, you can select and view all the keyframes and channels in your animation, but you do not see the animation curves. Instead, you see a track representation of each animation curve. Using the handles on the track, you can move the keyframes in time, but you cannot modify their value. This makes it easier to change animation timing to match on-screen occurrences.

In the Track Editor, channels are represented by horizontal bars called *tracks*. The channel name appears in the Channel hierarchy to the right of the Track Editor.



(a) Channel hierarchy (b) Track Editor (c) First keyframe in channel (d) A selected channel (e) Last keyframe in channel

The position and length of a track is determined by the first and last keyframes of the channel it represents. The track extends from the frame number of the first keyframe to the frame number of the last keyframe.

Light blue tracks represent selected channels. Dark blue tracks are for deselected channels. Grey tracks represent folders that contain at least one animatable channel.

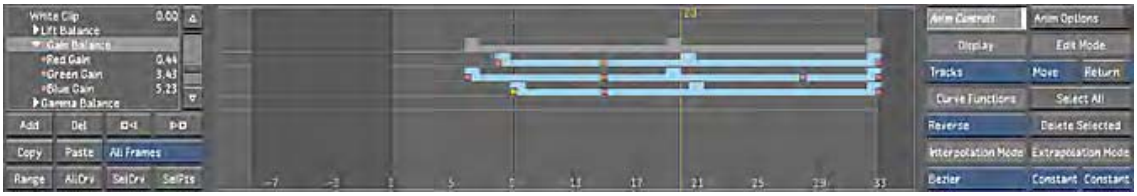
Superimposed over each track are small orange squares that represent the position of the channel's existing keyframes. You can adjust the position of these keyframes without affecting their channel value.

Accessing the Track Editor

Use the Track Editor to adjust animation timing for an existing animation.

To access the Track Editor:

1. Click Animation.
2. Enable the Animation Controls.
3. In the Display option box, select Tracks.
   The Track Editor appears.



Selecting Tracks

Before you can adjust an animation's timing, you must select the channel in the Channel hierarchy.

To select a channel in the Channel hierarchy:

1. Display the Track Editor (page 2201).
2. In the Channel hierarchy, either select a folder (to select all channels), or expand a folder and select individual channels.

   Selected channels are light blue. Deselected channels are dark blue.

Adjusting Animation Timing

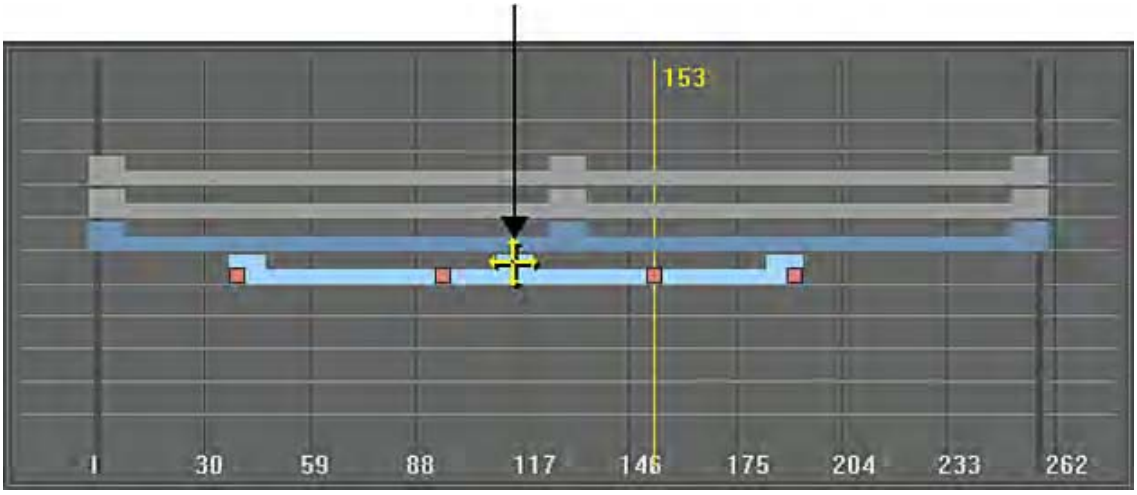
Using the Track Editor, you can adjust a keyframe's timing without adjusting its value. You can also slide the entire animation ahead or back in time, and change the duration of an animation.

To adjust the timing for an entire channel:

1. Display the Track Editor (page 2201).
2. In the Channel hierarchy, select the channel you want to modify.

   The corresponding track in the Track Editor turns light blue and its keyframes appear.

   TIP Click Range to see all three track handles.
3. In the Edit Mode box, select Move.
4. Move the mouse over the centre light-blue handle. When the four-direction arrow cursor appears, drag to the left or right to slide the entire animation forward or backward in time.



NOTE Adjusting timing for a folder changes the timing for all channels in that folder.

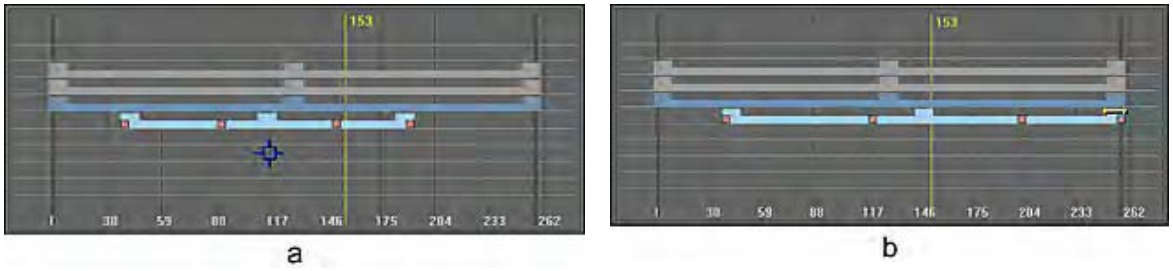
To adjust keyframe timing:

1. Display the Track Editor (page 2201).
2. In the Channel hierarchy, select the channel you want to modify.
   The corresponding track in the Track Editor turns light blue and its keyframes appear.
3. In the Edit Mode box, select Move.
4. Drag the keyframe to the left or right to slide the keyframe forward or backward in time.

   NOTE Dragging a keyframe in the Track Editor does not change the channel value.

Adjusting Animation Scaling

Using the Track Editor, you can independently modify the animation's start or end point. This changes the animation's overall duration and scales the keyframes. Dragging the start or end point compresses or stretches the keyframes in the selected channels. If you want to simply move the first or last keyframe to another point in time, drag the keyframe.



(a) Before scaling keyframes (b) After scaling keyframes

NOTE By repositioning the start or end point, you cannot delete any existing keyframes. This means that you can only adjust the start or end time as far as the adjacent keyframe.

To adjust an animated channel's start or end point:

1. Display the Track Editor (page 2201).
2. In the Channel hierarchy, select the channel you want to modify.
   The corresponding track in the Track Editor turns light blue and its keyframes appear.
3. In the Edit Mode box, select Move.
4. Select the first or last keyframe and drag it to change the start or end point of the animation.

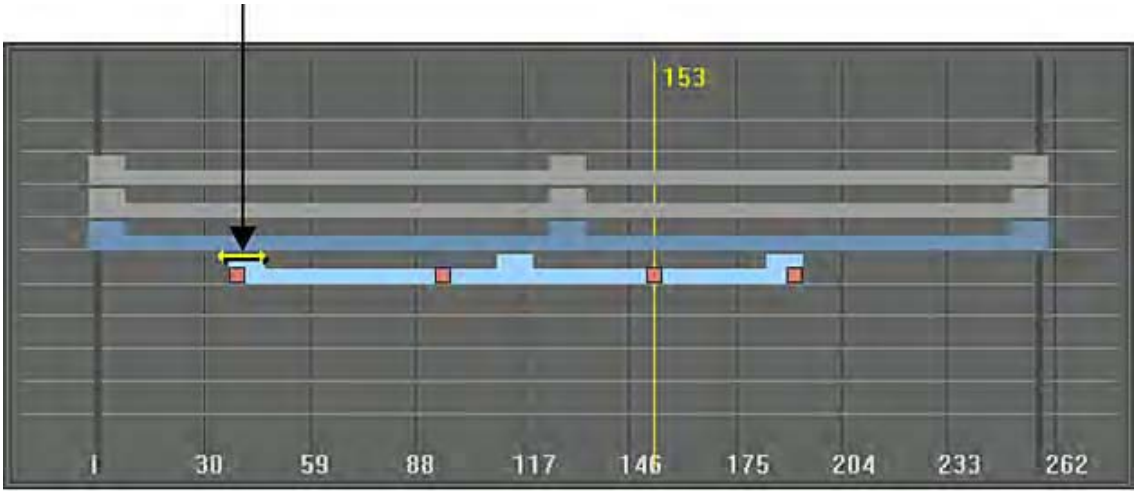
   NOTE Adjusting the timing for a folder changes the timing for all channels in that folder.

To scale an animated channel's timing:

1. Display the Track Editor (page 2201).
2. In the Channel hierarchy, select the channel you want to modify.
   The corresponding track in the Track Editor turns light blue and its keyframes appear.

3 In the Edit Mode box, select Move.

4 Move the mouse over the starting light-blue handle. When a two-directional arrow cursor appears, drag to the left or right to slide the start point forward or backward in time.



As you drag the handle, the keyframes in the track scale proportionally to the amount you move the handle.

NOTE Adjusting the timing for a folder changes the timing for all channels in that folder.

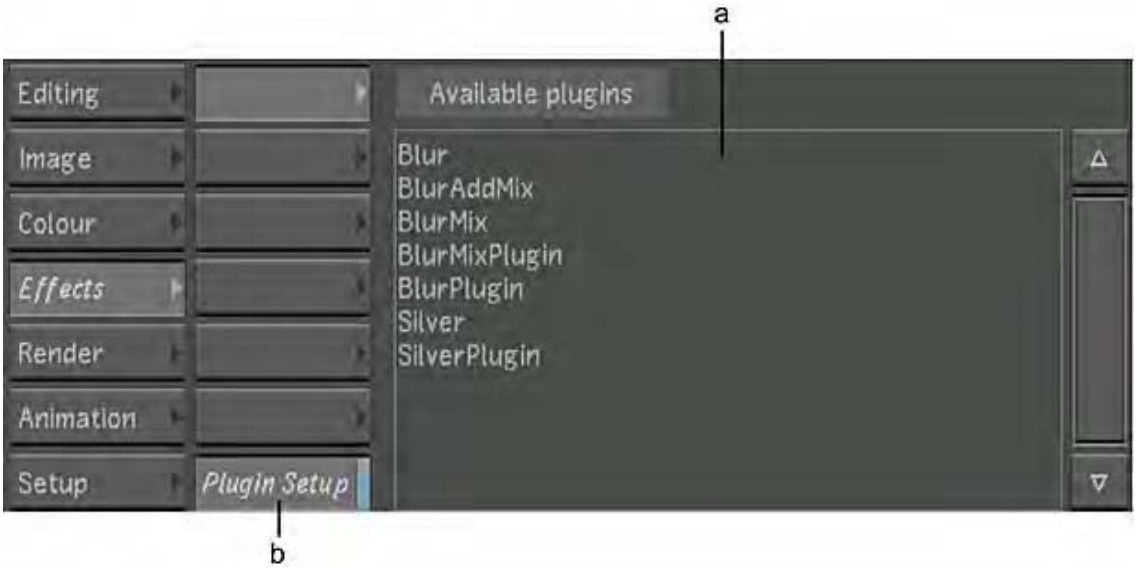
Animating in Other Menus

You can create animations directly in the other menus without having to access the Animation menu. For example, you can change the Grading settings directly in the Colour menu. In the following example, you create an animation using a blur plugin effect.

To create animations in the Effects menu:

1 Select a shot and scrub to the first frame.

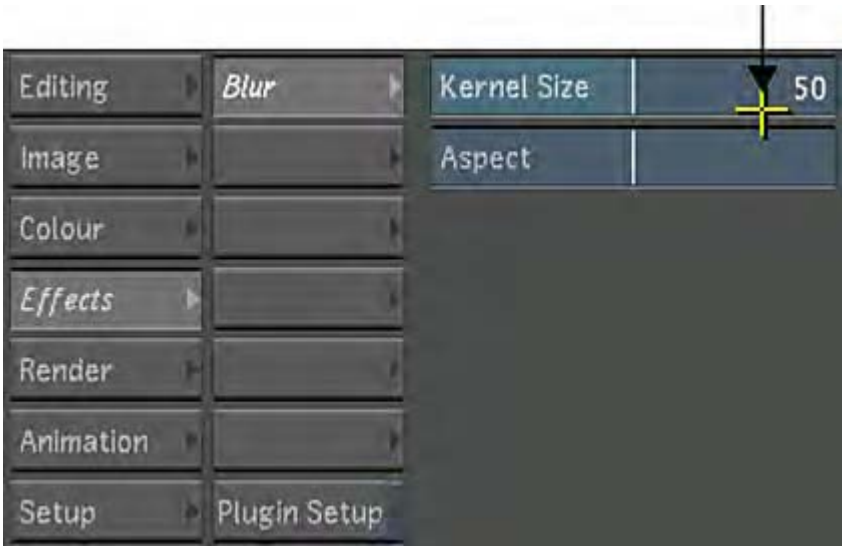
Click Effects, and then enable Plugin Setup.



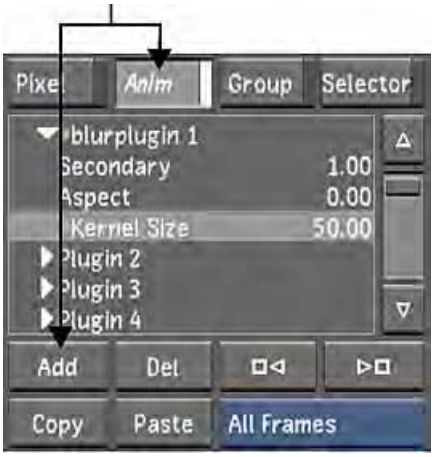
(a) Available plugins list (b) Plugin Setup button

The Available plugins list appears.

2 Select Blur from the plugin list, and then set the kernel Size to 50.



3 Click Anim, and then click Add.



A keyframe is added at the current frame. When adding keyframes in this way, the interpolation type is always set to Bezier. To change it, you have to use the Interpolation controls in the Animation menu.

4 Scrub to the last frame of the shot.

5 Set the Kernel Size to 0.

6 Click Add again to add another keyframe.

7 Scrub through the shot to see the animation.

8 To edit the shape of the animation curve, use the tools in the Animation menu. See Modifying Your Animation (page 2186).

Undoing and Redoing Operations

When working in the Animation menu, you can undo and redo all operations except the following:

- Canvas pan and zoom changes
- Changes made using the Align options

Stereoscopy

The Stereoscopy features of Lustre allow you to grade shots and see them in stereoscopic context. Stereoscopy allows you to take your shots and create the illusion of depth, within the image. In order to create this illusion of depth, you need to present each eye with a slightly different image. This is done by having two sets of footage or conformed timelines; one designated for the left point of view (Left Eye) and the other designated for the right point of view (Right Eye). Once you have one set of footage for the Left and Right Eye, you would use the encoding/decoding scheme to display the image. Encoding is done at the stereoscopic display level by the graphic SDI board and decoding is done by the glasses. This chapter shows you how to enable and use the Stereoscopy features.

Creating a Stereoscopic Project

When you create a stereoscopic project, the Left and Right folders are automatically created in the Library for your stereoscopic footage. See Project Settings (page 1738)

It is also possible to enable Stereoscopy from a monoscopic project, in which case you need to do the following:

- Enable the Stereoscopy feature. See Enabling Stereoscopy (page 2207).
- Load your left and right eye footage into your timeline. See Setting Up the Stereoscopic Footage (page 2209).

Enabling Stereoscopy

You can enable the Stereoscopy feature in one of the following ways:

- When you select your project settings before entering the Lustre application. See Project Settings (page 1738).
- By enabling the Stereoscopy button. See Enabling the Stereoscopy Button (page 2208).

Selecting a Raster for Stereoscopy

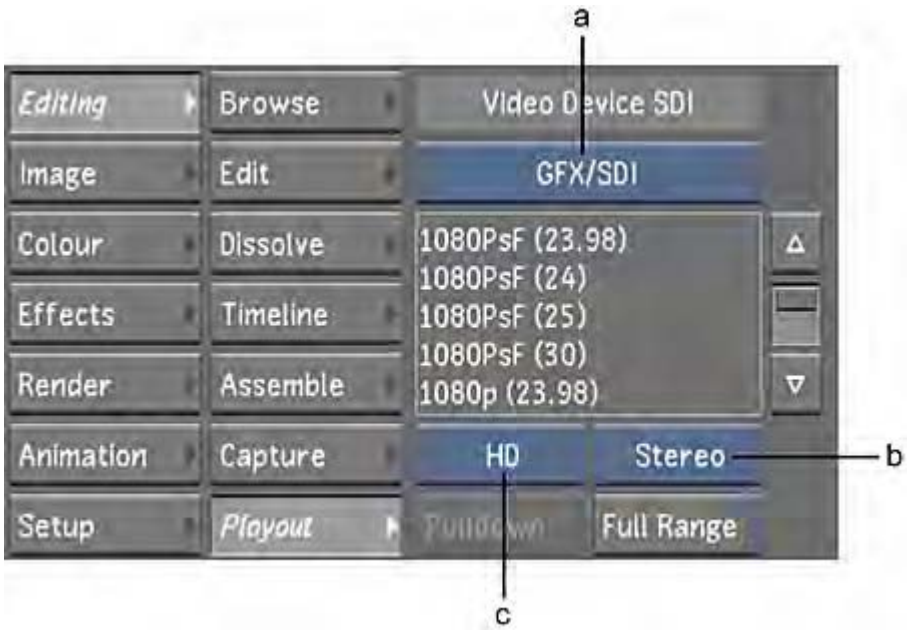
You want to only select a stereoscopic raster if you are connected to a proper stereoscopic device (e.g., Stereoscopy projectors (two projectors or one), or a 3D monitor). When you are playing out your footage to a dual link device, each eye is assigned to one link. The Left Eye always runs through the A Link and the Right Eye through the B Link. Refer to the *Autodesk Lustre Hardware Setup Guide*.

To select a stereoscopic raster:

1. Click Editing in the main menu and then click Playout.

   The Playout menu is displayed.
2. In the Video Device SDI group, toggle the Video/Graphics Raster button to display GFX/SDI, toggle the format option button to HD, toggle the link type button to Stereo, and select a Stereo raster from the list.

   NOTE The GFX/SDI raster outputs in 8-bit range when you are working in Stereo mode.



(a) Video/Graphics Raster button (b) Link Type button (c) Format button

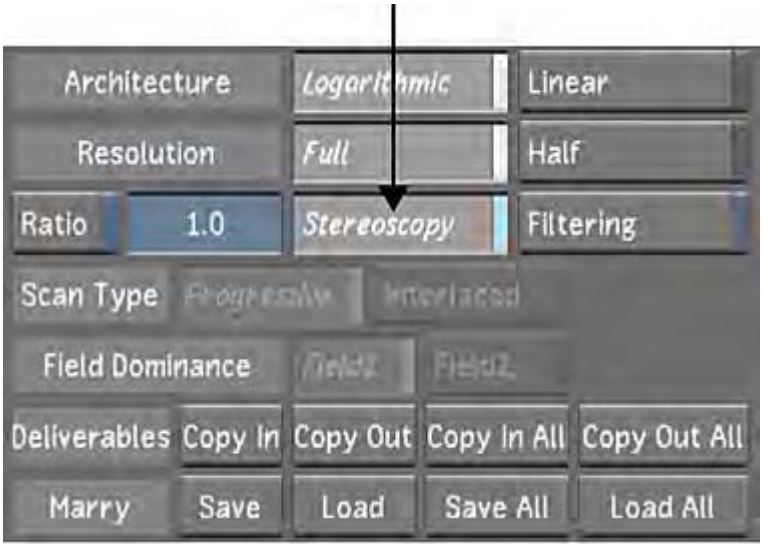
Stereoscopic outputs are now enabled.

Enabling the Stereoscopy Button

You can disable and re-enable the Stereoscopy feature by using the Stereoscopy button. You would enable the Stereoscopy button when you are assembling an EDL independently for each eye to create a two-layer timeline. You would then convert it to a Stereo timeline.

To enable the Stereoscopy feature:

1. Click Setup in the main menu and then click Grade.
   The Grade menu is now displayed.
2. Enable the Stereoscopy button.



   The Stereoscopy feature is now enabled.
3. (Optional) Click Editing and then click Playout. Within the Playout menu, select a raster from the Video Device SDI group



(a) Raster list

Setting Up the Stereoscopic Footage

Once you have created a Stereoscopic project, a Library with Left and Right folders, as well as a stereo timeline with a left and right layer is automatically created.

NOTE The stereoscopic timeline is locked as follows:

- The top layer is the left eye.
- The bottom layer is the right eye.

To load the stereoscopic footage to the timeline:

1. Drag your stereoscopic footage from the folder location to the Library. See, Loading Clips to the Library (page 1874).
2. Select and drop the footage for the Left eye into the Storyboard.
3. Click on the L/R button to enable the Right eye.
4. Select and drop the footage for the Right eye into the Storyboard.

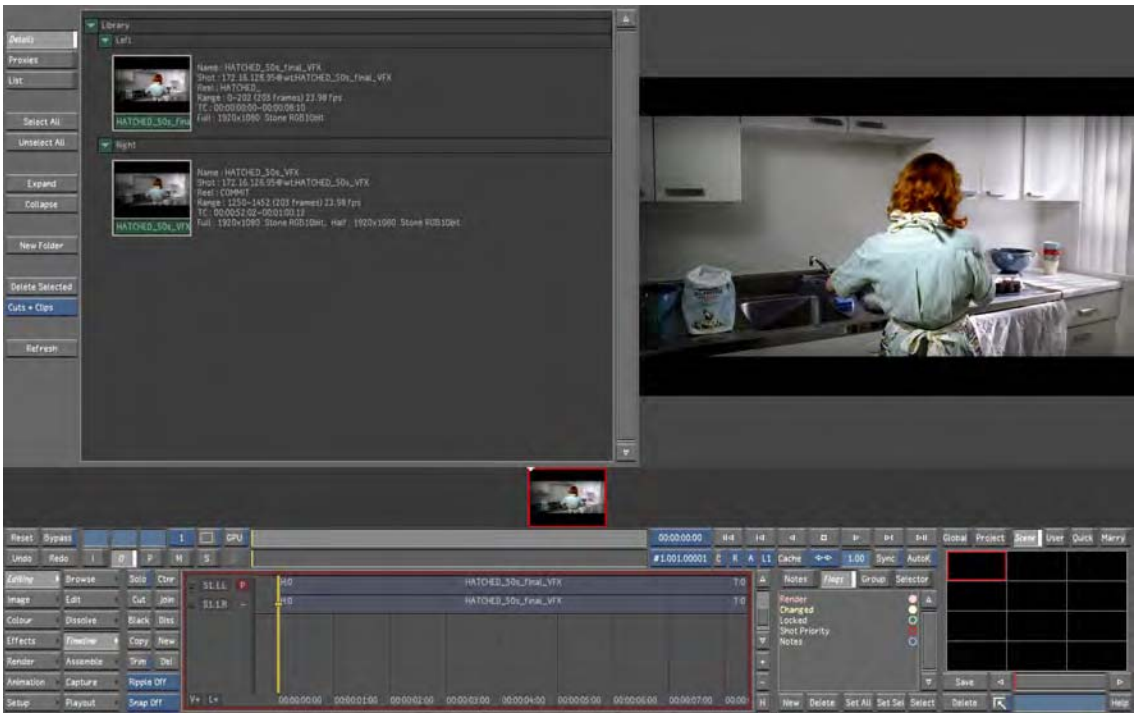


Image courtesy of EVN PRODUCTIONS

5. Save the new cut. See Managing Cuts (page 1773).

Assembling a Stereoscopic EDL

Use an EDL to rebuild a timeline using the original footage that matches the edited timeline from the offline editing stage of post-production.

To assemble a Stereoscopic EDL:

NOTE This procedure is for every type of media except RED media.

1. Import the Left and Right Eye media that you want to assemble into its corresponding Left or Right folder within the Library.
2. Click Editing in the Main menu, and then click Assemble.

   The Assemble menu is displayed.
3. Select the EDL you want to load from the EDL list and then click Load EDL.
4. Click Match Media.
5. Click Assemble to conform the EDL.

   A stereoscopic EDL is created.

To assemble a stereoscopic EDL for RED Stereo workflow:

1. Import the Left and Right Eye media that you want to assemble into its corresponding Left or Right folder within the Library.
2. Click Setup in the Main menu, and then click Grade.

   Make sure the Stereoscopy button is disabled.
3. Click Editing in the Main menu, and then click Assemble.
4. Select the Right Eye EDL from the EDL list and click Load EDL.
5. Within the Library, select the Right Eye folder.
6. Enable the Use Selected Folder button.
7. Click Match Media and then click Assemble to conform the EDL.
8. Enable the To Layer button.
9. Repeat steps #4-7 for the Left Eye media.
10. Click Setup in the Main menu, and then click Grade.
11. Enable the Stereoscopy button and then save the cut.

Repositioning the Left Eye and Right Eye

Repositioning is fully supported in a stereoscopic project. When Sync mode is enabled, you can reposition both eyes at the same time. (For example, the image in both eyes are offset horizontally by the same amount.)

You can also correct convergence discrepancies for the Left Eye and Right Eye by repositioning images at the same time in an equal and opposite manner. (For example, if you scale the image in the Left Eye to 0.80, the image in the Right Eye will be scaled to -1.20.)

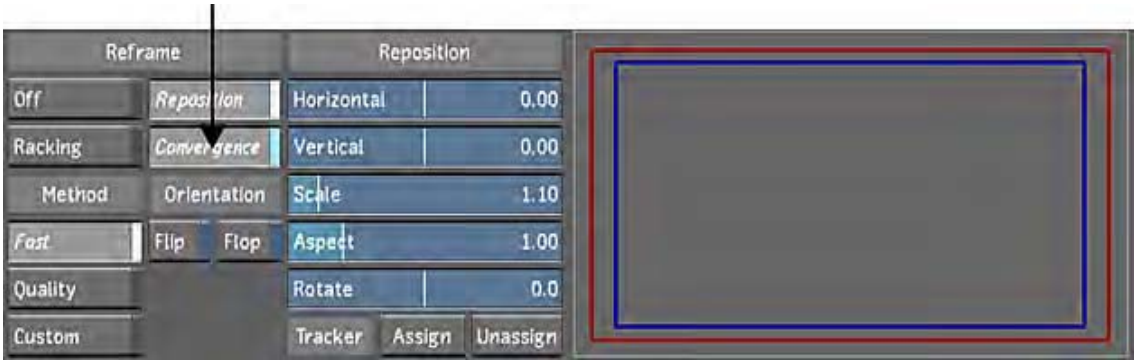
You can converge two images using the repositioning options of the Autodesk Control Surface. See Repositioning Shots (page 2449) See the Autodesk Control Surface User Guide.

TIP When converging a shot, select the Blend option in Dual View to overlay the images in the Player. See Viewing Stereoscopic Footage in Dual View (page 2212).

To converge Left Eye and Right Eye images:

1. Enable Sync mode. See Synchronizing the Left and Right Eye (page 2213).
2. In the timeline, place the positioner over the two shots that you want to converge.
3. Display the Reposition menu. See Accessing the Reposition Menu (page 2018).

4 Enable Convergence.



NOTE In the Preview window of the Reposition menu, the image reference (i.e., the blue rectangle) is for the assigned eye only.

5 Use the Reposition sliders to converge the shots. The other eye will be offset in the equal and opposite manner.

Viewing From One Eye to the Other

As you are grading your Left and Right Eye footage, there is an easy way for you to switch the footage that is being displayed in the Player. The Left and Right Eye button allows you to switch between the footage with ease.

To switch the footage displayed in the Player from one eye to the other:

1 Do one of the following:

- In the Player display controls, click on the Left/Right Eye (L/R) button.



NOTE This is the field 1 (F1)/field 2 (F2) button when you are not in Stereo mode. You can still check your field dominance when in Stereo mode by holding the Ctrl key and clicking on the Field 1/Field 2 buttons in the Setup/Grade menu.

- Press ; (semicolon).

If you are currently displaying the Right Eye timeline in the Player, the layer output selector is labelled R. When you click the R button, the layer output selector is now labelled L, the Left Eye is displayed in the Player, and the focus point is now on the Left Eye layer (and vice versa).

Switching the Player to display the Left or Right Eye footage does not affect what is playing out on the NVIDIA HDSDI output if you have selected a stereoscopic raster. If you have selected a mono (non-Stereo) raster, the layer (eye) you see on your monitor is what is displayed on your video output.

NOTE The position of the focus point is linked to the layer output selector.

Viewing Stereoscopic Footage in Dual View

Once a stereoscopic project is enabled, you can display each eye at the current frame at the same time in Dual View mode. As in A/B mode, you can view the images side by side (2-up mode), or a portion of each image in the same viewer (Vertical Wipe and Horizontal Wipe modes).

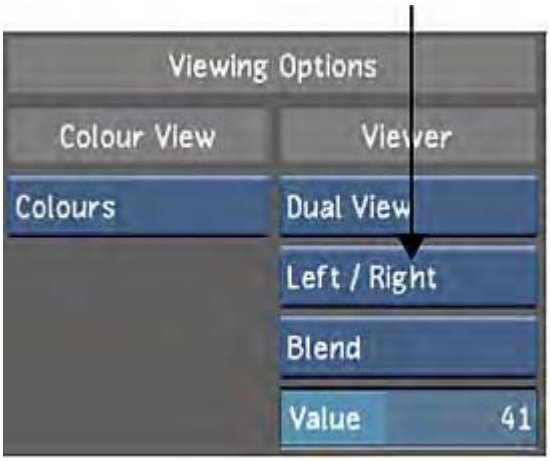
In a stereoscopic project, you can also view the Left Eye overlaid on the Right Eye, as a blended image.

To display the Dual View Blend mode:

1 Display the dual viewer. See Using Dual View (page 1986).



2 In the Dual View Display option box, select Left/Right.



3 Do one of the following:

- Select Blend.
- Press `Ctrl+F4` to cycle through display options until Blend is displayed.

  The viewer displays the image of the Right Eye overlaid over the Left Eye.



4 Set the Value slider to determine the percentage of opacity of the image in the Left Eye.

Synchronizing the Left and Right Eye

Since the Player only displays either the Left Eye or Right Eye, it is difficult and tedious to edit and grade one eye and create the same changes for the other eye. An easy way for you to make sure any editing or grading is automatically applied from one eye to the other is by using the Sync option. When this feature is enabled, any editing or grading you do on one eye is applied to the other.

To apply the editing and grading from one eye to the other:

1 From the Player display controls, click Sync.



Now all editing and grading operations on one eye will be duplicated on the other.

The Sync option only works if you do not break the sync. For example, if you enable Sync and draw a geometry, whatever you do to that geometry on one eye will be duplicated on the other. The moment you disable the Sync option and change the geometry on one eye (e.g., colour, position, vertex, etc.), that geometry will never be synced again when you re-enable Sync.

NOTE If you only break sync with the colour grading, the rest of the geometry (i.e., position, vertex, etc.) will still be in sync once you re-enable the Sync option.

When you re-enable Sync mode, the grading data that was applied separately on each eye will be retained on each eye. Re-enabling Sync mode will not copy the grading back to the other eye.

Editing the Stereoscopic Timeline

When the Sync button is enabled, all editorial operations (Cut, Trim, Copy, Delete, add a Dissolve, add black frames, Solo, etc.) and all structural operations (add a Track/Layer, add a Container, Collapse) are supported when working on stereoscopic timelines.

NOTE The sync button affects only the editorial operations. Structural operations are always performed in sync.

See Synchronizing the Left and Right Eye (page 2213).

Stereoscopy and Matte Containers/Multi-Channel Clips

When working in a stereoscopic project, entering the matte container or multi-channel segment shows both left and right eye content.

See Adding a Matte to a Matte Container (page 1945).

Swapping Left and Right Shots

Use the hotkeys ALT+L and ALT+R to swap all shots from one stereo eye to the other on the current layer. The layer indicator will not change location. It is also possible to swap shots on all layers of the primary video track with hotkeys CTRL+ALT+L and CTRL+ALT+R.

Replacing Shots in a Stereo Timeline

When working on a Stereoscopic project, you can perform a shot replace that applies to both eyes at the same time.

NOTE In Stereo, you must replace the shot for both eyes. It is not possible to use Replace Shot for only one eye.

1. Load the Timeline you want to modify.
2. In the Library, hold SHIFT key and select the Left clip. A "L" icon appears on the top right side of the thumbnail.
3. Navigate the Library to find the Right clip, hold SHIFT key and select the Right clip. A "R" icon appears on the thumbnail. Release SHIFT key.
4. Move the selection over the Storyboard or Timeline segment you want to replace and click on the thumbnail.



Both Left and Right shots have been replaced with the selection and the grading on the original shots is preserved.

Grading Stereoscopic Footage

When Sync mode is enabled, grading operations (including dragging and dropping) are applied to the respective destination eye. For example, grading data dragged from a Right Eye layer is always applied to a Right Eye destination layer, even if it is dropped on the Left Eye. See Synchronizing the Left and Right Eye (page 2213).

Shots can still be graded separately when Sync mode is disabled. After grading one eye, grading data can be copied to the other eye. See Copying Elements (page 1939). Separate grading can also be used to differentiate the grades on each eye.

During secondary colour grading, you can copy different grading data for each eye at the same time from one secondary layer to another, and the grading data for each eye that was applied when Sync mode was disabled will be copied to appropriate eyes.

Working with Grade Files

In a stereoscopic project, grade files contain data for both the Left Eye and Right Eye.

In the Grade bin, each storage container can include Left Eye and Right Eye grading data. The thumbnail of the grade corresponds to the currently assigned eye. In the Grade view, the expanded Grade bin and its player demonstrate the same behaviour. See Using Grade Bins (page 1821) and Using the Expanded Grade Bin (page 1828).

Disable Sync mode to apply data from a grade file to only the assigned eye: only the grade data for the current eye will be applied. When Sync mode is on, data from a grade file is applied to both eyes: if the grades are different for each eye, each grade is applied to the appropriate eye. See Synchronizing the Left and Right Eye (page 2213).

Bypassing Menu Parameters of an Eye

You can use the Bypass button to view the original shot of one eye or both eyes. See Bypassing Menu Parameters (page 1805).

Bypass results depend on the state of the Sync mode. See Synchronizing the Left and Right Eye (page 2213).

Set:	To:
Sync mode on	Display the original state of shots on the Left and Right Eye.
Sync mode off	Display the original state of shots on the assigned eye.

Rendering Stereoscopic Footage

The procedure for rendering your stereoscopic footage is the same as for rendering normal shots (i.e., mono mode). See Rendering Shots (page 2216).

The differences in rendering stereoscopic shots are as follows:

- The shots are rendered out to the folder you specified when creating your project (see Project Settings (page 1738)), but the Left and Right Eye footage is separated into two folders labelled 'Left' and 'Right'.
- You have to render both Left and Right Eye shots together. You cannot render each eye individually.

Once the shots have been rendered, Stereoscopy must be enabled in order for you to view your rendered footage in print (P) mode.

Rendering

Render individual shots, layer(s) in the timeline, or complete scenes in Lustre to create a new version of the footage with all colour grading, animation, repositioning, and dust removal data applied. Rendering is necessary for output and viewing both frames of interlaced footage at the same time. Furthermore, rendering 2K material with effects applied allows you to view your work in real-time.

WARNING

Due to the enhancements in the GPU, we do not recommend using Lustre to render grades created in Lustre 2011 or earlier if you are looking for pixel accuracy rendering.

You can render anytime during a colour grading project. For example, you might want to render an interim version part way through a project to show to a client, in addition to rendering the final version. Lustre has several options to make this easier—you can render full- or half-resolution versions of the grade, apply an output LUT, and render to a number of different file formats, regardless of the format of the original.

There are also several methods available for rendering your scenes—using the Lustre Render menus, or using Autodesk Burn® for Lustre to take advantage of the processing power of other machines. You can also use a Slave Renderer to render shots as you work on the grade.

In Lustre, there are two methods available for viewing rendered versions. To view any shot that has been rendered to disk, use Print view mode. See Setting the View Mode (page 1973). To quickly render a temporary version for playback, use memory caching. Frames are rendered and cached to your computer's memory and then made available for playback. See Caching Frames to Memory for Real-Time Playback (page 1996).

For increased speed of interaction on unrendered footage throughout the course of a project, you can opt to work on 1K proxies of the original footage.

Rendering Shots

You can use the Render menus to render your work. You can render selected shots, all shots in a cut, a specific layer in the multi-layer timeline, or the flattened result of a multi-layer timeline. To prepare for rendering, do the following:

- Specify the shots to render.
- Specify the layer to render.
- Set the resolution of the render files.
- Set resize options as needed.
- Specify the destination for the render files.
- Specify the output file format.
- Specify the file density (logarithmic or linear).
- (Optional) Change the output colour space or specify an output LUT.
- Set other options such as timecode burn-in and rendering with a viewing LUT.

Specifying the Shots to Render

Renders are done on a shot-by-shot basis—you can render one shot, several selected shots, or all shots in a scene. To specify a shot for rendering, you flag it. You can flag shots manually or have Lustre automatically flag any shot to which changes have been applied. By default, all shots are automatically flagged for rendering.

A file containing rendering information is created and stored in the Library directory found under each scene directory. This file contains a line for each shot that includes the shot path and name and the number 0 or 1, which indicates if a render flag is set for the shot (1) or not (0). These files are named `<grade name>`.*render* and are updated each time the grade is saved. Other applications, such as the command line renderer or scripts, can read or bypass this file setting.

If you are rendering a multi-layer timeline and Solo and Mute mode are disabled, only the flagged shots in the flattened timeline will be rendered. If you are rendering a multi-layer timeline and Solo or Mute mode is enabled, only the flagged shots in the active layer(s) will be rendered.

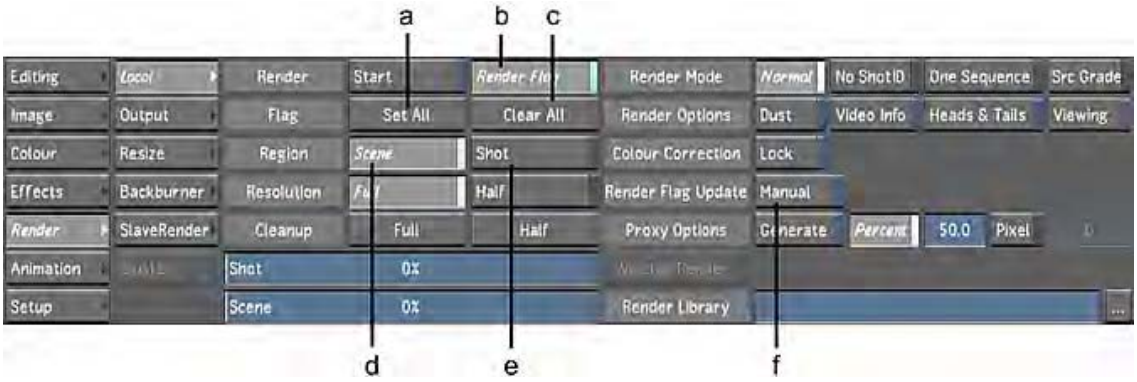
A shot that is flagged for rendering is identified by a red square in the upper-left corner of the screen.



To flag shots, use the Render Local menu.

To access the Render Local menu:

1 In the Main menu, click Render, and then click Local.
The Render Local menu appears.



(a) Set All button (b) Render Flag button(c) Clear All button (d) Render Scene button (e) Render Shot button (f) Render Flag Update button

NOTE If using the Linux version of Lustre, the Cluster menu will be part of the Render menu. This is for use with Lustre on an Incinerator high-speed network. If applicable, refer to your *Autodesk Incinerator Installation and User Guide*.

To manually flag one shot:

1 Navigate to any frame in the shot.
2 Enable Render Flag.

To manually flag all shots in the cut:

1 Click Set All.
The render flag is set for all shots in the cut and the Render Flag button is enabled.

To manually clear one flag:

1 Navigate to the shot.
2 Disable Render Flag.

To clear all render flags:

1 Click Clear All.

NOTE Render flags are not automatically cleared after rendering.

To flag shots automatically:

1 Enable Render Flag Update.
Shots will be flagged for render immediately after any grading or other effect has been applied.

After you set your render flags, you can still choose to render just the current flagged shot or all flagged shots in the timeline.

To set the shots to be rendered:

1 Do one of the following:
   - Enable Scene to render all flagged shots in the cut.
   - Enable Shot to render the current flagged shot.

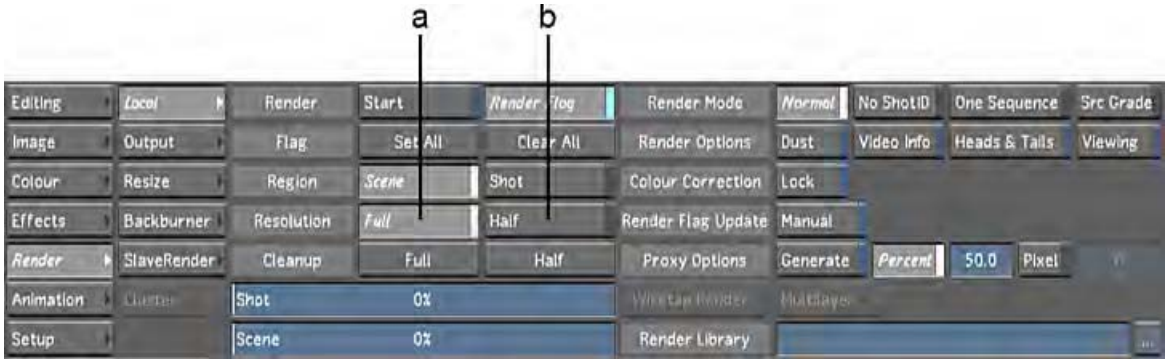
NOTE In both cases, rendering starts from the current frame and continues forward.

Setting the Resolution for the Render

You can render your original shots at full resolution, which is the same resolution as the original footage, or at half resolution. You can also render resized shots at either full or half resolution. If you use half-resolution scans as the source, they can affect the quality of the output but are usually faster to process. You may opt to do this if you want a quick, intermediate render as opposed to a final quality render.

To set the resolution, use the controls directly below the image window.

NOTE The resolution parameter in the Render Local menu is the same as that found in the Grade menu and in the user interface controls directly below the image window.



(a) Full Resolution button (b) Half Resolution button

To set the output resolution for full- or half-resolution output:

1 Specify the output resolution you want to use by enabling the Full Resolution or Half Resolution toggle, below the Viewport.

To set the output resolution to a video format:

1 Specify the source resolution you want to use for your render files by enabling the Full Resolution or Half Resolution toggle, below the Viewport.
2 Select a video format in the Render Resize menu. See Resizing Shots (page 2218).

Resizing Shots

Resize shots prior to rendering them to conform to a different destination output format. For example, if you plan on playing out a cut to a video format, use the controls in the Resize menu to select the appropriate video format and then render the cut at that format to use as the version you play out to a VTR. After you have selected the output format, you can set crop and resize options to control how the image fits into the new format.

NOTE Support for resizing depends on the version of the graphics card installed on your system. See your release notes for details.

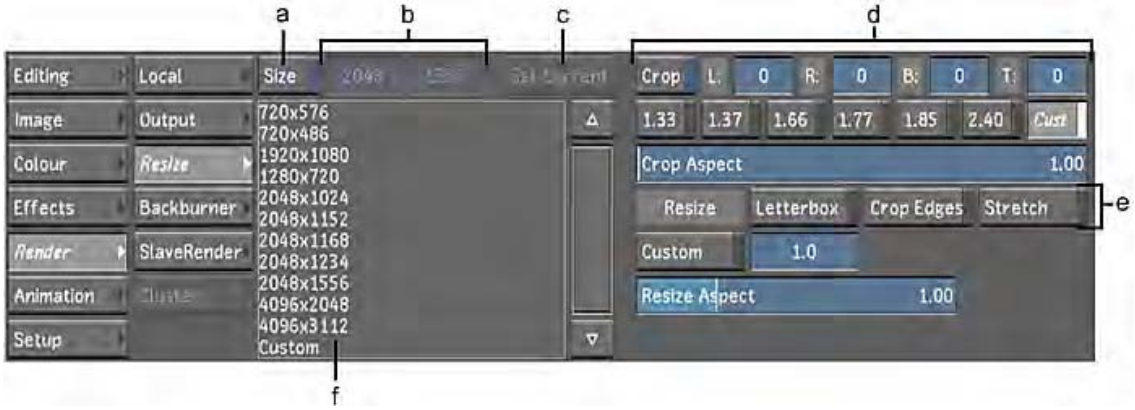
Resize data is saved with the grade in a Resize preset file that uses the grade name.

When you resize shots using the Resize menu, all shots in the timeline are resized. To resize individual shots, use the tools in the Reposition menu. See Repositioning an Image (page 2020).

To resize shots:

1. Load the grade linked to the cut containing the shots you want to resize. See Loading Grades for a Cut (page 1778).
2. Click Render and then click Resize.

   The Resize menu appears.



(a) Size button (b) Custom Width and Height fields (c) Set Current button (d) Crop options (e) Resize options(f) Video and film format list

3. To resize to a new format, such as a video format, enable Size, and then select a format from the list. If you select Custom, you can enter a width and height in the Width and Height fields, or click Set Current to set the format to the current frame size.

   If the selected format is larger than the original image size, by default a black border appears around the image, showing the final output size. If the selected format is smaller than the original image size, the image is cropped, showing the final output size.

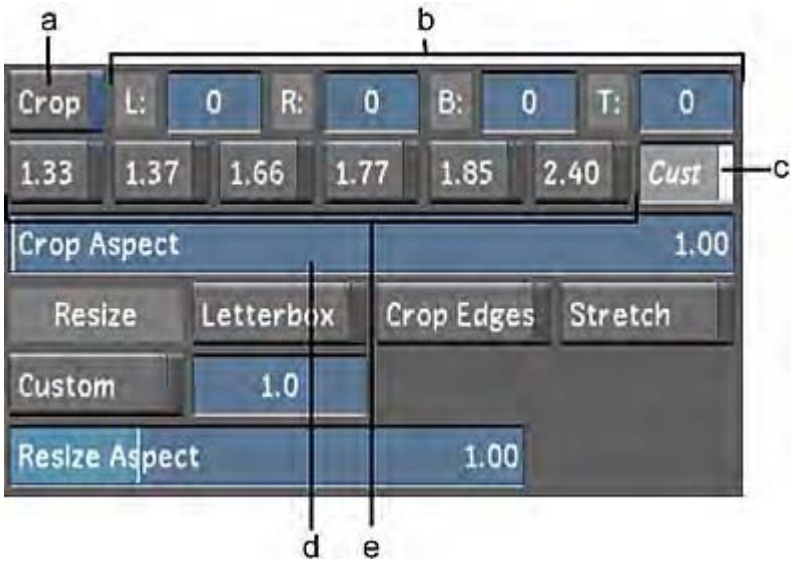
   NOTE You can check the resolution of the original scans in the Library, accessible from the Editing Browse menu. Source resolution is important for quality resizing. For example, you obtain a better quality HD output with 2K full scan sources than with 1K half scan sources.

4. Select a Resize option to specify how the image will fit into the specified format.

Select:	To:
Letterbox	Fit the entire image into the specified output format. Vertical or horizontal black borders accommodate any difference in the aspect ratio of the original image and final output format.
Crop Edges	Fill the new format size with the image, cropping the edges of the original image as needed to accommodate differences in the aspect ratio of the original image and final output format.
Stretch	Stretch the original image to fit the specified output format. If the aspect ratio of the original image and final output format is different, the image is distorted.

5 Set the crop size. Enable Crop and then do one of the following:

- Enter custom crop values in the Left, Right, Bottom, and Top fields.
- Enable one of the standard Crop Preset buttons.
- Enable Cust and then use the Crop Aspect slider to set the aspect ratio of the crop.



(a) Crop button (b) Left, Right, Bottom, and Top fields (c) Cust (Custom) button (d) Crop Aspect slider (e) Crop Preset buttons

NOTE Cropping in Lustre affects both output resolution and how the source file is read. For example, if you crop a 2K (2048x1556) scan to 1:85, the render size becomes 2048x1107. Lustre also reads the scan more quickly since there are fewer lines of pixels to go through.

6 To proportionally scale the image, enable Cust and then enter a scaling value in the Cust field.

7 If needed, you can squeeze or stretch the image horizontally using the Resize Aspect slider.

The Resize Aspect value you set is displayed in the field to the right of the slider.

Specifying the Destination for Local Render Files

The render files are saved in the location specified in the Renders Full Home or Renders Half Home, in the Project Settings. The sub-folders are named according to the render mode used, (i.e. Normal, One Sequence, etc.), which Lustre automatically creates when you render your first shots.

You can use the default path structure defined in the RenderNaming section of the init.config file. Or you can customize your renders by modifying the fields in the Render Naming menu, in the Project settings.

See

Recommended Directory Structure for Projects (page 1734).

Render Naming Settings (page 1747)

NOTE Dust removal render files created using the Rendering controls in the Dust menu are not saved with the rendered files. See Rendering the Result (page 2007).

You can specify how you want render files to be organized using the Render Place controls in the Render Local menu.



(a) Render Place controls

These controls affect the naming scheme of the render files, the location where they are saved, and the way that multiple instances of the same shot in the timeline are saved. For example, when Normal is selected, Lustre saves all the render files of each given shot to an individual folder based on the original filename and grade.

To set how render files are organized:

NOTE The following explains the render modes when using the DEFAULT render naming structure.

1 Enable Normal, No ShotID, One Sequence, or Src Grade.

Normal Individually graded shots are rendered to a folder called "Normal", in the Renders Home location and the UID is included in the filename.

Default render structure for Normal mode (ignore any line breaks):

```
[RENDERS_HOME]/Normal/[SCENE]/[GRADE_NAME]/[REEL_NAME]/
[RESOLUTION]/[UID]_[ORIGINAL_FILENAME]_[GRADE_NAME]
```

Black frames are rendered out to a separate folder in the Renders Home location.

No ShotID Individually graded shots are rendered to a folder called "NoShotID", in the Renders Home location.

Default render structure for No ShotID (ignore any line breaks):

```
[RENDERS_HOME]/NoShotID/[SCENE]/[GRADE_NAME]/[REEL_NAME]/
[RESOLUTION]/[ORIGINAL_FILENAME]_[GRADE_NAME]
```

Black frames are rendered to a separate folder in the Renders Home location.

One Sequence Individually graded shots are rendered as a single sequence in the Renders Home location. Black frames are rendered to the same single sequence in the same scene folder.

Default render structure for One seq mode (ignore any line breaks):

```
[RENDERS_HOME]/OneSequence/[SCENE]/[GRADE_NAME]/[RESOLUTION]
```

Src Grade

Individually graded shots are rendered to a folder called "SrcGrade" in the Renders Home location and UID is included in the filename.

When used locally, this render mode bypasses dissolve transitions and retime parameters. Dissolves become hard cuts and retimes are rendered with a default value of 100%.

This render mode is also used in Interoperability workflows. When rendering with a Wiretap path, Src Grade will retain transition effects and soft effects.

Black frames are rendered to a separate folder in the Renders Home location.

Rendering the Multi-Layer Timeline

Lustre renders out what is displayed in the multi-layer timeline and Player, according to:

- Top vertical priority
- The presence or absence of priority shots and muted layers
- The Solo status

If Solo mode is disabled, Lustre renders the topmost layer by default. If there are gaps, Lustre takes into account top vertical priority behaviour and render out all the shots from the lower layers that are visible through the transparent gaps that are above.

If you do not want a shot in a lower layer to be rendered that is located beneath a gap, you can replace the gap with a virtual black clip. See Adding a Virtual Black Clip using the Storyboard (page 1931) and Adding Black Media to the Timeline (page 1940).

If you wish to render out the shots on a specific layer, you either must Solo the active layer, or Mute the unwanted layers. If there are gaps in this layer, the gaps are skipped by the rendering process .

If you wish to render out a shot that normally would not be rendered because top vertical priority renders out a shot on a different layer, you can use the shot priority function to prioritize this shot.

For more information, refer to the following links:

- Top Vertical Priority (page 1958)
- Muted Layers (page 1959)
- Shot Priority (page 1960)
- Solo Mode (page 1963)

Rendering the Multi-Layer Timeline with Normal

When the Normal button is selected, Solo mode is disabled, and there are no muted layers, Lustre interprets gaps as being transparent and flattens layers in a top-down manner. Lustre renders out what is visible in the multi-layer timeline in accordance with top vertical priority. The individually graded shots with frames that are visible are rendered and saved to their own folder in the Renders Home location. If there are no shots beneath the gap, the rendering process skips the gap.

You can render one specified layer at a time by muting unwanted layers or soloing a desired layer. Lustre renders individually graded shots with frames that are visible as individual shots and saves them to their own UID directories in the Renders Home location.

Rendering the Multi-Layer Timeline with No ShotID

Renders the same way as Normal mode. However, when rendering with No ShotID, you can render one layer only for each grade. Consequently, when rendering out L1 first, then rendering out L2 within the same grade, the L2 render overwrites the previous L1 render files.

To render out L2 without overwriting the previous L1 render, you can do one of the following:

- Save a new grade (i.e., grd02).
- Use Normal render mode.
- Add a UID token in the filename. See Render Naming Settings

Rendering the Multi-Layer Timeline with One Sequence

When the One Sequence button is selected, Solo mode is disabled, and there are no muted layers, by default Lustre sees the gaps as transparent and flattens the layers in a top-down manner. Lustre renders out what is visible in the multi-layer timeline in accordance with top vertical priority. Lustre renders the individually

graded shots with frames that are visible to the scene folder as a single sequence in the Renders Home location. If there are no shots beneath the gap, the gap is rendered as black to the same single sequence in the same scene folder.

When the One Sequence button is selected, you can render one specified layer at a time by soloing the desired layer or muting the unwanted layers. Lustre renders the individually graded shots with frames that are visible to the scene folder as a single sequence in the Renders Home location. Gaps are rendered as black to the same single sequence in the same scene folder.

NOTE When rendering with One Sequence, you can render one layer only for each grade. That is, if you render out L1 first, and then within the same grade renders out L2, the L2 render will overwrite the previous L1 render files. To render out L2 without overwriting the previous L1 render, you must save a new grade (grd02).

Rendering the Multi-Layer Timeline with Src Grade

Src Grade works just like Normal render in that it renders each shot to its own folder under the SrcGrade directory and the rendering process will skip gaps if there are no shots beneath them. When this option is enabled, the rendering process bypasses any dissolve transitions and retime parameters. In other words, dissolves become hard cuts and retimes are rendered with a default of 100% value.

Rendering Dust Metadata

You can render shots with or without dust metadata. This is a workflow decision. You can render dust metadata either from the Render Local menu or from the Image Dust menu. See Rendering the Result (page 2007).

To render shots with dust metadata, you must first use the dust removal tools to remove dust artefacts. See Removing Dust (page 1998).

To render shots with or without dust metadata:

1. In the Render Local menu, do one of the following:
   - To render shots without dust metadata, disable Dust.
   - To render shots with dust metadata, enable Dust.

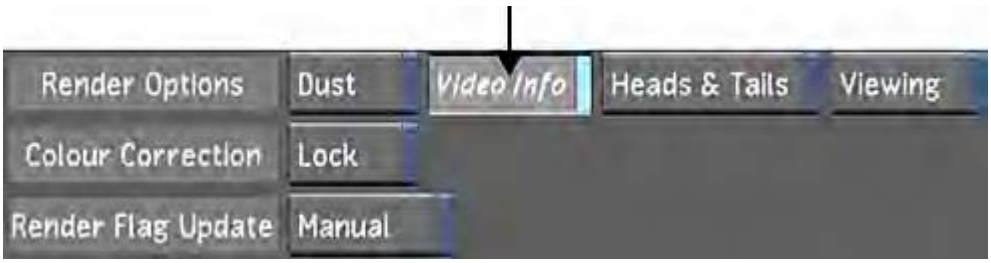


Burning Timecode into the Render

You have the option of burning the timecode into the render files. For example, burn a timecode into the black border of a letterboxed shot to identify the location of the shot.

To set timecode burn-in:

1. In the Render Local menu, enable Video Info.



Rendering with the Viewing LUT

For linear work, you can render shots to 8-bit BMP files using the currently loaded viewing LUT with the Viewing option. Use this option when you want to render the image displayed in the monitor.

When this option is enabled, the output consists of exactly what you see on the monitor, namely, the image resulting from the applied monitor calibration setting, viewing LUT, and colour space settings applied in the Render Output menu.

When this option is disabled, the currently applied viewing LUT is not taken into account in the output image, and the files are rendered to the file format specified in the Render Output menu rather than to 8-bit BMP files.

For information on monitor calibration, see Calibrating a Monitor (page 1849). For information on output colour space settings, see Setting Colour Space Options (page 2226).

NOTE You can also render shots to DPX files with a viewing LUT applied. This is performed in the Render Output menu. See Setting Colour Space Options (page 2226).

To set rendering with the viewing LUT:

1 In the Render Local menu, enable Viewing.



Rendering Head and Tail Frames

When rendering shots or timelines, you can render the grade on the head and tail frames. This can be useful if, for example, the graded shot needs to be changed to include a longer dissolve and requires additional frames.

NOTE Rendering the head and tail frames is only applicable to the Normal, No ShotID, and Src Grade rendering options.

To render shots with head and tail frames:

1 In the Render Local menu, enable Heads & Tails.



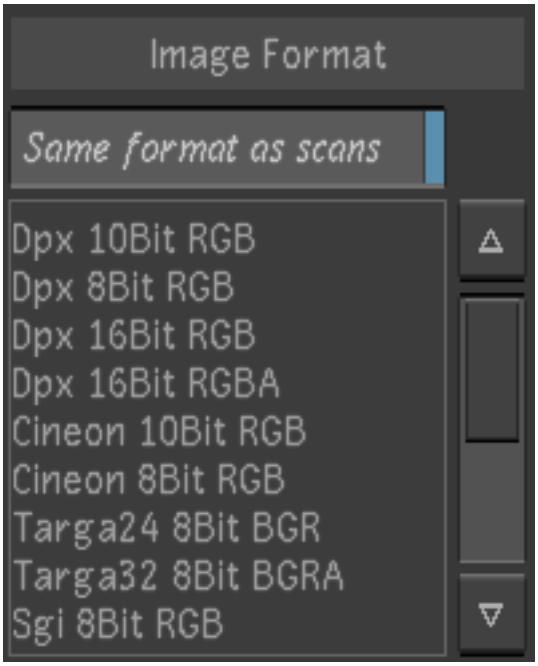
Specifying the Output Format of the Render Files

You can render your shots to the same file format as the original footage or select a different one. By default, shots are rendered to the same format as the original footage. Currently supported formats are DPX and Cineon at 10 and 8 bits per channel, TIFF at 8 and 16 bits per channel, TARGA®, SGI, and BMP at 8 bits per channel and OpenEXR . When OpenEXR is set as the output format, the resulting files are encoded as single channel RGB OpenEXR 16-bit half float.

NOTE The ability to render OpenEXR files is only available on the Linux version of Lustre. Also, it is not possible to render OpenEXR files through BrowseD.

To specify the output format:

1. In the Main menu, click Render and then click Output.
   The Render Output menu appears.
2. Do one of the following:
   - To render to the same format as the originals, enable Same format as scans.
   - To render to another format, disable Same format as scans and select an output format from the list.



NOTE If you applied a colour transform that converts between integer and floating-point data or vice versa in the Transcode menu or elsewhere, you should disable Same format as scans and specify the output format manually.

NOTE If you output to a floating-point format, the float conversion LUT is applied to the output buffer for display. See Calibration Settings (page 1741).

Setting Colour Space Options

You can convert the colour space of the output shots from Log to Lin or vice versa. Whether you are outputting in Log or Lin colour space, you have several options for setting a LUT or colour transform used on output. You can:

- Use the default colour mapping. The default mapping differs according to the grading architecture used on the project (as set in the Grade menu).

Grading Architecture	Output Colour Space	Colour Mapping Applied
Log	Log	None
Log	Lin	Standard log-to-lin (95-685)
Lin	Lin	None
Lin	Log	Standard lin-to-log (95-885)

- Select an output LUT or colour transform.

The output colour space and colour mapping affect the image you see on the monitor. These settings are saved in the output buffer, which is applied to the output on the monitor before the viewing LUT and *monitor.calib* correction (if used) are applied.

Specifying the Output Colour Space

Using the Default Colour Mapping

1. Access the Render / Output menu.
2. From the Colour Space box, select either Logarithmic or Linear.
   You are set up to use the default colour mapping.

Using a LUT or Colour Transform

1. Access the Render / Output menu.
2. From the LUT Type box, select 1D Lut, 3D Lut or Colour Transform.
3. Click Enable.
4. Select an item from the list, or click the Browse (...) button to locate and select a different LUT or colour transform.
5. Click the Enable LUT button.
   The selected LUT will be applied to your render.

To create a custom colour transform, see Creating a Custom Colour Transform (page 1885).

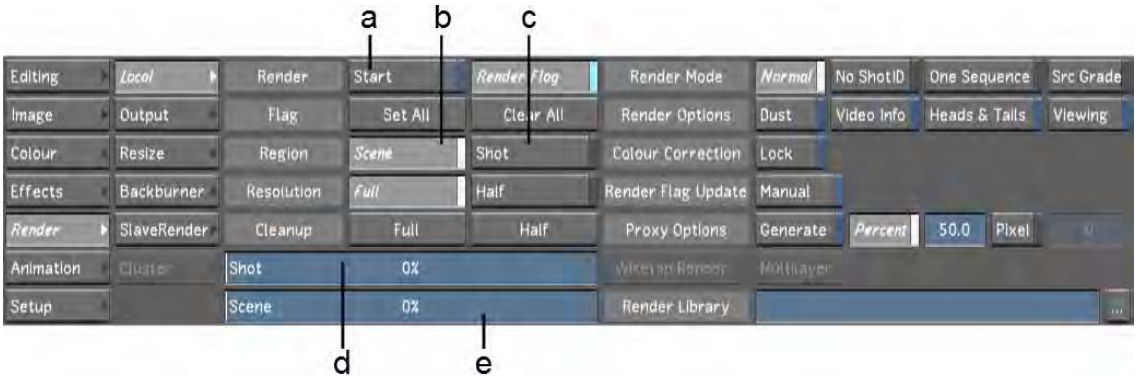
NOTE If you apply a colour transform that has floating-point inputs, the inverse float-conversion LUT is applied automatically first. See Calibration Settings (page 1741).

Rendering the Selected Region

Once you have set all the rendering options, you can render flagged shots within a scene region using the Render Local menu, or just render one shot at a time by enabling the shot region.

To render the selected region:

1 Access the Render Local menu.



(a) Start button (b) Region Scene button (c) Region Shot button (d) Shot progress bar (e) Scene progress bar

2 Click Start.

The render starts and its progress is shown in the Shot and Scene progress bars. The Start button label changes to Stop. When the Shot progress bar reaches 100% and the Render button is no longer highlighted (it again reads “Start”), the render is complete.

3 If needed, you can abort the process by clicking Stop.

Generating and Viewing Proxies

When working with high-resolution material in Lustre installations that do not include Incinerator, the speed of interaction can decrease substantially as grading effects are added, compromising real-time playback. For faster interaction with the application, you can generate lower resolution proxies of your footage and perform the colour grading on the proxies. After the grading is complete, you can easily render the colour grading data to the full-resolution version.

If you plan on using proxies on a project, it is recommended that you generate them at the beginning of the project so that you will have them available when needed. Proxies are stored in a sub-directory of the scans directory, at the same directory level as that of the original footage.

You can specify the proxy generation filter by selecting the one you need in the Rendering menu. See Rendering Settings (page 1743).

An alternative to generating proxies in Lustre is to create them using another means outside the application, and place them in a directory with a name composed of the resolution (for example, *1024x778*). See Recommended Directory Structure for Projects (page 1734).

NOTE Proxy generation differs from rendering a half-resolution render using the Half Resolution button in that grading and other effects data is not applied to proxies as it is when rendering the images.

To generate proxies:

1 Specify the proxy generation filter in the Rendering menu for the project configuration.

To override the filter specified in the configuration file, enter the appropriate proxy generation filter in the Command Line Renderer. See Command Line Renderer (page 2237).

2 Load the scene and grade containing the cut for which you want to generate proxies. See Selecting Scenes (page 1770) and Loading Grades for a Cut (page 1778).

3 Navigate to the first shot in the cut for which you need proxies. You do not have to go to the beginning of the shot, as generation automatically starts at the first frame of the current shot.

4 In the Render Local menu, set the proxy size.

Enable:	To enter:
Percent	The size of the proxy as a percentage in the Percent field.
Pixel	The width of the proxy in pixels in the Pixel field.



(a) Percent button (b) Percent field (c) Pixel button (d) Pixel field

Modifying a value in one field automatically updates the value in the other field.

NOTE No other options in the Render Local menu affect generated proxies.

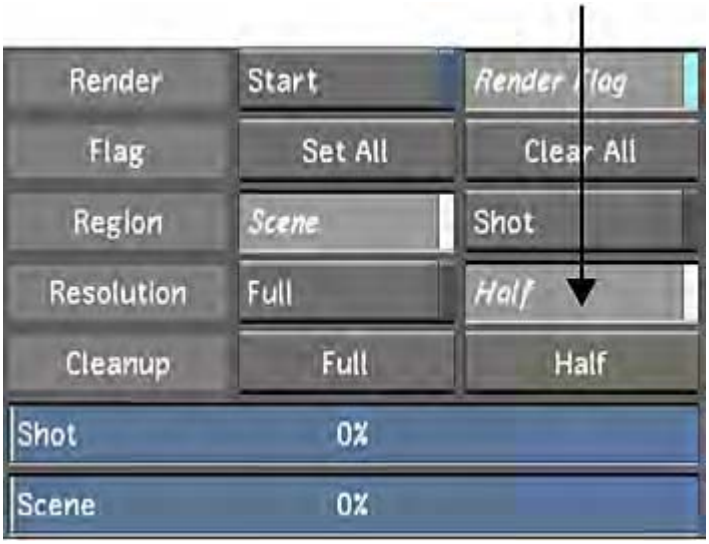
5 Click Generate.

The Generate button changes to a Stop button, and proxy generation starts at the first frame of the current shot. It continues through to the end of the cut unless you abort the process by clicking the Stop button.

To view proxies:

1 Do one of the following:

- Enable the Half Resolution button in the Render Local menu or in the Grade menu.



- Set the Full/Half Resolution button to 1/2.



- Press F9.

NOTE An ungenerated proxy appears as a red X with a black background.

Generating Proxies for Matte Media in Multi-Channel Clips and Matte Containers

It is now possible to generate proxy media for matte clips inside a Multi-Channel clip and Matte Container. When generating proxies, only the matte defined in the Multi-Channel clip & Matte Container (the S track) is generated. If you enable the Render/Proxy Options/All option, all mattes located inside the Multi-Channel clip/Matte Container are generated, including any shots located on any timeline layers of the primary video track.

To automatically generate matte proxies for the secondary track (S):

1. Access the Render/Local menu.
2. From the Proxy options section, enable Generate.

To generate proxies for all mattes

1. Access the Render/Local menu.
2. From the Proxy options section, enable Generate and All.

Locking Colour Grading

You can lock the colour grading on the current shot so that no colour changes can be made to the shot. This is useful to avoid accidental modification once the grading is complete. For example, you might want to lock the colour grading after it has received final client approval.

NOTE Sparks plugin effects are also locked, but dust removal is not.

To lock colour grading:

1. Navigate to the shot for which you want to lock the colour grading.
2. Do one of the following:
   - In the Render Local menu, enable the Colour Correction Lock button.



   - Enable the Locked flag in the Flags section by right-clicking on the Locked flag. A filled green circle indicates that the flag is enabled. See Flagging Shots (page 1808).

   The word “Locked” appears in the Printer Light fields and the shot can no longer be modified in any of the Colour menus until either the Lock button or Lock flag has been disabled.

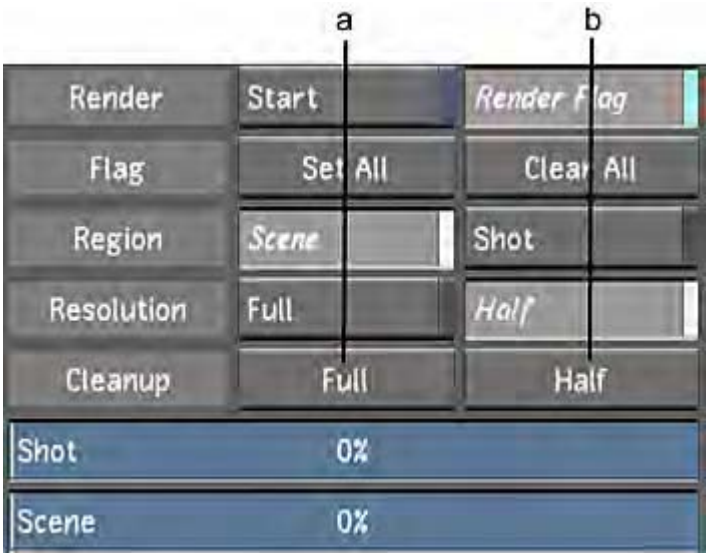
Cleaning Up Render Files

You can remove unneeded render files. Because renders are created separately for each grade (that is, grd01 to grd99), you may want to delete renders for grades that are not used anymore to free up storage.

To remove unneeded render files:

1. Load the grade that has associated unwanted render files. See Loading Grades for a Cut (page 1778).
2. In the Main menu, click Render, and then click Local to display the Render Local menu.
3. Do one of the following.

Click:	To:
Cleanup Full and confirm	Remove all full-resolution render files associated with the grade.
Cleanup Half and confirm	Remove all half-resolution files associated with the grade.



(a) Cleanup Full button (b) Cleanup Half button

Rendering Shots as You Work

You can render shots as you work on them using a slave rendering machine. As you move from shot to shot in the timeline, the previous shot is rendered using the Slave Renderer, so your work is not interrupted during rendering. If several shots are changed before Lustre has had time to render the last one, the changed shots are sent to a job queue.

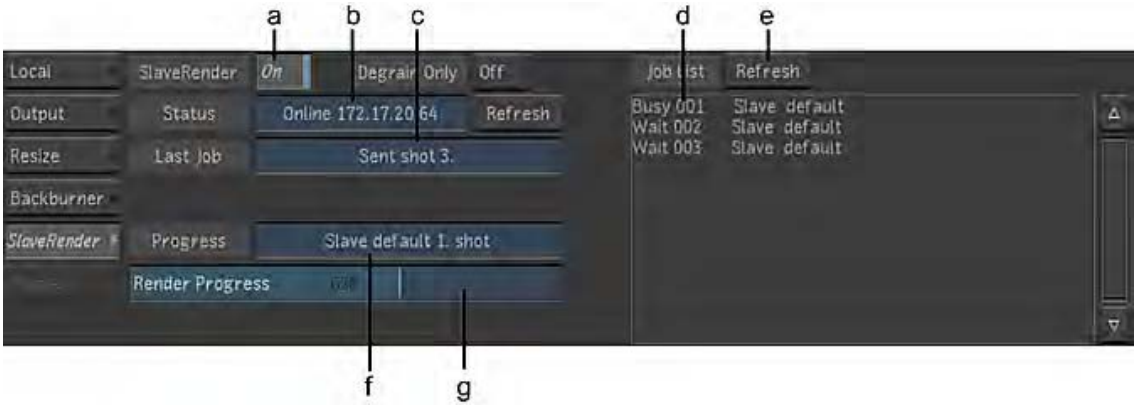
NOTE When rendering a multi-layer timeline, Slave renders the shots that are visible from a flattened timeline, or the shots from the active layer in the multi-layer timeline, depending on the mode you are working in. For Solo mode, see Multi-Layer Timeline (page 1932) and for Mute mode, see Muted Layers (page 1959).

Files rendered with the Slave Renderer are stored in the same directories as those used for renders created using the Render Local menu. Directories are specified when you first create the project. To view the renders, use P (Print) view mode. See Setting the View Mode (page 1973).

NOTE For information on setting up the Slave Renderer, refer to the *Autodesk Lustre Software Installation Guide* for your platform.

To render shots with the Slave Renderer:

1. In the *init.config* file, set the IP address for the slave render machine. To render at full resolution, enable Renders Full Res in the Network Rendering page of the project configuration. See Network Rendering Settings (page 1754).
2. Start Lustre.
3. Access the Render SlaveRender menu.



(a) SlaveRender On/Off button (b) Status field (c) Last Job field (d) Job List (e) Job List Refresh button (f) Progress field (g) Render progress bar

In the Status field, the status and IP address of the slave render machine are displayed.

TIP Click Refresh to update the Status field.

4 Turn on slave rendering. Set the SlaveRender On/Off button to On.

The letters BR in the upper-left corner of the screen indicate that slave rendering is activated.



5 In the Render Local menu, specify the region to render. See Specifying the Shots to Render (page 2216).

6 When you want to render a shot with the Slave Renderer, modify a parameter and then move to the next shot.

7 To view the render status, display the SlaveRender menu.

The last job sent to the Slave Renderer is displayed in the Last Job field. The last job sent is also the shot currently being rendered, and its name is displayed in the Progress field. The shot's render progress is indicated by the Render progress bar.

TIP The Render progress bar shows the shot's render progress as a percentage and updates every second. You can change the update interval using the Refresh in Seconds slider in the Network Rendering tab of the project configuration. See Network Rendering Settings (page 1754).

8 When you have finished, you can turn off rendering with the Slave Renderer by setting the SlaveRender On/Off button to Off.

Any jobs remaining in the job queue are cancelled.

To play shots rendered by the Slave Renderer:

1 Navigate to a shot that has been rendered.

2 Enable P (Print) view mode.



3 Use the playback controls to play the shot.

NOTE If you have rendered files on the filesystem but Lustre does not "see" them, you can force a refresh by pressing `Shift` while enabling P in the user interface. The force refresh function is especially useful when background rendering processes are involved.

Rendering with Burn for Lustre

Burn for Lustre allows you to render shots using Linux servers set up on the network, speeding up the rendering process and freeing up the Lustre stations. On a 2K film project, you might, for example, want to render half-resolution versions using the local machine and render 2K final versions using Burn for Lustre, as this takes much longer. Or, when you are satisfied with the grade of a reel, use the render farm to render 1K or 2K versions of it.

NOTE When rendering a multi-layer timeline, Burn only renders the flattened timeline. It does not take into account the Solo (see Multi-Layer Timeline (page 1932)) or Mute (see Muted Layers (page 1959)) status, since these modes are not saved in the grade.

Files rendered using Burn for Lustre are stored in the same directories as those used for renders created using the Render Local menu. Render directories are set up in the Render Backburner menu. See the *Autodesk Lustre Software Installation Guide* for your platform.

To view and manage jobs sent to the render farm, you use Autodesk Backburner™ Manager and Backburner Monitor.

Remote Rendering with Burn and Wiretap

Burn for Lustre allows you to render footage stored in the local storage array, or footage imported via Wiretap, to a destination specified in the Burn Render Fulls Home. The destination could be the local storage array, shared network storage, or the Stone filesystem via Wiretap.

NOTE If rendering footage imported via Wiretap, the footage must be located on the Stone filesystem.

Remote Rendering to the Local Storage Array or Shared Network Storage with BrowseD

Enter the Wiretap path in the Renders Full Home field in the Render Backburner menu. See Submitting a Remote Render Job (page 2234).

Example:

BURN_PROJECT_HOME 192.23.200.76:/L:/lustre_project

BURN_RENDERS_FULL_HOME 192.23.200.76:/L:/renders/full

Command Line Render

./render -s `<scene_name>` **-g** `<grd_name>` **-r** `<output_resolution>` **-i -1**

192.23.200.76:/L:/lustre_project -2 192.9.200.19@wt:/stonefs/project/library -3

192.23.200.76:/L:/renders/full

NOTE Use the -i option to render, taking trim information into account.

For more information, see Command Line Renderer (page 2237).

Starting Backburner Manager

Before you can start a network render, you must start Backburner Manager on a Windows or Linux workstation designated as the Manager, and start Backburner server on each rendering server.

supports Autodesk Backburner. In Backburner, the Backburner Web Monitor has all the features of the Windows-based Backburner Monitor. The Backburner Web Monitor runs in a Web browser from any workstation on the network. You can use it to view and control jobs currently being processed. Jobs in the Distributed Queueing System can be stopped, restarted, reordered, archived, or removed. You can monitor the overall health of the Distributed Queueing System and identify any Render Nodes that are not working. See the *Autodesk Backburner User Guide*.

For details on starting and configuring Backburner Manager, see the Backburner Installation Guide.

Submitting a Remote Render Job

You can render one shot, several selected shots, or all shots in a scene using Burn for Lustre. Jobs are submitted from the Render Backburner menu, where you can also monitor job progress.

To render a selected region using Burn for Lustre:

1. Load the cut containing the shots you want to render. See Managing Cuts (page 1773).
2. Apply the required colour grading changes, and then save the grade.
3. If you do not wish to render out the entire timeline, flag the shots you wish to render. See Flagging Shots (page 1808).
4. Click Render and then click Backburner.

   The Render Backburner menu appears.



(a) Burn button (b) Task Size button (c) Frames field (d) Job list

5. If needed, change the size of the task (in number of frames) sent to individual Linux systems by enabling the Task Size button and then changing the value in the Task Size field.

   NOTE It is recommended that you save the grade file before proceeding to the next step. If not, changes made in the Render menu might not be submitted to the render farm. Press S on the keyboard to save your changes.

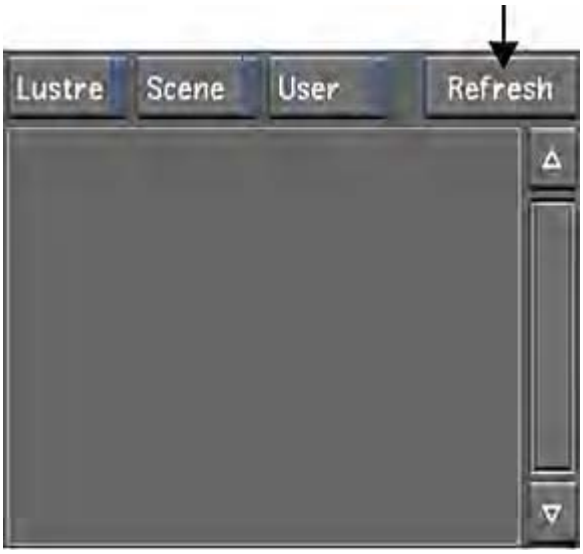
6. Click Burn.

   The job is submitted to Backburner, which then queues and distributes it to the render farm. The last successfully submitted job appears in the Last Job Name field. If the job submission was unsuccessful and nothing appears in the Last Job Name field, check for errors in the Lustre console. Otherwise, check for errors in the Backburner Monitor.

   NOTE The total number of frames in the cut is displayed in the Frames field, for reference.

To monitor job progress:

1 In the Render Backburner menu, click Refresh.



2 All submitted jobs appear in the Job list. This is the same list of jobs shown in the upper-left panel of the Backburner Monitor. See Monitoring the Render Tasks with Backburner Monitor (page 2235).

3 To view a subset of the jobs, apply a filter to the job list by enabling one or more of the filter buttons above the job panel.

Enable:	To view:
Lustre	Only jobs submitted from Lustre (as opposed to other Autodesk applications such as Flame or Combustion).
Scene	Only jobs for the currently loaded scene.
User	Only jobs submitted by the current user.

Monitoring the Render Tasks with Backburner Monitor

During an active render session, you can use Backburner Monitor on any Windows workstation on the network to view and edit jobs that are in process or in the job queue.

To start Backburner Monitor:

1 On the workstation designated as the render monitor, choose Start | Programs | Autodesk | Backburner | monitor.

The Monitor window appears.

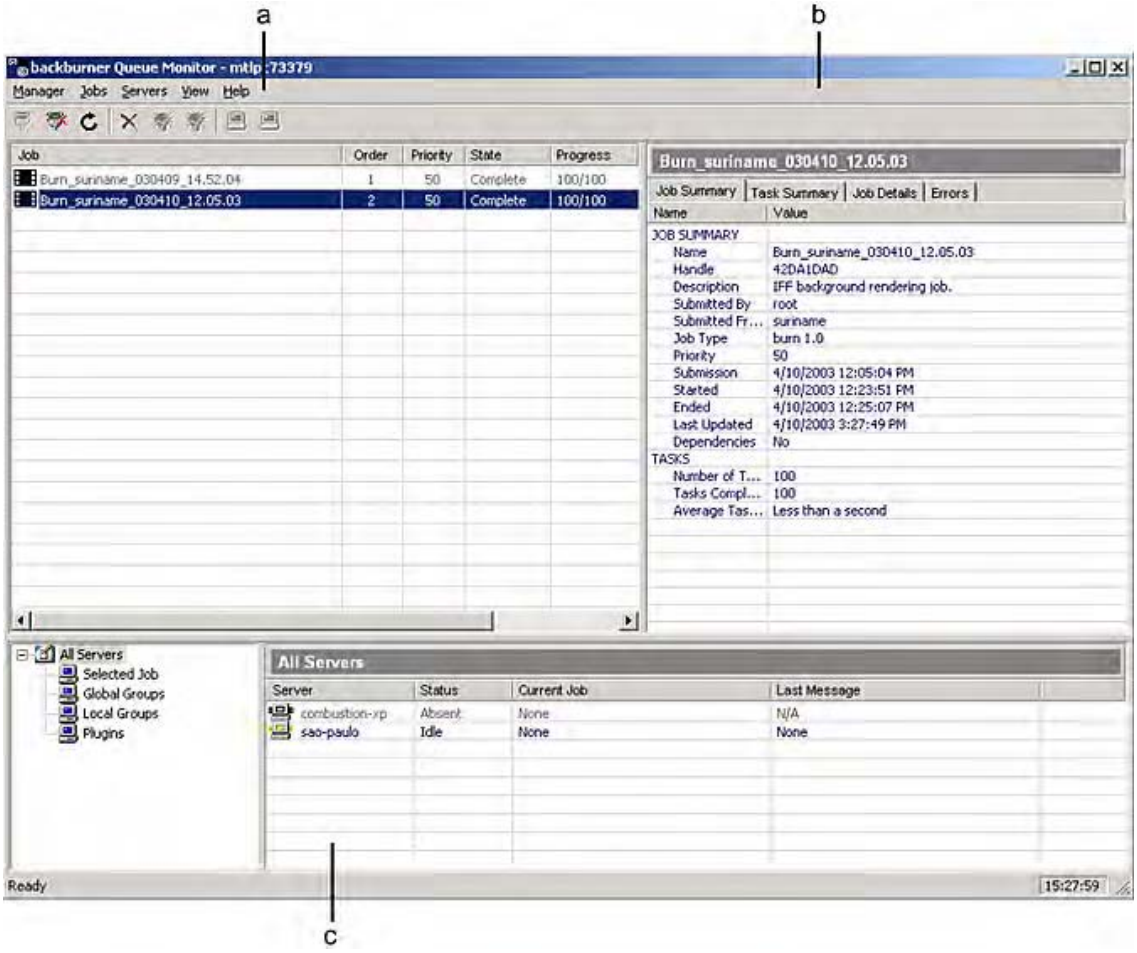
2 If no servers or data appear in the Backburner Queue Monitor, the Backburner Manager was not found with the current settings. Choose Manager | Connect to connect to the Backburner Manager.

The Connect to Manager dialog appears.



3 Enter localhost if you are using Backburner Monitor on the same workstation as Backburner Manager. Otherwise, enter the IP address of the workstation on which Backburner Manager is running.

When Backburner Manager is found, its name appears in the title bar and the list of servers is updated.



(a) Job list (b) Job panel (c) List of servers

To view and edit jobs:

1 In the Job list, select a job.

The columns in the Job list include Order, Priority, State, Progress, and Owner. The Job Summary, Task Summary, and Job Details panels are updated to reflect the selected job.

2 In the Job list, right-click a selected job to choose from a list of operations. For example, activate, suspend, delete, or restart the current job.

Monitoring Results Using rdesktop

You can access Backburner Monitor by running a Windows session on the Linux system using *rdesktop*—a remote desktop protocol client. The *rdesktop* application is an open source client for Windows NT® Terminal Server and Windows 2000 Terminal services. The *rdesktop* application displays your NT desktop on the Linux platform. For more information, see *GUID-690963B8-74BD-4190-BB0A-88696222C2ED*.

Command Line Renderer

The Command Line Renderer allows you to render shots by entering a command in a Linux shell instead of using the Render Backburner menu.

NOTE Use the *bbserver.log* file describing the Command Line Renderer output to view error messages and other pertinent information.

Run the Command Line Renderer on the machine you want to render on. Files rendered using the Command Line Renderer are stored in the same directories as those used for renders created using the Render Local menu. Render directories are set up in the Project menu.

The Project Home directory must be specified when you use the Command Line Renderer. The Scans home and Renders home directories must also be specified if they are not sub-directories of the Project Home directory. You can specify these directories in two ways. You can either use the -1, -2, and -3 parameters in the command line, or you can set them as environment variables on the Linux server.

To use the Command Line Renderer, Burn for Lustre must be installed and licensed on each machine (node). For information, refer to the *Autodesk Lustre Software Installation Guide* for your platform.

Command Line Renderer Syntax

The Command Line Renderer uses the following syntax:

./render -s `<scene_name>` **-g** `<grade_name>` **-o** `<output_file_format>` **-r** `<output_resolution>`

NOTE This syntax example shows required parameters and modifiers when environment variables for project directory locations have been set on the local machine. For a complete list of parameters, refer to the following table.

Par.	Modifier	Description	Req'd	Comments
`-s`	`<scene_name>`	The name of the scene containing the grade and associated cut you want to render.	Yes	
`-g`	`<grade_name>`	The name of the grade and associated cut you want to render.	Yes	
`-o`	`<output_file_format>`	The file format of the render files. Valid modifiers: cin, dpx, tga, sgi, tif or bmp.	Yes	

Par.	Modifier	Description	Req'd	Comments
-r	<output_ resolution>	The resolution of the render files. Valid modifiers: full, half.	Yes	
-l	<video_format>	Renders to a specified video format. Valid modifiers: HD, PAL, or NTSC.	No	
-1	<pro- ject_home_folder>	The path and directory name of the Project Home directory, including the drive letter. Example: G:\Pro- jectAB	No	Not required if set as envir- onment variables on the local machine. See Project Configuration (page 1730).
-2	<foot- age_home_folder>	The path and directory name of the original footage directory, including the drive letter.	No	Not required if set as an environment variable on the local machine.
-3	<renders_home_folder>	The path and directory name of the render files directory, including the drive letter.	No	Not required if set as an environment variable on the local machine.
-n	<number_shots>	The number of shots, starting from the first shot in the cut, to be rendered.	No	Use only when rendering to the Slave Renderer.
-f	<first_frame last_frame>	Renders a range of frames within the set of shots defined for render- ing. Example: -f 120 340	No	Can be used to specify a new range when restarting a cancelled render.
-a	<start_fr_no end_fr_no>	Renders a range of frames in the timeline, between the Start Frame number and End Frame number. Example: -a 140 480	No	Do not use this parameter with other parameters that specify a subset of shots.
-i	None	Renders shots taking trim informa- tion into account.	No	
-e	<preset>	Name of output colour space pre- set, if used.	No	
-t	<number_of _threads>	Specifies the number of threads, or processors, used for processing. You can opt to use only one of two processors. Example: -t 1	No	On Linux machines, you must set this parameter to -t 2.
-w	None	Renders wedges to the same place.	No	Same as No Wedge option. See Specifying the Destina- tion for Local Render Files (page 2220).

Par.	Modifier	Description	Req'd	Comments
-q	None	Renders to one sequence.	No	Same as One Sequence option. If neither -w nor -q are specified, Normal is used. This parameter should be used with -i.
-p	<part_of_ frame>	Renders a fraction of the shots in the cut. Express the modifier as a fraction, for example, <1/4> would render the first 25% of the shots.	No	You can use this parameter to divide the job between several machines.
-u	None	Renders dust removal data.	No	
-k	None	Renders with timecode burned in.	No	
-L	None	Render using the Normal filter for reposition.	No	See Rendering Settings (page 1743).
-M	None	Render using the Mitchell filter for reposition. There are three parameters (s = size; B = blur; C = ringing).	No	See Rendering Settings (page 1743).
-T	None	Render using the CatmullRom filter for reposition.	No	See Rendering Settings (page 1743).
-O	None	Render using the new type Lanczos3lobe filter for reposition.	No	See Rendering Settings (page 1743).
-d	None	Debug level.	No	
-h	None	Print help to command prompt window.	No	Use this parameter by itself. Example: render -h.

Example of Use

render -s alabama -g grd21 -o dpx -r half -i -q -1 g:/project1 -n 1

Where **alabama** is the scene name, **grd21** is the grade name, **dpx** is the output format, and **g:/project1** is the Project Home directory as mapped from the local (rendering) machine.

To render one or more shots with the Command Line Renderer in Windows:

1. Open a Command Prompt window on the machine on which you want to render the shots by choosing Start | Programs | Accessories | Command Prompt.
2. Go to the Lustre application directory by typing:

   cd C:\Program Files\Autodesk\Lustre <version_number>

   where C is the local drive.
3. Enter the render command using the syntax provided in the previous table.

To render one or more shots with the Command Line Renderer in Linux:

1. Open a Linux shell.
2. Go to the Lustre application directory by typing:
   cd /usr/autodesk/lustre_<version_number>
3. Enter the render command using the syntax provided in the previous table.

Rendering Grades Created in Releases Prior to Lustre 2011 Extension 1

Due to the enhancements in the GPU, we do not recommend using Lustre to render grades created in Lustre 2011 or earlier if you are looking for pixel accuracy rendering.

To be able to render grades created before Lustre 2011 Extension 1 using the older HLS/Blur algorithm:

1. In a text editor, open the init.config configuration file.
2. Add the following keyword in the Miscellaneous section:

```
<GPUHLSKeyerBackwardCompatibility state= "On" />
```

3. Restart Lustre.
4. Load and render the grade.

NOTE

This keyword must be enabled in any init.config configuration file used by Lustre or Lustre render services (slave renderer, burn, etc). Also, verify that the keyword is disabled from init.config configuration files before creating new content in Lustre 2011 Extension 1. If not, all newly created grades will use the older HLS/Blur algorithm.

To disable the keyword:

1. Do one of the following:
   - Remove the keyword from the init.config configuration file
   - Set the keyword to:

```
<GPUHLSKeyerBackwardCompatibility state= "Off" />
```

NOTE

When using Incinerator, this keyword does not apply. Incinerator uses CPU rendering only and does not use the GPU to accelerate the keyers and Blur.

Media Export (Linux Only)

About Background Media Export

Lustre 2013 supports a media encoding service called Media Converter. The Media Converter allows Lustre to offload encoding tasks to Wiretap Gateway servers (Linux and Mac OS X). This means that you can continue working on your project while media files are being generated in the background, even if you are

using your local Wiretap Gateway server. It offers the ability to encode streaming formats such as Quicktime, MFX, graphic files like Jpeg, DPX and Tiff. and audio files like AIFF, Wave and mp3. It also adds support for formats and codecs only available on specific operating systems (Apple ProRes) and leverages your networking infrastructure (InfiniBand, 10-Gig, etc). The tasks are managed by Backburner Manager, the Autodesk queue management system. This means that facilities can centralize the management of these tasks without disturbing the creative suite and use a web browser to monitor and edit the tasks.

The Media Export workflow in Lustre is based on Export Versions: export configurations that are linked to a grade. Consequently, all the exports you create can be recalled for future use. Numerous options are available to suit multiple file delivery workflows.

You can define the Wiretap Gateway server used for background media export. It can be local or remote. Unless otherwise specified, the default Wiretap Gateway Server is set to "localhost".

To set up a stand-alone Wiretap Gateway server for media encoding, see Stand-Alone Wiretap Gateway Server Configuration for Media Encoding (page 2257).

Setting the Render Location during Project Creation

Define the location of the exported media files, in the Rendering tab, when you create the project or after the project is created in the Project Settings. The path can be local or remote.

- To export locally, set the path to:

/[PROJECT_HOME]

- To export to a remote location, the path must be in the following format:

Remote_Host_Name@wtg:/[PROJECT_HOME]

For example: Upton@wtg:/[PROJECT_HOME]

The Export Panel

The Export Panel is a sub-menu of the Render menu. It is divided into three sub-menus:

Sub-menu:	Description:
Format Options	Use to define file format options used for export, media location and the Wiretap Gateway Server used for the export tasks.
Resize	Use to define the output resolution and the Resize options of the exported media files.
LUT Options	Use to apply a LUT to the media files to be exported.

Settings Common to the Three Export Sub-Menus

The first column of options, along with the Export Versions and Export Templates boxes, are common to all Export sub-menus.



Media Origin: Defines the type of media used for export. The options are:

- **Source Grade Renders:** Source Grade rendered files are used to create media files on a shot basis.

- **One Sequence Renders:** One Sequence rendered files are used to create one media file for the entire grade.
- **Original Media:** Uses the original media files for the export. No grades are applied.

NOTE Because the Export service requires media files to be saved to disk, you must render your grades in Lustre prior to exporting. If you set off an export prior to rendering you grades, Lustre automatically starts the rendering process in the mode corresponding to your export options (One Sequence or Source Grade) and exports the file once rendering is complete.

Region: Enable Scene to export the entire timeline or enable Shot to export only the current shot. Available when Media Origin is set to Source Grade rendered or Original Media.

Media Selector: Use to define attributes of the exported media. Options are:
- **Video Only:** Only video stream is exported.
- **Video & Audio:** Video and audio stream is exported.
- **Audio Only:** Only audio stream is exported.

Export All Versions: If you created multiple Export versions, enabling this option will add them all as jobs in the queue.

See Export Versions (page 2251).

Export: Click to start the Export tasks.

Export Version: Use to create, update, load and delete Export Versions.

You can create multiple Export Versions, with different export settings for the same grade. Export Versions contain all the data required to define an export job and are stored in the Library folder, alongside the grade metadata file. Export Versions are named after the grade they are associated to:
- grade01_export01, grade01_export02, etc.

You can add a comment to an Export Version, which appears alongside the export name:
- Export01 Final Version

See Export Versions (page 2251).

File Naming: Use tokens to define the filename of the exported file(s). The default value is [GRADE_NAME] for One Sequence Renders and [ORIGINAL_FILENAME]_[GRADE_NAME] for Source Grade Renders.

To create a folder, type a token in the file name field followed by "/". This creates a folder in the destination location.

The following tokens can be used:
- [PROJECT_NAME]: The name of the current project.
- [PROJECT_HOME]: The home folder of the project.
- [SCENE]: The current scene.
- [USER]: The current user settings.
- [GRADE_NAME]: grd01 is the grade name.
- [GRADE_NOTE]: Text data entered by user and appearing to the right of the grade in the grade list.
- [CUT]: The name of the cut.
- [UID]: The shot-based UID.
- [RESOLUTION]: The resolution of the exported media.
- [REEL_NAME]: The reel/tape name of the source clip.
- [STEREO_LAYER]: Left or Right.

- [ORIGINAL_FILENAME]: The original file name of the source clip (without the file extension).
- [DATE]: Current date in YY_MM_DD format (i.e. 2011_11_29).
- [TIME]: Current job starting time in HH_MM_SS format (i.e. 22_12_10).
- [EXPORT_VERSION]: Current export version name.
- [EXPORT_NOTE]: Current export version note.

NOTE For file sequence exports, you can define the file padding value. If the selected file format is not a file sequence format, the padding field is greyed out.

Destination: Displays the destination folder for the exports, as defined by the project settings. You can change the destination folder by clicking on the browse button. When browsing for a destination folder, only Wiretap Gateway servers are displayed. You can select different Wiretap Gateway servers than your default server. The path must be in the following format:

[EXPORT_HOST]@wtg/[PROJECT_HOME]



NOTE When you select a different Wiretap Gateway server, the Wiretap Gateway Server option field is updated. This action also refreshes the available codecs list.

Once you select a Wiretap Gateway server, you can select an existing folder or create a new one as your destination folder.

Export Templates: Use to save, load and delete Export Templates. An Export Template contains one or more Export Versions.

See Export Templates (page 2252).

The Format Options Tab

Adjust the values in the Format Options tab to define your Wiretap Gateway server and the file format export options.

Format Options Tab - Video Settings



Video File Format: Use to select the file type for the export of the video stream.

Compression: Displays the list of available codecs based on the selected Wiretap Gateway server.

Options for these two fields are:

Streaming File Formats (Audio and Video):

- QuickTime:
  - proRes (if using an OSX Wiretap Gateway Server)
  - 8/10 bit Packed YUV
  - DNxHD
  - DV
  - DVCPro
  - H264
  - IMX 30/40/50
  - MPEG-4
  - PNG
  - RLE
  - Uncompressed

  - MXF:
    - AVC-Intra 100/50
    - DNxHD
    - DV
    - DVCPro 50/HD
    - XDCAM HD422

File Sequence Formats (Video only):

- Alias:
  - RLE

- Sgi:
  - RLE
  - Uncompressed

- Tiff:
  - LZW
  - RLE
  - Uncompressed

- Wavefront
  - RLE

- Uncompressed

- Open EXR:
  - B44
  - B44A
  - PIZ
  - PXR24
  - ZIP multi-scanline
  - ZIP scanline

- Dpx:
  - Z depth homogeneous
  - Z depth Linear
  - Pal
  - NTSC
  - CCIR 601 (525/625)
  - CCIR 709-1
  - SMPTE 204M
  - Unspecified
  - Logarithmic
  - Linear
  - Print Density

- Cineon:
  - Uncompressed

- Jpeg
  - Compression (0 - 100)

- Pict
- Pixar
- SoftImage
- Targa
- Maya

NOTE The list only displays the available codecs that are compatible with the resolution, frame rate and bit depth of the current grade.

Include YUV Headroom Enable to include YUV headroom on export for Quicktime and MXF files.

Profile: You can select a pre defined profile for certain video file formats. Selecting a profile populates all the relevant parameters for the export.

Each codec can be configured to produce different results. Data rate, video range, etc. can be edited to account for many file delivery requirements. The codec profiles are located in:

/usr/discret/mediaconverter/2013.0.0/profiles/<file format>/<essence>/<compression>

File Format:

- QuickTime
- MXF

Essence:

- Video
- Audio

Compression:

- The codec in question.

Each codec has a default value, located in the <default.xml> file.

H264 Codec Profiles

For H264, multiple preset profiles are available:

H264 Codec Profile:	Description:
Baseline_1SEG_384Kbits	H264_CIF 352x288 or 352x240 at 384 kbit/s, Baseline profile.
Baseline_3GP_256Kbits	H264_FLASH_LOWRES 320x240 at 300 kbit/s, Baseline profile.
Baseline_600Kbits	H264_BASELINE 320x240 at 600 kbit/s, Baseline profile.
Baseline_Adobe_300Kbits	H264_FLASH_LOWRES 320x240 at 300 kbit/s, Baseline profile.
Baseline_Apple_1_5Mbits	H264_iPOD Apple iPod, 320x240 at 1.5 Mbit/s, Baseline profile.
Baseline_Apple_400Kbits	H264_iPOD Apple iPod, 320x240 at 400 kbit/s, Baseline profile.
Baseline_Apple_600Kbits	H264_iPOD Apple iPod, 320x240 at 600 kbit/s, Baseline profile.
Baseline_Apple_970Kbits	H264_iPOD Apple iPod, 320x240 at 970 kbit/s, Baseline profile.
Baseline_CIF_600Kbits	H264_CIF 352x288 or 352x240 at 600 kbit/s, Baseline profile.
Baseline_RIM_12Mbits	H264_BASELINE 1920x1080 at 12 Mbit/s, Baseline profile.
Baseline_RIM_20Mbits	H264_BASELINE 1920x1080 at 20 Mbit/s, Baseline profile.
Baseline_RIM_4Mbits	H264_BASELINE 1920x1080 at 4 Mbit/s, Baseline profile.
HDTV_1080i_10Mbits	H264_HDTV_1080i 1920x1080 at 10 Mbit/s, High profile, interlaced.
HDTV_720p_8Mbits	H264_HDTV_720p 1280x720 at 8 Mbit/s, High profile.

H264 Codec Profile:	Description:
High_1080i_6Mbits	H264_HIGH 1920x1080 at 6 Mbit/s, High profile, interlaced.
High_AVC_HD_20Mbits	H264_AVCHD AVCHD, 1920x1080 at 20 Mbit/s, High profile, interlaced.
High_AVC_Intra_111Mbits	H264_INTRA_CLASS_100 AVC Intra Class 100, 1920x1080 at 111 Mbit/s, High 10 profile, interlaced.
High_AVC_Intra_54Mbits	H264_INTRA_CLASS_50 AVC Intra Class 50, 1440x1080 at 54 Mbit/s, High 10 profile, interlaced.
High_Blu_Ray_20Mbits	H264_BD_HDMV Blu-ray HD, 1920x1080 at 20 Mbit/s, High profile, interlaced.
High_Blu_Ray_8Mbits	H264_BD Blu-ray SD, 720x576 or 720x480 at 8 Mbit/s, High profile, interlaced.
High_Divx_2Mbits	H264_DIVX DivX+, 1920x1080 at 2 Mbit/s, High profile.
High_DVD_3Mbits	H264_DVD 720x576 or 720x480 at 3 Mbit/s, High profile, interlaced.
High_HD_DVD_20Mbits	H264_HD_DVD 1920x1080 at 20 Mbit/s, High profile, interlaced.
High_Microsoft_10Mbits	H264_SILVERLIGHT Microsoft Silverlight, 1920x1080 at 10 Mbits/s, High profile.
High_Microsoft_500Kbits	H264_SILVERLIGHT Microsoft Silverlight, 640x480 at 500 kbits/s, High profile.
Main_3Mbits	H264_MAIN 704x576 or 704x480 at 3 Mbit/s, Main profile.
Main_Adobe_670Kbits	H264_FLASH_HIGHRES 640x480 at 670 kbit/s, Main profile.
Main_Apple_1_8Mbits	H264_MAIN 1024x576 at 1.8 Mbit/s, Main profile.
Main_Apple_4_5Mbits	H264_MAIN 1280x720 at 4.5 Mbit/s, Main profile.
Main_D1_3Mbits	H264_D1 720x576 or 720x480 at 3 Mbit/s, Main profile, interlaced.
Main_Sony_2Mbits	H264_PSP_640x480 Sony PSP Level 3, 640x480 at 2 Mbit/s, Main profile.
Main_Sony_700Kbits	H264_PSP Sony PSP, 320x240 at 700 kbit/s, Main profile.
Main_Sony_900Kbits	H264_PSP_480x270 Sony PSP Level 2, 480x272 at 900 kbit/s, Main profile.
Main_SVCD_1_15Mbits	H264_SVCD 480x576 or 480x480 at 1.15 Mbit/s, Main profile, interlaced.

The H264 codec profiles are located in:

/usr/discreet/mediaconverter/profile/quicktime/video/h264

NOTE Some H264 codec profiles are also used by some Export Templates. Keep this in mind if you plan to modify some of the parameters within the codec profiles. Your changes will be reflected in the Export Templates.

Format Options Tab - Audio Settings



Audio File Type: For streaming formats, like QuickTime and MXF, the Audio File Type menu is automatically set to the Video File Type. There may be situations in which you require a separate audio stream. For example, if you export the video stream as a DPX file sequence, you can select a standalone audio file format (.aiff, .wav, .mp3, etc.).

The supported standalone audio formats are:

- AIFF-C
- AIFF
- Nextsnd
- MP3
- WAVE
- Broadcast Wave
- Berkeley/IRCAM/CARL Sound (BISCF)
- Audio Visual Research (AVR)

Compression: Use to select the audio compression settings. Options are:

For QuickTime: (Only 16-bit is available. There are no options in the UI.)

- Alaw
- ima4
- Linear PCM (QT 7)
- Mpeg Layer 2 Audio
- Sowt
- Twos
- Ulaw

For MXF:

- 16bit Linear PCM
- 24bit Linear PCM

Sampling: Use to define the sampling frequency of exported audio files. Options are:

- Native
- 11Khz
- 22Khz

- 44.1Khz
- 48Khz

Lustre's behavior when exporting audio depends on the type of render mode selected. To export media files with audio, Lustre uses the grade-based audio file when the Media Origin is set to OneSequence. If you select Source Grade or Original Media and you are working with media files that contain audio media, such as Quicktime, MXF, R3D, the exported audio will come from the source media files.

Profile: When exporting Quicktime movies with PCM audio, you can select an audio encoding profile to define the bit depth and the bit ordering. The following profiles are available:

- 16-bit (LE)
- 16-bit (BE)
- 24-bit (LE)
- 24-bit (BE)
- 32-bit fp (LE)
- 32-bit fp (BE)

NOTE LE: little endian; BE: big endian. Endianness has no impact on performance, and little endian should work in most cases. Output in big endian mode if you expect compatibility issues with the target application / operating system.

Summary:

- **Source Grade Renders:** Lustre uses the source audio of each clip for the render and re-encodes it to the selected format.
- **One Sequence Renders:** Lustre uses the timeline's audio and re-encodes it as one multi-channel audio track (the number of channels depends on the selected output format).
- **Original Media:** Lustre uses the source audio of each clip for the render and re-encodes it to the selected format.

NOTE

- You must select an audio output format that supports the number of audio channels you are exporting. For example, if you are exporting a four channel audio file to mp3, channels 3 and 4 will be discarded during the encoding as the mp3 format only supports up to two channels.
- When exporting audio, Lustre will re-encode to the bit depth of either your source audio or your timeline audio, based on your export mode, as long as the selected output format supports that bit depth. If it does not support the bit depth, the export defaults to a lower bit depth, supported by the selected output format.

The Resize Tab

Adjust the values in the Resize Options tab to define the destination resolution and the framing of the exported media.



Output Resolution: Use to define the output resolution, aspect ratio, bit depth and scan mode of the exported media.

Resolution Field: Use to set custom resolution values.

Aspect Ratio Field: Select from a list of preset aspect ratios or set your own value.

Bit Depth Field: Use to define the bit depth of exported media.

Scan Mode Field: Use to define the scan mode of exported media.

Resize Options: Use to define the Fit Method, Resize Filter type, the Precision and the Crispness/Softness of the exported media.

Fit Method Box: Use to select the Fit Method applied to exported media. Options are:

- Fill (the default option)
- Crop Edges
- Letterbox
- Center/Crop

Resize Filter Type Box: Use to select the Resize Filter type applied to exported media. Options are:

- Impulse
- Triangle
- Mitchell
- Bicubic
- Quadratic
- Gaussian
- Shannon
- Lanczos

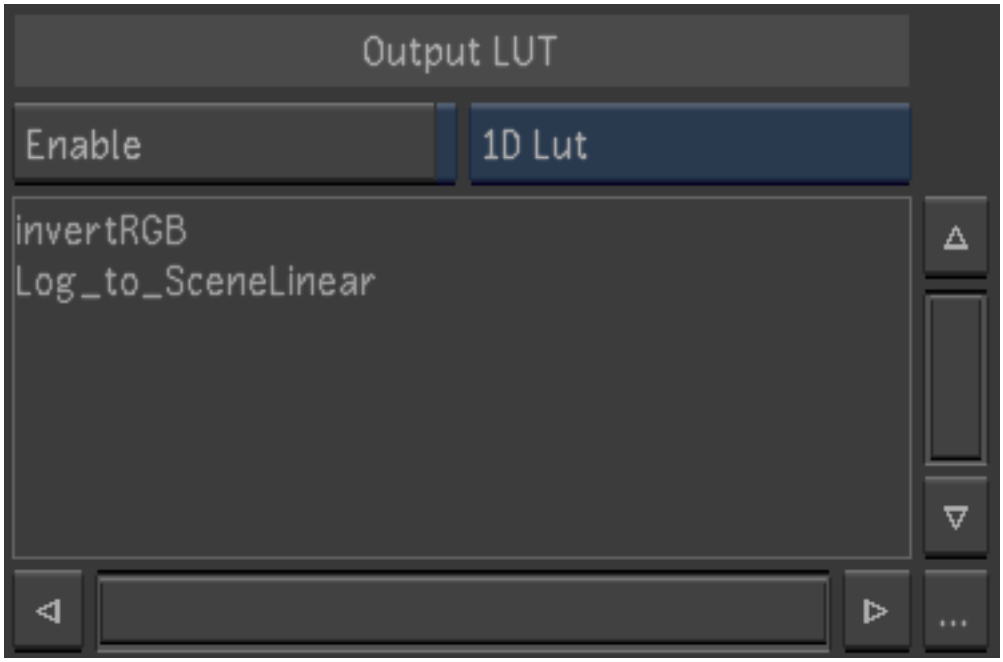
Aspect Button: Enable to use non-square pixel formats. This option is automatically enabled when Fit Method is set to Crop Edges or Letterbox.

Precision Field: Use to set the frequency cut-off point used during the Resize.

Crisp/Soft Field: Use to set the amount of blur used during the Resize.

The LUT Options Tab

Use the Render / Export / LUT Options menu to apply a LUT or colour transform to exported media.



Enable: Apply the selected LUT or colour transform when exporting.

Output LUT type: Select 1D LUT, 3D LUT, or Colour Transform.

Output LUT list: Select a LUT or colour transform to apply to exported media.

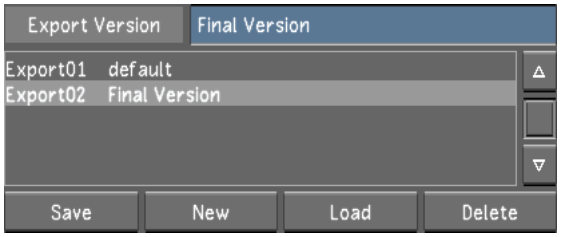
Browse (...) button: Press to open the file browser to navigate to a LUT or colour transform file.

NOTE The Wiretap Gateway supports Flame-compatible 1D and 3D LUT files only. Lustre-compatible 1D and 3D LUT files are not supported.

Export Versions

Export Versions contain all the data required to define an export job and are stored alongside the grade metadata files.

You can create, update, load and delete Export Versions.



To Create an Export Version

1. Define your export settings in the Export tabs.
2. Type a comment to identify your Export Version in the Export Version Comment field.
3. Click New.

Your Export Version is saved and displayed in the Export Version List.

To Load an Export Version

1. Select the Export Version you want to load from the list.
2. Click Load.

Your Export Version is loaded and the export settings are updated.

To Update an Export Version

1. Make sure the Export Version you want to update is loaded.
2. Modify the parameters as needed.
3. Click Save.

The Export Version is updated.

To Delete an Export Version

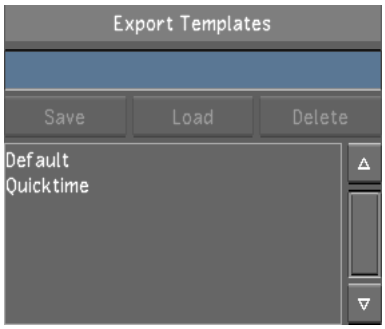
1. Make sure the Export Version you want to deted is loaded.
2. Click Delete.

Your Export Version is deleted.

Using Export Templates

Use templates to automatically define one or multiple export versions. If for a given project you need to consistently provide the same file types, use Templates to easily create the required versions. You can create, update, load and delete Export Templates.

Export Templates contain one or more Export Versions. When you load an Export Template, the Export Versions contained within the template are appended to the currently displayed Export Versions. If you wish to overwrite the currently displayed Export Versions when you load your template, press the Alt key when clicking the Load button.



To Create an Export Template

1. Make sure you have saved at least one Export Version.
2. Type a name to identify your Export Template in the Export Template Name field.
3. Click Save.

Your Export Template is saved and is displayed in the Export Template List.

To Load an Export Template (Append)

1. Select the Export Template you want to load from the list.
2. Click Load.

Your Export Template is loaded and the Export Version list is updated.

To Load an Export Template (Overwrite)

1. Select the Export Template you want to load from the list.
2. Hold the Alt key.
3. Click Load.

Your Export Template is loaded and the Export Version list is updated.

To Update an Export Template

1. Make sure the Export Template you wish to update is loaded.
2. Modify the export parameters and save one or more new Export Versions.
3. Click Save.

The Export Template is updated to contain the new Export Versions.

To Delete an Export Template

1. Make sure the Export Template you wish to delete is loaded.
2. Click Delete.

Your Export Template is deleted.

Preset Export Templates

You can also choose from a list of preset export templates to use for the following applications and devices:
:

- Avid® Media Composer® software
- Apple® Final Cut Pro® application
- Adobe® Flash®
- Apple® iPod®
- Apple® iPad®
- Apple® iPad® 2
- Apple® iPhone®
- Apple® iPhone® 4
- Apple® iPhone® 4S
- Apple® iPod® Touch
- Apple® TV
- Apple® TV 2
- Blu-ray™
- BlackBerry® PlayBook™ tablet
- DivX®

- Microsoft® Silverlight®
- Sony® PSP®

Preset Export Template:	File Size Estimate (30 second clip with two audio tracks at 23.97 fps):
Apple_FinalCutpro_1080p_online	431M
Apple_FinalCutpro_1080p_online_444	906M
Apple_FinalCutpro_1080p_online_HQ	614M
Apple_FinalCutpro_1080p_online_HQ	210M
Apple_FinalCutpro_720p_online_444	462M
Apple_FinalCutpro_720p_online_HQ	319M
Apple_FinalCutpro_960p_offline	123M
Adobe_Flash_320x240_300Kbits	2.7M
Adobe_Flash_640x480_670Kbits	4.0M
Apple_iPad_1024x576_1_8Mbits	13M
Apple_iPad_1280x720_1_8Mbits	13M
Apple_iPhone_640x480_1_5Mbits	5.1M
Apple_iPod_320x240_600Kbits	3.7M
Apple_iPod_640x480_970Kbits	5.1M
Apple_TV_1280x720_4_5Mbits	13M
AVC_HD_1920x1080_20Mbits	76M
AVC_Intra_1440x1080_54Mbits	166M
AVC_Intra_1920x1080_111Mbits	330M
Avid_MediaComposer_1080p_offline	141M
Avid_MediaComposer_1080p_online_10bit	648M
Avid_MediaComposer_1080p_online	648M

Preset Export Template:	File Size Estimate (30 second clip with two audio tracks at 23.97 fps):
Avid_MediaComposer_720p_offline	220M
Avid_MediaComposer_720p_online	333M
Avid_MediaComposer_720p_online	333M
BlackBerry_PlayBook_1920x1080_12Mbits	11M
BlackBerry_PlayBook_1920x1080_20Mbits	11M
BlackBerry_PlayBook_1920x1080_4Mbits	11M
Divx_1920x1080_2Mbits	77M
HDTV_1080i_10Mbits	38M
HDTV_720p_8Mbits	35M

Export Workflows

Example workflows used with the Background Media Export feature in Lustre 2013.

Final Project Delivery Workflow

When your project is finished and ready to be digitally delivered, you can use the following workflow.

1. Render your project in One Sequence mode.
2. In the Render/Export menu, create a new Export version. **See** Export Versions (page 2251).
3. Set the Media Origin field to One Sequence Renders.
4. Set the Region field to Scene to export all shots.
5. If the sequence has audio media and that you want to be included in the export, enable the Video & Audio option.
6. Set the Video File Format and the Compression type or select a profile, if available.
7. If the Media Origin field is set to Audio & Video, set the Audio File Format and Compression type.
8. Define the File Name using tokens or an actual alphanumeric filename.
9. Define the Export Destination.
10. Save your Export Version.
11. Click Export.

The Export process is started and the resulting media - in this case one file sequence, with audio if selected - will be saved at your defined destination.

If the grade is not rendered before you click Export, Lustre warns you and automatically starts the render, and then the export.

If your workstation is setup with a Burn configuration, you can now send Media Export jobs without first having to render your grades in the foreground. Both processes take place in the background. After the grades are rendered, the Media Export jobs start. This allows you to send multiple jobs for Media Export without having to wait for foreground grade rendering. See Remote Rendering with Burn and Wiretap (page 2233).

If you need to deliver the same grade in other file formats, create a new Export Version and define the various parameters for the chosen file format.

If you want to export multiple versions, enable the Export All Versions option and press Export.

Production Dailies for Editorial Workflow

When working with digitally acquired content like RED and ARRIRAW, post facilities are often asked to deliver graded production material to Editorial, often in various file formats. The following workflow can accommodate this situation.

1. Import the source clips you need to grade.
2. Create a new Cut and a new Grade.
3. Grade the content.
4. In the Render/Export menu, create a new Export version. **See** Export Versions (page 2251).
5. Set the Media Origin field to Source Grade Renders.
6. Set the Region field to Scene to export all shots.
7. If the sequence has audio media and that you want to be included in the export, enable the Video & Audio option.
8. Set the Video File Format and the Compression type or select a profile, if available.
9. If the Media Origin field is set to Audio & Video, set the Audio File Format and Compression type.
10. Define the File Name using tokens or an actual alphanumeric file name.
11. Define the Export Destination.
12. Save your Export Version.
13. Press Export.

The Export process is started and the resulting media - in this case the individual source clips that comprise your sequence, with audio if selected - will be saved at your defined destination.

If the grade is not rendered before you click Export, Lustre warns you and automatically starts the render, and then the export.

If you need to deliver the same grade in other file formats, create a new Export Version and define the various parameters for the chosen file format.

If you want to export multiple versions, enable the Export All Versions option and press Export

Media Conversion Workflow

When you need to deliver media files of a given format into another one, with or without audio, the following workflow can accommodate this situation.

This workflow is similar to the Production Dailies for Editorial workflow but does not require media rendering since the exported media is not graded. Only the original media is exported.

The steps are identical to the Production Dailies for Editorial workflow except that you need to set the Media Origin to Original Media.

Stand-Alone Wiretap Gateway Server Configuration for Media Encoding

To use a stand-alone Linux or Mac OSX remote Wiretap Gateway server for media encoding, follow the instructions below.

1. Install Backburner 2013.
   - Add the name of your Backburner Manager in /usr/discreet/backburner/cfg/manager.host. In most cases, the Backburner Manager to use is the one installed on your creative workstation.
   - Restart Backburner manager.
2. Install Wiretap Gateway 2013.
3. Restart the Wiretap Gateway Server.

Media Export in Lustre with Incinerator

Whne using the Media Export feature in Lustre with Incinerator, it is important to keep certain considerations in mind.

The Incinerator configuration uses Browsed and Lustre Media Server (LMS) for media access over InfiniBand. In this configuration, the Backburner Manager runs on the LMS. You must configure the remote server (Linux / Mac OS X) to use the LMS as the Backburner Manager.

To speed up media transfers between the LMS and the Linux servers used for media export, make sure the Wiretap Gateway configuration file is set to use InfiniBand as the main network interface. To acheive this:

1. As root, open the following file: */usr/discreet/wiretapgateway/cfg/wiretapgateway.cfg*.
2. Define the server IP address in the IP0 line: *IP0=10.0.10.1*
3. Save the file.
4. Restart the Wiretap Gateway server with the following command: */etc/init.d/wiretapgateway restart*.

Your Wiretap Gateway server is now configured to use InfiniBand as the main network interface.

Video Capture and Video Playout

During the colour grading process, you may need to capture or playout video and audio from, or record to, an external device, such as a VTR.

For example, you may receive an edited HD or SD video tape to grade with Lustre. Or, you may be given raw footage on video tapes and an EDL to auto-conform in Lustre, and then colour grade.

During capture, the files are created on-the-fly and are available after capturing is completed.

NOTE When capturing video from an HD or SD tape and converting to a DPX file, Lustre writes the SMPTE timecode to the DPX header. Once captured, the timecode appears as part of the DPX file information and is available for use in EDL assembly.

After grading a scene, you play it out to a VTR for client approval or for final delivery on HD or SD tapes.

NOTE The availability of SDI video rasters, which are required to control a VTR, depends on the type of license you have purchased. Contact Customer Support for details.

But before you can capture from, or play out to, an external device, you set up options such as the capture path, file type, and video resolution, as well as options for transferring shots between Lustre and a VTR.

About Audio Tracks

When capturing and recording, you can transfer one video track and up to 16 audio tracks (eight tracks of DVS/AJA AES audio, or 16 tracks of embedded AJA audio). In Lustre, audio tracks are identified as A1 through A16. You can select non-consecutive tracks (for example, you can select tracks A1, A5, and A8).

You can capture, playback, and play out audio in two different ways:

- Audio embedded with video, coming from or going to the SDI connectors ('Embedded' option)
- Audio going through XLR cables from or to the DVS or AJA breakout box ('AES' option)

For more information on capturing, playing back, and playing out audio, see Audio (page 2306).

Capture Menu Options

Use the Capture menu to perform your VTR capture operations.

NOTE Capture can only be done using the DVS or AJA video cards. The GFX/SDI options listed below are provided as reference only and should not be used during the capture.

The following three graphics represent the Capture menu.







(a) Video/Graphics option box (b) Raster list (c) Format option box (d) Link Type option box (e) Audio File Type option box (f) Sampling Rate/Bit Depth option box (g) Audio Input button (h) Audio Track Selection option box(i) VTR Status panel (j) Capture Error status

Video/Graphics option box Toggle between the video card (AJA or DVS) and the graphics card (GFX/SDI). The Raster list is then populated with rasters for the selected card.

WARNING When capturing, do not select a raster for the GFX/SDI card. Doing so is not supported and can result in failed capture.

Raster list With a video card selected in the Video option box (AVIO), select one of the listed rasters to set the resolution and timing for capture. For more information about rasters, see Supported Video SDI and GFX SDI Rasters (page 2298).

Format option box Toggle to display only rasters belonging to a particular footage format in the Raster list.

Select:	To display:
SD	Standard definition rasters.
HD	High definition rasters.

Select:	To display:
Film	Film rasters.
Audio	Audio rasters (only applicable to GFX/SDI).

NOTE Enable an Audio Only raster when monitoring the GFX/SDI output and working with audio signal coming out of the AJA/DVS breakout box. This is similar to working with audio media or to using the LTC chase feature with a DVS.

Link Type option box Toggle to display only rasters belonging to a particular link type in the Raster list.

Select:	To display:
Single	The single link rasters.
Dual	The dual link rasters.
Stereo	The stereoscopy rasters (only applicable to GFX / SDI).

Pulldown button Enable to remove 3:2 pulldown from Film-to-video transferred material. See Removing Pulldown when Capturing (page 2265).

Full Range button Enable to configure Lustre to capture the full video signal from the video board. If disabled, Lustre captures only the legal portion of the video signal.

Sync Mode list Select one of the sync modes.

Select:	To:
InternalSync	Set the sync mode to a free running internal sync (SD and HD).
ExternalSync	Genlock to an analog sync connected to the sync input (SD only).
TrilevelSync	Genlock to a trilevel sync connected to the sync input (HD only).

Video Capture Format list Select a video capture format to set file type, colour space, and bit depth of the captured material.

Select:	To capture only:
Dpx 10Bit RGB	DPX files with an RGB colour space at a bit depth of 10-bit.
Tiff 8Bit RGB	TIFF files with an RGB colour space at a bit depth of 8-bit.
AVI 8Bit RGB	AVI files with an RGB colour space at a bit depth of 8-bit (available only on Windows workstations.)

Audio File Type option box Choose whether to save audio captured from a VTR/ATR, or imported from Wiretap as a Wave (.WAV) or AIFF (.AIF) file.

Audio Sampling Rate/Bit Depth option box Select to capture the audio tracks at a sampling rate of 48 kilohertz (kHz) and a bit depth of 16-bit, or 48 kHz and a bit depth of 24-bit.

Single File Capture button When enabled, all the selected audio tracks are saved in a single file. When disabled, each audio track is saved as an individual file. This option is enabled by default.

Log panel Displays status messages during capture.

TC button Initiates a capture from VTR using timecode in and out points. See Capturing from Timecode (page 2267).

EDL button Initiates a capture from VTR using an EDL. See Auto-conforming an EDL (page 2273).

NOTE EDL Capture is video only; audio events are ignored.

Capture TC Break button Initiates a capture from VTR using the timecode in and out points, regardless of the number of timecode breaks. See Capturing Media with Timecode Breaks (page 2270).

Timecode In Point field Enter the start timecode. Press Set next to the field to set the current timecode (read from the VTR) as the in point.

Timecode Out Point field Enter the end timecode. Press Set next to the field to set the current timecode (read from the VTR) as the out point.

EDL Log Add button Logs the In and Out timecode points as an EDL for capture. See Logging and Batch Digitizing (page 2271).

NOTE Only video events are supported for EDL logging.

Handle button Enable to capture heads and tails. This feature is used when capturing from an EDL or from timecode.

NOTE You must enable the Handle button in order to use the Handle slider.

Handle slider Set the number of head and tail frames to capture.

V button Enable to capture the video track when capturing from a video source. Disable to capture audio only.

A1 - A16 buttons Enable the audio track(s) (A1-A16) to capture from an audio source.

VTR Status panel Displays the VTR status messages when Lustre is connected to a VTR.

Audio Track Selection option box Used in correlation with the A1-A16 buttons. Toggle to determine whether tracks 1-8 or 9-16 are visible. Tracks 9-16 are only available with embedded AJA audio. If only eight audio tracks are available, this option is disabled (greyed out).

Audio Input option box Toggle to specify the source of the captured audio.

Select:	To specify the connection to the audio source:
Embedded	The SDI connection.
AES	The XLR connections on the DVS/AJA breakout box.

Video Delay field Enter a frame number to delay the video capture for that number of fields. For example, if you set the field to 10, the capture begins 10 frames after the In point. If you set the field to -10, capture begins 10 frames before the In point.

Audio Delay field Enter a frame number to delay the audio capture for that number of fields. For example, if you set the field to 10, the audio capture begins 10 frames after the In point.

NOTE The Video and Audio Delay fields compensate for the frame delays that may exist when capturing to Lustre.

Capture Error status Displays a message when an error has occured while capturing from a VTR.

Pulldown Field buttons Enable a Pulldown Field button to select the frame used for the 3:2 pulldown. Only available when Pulldown is enabled. See About 3:2 Pulldown (page 2297).

Standby Off button Removes the VTR heads from the tape.

Eject button Ejects the tape in the VTR.

Home field Displays the project home directory. This is the destination path for captured footage. You cannot change this value without modifying your project settings.

Scene field Displays the current scene you are working in during capture. You cannot change this value without modifying your project settings.

Capture field Enter the name of the folder where video media is saved.

Tape field Displays the tape name. When capturing in TC mode, enter a tape name. When capturing from an EDL, the default tape name is determined by that EDL. If you capture an EDL with multiple tapes, a folder structure (<tape>/<resolution>/) is created for each tape (e.g., *0001/1920x1080*).

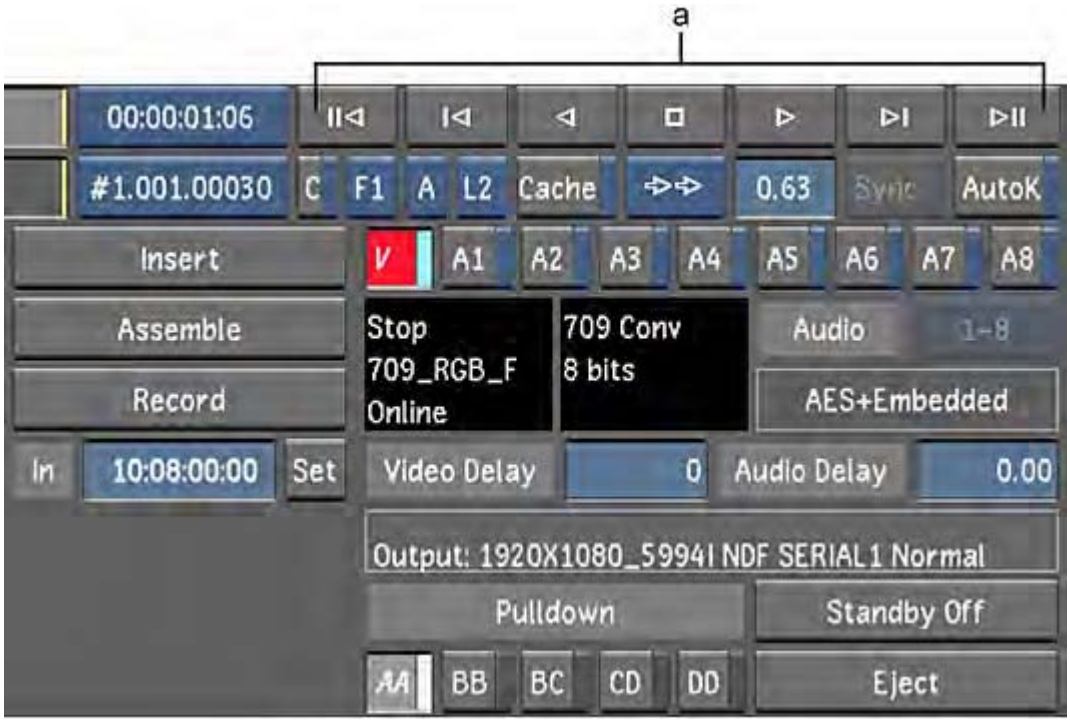
Resolution field Defaults to the resolution of the captured files, and is used in the folder structure. Do not change this field.

File field Displays the filename, which is the frame offset value based on the source timecode (e.g., *84600.dpx* is 1:00:00:00 at 24 fps). <NUM7> defines the amount of padding for the file name (for example, *084600.dpx* reflects six digits of padding before the extension). You can add a prefix before the <NUM7> number, but to ensure consistent frame counting, do not erase the <NUM7> number (e.g., *Mymovietitle.084600.dpx*).

Controlling the VTR

When working with a VTR, you can use the VTR transport controls to play through the video tape and locate your footage. As you find your footage, you can capture it immediately, or add shots to a list that you can batch digitize all at once.

You can also use the transport controls to cue up a VTR to the record in point when playing out to the VTR.



(a) VTR transport controls

NOTE To control the VTR remotely, you must connect a 9-pin RS-422 cable between the VTR and Lustre. For information, refer to the *Autodesk Lustre Software Installation Guide* for your platform.

To control the VTR:

1 Do one of the following:

- If performing capture operations in the Capture menu, click the Source Timecode field. The capture controls turn blue to indicate that you can use them as VTR transport controls.
- If performing playout operations in the Playout menu, click the Source Timecode field. The playback controls turn blue to indicate that you can use them as VTR transport controls.

  NOTE In the Playout menu, you can toggle between the timeline and VTR by clicking on the Source Timecode field. If the Source Timecode field is blue, the controls can be used as VTR transport controls. If the controls are white, they are timeline controls.



(a) Source Timecode field

The transport controls are as follows.

Click:	To:
	Fast rewind.

Click:	To:
	Cue to TC In. Right-click to jog back one frame.
	Play in reverse.
	Stop the VTR.
	Play forward.
	Cue to TC Out. Right-click to jog ahead one frame.
	Fast forward.

2 To eject the tape in the VTR, click Eject in the Capture menu.

NOTE You can also control the VTR from the Playout menu. See Playing Out to a VTR (page 2280).

Capturing Material

Use the Capture menu to digitize material from an external source. You can digitize in any one of the following ways:

- Digitize one event at a time by capturing from timecode values. See Capturing from Timecode (page 2267).
- Digitize events from an external source that has various timecode breaks. See Capturing Media with Timecode Breaks (page 2270).
- Batch digitize from a Log file. See Logging and Batch Digitizing (page 2271).
- Batch digitize from an imported EDL. See Auto-conforming an EDL (page 2273).
- Perform a live capture from non-controlled sources. See Performing a Live Capture from a Non-Controlled Source (page 2276).

For certain video formats, you can remove pulldown when capturing. See Removing Pulldown when Capturing (page 2265).

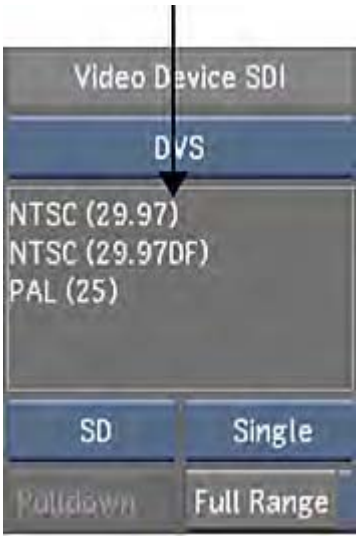
Selecting a Raster for Capture

Whenever capturing material, you must select a raster. For a list of the supported rasters, see Supported Video SDI and GFX SDI Rasters (page 2298).

NOTE When you select a drop frame (DF) video format, the timecodes are calculated in Drop Frame mode.

To select a raster for capture:

1. In the Video Device SDI group, toggle the Video/Graphics option box to display your video card (AVIO). See Capture Menu Options (page 2258).
2. Select an option from the Link Type box. Select Single for a 4:2:2 video signal, and Dual for a 4:4:4 video signal.
3. Toggle the Format option box to the required raster format (SD, HD, Film, or Audio).
4. In the Raster list, select the raster that matches the material to capture.



NOTE If clicking on a raster in the Raster list does not select the item, that format is not available for capture on that system.

5. (Optional) Remove 3:2 pulldown frames on capture. See Removing Pulldown when Capturing (page 2265).

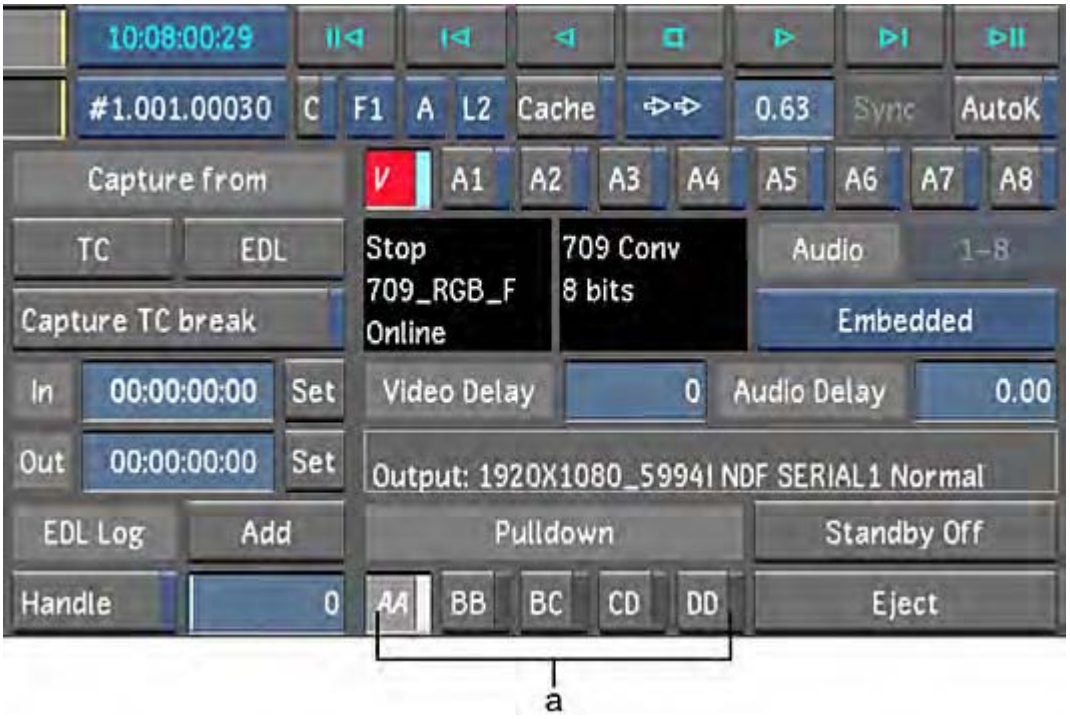
Removing Pulldown when Capturing

Depending on the type of source material you are working from, you may need to remove 3:2 pulldown frames at capture time. For more information about 3:2 pulldown, see About 3:2 Pulldown (page 2297).

NOTE Inconstant cadence material (i.e., re-edited material) may produce jittery results.

To remove 3:2 pulldown during capture:

1. Click Pulldown.
2. Use the VTR to shuttle to the first frame you want to capture.



(a) Pulldown Field buttons

3 Enable the Pulldown Field button that matches the field composition of that frame.

Setting In and Out Points

You can set in and out points to select the material to capture from an external video source.

To set in and out points:

1 Set the in point. Use one of the following techniques:
- Enter the in point timecode by typing the value directly in the In field.
- Navigate to the in point using the VTR transport controls, and then, in the Capture from group, click Set next to the In field.



2 Set the out point. Use one of the following techniques:
   - Enter the out point timecode by typing the value directly in the Out field.
   - Use the VTR transport controls to navigate to the out point, and then, in the Capture from group, click Set next to the Out field.

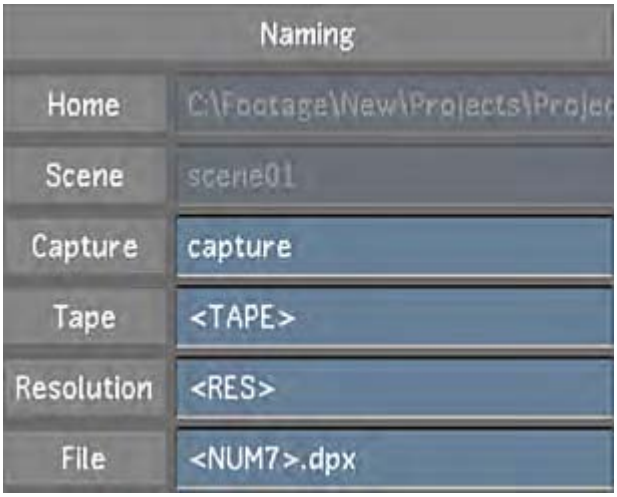


Capturing from Timecode

In capturing from timecode mode, you capture one shot by marking the length of the shot with in and out timecode values. You can either enter the timecode values manually or scrub the VTR to the in and out points. You also have the option of capturing video only, audio only, or audio along with the video.

To capture from timecode:

1 Click Editing, and then click Capture.
   The Capture menu appears.
2 In the Naming group, enter values in Capture and Tape. See Capture Menu Options (page 2258).



3 Select a raster for capture. See Selecting a Raster for Capture (page 2264).

4 In the Sync Mode list, select a synchronization option. See Capture Menu Options (page 2258).

5 In the Video Capture Format list, select a file format. See Capture Menu Options (page 2258).



NOTE The AVI format is only available on Windows workstations.

6 Select the audio file type, bit depth, and determine if you want to capture the audio tracks as a single or multiple files. Single File Capture is enabled by default. For more information about capturing audio, see Capturing an Audio File (page 2309).

7 (Optional) Set video and audio capture delays. See Capture Menu Options (page 2258).

8 Select the video and audio tracks to capture. See Capture Menu Options (page 2258).

9 Set the in and out points. See Setting In and Out Points (page 2266).

10 In the Capture from group, click TC. When the button turns red, click it again to confirm the action.

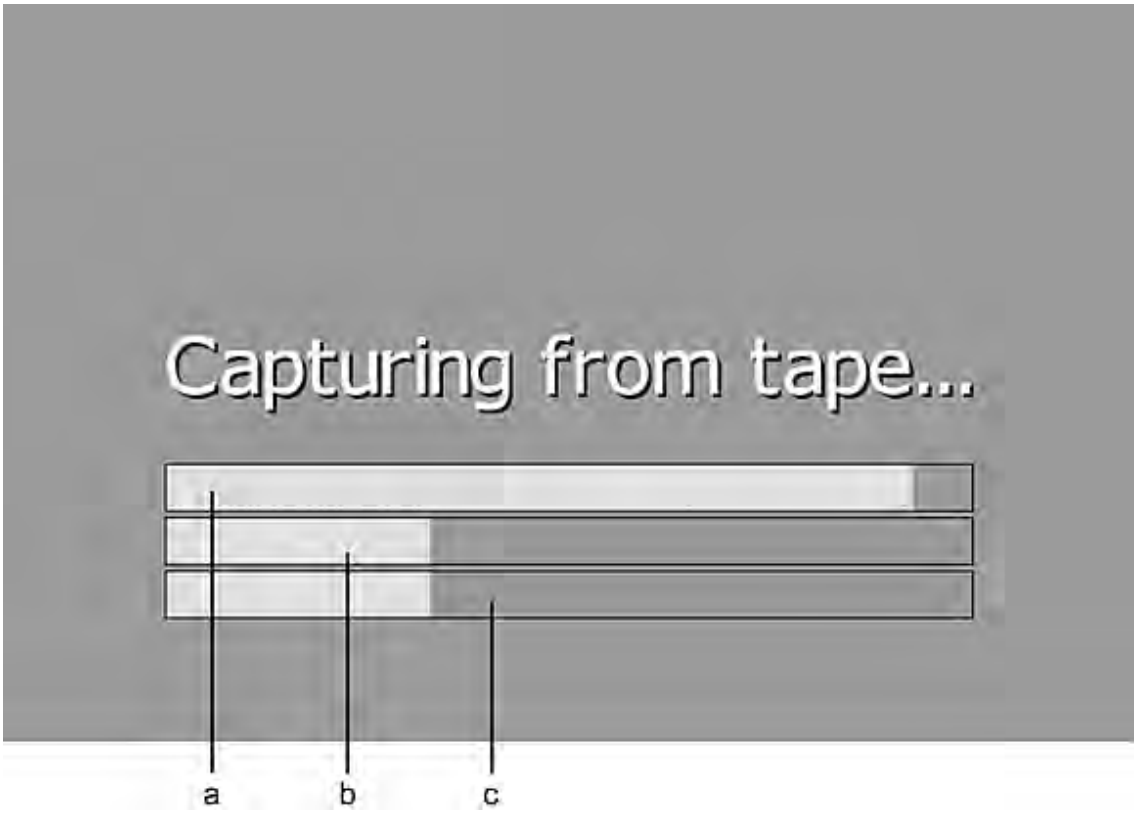


The capture starts. The VTR shuttles back to the in point and begins capturing. The VTR's current timecode and remaining timecode are displayed in a field located to the left of the playback controls. While capturing, both timecodes update and appear in red.



(a) VTR timecode remaining (b) Current VTR timecode

During capture, the message “Capturing from tape...” appears in the Player, and progress bars display shot, tape, and overall progress. There is no video displayed in the Player during capture.



(a) Shot progress (b) Tape progress (c) Overall progress

11 (Optional) To pause the capture process to access additional capture options, press `Esc`, and then do one of the following:

- Press `Esc` to abort the capture.
- Press `E` to restart the capture.
- Press `S` to skip the current tape.

Capturing Media with Timecode Breaks

The Capture with Timecode Breaks feature allows you to capture all the content of a tape, regardless of the number of timecode breaks. When you use this option, Lustre begins capturing from the given in point until the duration point has been reached. A file sequence is created for each continuous timecode section.

To use this feature properly, you need to make sure you are entering the correct values for the in and out point. The in point is the source timecode of where you want the capture to begin. In the following procedure, the in point is set at 10:00:00:00 (as an example). The out point is used to compute the duration of capture. In the procedure, the out point is set at 10:10:00:00. This represents that ten minutes of footage will be captured, regardless of the number of timecode breaks within this time frame.

NOTE The pulldown removal option and audio capture feature are not supported when capturing with timecode breaks.

To capture media that contains timecode breaks:

1. Click Editing, and then click Capture to display the Capture menu.
2. Enable the Capture TC break button.



3 Make sure the video track button is enabled.

4 In the timecode in point field, enter the point of where you want the capture to begin (e.g., 10:00:00:00).

5 In the timecode out point field, enter a timecode that is used to compute the duration of the capture (e.g., 10:10:00:00, to represent ten minutes of capture).



6 Click TC. When the button turns red, click it again to confirm the action. The capture starts.

Logging and Batch Digitizing

You can log shots and batch digitize them. To do this, you shuttle through the tape, mark in and out points that correspond to the sections you want to digitize, and then add those sections to a list of shots to digitize all at once.

NOTE This operation only logs and captures video.

To batch digitize shots from a video tape:

1 Click Editing, and then click Capture.

The Capture menu appears.

2 Select a raster for capture. See Selecting a Raster for Capture (page 2264).

3 In the Video Capture Format list, select the file format that you want to create during the capture operation.

4 In the Naming group, enter values in the Capture and Tape fields. See Capture Menu Options (page 2258).

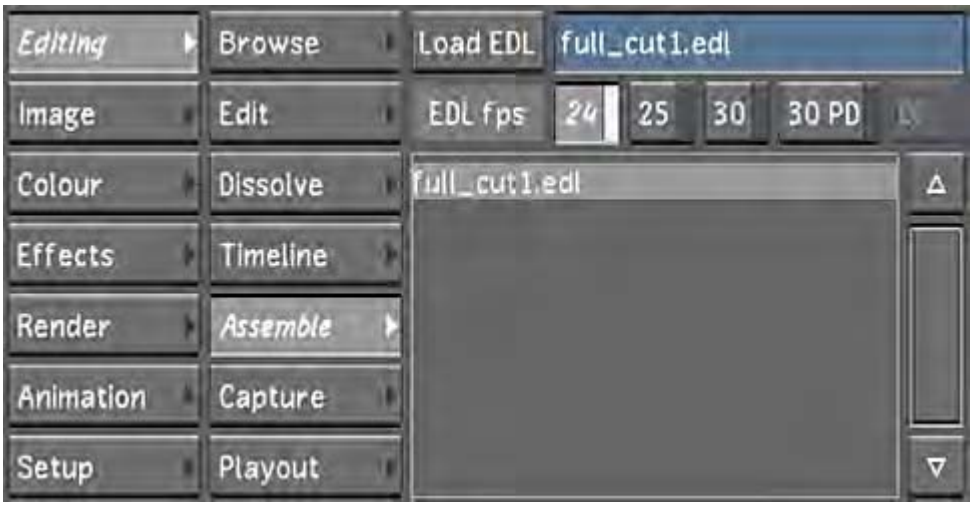
WARNING It is important that you enter a tape name to avoid overwriting previously captured files.

5 Set the in and out points. See Setting In and Out Points (page 2266).

6 In the EDL Log group, click Add to add the selected material to the events list.



7 To view the events list, click Editing, and then Assemble.



You need to mark the captures before returning to the Capture menu: click in the Event column to select an entry, or click Select All, and then click Mark for Capture.The entries marked for batch digitization appear in the Assemble menu.



8 Continue adding entries to the events list as needed.

9 (Optional) To delete an entry from the events list, select it from the Assemble window and click Delete Selected. Click it a second time to confirm the action. To clear the list, click Delete All.



10 When you are ready to digitize, return to the Capture menu.

11 In the Capture from group, click EDL once and then confirm the action by clicking EDL a second time.



Auto-conforming an EDL

You can automatically digitize material using instructions contained in an EDL, and then assemble the digitized events into a cut which appears in the Storyboard and the timeline.

You can batch digitize from an EDL that has been placed in the current scene's Library directory. This directory is located under *<HOME>\<scenename>*.

To capture from an EDL:

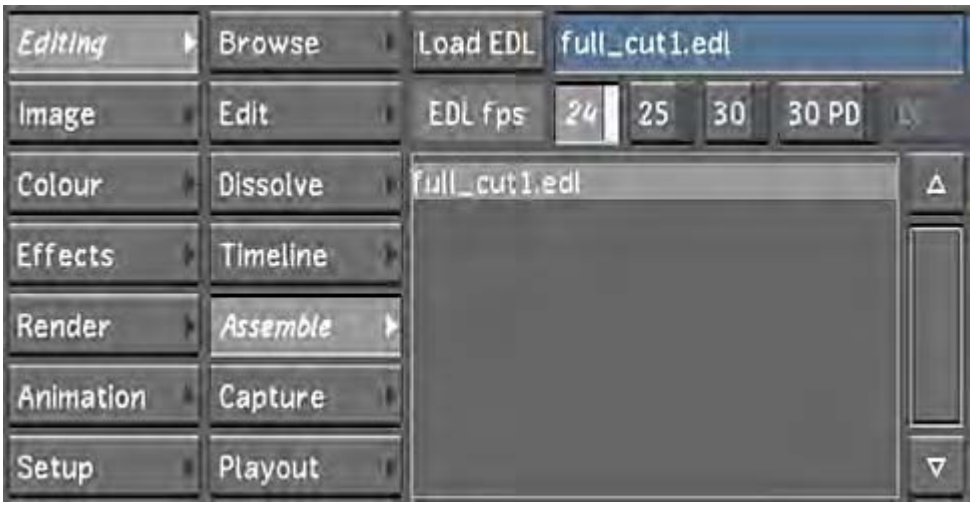
1 Make sure your EDL file is in the scene's Library directory.

NOTE If the file is in the wrong location, it will not show up in the EDL list.

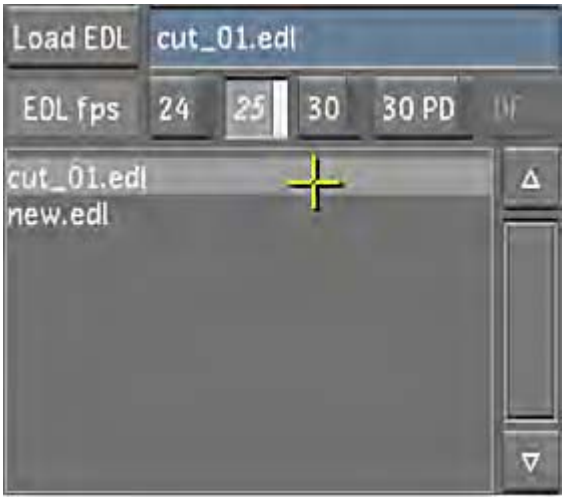
2 Click Editing, and then click Capture.

3 Select a raster for capture. See Selecting a Raster for Capture (page 2264).

4 Click Assemble to access the Assemble window.



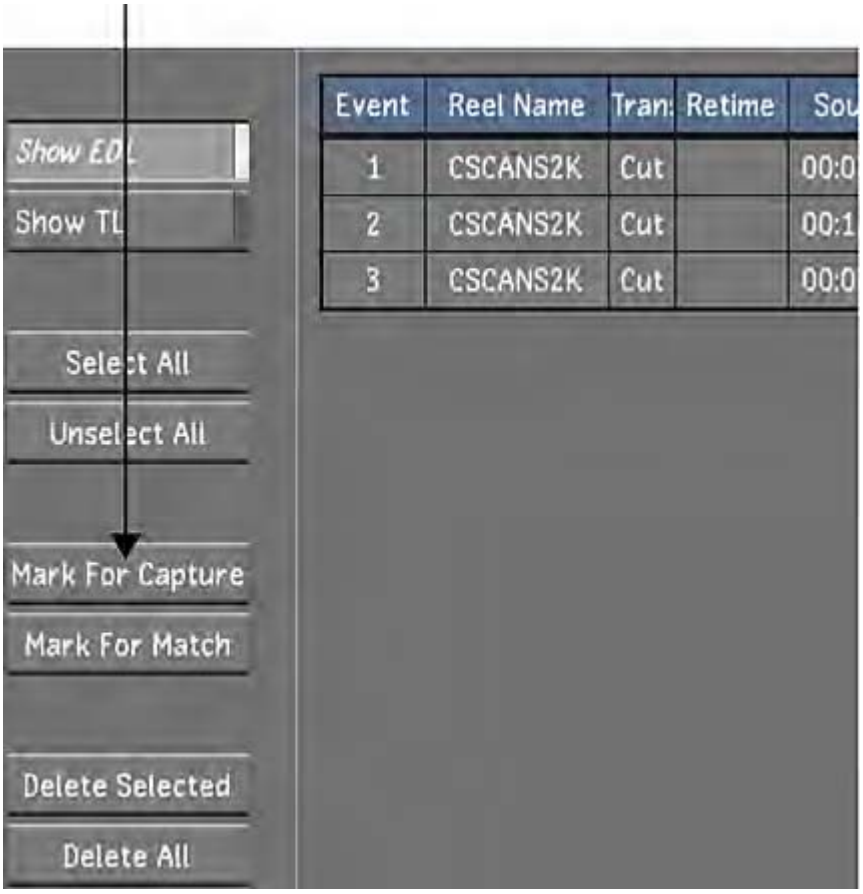
5 Select an EDL file from the EDL list. `Ctrl`-click the EDL list to refresh it.



6 Click Load EDL to display the EDL contents in the Assemble menu.

7 Select individual events to capture (`Ctrl`+click), a group of events (`Shift`+click), or click Select All.

8 Click Mark for Capture.



The selected events are marked as Need Capture under the Media column of the EDL events.

9 Click Capture.

10 In the Video Capture Format list, select the file format that you want to create during the capture operation. See Capture Menu Options (page 2258).

11 In the Naming group, enter values in the Capture and Tape fields. See Capture Menu Options (page 2258).

12 Click EDL once, and then confirm the action by clicking EDL a second time.



The Capture starts.

13 When capture is completed, add the new shots to the Library. See Loading Clips to the Library (page 1874).

14 Go back to the Assemble menu and assemble the EDL. See Assembling an EDL (page 1898).

15 Click Browse to display the Browse menu.

16 Enter the name for the new cut in the Cut name field.

17 Click New.

The new cut is saved and named after the captured EDL.

Performing a Live Capture from a Non-Controlled Source

You can capture live from any non-controlled source. A non-controlled source is any source with an SDI Out, such as another Lustre workstation, or a Smoke or Flame workstation. For example, you have footage residing on a Flame workstation that you need. You can connect to that workstation and input the footage directly, without going through tape output and input.

To perform a live capture from a non-controlled source:

1 Connect the SDI Out of the non-controlled source to the SDI In on the video card of the Lustre workstation.

2 Click Editing, and then click Capture.

3 Select a raster for capture. See Selecting a Raster for Capture (page 2264).

4 In the Capture from group, enter in and out points in the In and Out fields.

This is done in order to set the duration of the capture. The values can be anything, as long as they reflect the desired duration.



5 To start the capture, `Ctrl`+double-click TC.

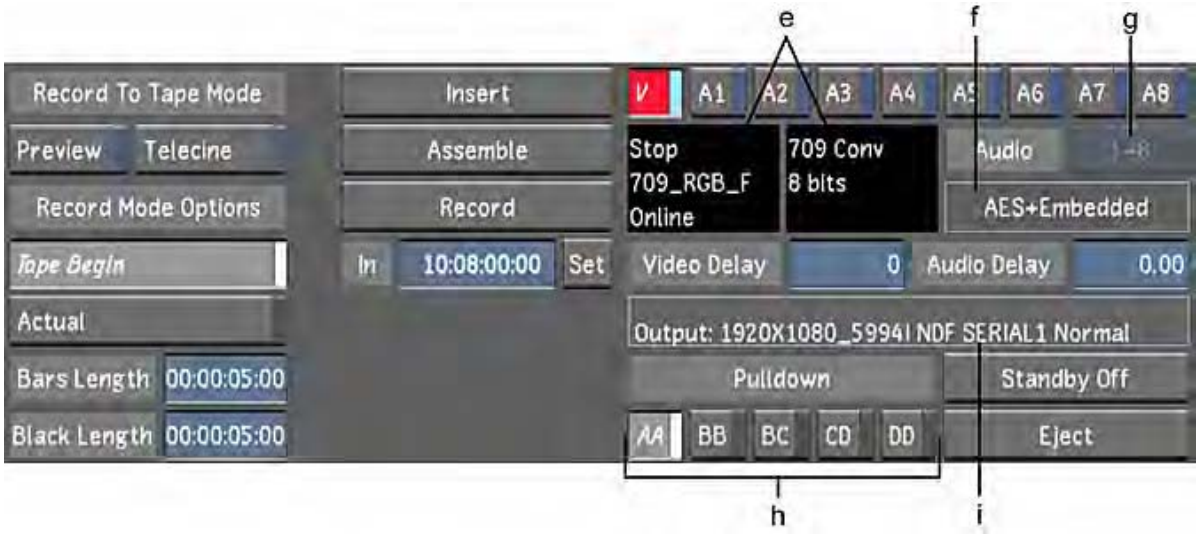


Playout Menu Options

Use the Playout menu to play out media to a VTR.

The following two graphics represent the Playout menu.





(a) Video/Graphics Raster option box (b) Raster list (c) Format option box (d) Link Type option box (e) VTR Status panel (f) Audio Output status (g) Audio Track Selection option box (h) Pulldown Field buttons (i) Playout Error status

Video/Graphics option box Toggle between the video card (AVIO) and the graphics card (GFX/SDI). The Raster list is then populated with rasters for the selected card.

Raster list With a video card selected in the Video option box (AVIO), select one of the listed rasters to set the resolution and timing for capture. See Supported Video SDI and GFX SDI Rasters (page 2298).

Format option box Toggle to display only rasters belonging to a particular footage format in the Raster list.

Select:	To display:
SD	Standard definition rasters.
HD	High-definition rasters.
Film	Film rasters.
Audio	Audio rasters (only applicable to GFX/SDI).

Link Type option box Toggle to display only rasters belonging to a particular link type in the Raster list.

Select:	To display:
Single	The single-link (4:2:2 signal) rasters.
Dual	The dual-link (4:4:4 signal) rasters.
Stereo	The stereoscopy rasters (only applicable to GFX/SDI).

Pulldown button Enable to configure Lustre to add a pulldown frame when playing out. See About 3:2 Pulldown (page 2297).

Full Range button Enable to configure Lustre to play out the full video signal from the video board. If disabled, Lustre plays out only the legal portion of the video signal.

Sync Mode list Select one of the sync modes.

Select:	To:
InternalSync	Set the sync mode to a free-running internal sync (SD and HD).
ExternalSync	Genlock to an analog sync connected to the sync input (SD only).
TrilevelSync	Genlock to a trilevel sync connected to the sync input (HD only).

Source Playout From buttons Select a starting point for the playout.

Enable:	To select:
Begin	The start of the timeline as the starting point for playing out to tape.
Actual	The current position of the positioner as the starting point for playing out to tape.
Markers	The marker position as the starting point for playing out to tape.

NOTE Markers are In/Out marks that you can set to define the playout operation. See Defining In and Out Markers when Playing Out to Tape (page 2282).

Playout From Timeline button When enabled it allows you to play out source material containing secondary grading without having to perform a render first. See Playout from Timeline with GPU Acceleration (page 2295).

Record To Tape Mode buttons Allow you to configure Lustre to display what the edited footage would look like without recording to VTR or to play out the current timeline and send the source timecode to the VTR.

Enable:	To:
Preview	Preview the footage that will be recorded without recording to tape.
Telecine	Record the footage to tape along with its original source timecode and black media between shots. See Writing Telecine-Style Tape (page 2290).

Record Mode Options buttons Define where on the tape Lustre begins recording when using the Record playout mode. For more information about Record Mode options, see Playing Out to a VTR (page 2280).

Enable:	To begin recording:
Tape Begin	At the beginning of the tape.
Actual	At the position of the current frame on the tape.

Bars Length field Defines the duration (in seconds) of the colour bars recorded to the video tape before the recorded footage begins.

Black Length field Defines the duration (in seconds) of the leader black recorded to the video tape before the recorded footage begins.

Insert button Initiates recording when inserting video and/or audio onto a tape that already contains material.

Assemble button Initiates recording of both video and audio signals on all tracks to a tape when you need to record beyond the point where the valid control track ends.

Record button Initiates recording of both video and audio signals on all tracks when recording an entire cut to tape.

NOTE Use the Record button to write to a new tape with no content. When you record in Record mode, Lustre writes bars and black media before the playout.

In Point field Enter the timecode of the tape where the recording will begin.

NOTE When you configure the in point, the out point is automatically set to the timecode that equals to sum of the clip duration added to the in point.

In Point Set button Sets the current timecode of the tape as the in point.

V button Enable to play out the video track when playing out to a video source. Disable to play out audio only.

A1 - A16 buttons Enable the audio tracks (A1-A16) for playout.

NOTE You can only specify specific video and audio tracks to play out if you are in Insert mode.

VTR Status panel Displays VTR status messages when Lustre is connected to a VTR.

Audio Track Selection option box Used in correlation with the A1-A16 buttons. Toggle to determine whether tracks 1-8 or 9-16 are visible. Tracks 9-16 are only available with embedded AJA audio. If only eight audio tracks are available, this option is disabled (greyed out).

Audio Output status Audio is always outputted to both AES and Embedded audio.

Video Delay field Enter a frame number to delay the video playout for that number of fields. For example, if you set the field to 10, playout begins 10 frames after the in point. If you set the field to -10, playout begins 10 frames before the in point.

Audio Delay field Enter a frame number to delay the audio playout for that number of fields. For example, if you set the field to 10, audio playout begins 10 frames after the in point. If you set the field to -10, audio playout begins 10 frames before the in point.

NOTE The Video and Audio Delay fields compensate for frame delays that may exist while transferring shots from Lustre to the output device.

Playout Error status Displays a message if an error occurs while performing a playout.

Pulldown Field buttons Enable a Pulldown Field button to select the frame used for the 3:2 pulldown. Only available when Pulldown is enabled. See About 3:2 Pulldown (page 2297).

Standby Off button Removes the VTR heads from the tape.

Eject button Ejects the tape in the VTR.

Playing Out to a VTR

Use the Playout tool to play out frames from Lustre so that they can be recorded to an external device.

NOTE If you play out Telecine-style to tape, you can only play out in Assemble mode.

In the majority of cases, you will play out rendered frames to a VTR. See Rendering Shots (page 2216). There are exceptions:

- The creation of real-time deliverables using source material. See Creating Real-Time Deliverables from the Timeline or Rendered Material (page 2291).
- GPU playout from the timeline. See Playout from Timeline with GPU Acceleration (page 2295).

There are three modes for playing out to a VTR:

- In Insert mode, you record video only, audio only, or a combination of both, to a tape that already contains material (either pre-striped with black or actual video material with a valid control track).
- In Assemble mode, you can add video to the end of previously-recorded material.
- In Record mode, you can play out video to a blank tape or a tape that contains existing material. Everything on the tape is overwritten when recording.

For some video formats, you can add 3:2 pulldown during the playout operation. See About 3:2 Pulldown (page 2297).

Playout Workflow

The recommended steps for playing out video to a VTR are as follows.

Step:	Refer to:
1. Render the timeline in the appropriate resolution for the VTR to which you want to play out.	Rendering Shots (page 2216).
2. Use the transport controls to cue up the VTR to the in point.	Controlling the VTR (page 2262).
3. (Optional) Cue up the timeline.	Navigating through Shots (page 1975).
4. Select a playout mode.	Playing Out to a VTR (page 2280).

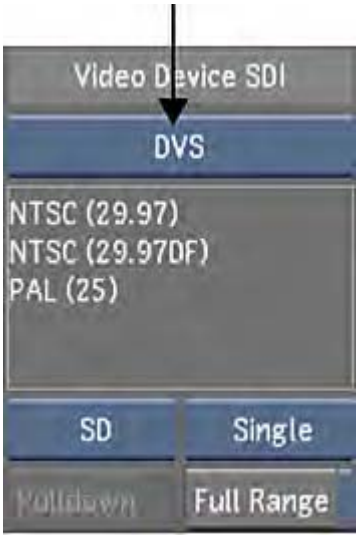
Selecting a Raster for Playout

When playing out to an external device, you must select a raster. See Supported Video SDI and GFX SDI Rasters (page 2298).

NOTE When you select a drop frame (DF) video format, the timecodes are calculated in Drop Frame mode.

To select a raster for playout:

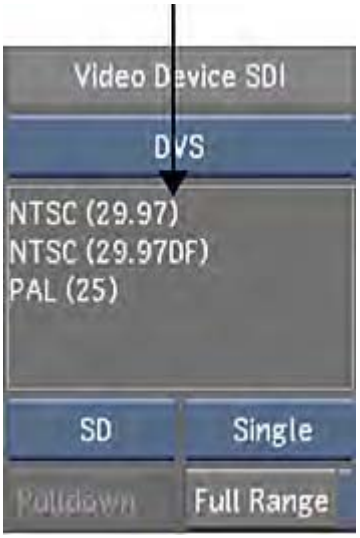
1 In the Video Device SDI group, toggle the Video/Graphics raster option box to display your video card (AVIO). See Playout Menu Options (page 2277).



2 Select an option from the Link Type box. Select Single for a 4:2:2 video signal, and Dual for a 4:4:4 video signal.

3 Toggle the Format option box to the required raster format (SD, HD, Film, or Audio).

4 In the Raster list, select the raster for playout.



NOTE If clicking on a raster in the Raster list does not select the item, that format is not available for capture on that system.

5 (Optional) If you need to add 3:2 pulldown frames when playing out, enable Pulldown.

The pulldown is added to the played out video. The first frame played out becomes the AA frame in the pulldown sequence. See About 3:2 Pulldown (page 2297).

Defining In and Out Markers when Playing Out to Tape

Users can create a video tape from a specific segment of the timeline by setting in and out points for playing out to tape. This is done using the same hot keys as for setting in and out points for playing shots in the

Player. A new option has been added to the Editing > Playout menu that lets you select the marked section for playout to tape.

To define in and out markers when playing out to tape:

1 Go to the start and end frame and mark the in and out points.

Press:	To:
Shift+I	Mark an in point.
Shift+O	Mark an out point.
Shift+L	Clear in and out points.

2 Click Editing, and then click Playout to display the Playout menu.

3 Click Markers.



NOTE If you enable an in point only, playout begins at that point and goes to the end of the timeline. If you enable an out point only, playout begins at the start of the timeline and ends at the out point.

Recording in Insert Mode

Use Insert mode when you want to insert video only, audio only, or both video and audio onto a tape that already contains material. The entire area to which you are recording must have a valid, continuous control track, as this mode only plays out the video (existing timecode and audio are not affected).

NOTE When recording to tape, you can only configure specific video and audio tracks to play out if you are in Insert mode.

To record in Insert mode:

1 Set the VTR to Regen timecode.

2 Before you can play out to a VTR, you must render the timeline at the appropriate video resolution. See Rendering Shots (page 2216) and Playout Workflow (page 2281).

3 Click Editing, and then click Playout to display the Playout menu.

4 Select a raster for playout. See Selecting a Raster for Playout (page 2281).

5 In the Sync Mode list, select the synchronization mode. See Playout Menu Options (page 2277).

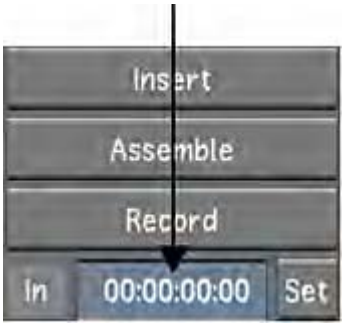
6 (Optional) To record only a portion of the timeline, use markers to define where, in the timeline, you wish to start and stop recording to tape. See Defining In and Out Markers when Playing Out to Tape (page 2282).

7 In the Source Playout From group, determine where the playout operation starts.

Click:	To start the playout:
Begin	From the start of the current timeline.
Actual	At the current frame.
Markers	At the in marker in the timeline, and stop at the out marker.



8 In the In field, enter the tape timecode where the video is to be inserted.



TIP To find the desired timecode more quickly, click the Source Timecode field next to the playback controls while still in the Playout menu. The playback controls and Source Timecode field turns blue to indicate that they can be used to control the VTR. Scrub to the desired timecode, then enter it in the In field.

9 (Optional) In the Record To Tape Mode group, enable Preview.

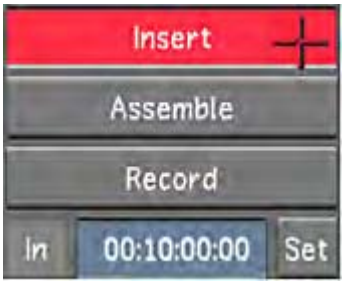


With Preview enabled, your footage is played out in the Player, but nothing is written to tape. The Insert button turns green after the first click to show that the action is non-destructive.

10 (Optional) Configure video and audio record delay. See Playout Menu Options (page 2277).

11 Configure Lustre to record specific video and audio tracks to tape. See Playout Menu Options (page 2277).

12 Click Insert, and then click it again to confirm the action.



TIP To record a single shot as opposed to the entire timeline, select the shot you want to play out and hold down SHIFT while clicking Insert.

The playout operation begins.

Recording in Assemble Mode

Use Assemble mode when recording on a tape that contains a valid control track up until a certain point, but you need to record beyond that point. In Assemble mode, you are attaching video and audio to existing material. For example, yesterday you played out your work onto a blank tape using Record mode. Today, you have completed additional work and would like to add it to the end of the tape. In this case, you cue the tape to the in point (at a point where there is a valid control track) and begin playing out in Assemble mode. The timecode is continued from the previously recorded material.

NOTE When recording to tape, you can only configure specific video and audio tracks to play out if you are in Insert mode.

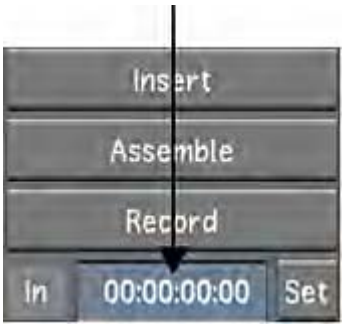
To record in Assemble mode:

1. Set the VTR to Regen timecode.
2. Before you can play out to a VTR, you must render the timeline at the appropriate video resolution. See Rendering Shots (page 2216) and Playout Workflow (page 2281).
3. Click Editing, and then click Playout to display the Playout menu.
4. Select a raster for playout. See Selecting a Raster for Playout (page 2281).
5. In the Sync Mode list, select the synchronization mode. See Playout Menu Options (page 2277).
6. (Optional) To record only a portion of the timeline, use markers to define where, in the timeline, you wish to start and stop recording to tape. See Defining In and Out Markers when Playing Out to Tape (page 2282).
7. In the Source Playout From group, determine where the playout operation starts.

Click:	To start the playout:
Begin	From the start of the current timeline.
Actual	At the current frame.
Markers	At the in marker in the timeline, and stop at the out marker.



8 In the In field, enter the tape timecode where the video is to be edited.



TIP To find the desired timecode more quickly, click the Source Timecode field next to the playback controls while still in the Playout menu. The playback controls and Source Timecode field turns blue to indicate that they can be used to control the VTR. Scrub to the desired timecode, then enter it in the Tape Rec IN field.

9 (Optional) In the Record To Tape Mode group, enable Preview.



With Preview enabled, your footage is played out in the Player, but nothing is written to tape. The Insert button turns green after the first click to show that the action is non-destructive.

10 (Optional) Enable Telecine. See Writing Telecine-Style Tape (page 2290).

11 (Optional) Configure video and audio record delay. See Playout Menu Options (page 2277).

12 Click Assemble, then confirm the action.



TIP To record a single shot as opposed to the entire timeline, select the shot you want to play out and hold down SHIFT while clicking Assemble.

The playout operation begins.

Recording in Record Mode

Use Record mode to record an entire cut to a VTR. Record mode overwrites the entire contents of the tape (including timecode). To use Record mode, you must set the VTR timecode to Preset or Auto timecode.

You do not have to pre-stripe the tape with timecode when using Record mode.

You have the option of specifying a start timecode value, and including leader black and colour bars. Leader black and colour bars are added to the start of the program (on tape).

NOTE In Record mode, it is not possible to record a subset of the total available tracks to tape; all tracks are recorded to tape in this mode.

At least 10 seconds of black are always recorded to the video tape before the footage. Also, footage always starts on the minute. Lustre pads the leader with additional black and colour bars to ensure that the video always begins on the minute. The following examples illustrate how this is calculated.

Example 1

- Start Timecode: 00:59:00:00
- Min Colourbar Length: 15 seconds
- Black Length: 15 seconds

To begin, Lustre always records 10 seconds of black from 00:59:00:00 to 00:59:10:00 (this length is locked and is always recorded).

After the initial black, colour bars are recorded from 00:59:10:00 to 00:59:45:00 (the Minimum Colourbar Length of 15 seconds is surpassed).

Finally, black is recorded from 00:59:45:00 to 01:00:00:00 (the Black Length value of 15 seconds must be respected) and the video material begins exactly at 01:00:00:00.

Example 2

- Start Timecode: 00:59:30:00
- Min Colourbar Length: 20 seconds
- Black Length: 10 seconds

To begin, Lustre always records 10 seconds of black from 00:59:30:00 to 00:59:40:00 (this length is locked and is always recorded).

After the initial black, colour bars are recorded from 00:59:40:00 to 01:00:50:00 (the Minimum Colourbar Length of 20 seconds is surpassed).

Finally, black is recorded from 01:00:50:00 to 01:01:00:00 (the Black Length value of 10 seconds must be respected) and the video material begins exactly at 01:01:00:00.

In this case, the Minimum Colourbar Length and the Black Length (which combine to add up to 30 seconds) does not fit into the gap between 00:59:40:00 and 01:00:00:00 (20 seconds). Because video must always start on the minute, additional colour bars are added so that the material starts at the next minute.

To record to a VTR in Record mode:

1. Before you can play out to a VTR, you must render the timeline at the appropriate video resolution. See Rendering Shots (page 2216) and Playout Workflow (page 2281).
2. Click Editing, and then click Playout to display the Playout menu.
3. Select a raster for playout. See Selecting a Raster for Playout (page 2281).

4 In the Sync Mode list, select the synchronization mode. See Playout Menu Options (page 2277).

5 (Optional) To record only a portion of the timeline, use markers to define where, in the timeline, you wish to start and stop recording to tape. See Defining In and Out Markers when Playing Out to Tape (page 2282).

6 In the Source Playout From group, determine where the playout operation starts.

Click:	To start the playout:
Begin	From the start of the current timeline.
Actual	At the current frame.
Markers	At the in marker in the timeline, and stop at the out marker.



7 (Optional) In the Record To Tape Mode group, enable Preview.



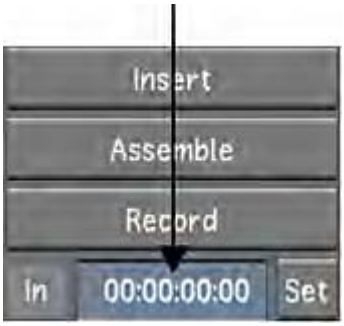
With Preview enabled, your footage is played out in the Player, but nothing is written to tape.

8 In the Record Mode Options group, determine where on the video tape you want to record to:

- Enable Tape Begin to rewind to the start of the tape, and then begin the playout.
- Enable Actual to start the playout operation at the current frame of the tape.



9 Enter the start timecode in the In field.



10 Enter the minimum colour bar length and the minimum Black length in the Bars Length and Black Length fields, respectively.



11 (Optional) Configure the video and audio record delay. See Playout Menu Options (page 2277).

12 Click Record, and then click it again to confirm the action.



The playout operation begins.

VTR Emulation

When configured to emulate a VTR, Lustre can be controlled, using an RS-422 interface, by a third-party application or device. You control Lustre VTR emulation from the application or device that treats Lustre as a VTR device. Lustre supports SD and HD emulators.

The RS-422 commands that you can use for VTR emulation are summarized as follows:

- Play
- Stop
- REW
- FF
- PREROLL
- PREROLL TIME PRESET
- SHUTTLE
- JOG
- VARPLAY
- Reset TC IN
- Reset TC OUT
- Mark TC IN
- Mark TC OUT
- Set TC IN
- Set TC OUT
- AUTO mode ON
- AUTO mode OFF
- TC IN+
- TC IN-
- TC OUT+
- TC OUT-
- CUE to (given) TC
- CUE to TC OUT
- EDIT channels preset
- Eject

NOTE Unless otherwise configured by the controller device, the pre-roll time on the Lustre VTR emulator defaults to seven seconds with each video initialization.

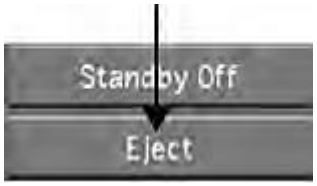
To operate Lustre as a VTR:

1. Connect Lustre to the breakout box by doing one of the following:
   - If Lustre is running on the IBM® IntelliStation® Z Pro 6223 or the HP® xw8400, connect to the B port on the DVS Centaurus breakout box.
   - If Lustre is running on the HP xw8600, connect to the B port on the AJA breakout box.
2. Connect the master application or device to Lustre on the patch panel using a straight pin-to-pin serial cable. A cross-cable will not work.

   WARNING Save your grade before editing your project, or you will lose any unsaved work.
3. Launch Lustre and navigate to the Engineering page of the Project Management settings (See Engineering Settings (page 1749)).
4. Enable VTR Emulator.
5. Use the Offset in Frames slider to set the number of frames you wish to shift ahead as footage is transferred from the controlling device to Lustre. Enter a negative number of frames to shift the clip backwards.
6. Click Save Project.
7. Click Editing, and then click Playout to display the Playout menu.
8. Select a raster for playout. See Selecting a Raster for Playout (page 2281).
9. Prior to playout from the VTR emulator, perform the following tasks:
   - Add a header of at least five seconds to the start of the Storyboard.
   - Verify that the start Timecode begins at the end of the header, and not at 00:00:00:00. This is necessary because the Lustre VTR emulation does not provide a pre-roll outside of the boundaries of the existing Storyboard.

To disable VTR emulation:

1. Click Eject.



2. To re-initialize, either switch between the Editing Capture and Playout menus, or re-select the DVS raster.

Writing Telecine-Style Tape

The Telecine feature allows you to emulate grading with telecine tape. The original source timecodes are written to tape, and each shot is separated by a black sequence of duration equal to Black Length value field.

Before you can use the Telecine feature, you must configure the VTR and the breakout box.

To configure the VTR:

1 Set the VTR's timecode configuration to:
   - Record run.
   - Preset Timecode Generation.

   NOTE The timecode data is transferred through the RS-422 connection.

To output telecine-style tape:

1 Enable Telecine prior to starting the playout operation.



2 Sort the timeline in C mode (with or without Head/Tail) using the C-Mode option found in the Editing > Assemble > Sort Modes menu.

3 (Optional) To resize *rendered* 2K or HD content to NTSC or PAL resolutions:
   - Select an NTSC or PAL raster.
   - Enable the Pulldown option.

4 Click Assemble once, and then a second time to confirm the action.

Playout and GPU Acceleration

It is possible to play out real-time content to tape without having to first process by using the power of the graphical processing unit, or GPU.

- Use Real-time deliverables. See Creating Real-Time Deliverables from the Timeline or Rendered Material (page 2291).
- Playout from the timeline. See Playout from Timeline with GPU Acceleration (page 2295).

Creating Real-Time Deliverables from the Timeline or Rendered Material

You can record 10-bit, multi-format video deliverables to tape from existing source or rendered footage without having to perform additional render passes. These deliverables are processed in real time and are subject to high-quality resizing.

NOTE Support for real-time deliverables depends on the version of the graphics card installed on your system.

Real-time deliverables can include specific primary grading (excluding curves), most reposition types, and hardware LUTs. Animation on primary grading, and on panning and scanning is also available and is performed in real time during output to tape. Modifications for real-time deliverables are displayed in Deliverable (D) view and do not affect the master version's Output view or rendered version's Print view. When playing out, you can use either source or rendered material.

NOTE Real-time deliverables playout cannot include output primary grading, secondary grading, or rotations. These types of modifications must be rendered or copied to the real-time deliverables module prior to playout. See Sharing Modifications Between Setups and Grades (page 2295).

For example, you could use 2K rendered footage to create an HD tape and a DVD version without having to render again. Alternately, you could use source material and apply primary grading and repositioning that only affects the real-time deliverable, and does not require a render pass prior to playing out.

Creating Setups for Real-Time Deliverables

After you switch to Real-Time Deliverables mode, the work you perform on your shot is no longer saved to the grade. Instead, it is saved to a real-time deliverables setup file. This file contains grading and reposition information that is separate from the grade file. The setup file is an XML-based grade file, which allows you to use it with other systems.

NOTE You can share grading between real-time deliverables setups and grade files. See Sharing Modifications Between Setups and Grades (page 2295).

Although real-time deliverables setups are linked to the grade file, you can apply a setup to a grade linked with a different cut.

After the setup has been created and loaded to a grade, you can switch back to the Playout menu and play out to tape in either Insert or Assemble mode.

TIP Enabling the D button also allows you to send a 10-bit buffer to the SDI output.

To create a real-time deliverables setup:

1. In the Main menu, click Setup, and then click Grade.
   The Grade menu appears.
2. From the Grade list, load the grade from which you want to create a real-time deliverables setup.



3. Click D.



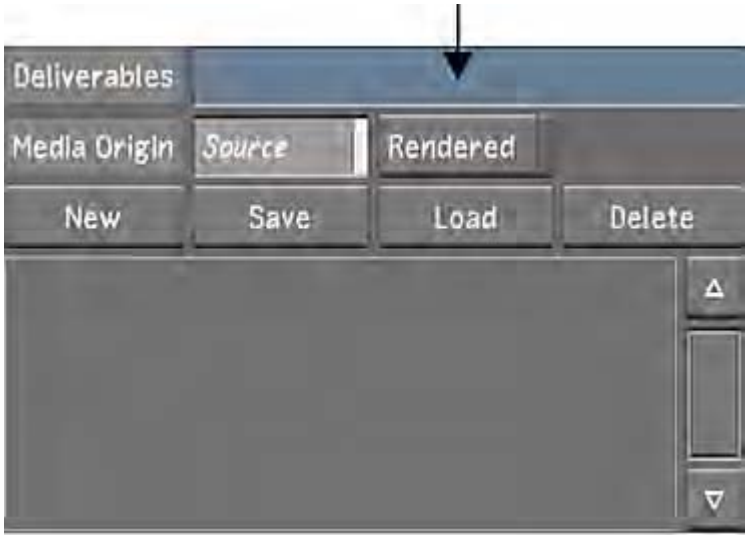
The button automatically switches you to Real-Time Deliverables view mode, and enables GPU processing See GPU Acceleration (page 2043).

NOTE If the screen shows a red X instead of an image, the enabled Media Origin contains no media. The following step tells you what to do.

4 Select the media origin to be used during playout. Enable Source for original, unrendered material. Enable Rendered for rendered material.

NOTE Unrendered changes made to source material will not be applied to setups. Also, when you select Source (O scans) as the media origin, Grade file settings are bypassed.

5 In the Deliverables field, enter a name for the setup.



6 Click New. The setup name is added to the list of setups for the grade.

7 (Optional) Perform reposition modifications such as panning and scanning, resizing, and repositioning. See Repositioning Images (page 2016).

NOTE Rotations cannot be included in real-time deliverables setups.

8 (Optional) Perform input primary colour grading. See Primary Colour Grading (page 2058).

NOTE Only input primary grading is permitted for real-time deliverables setups.

9 (Optional) Apply a HW LUT, using a grade file to store it. Real-time deliverables do not save Print Lut reference. See Applying LUTs for Viewing (Print LUTs) (page 1853).

NOTE When creating real-time deliverables setups, you can only apply a HW LUT to the main monitor. In other words, you can only enable the button labelled 1.

10 Return to the Setup Grade menu.

11 Click Save to save the setup.

The setup is saved to the scene's Library folder as an XML file. The file uses the real-time deliverables setup name, preceded by the name of the grade on which it is based. For example, if the grade name is *grd01*, and the setup name is *My_HD*, the XML grade file is *[grd01]My_HD.deliv.xml*.

TIP You can save different versions of the same setup according to your mastering requirements.

Once a setup exists, it can either be loaded to the grade for which it was created, or loaded to a different grade.

To load a setup to the grade for which it was created:

1 In the Deliverables section of the Grade menu, select a setup from the list and click Load.
This loads both the grade and the setup.

To load a setup to a different grade:

1 In the Grade menu, load the grade on which you want to apply the existing deliverables.
2 Select a setup from the list, and press `Ctrl` while clicking Load.
This loads the setup to the grade.

To rename a setup:

1 Middle-click the setup name.
2 Enter the new name, and then press `Enter`.

To delete a setup:

1 Select a setup from the list, and click Delete.

Resizing Shots Prior to Playing Out

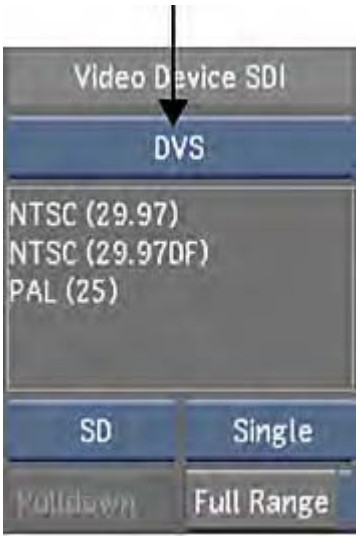
Before you play out real-time deliverables, resize your shots to ensure that they properly conform to the destination output format. See Resizing Shots (page 2218).

Playing Out Real-Time Deliverables to Tape

After a setup has been created and loaded to the desired grade, and resizing has been performed, you are ready to play out to tape.

To play out real-time deliverables to tape:

1 In the Main menu, click Editing, and then click Playout.
The Playout menu appears.
2 Toggle the Video/Graphics raster option box to display the graphics card (GFX/SDI).



3 Toggle the Link Type option box to the required link type (Single or Dual).

4 Toggle the Format option box to the required raster format (SD, HD, Film, or Audio).

5 In the Raster list, select a raster.

6 Play out to tape in either Insert or Assemble mode. See Recording in Insert Mode (page 2283) and Recording in Assemble Mode (page 2285).

Sharing Modifications Between Setups and Grades

By default, unrendered changes to a grade file are not applied to real-time deliverables setups, and modifications made to a setup are not copied back to the grade. However, you can force changes to be copied back and forth either on a shot-by-shot basis or for the entire scene.

To share modifications between setups and grades:

1 In the Grade menu, click one of the Deliverables buttons.



Click:	To transfer:
Copy In	Modifications made to the current shot from the grade to the setup.
Copy In All	Modifications made to the scene from the grade to the setup.
Copy Out	Modifications made to the current shot from the setup to the grade.
Copy Out All	Modifications made to the scene from the setup to the grade.

Playout from Timeline with GPU Acceleration

NOTE When playing out from source, playback performance may be impacted depending on the grading applied to the current shot. If frames are dropped during play out, Lustre recues the VTR to the last known frame laid down to tape.

You can now use GPU acceleration to play out source material in real time with Print LUT, primary and secondary grading, and GPU-accelerated plugins applied. While Real-Time Deliverables mode allows you to play out directly to tape, it only permits primary grading, resizing, pan and scan, and LUT processing. The Playout From Timeline option lets you play out source material containing secondary grading without having to first perform a rendering pass.

By enabling the Playout From Timeline function in the project settings or the Playout menu, and by enabling GPU acceleration, you can play out from Output view mode and use all features currently supported by enhanced GPU acceleration.

NOTE If you enable the Playout From Timeline function in project settings, playing out from the Output view is the default for this project. If you enable the Playout From Timeline function from within the Playout menu, this setting applies only for the current session.

- For more information about GPU acceleration, see GPU Acceleration (page 2043).
- For more information about Output view mode, see Setting Image View Options (page 1973).

If you want to play out at a different resolution than your timeline, use the real-time deliverable solution. But if you want to play out content with complex grading, use Play out from Timeline, in which case you are limited to using the same resolution as the source.

To play out source material to tape with all features supported by GPU acceleration mode:

1. Enable the Playout From Timeline setting by doing one of the following:
   - In your project settings, enable Playout from Timeline in the Engineering menu. See Engineering Settings (page 1749).
   - In the Playout menu, enable Playout from Timeline. See Playout Menu Options (page 2277).

   NOTE When you define settings in Project Management, these settings become the default settings whenever this project is loaded. When you define settings in the application (that is, not in the project settings), Lustre reverts to the project setting defaults upon reboot.

2. Set the Player to the O view by clicking O.



   NOTE You must set the Player to the O view to be able to record unrendered media to tape and to use GPU acceleration.

3. Enable GPU acceleration by clicking GPU or by pressing Y on your keyboard.
4. Play out to the VTR. See Playing Out to a VTR (page 2280).

Expected Performance

Real-time playback is not guaranteed when you play out directly to tape with GPU acceleration. The playback speed indicator in the GUI is a good indicator of what to expect when you lay down the timeline to tape. If frames are dropped when you play back your timeline on the SDI monitor, the same is likely to occur when you play out to tape.

You can expect good playback with no dropped frames when you ensure the following:

- HD or SD format is used for both capture and output (no resize applied).
- Up to four secondaries with keys are used, with a maximum of one geometry per secondary.
- A 3D Mesh LUT is enabled.
- GFX/SDI is enabled in the Playout menu.

Regardless of whether or not you conform to all conditions stated above, some animations may not play back in real time.

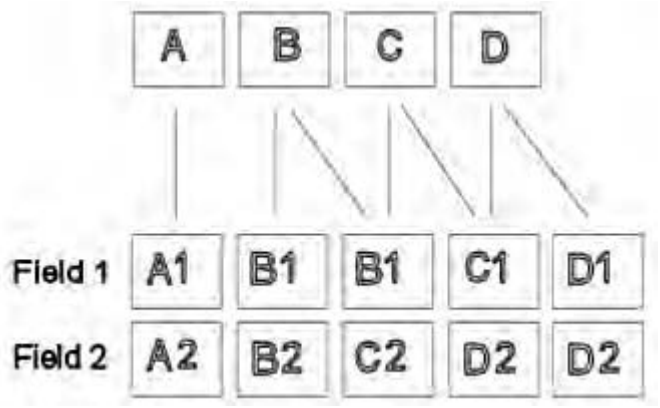
NOTE Support for direct playout to tape from GPU acceleration depends on the version of the graphics card installed on your system.

About 3:2 Pulldown

When capturing from or playing out to a VTR, you may want to convert footage using a 3:2 pulldown raster if the incoming video is NTSC or 1080i HD.

When film, which plays at 24 fps, is telecined to either NTSC or 1080i HD video, which plays at 29.97 fps, an extra 'pulldown frame' is added for every four film frames. This is because a straight 1:1 playback of film frames at NTSC or interlaced HD speeds results in a faster than appropriate playback. The telecine machine used to transfer film to video adds the pulldown frame by taking four consecutive film frames (A, B, C, and D) and performing a telecine to five interlaced video frames (A1A2, B1B2, B1C2, C1D2, and D1D2). Consequently, the pulldown insertion process creates two hybrid frames (B1C2 and C1D2), as shown in the following diagram.

NOTE The '1' and '2' suffixes after the frame letters denote the field number in each interlaced frame of NTSC or 1080i HD video. They do not represent a sequence over time.



When you capture NTSC or 1080i HD video previously telecined from film, you may want to conform with 24 fps progressive footage. In this case, you should capture the NTSC footage and simultaneously remove the 3:2 pulldown frame. When a 3:2 video SDI raster is selected, you can select the first pulldown field in the footage by clicking on the appropriate Pulldown Field button in the Capture menu. Then, the five interlaced NTSC frames are reconfigured into four progressive frames.

When you are ready to record to a VTR, you might need to record to NTSC or 1080i HD video. If your cut runs at 24 fps, you need to configure Lustre to add a 3:2 pulldown frame to bring the framerate up to 29.97 fps. In this case, use the Pulldown Field button in the Playout menu to specify which of the five pulldown fields will be the first in the sequence to be recorded to tape. Lustre will add the additional 3:2 pulldown frame for every four frames starting at the specified field.

Converting Logarithmic to Linear

You can output your film scans to video once you have completed the film grading. There are two general workflows you can use to convert logarithmic images to linear ones:

- Using the Render Viewing option
- Creating manually a Log-to-Lin conversion LUT

To convert logarithmic images to linear ones using the Render Viewing option:

1. Use the same project and the same source footage.
2. Switch to Lin mode. See Setting Logarithmic or Linear Mode (page 1786).
3. Colour grade the footage for video.
4. In the Render > Local menu, enable Viewing in the Rendering group, and then adjust the black, white, and gamma as needed.
5. Render the files in the correct video resolution. See Rendering Shots (page 2216).

   NOTE When Render Viewing is ON, the output render format is set automatically to 8-bit BMP files.
6. Play out the rendered frames to record them on an appropriate VTR.

To convert logarithmic images to linear ones by manually creating a Log-to-Lin conversion LUT:

1. Use the same project and the same source footage.
2. Switch to Lin mode. See Setting Logarithmic or Linear Mode (page 1786).

   When you switch to Lin mode, a Log-to-Lin conversion LUT is applied to the images automatically.
3. If needed, access the Input LUT menu and adjust the parameters of the default Log-to-Lin conversion.
4. Switch off monitor calibration (since the final render will also be seen on a monitor) and turn off any Print LUTs.
5. Use the Render Output menu to adjust the white, black, and gamma as needed.
6. Colour grade the footage for video.
7. Render the files in the correct video resolution. See Rendering Shots (page 2216).
8. Play out the rendered frames to record them on an appropriate VTR.

Supported Video SDI and GFX SDI Rasters

When capturing from or recording to a VTR, you must select the appropriate raster. The raster defines the video capture/playout SDI timing and the graphic SDI timing. Lustre supports many DVS and AJA video SDI rasters and GFX SDI graphics rasters. See Video SDI Rasters (page 2298) and GFX SDI Rasters (page 2301).

In past releases of Lustre, you had to define a colour conversion matrix in a configuration file. You no longer need to do this. The correct conversion matrix is automatically configured when you select a video SDI raster. The matrix name will be displayed in the VTR status box located in the Editing > Capture and Editing > Playout menus.

NOTE You cannot enable both a GFX and a Video raster, except when working with audio, when you can use a DVS/AJA raster for audio, and a GFX for graphics.

Video SDI Rasters

The following table lists the video SDI rasters that are available for capture and playout in Lustre.

NOTE The Lustre HD Station does not support bit depths greater than 10-bit, input resolutions greater than 2K, or output resolutions greater than 1920x1080.

Video Type:	Pulldown:	Link Type:	Video Signal:
NTSC (29.97)	No	Single	29.97 fps, 59.94 Hz, interlaced, 720x486, NDF timecode, YUV 4:2:2 data.
NTSC (29.97)	Yes	Single	29.97 fps, 59.94 Hz, interlaced, 720x486, pulldown insert on playout and removal on capture, NDF timecode, YUV 4:2:2 data.
NTSC (29.97DF)	Yes	Single	29.97 fps, 59.94 Hz, interlaced, 720x486, DF timecode, YUV 4:2:2 data.
NTSC (29.97DF)	No	Single	29.97 fps, 59.94 Hz, interlaced, 720x486, pulldown insert on playout and removal on capture, DF timecode, YUV 4:2:2 data.
PAL (25)	No	Single	25 fps, 50 Hz, interlaced, 720x576, YUV 4:2:2 data.
720p (50)	No	Single	50 fps, progressive, 1280x720, YUV 4:2:2 data.
720p (50)	No	Dual	50 fps, progressive, 1280x720, RGB 4:4:4 data.
720p (59.94)	No	Single	59.94 fps, progressive, 1280x720, NDF timecode, YUV 4:2:2 data.
720p (59.94DF)	No	Single	59.94 fps, progressive, 1280x720, DF timecode, YUV 4:2:2 data.
720p (59.94)	No	Dual	59.94 fps, progressive, 1280x720, NDF timecode, RGB 4:4:4 data.
720p (59.94DF)	No	Dual	59.94 fps, progressive, 1280x720, DF timecode, RGB 4:4:4 data.
720p (60)	No	Single	60 fps, progressive, 1280x720, YUV 4:2:2 data.
720p (60)	No	Dual	60 fps, progressive, 1280x720, RGB 4:4:4 data.
1080PsF (23.98)	No	Single	23.98 fps, 72 Hz, segmented frames, 1920x1080, YUV 4:2:2 data.
1080PsF (23.98)	No	Dual	23.98 fps, 72 Hz, segmented frames, 1920x1080, RGB 4:4:4 data.
1080p (23.98)	No	Single	23.98 fps, progressive, 1920x1080, YUV 4:2:2 data

Video Type:	Pulldown:	Link Type:	Video Signal:
1080p (23.98)	No	Dual	23.98 fps, progressive, 1920x1080, RGB 4:4:4 data
1080PsF (24)	No	Single	24 fps, 48 Hz, segmented frames, 1920x1080, YUV 4:2:2 data.
1080PsF (24)	No	Dual	24 fps, 48 Hz, segmented frames, 1920x1080, RGB 4:4:4 data.
1080p (25)	No	Single	25 fps, progressive, 1920x1080, YUV 4:2:2 data
1080p (25)	No	Dual	25 fps, progressive, 1920x1080, RGB 4:4:4 data
1080p (29.97)	No	Single	29.97 fps, progressive, 1920x1080, NDF timecode, YUV 4:2:2 data
1080p (29.97)	No	Dual	29.97 fps, progressive, 1920x1080, NDF timecode, RGB 4:4:4 data
1080p (29.97DF)	No	Single	29.97 fps, progressive, 1920x1080, DF timecode, YUV 4:2:2 data
1080p (29.97DF)	No	Dual	29.97 fps, progressive, 1920x1080, DF timecode, RGB 4:4:4 data
1080p (30)	No	Single	30 fps, progressive, 1920x1080, NDF timecode, YUV 4:2:2 data
1080p (30)	No	Dual	30 fps, progressive, 1920x1080, NDF timecode, RGB 4:4:4 data
1080i (25)	No	Single	25 fps, 50 Hz, interlaced, 1920x1080, YUV 4:2:2 data.
1080i (25)	No	Dual	25 fps, 50 Hz, interlaced, 1920x1080, RGB 4:4:4 data, RGB 4:4:4 data.
1080i (29.97)	No	Single	29.97 fps, 59.94 Hz, interlaced, 1920x1080, NDF timecode, YUV 4:2:2 data.
1080i (29.97)	No	Dual	29.97 fps, 59.94 Hz, interlaced, 1920x1080, NDF timecode, RGB 4:4:4 data.
1080i (29.97DF)	No	Single	29.97 fps, 59.94 Hz, interlaced, 1920x1080, DF timecode, YUV 4:2:2 data.
1080i (29.97DF)	No	Dual	29.97 fps, 59.94 Hz, interlaced, 1920x1080, DF timecode, RGB 4:4:4 data.

Video Type:	Pulldown:	Link Type:	Video Signal:
1080i (29.97)	Yes	Single	29.97 fps, 59.94 Hz, interlaced, 1920x1080, pull-down insert on playout and removal on capture, NDF timecode, YUV 4:2:2 data.
1080i (29.97)	Yes	Dual	29.97 fps, 59.94 Hz, interlaced, 1920x1080, pull-down insert on playout and removal on capture, NDF timecode, RGB 4:4:4 data.
1080i (29.97DF)	Yes	Single	29.97 fps, 59.94 Hz, interlaced, 1920x1080, pull-down insert on playout and removal on capture, DF timecode, YUV 4:2:2 data.
1080i (29.97DF)	Yes	Dual	29.97 fps, 59.94 Hz, interlaced, 1920x1080, pull-down insert on playout and removal on capture, DF timecode, RGB 4:4:4 data.
1080i (30)	No	Single	30 fps, 60 Hz, interlaced, 1920x1080, NDF timecode, YUV 4:2:2 data.
1080i (30)	No	Dual	30 fps, 60 Hz, interlaced, 1920x1080, NDF timecode, RGB 4:4:4 data.
1080i (30)	Yes	Single	30 fps, 60 Hz, interlaced, 1920x1080, pulldown insert on playout and removal on capture, NDF timecode, YUV 4:2:2 data.
1080i (30)	Yes	Dual	30 fps, 60 Hz, interlaced, 1920x1080, pulldown insert on playout and removal on capture, NDF timecode, RGB 4:4:4 data.
1556PsF (14/HSDL)	No	Single	14 fps, segmented frame, 2048x1556, allows HSDL transfer, YUV 4:2:2 data.
1556PsF (15/HSDL)	No	Single	15 fps, segmented frame, 2048x1556, allows HSDL transfer, YUV 4:2:2 data.
1556PsF (14/HSDL)	No	Dual	14 fps, segmented frame, 2048x1556, allows HSDL transfer, RGB 4:4:4 data.
1556PsF (15/HSDL)	No	Dual	15 fps, segmented frame, 2048x1556, allows HSDL transfer, RGB 4:4:4 data.

GFX SDI Rasters

The following table lists the GFX SDI rasters that are available for capture and playout in Lustre.

NOTE The Lustre HD Station does not support bit depths greater than 10-bit, input resolutions greater than 2K, or output resolutions greater than 1920x1080.

Video Type:	Pulldown:	Link Type:	Stereo:	Video Signal:
NTSC (29.97)	No	Single	No	29.97 fps, 59.94 Hz, interlaced, 720x486, NDF timecode, YUV 4:2:2 data.
NTSC (29.97)	Yes	Single	No	29.97 fps, 59.94 Hz, interlaced, 720x486, pulldown insert on playout and removal on capture, NDF timecode, YUV 4:2:2 data.
PAL (25)	No	Single	No	25 fps, 50 Hz, interlaced, 720x576, YUV 4:2:2 data.
720p (60)	No	Single	No	60 fps, progressive, 1280x720, YUV 4:2:2 data.
720p (60)	No	Dual	No	60 fps, progressive, 1280x720, RGB 4:4:4 data.
720p (59.94)	No	Single	No	59.94 fps, progressive, 1280x720, NDF timecode, YUV 4:2:2 data.
720p (59.94)	No	Dual	No	59.94 fps, progressive, 1280x720, NDF timecode, RGB 4:4:4 data.
1080PsF (23.98)	No	Single	No	23.98 fps, 72 Hz, segmented frames, 1920x1080, YUV 4:2:2 data.
1080PsF (23.98)	No	Dual	No	23.98 fps, 72 Hz, segmented frames, 1920x1080, RGB 4:4:4 data.
1080PsF (23.98)	No	Dual	Yes	23.98 fps, 72 Hz, segmented frames, 1920x1080, RGB 4:4:4 data.
1080PsF (24)	No	Single	No	24 fps, 48 Hz, segmented frames, 1920x1080, YUV 4:2:2 data.
1080PsF (24)	No	Dual	No	24 fps, 48 Hz, segmented frames, 1920x1080, RGB 4:4:4 data.
1080PsF (24)	No	Dual	Yes	24 fps, 48 Hz, segmented frames, 1920x1080, RGB 4:4:4 data.
1080PsF (25)	No	Dual	Yes	25 fps, 48 Hz, segmented frames, 1920x1080, RGB 4:4:4 data.

Video Type:	Pulldown:	Link Type:	Stereo:	Video Signal:
1080PsF (30)	No	Dual	Yes	30 fps, 48 Hz, segmented frames, 1920x1080, RGB 4:4:4 data.
1080p (23.98)	No	Dual	Yes	23.98 fps, progressive, 1920x1080, NDF timecode, RGB 4:4:4 data.
1080p (24)	No	Single	No	24 fps, progressive, 1920x1080, NDF timecode, YUV 4:2:2 data.
1080p (24)	No	Dual	No	24 fps, progressive, 1920x1080, NDF timecode, RGB 4:4:4 data.
1080p (24)	No	Dual	Yes	24 fps, progressive, 1920x1080, NDF timecode, RGB 4:4:4 data.
1080p (25)	No	Dual	Yes	25 fps, progressive, 1920x1080, NDF timecode, RGB 4:4:4 data.
1080p (29.97)	No	Dual	Yes	29.97 fps, progressive, 1920x1080, NDF timecode, RGB 4:4:4 data.
1080PsF (25)	No	Single	No	25 fps, 50 Hz, segmented frames, 1920x1080, YUV 4:2:2 data.
1080PsF (25)	No	Dual	No	25 fps, 50 Hz, segmented frames, 1920x1080, RGB 4:4:4 data.
1080i (25)	No	Single	No	25 fps, 50 Hz, interlaced, 1920x1080, YUV 4:2:2 data.
1080i (25)	No	Dual	No	25 fps, 50 Hz, interlaced, 1920x1080, RGB 4:4:4 data, RGB 4:4:4 data.
1080i (25)	No	Dual	Yes	25 fps, 50 Hz, interlaced, 1920x1080, RGB 4:4:4 data, RGB 4:4:4 data.
1080p (29.97)	No	Single	No	29.97 fps, progressive, 1920x1080, NDF timecode, YUV 4:2:2 data.
1080p (29.97)	No	Dual	No	29.97 fps, progressive, 1920x1080, NDF timecode, YUV 4:2:2 data.
1080i (29.97)	No	Single	No	29.97 fps, 59.94 Hz, interlaced, 1920x1080, NDF timecode, YUV 4:2:2 data.

Video Type:	Pulldown:	Link Type:	Stereo:	Video Signal:
1080i (29.97)	No	Dual	No	29.97 fps, 59.94 Hz, interlaced, 1920x1080, NDF timecode, RGB 4:4:4 data.
1080i (29.97)	No	Dual	Yes	29.97 fps, 59.94 Hz, interlaced, 1920x1080, NDF timecode, RGB 4:4:4 data.
1080i (29.97)	Yes	Single	No	29.97 fps, 59.94 Hz, interlaced, 1920x1080, pulldown insert on playout and removal on capture, NDF timecode, YUV 4:2:2 data.
1080i (29.97)	Yes	Dual	No	29.97 fps, 59.94 Hz, interlaced, 1920x1080, pulldown insert on playout and removal on capture, NDF timecode, RGB 4:4:4 data.
1080i (30)	No	Dual	Yes	30 fps, 59.94 Hz, interlaced, 1920x1080, NDF timecode, RGB 4:4:4 data.
1080PsF (30)	No	Single	No	29.97 fps, segmented frame, 1920x1080, NDF timecode, YUV 4:2:2 data.
1080PsF (30)	No	Dual	No	29.97 fps, segmented frame, 1920x1080, NDF timecode, RGB 4:4:4 data.
1080PsF (30)	Yes	Single	No	29.97 fps, 59.94 Hz, segmented frame, 1920x1080, pulldown insert on playout and removal on capture, YUV 4:2:2 data.
1080PsF (30)	Yes	Dual	No	29.97 fps, 59.94 Hz, segmented frame, 1920x1080, pulldown insert on playout and removal on capture, RGB 4:4:4 data.
1080i (30)	No	Single	No	30 fps, 60 Hz, interlaced, 1920x1080, NDF timecode, YUV 4:2:2 data.
1080i (30)	No	Dual	No	30 fps, 60 Hz, interlaced, 1920x1080, NDF timecode, RGB 4:4:4 data.
1080i (30)	Yes	Single	No	30 fps, 60 Hz, interlaced, 1920x1080, pulldown insert on playout and removal on capture, NDF timecode, YUV 4:2:2 data.
1080i (30)	Yes	Dual	No	30 fps, 60 Hz, interlaced, 1920x1080, pulldown insert on playout and removal

Video Type:	Pulldown:	Link Type:	Stereo:	Video Signal:
				on capture, NDF timecode, RGB 4:4:4 data.
2Kp (23.98)	No	Single	No	23.98 fps, 60 Hz, interlaced, 2048x1080, NDF timecode, YUV 4:2:2 data
2Kp (23.98)	No	Dual	No	23.98 fps, 60 Hz, interlaced, 2048x1080, NDF timecode, RGB 4:4:4 data
2Kp (24)	No	Single	No	24 fps, 60 Hz, progressive, 2048x1080, NDF timecode, YUV 4:2:2 data
2Kp (24)	No	Dual	No	24 fps, 60 Hz, progressive, 2048x1080, NDF timecode, RGB 4:4:4 data
2Ki (25)	No	Single	No	25 fps, 60 Hz, interlaced, 2048x1080, NDF timecode, YUV 4:2:2 data
2Ki (25)	No	Dual	No	25 fps, 60 Hz, interlaced, 2048x1080, NDF timecode, RGB 4:4:4 data
2Kp (25)	No	Single	No	25 fps, 60 Hz, progressive, 2048x1080, NDF timecode, YUV 4:2:2 data
2Kp (25)	No	Dual	No	25 fps, 60 Hz, progressive, 2048x1080, NDF timecode, RGB 4:4:4 data
2Ki (29.97)	No	Single	No	29.97 fps, 60 Hz, interlaced, 2048x1080, NDF timecode, YUV 4:2:2 data
2Ki (29.97)	No	Dual	No	29.97 fps, 60 Hz, interlaced, 2048x1080, NDF timecode, RGB 4:4:4 data
2Kp (29.97)	No	Single	No	29.97 fps, 60 Hz, progressive, 2048x1080, NDF timecode, YUV 4:2:2 data
2Kp (29.97)	No	Dual	No	29.97 fps, 60 Hz, progressive, 2048x1080, NDF timecode, RGB 4:4:4 data
2Ki (30)	No	Single	No	30 fps, 60 Hz, interlaced, 2048x1080, NDF timecode, YUV 4:2:2 data
2Ki (30)	No	Dual	No	30 fps, 60 Hz, interlaced, 2048x1080, NDF timecode, RGB 4:4:4 data

Video Type:	Pulldown:	Link Type:	Stereo:	Video Signal:
2Kp (30)	No	Single	No	30 fps, 60 Hz, progressive, 2048x1080, NDF timecode, YUV 4:2:2 data
2Kp (30)	No	Dual	No	30 fps, 60 Hz, progressive, 2048x1080, NDF timecode, RGB 4:4:4 data

Audio

There are a few ways you can hear audio during a timeline playback:

- You can import audio files to the system.
- You can capture audio from a video tape recorder (VTR) or audio tape recorder (ATR).
- You can have an external device chase the audio timecode by using the longitudinal timecode (LTC) chase option.

The audio signal is captured and played back from the DVS or AJA breakout box (BOB). For information on connecting the external audio devices, refer to your hardware guide. When you are capturing, playing back, or playing out audio tracks, you can select up to 16 tracks (eight tracks of DVS/AJA AES audio, or 16 tracks of embedded AJA audio) from the track selector at a sampling rate of 48 kHz (16-bit or 24-bit). See Capture Menu Options (page 2258) or Playout Menu Options (page 2277).

NOTE To monitor the audio from the DVS/AJA BOB, the AES audio needs to be converted to an analog audio signal. To do this, use a digital audio mixer or a digital to analog converter (e.g., the Lucid converter). Refer to your hardware guide for information about the Lucid converter.

Audio Workflow

Use the following typical workflow to add audio to your video playback.

Step:	Refer to:
1. Import or capture the audio file.	Importing an Audio File (page 2306), Importing Audio From Wiretap (page 2308), or Capturing an Audio File (page 2309).
2. Playback the selected audio file/tracks with the video.	Playing Back Audio with the Timeline (page 2310) or Playback Using the LTC Chase Option (page 2313).
3. Playing out the audio file/tracks with the video.	Playout of Audio and Video (page 2315).

Importing an Audio File

Lustre can use the audio files that you import into the scene's *Library\audio* directory or embedded in media imported from the Wiretap or Wiretap Gateway server.

NOTE The audio folder is automatically created when you create a scene.

An imported audio file can comprise of a single track or up to 16 interleaved tracks.

There are three ways to import audio into Lustre:

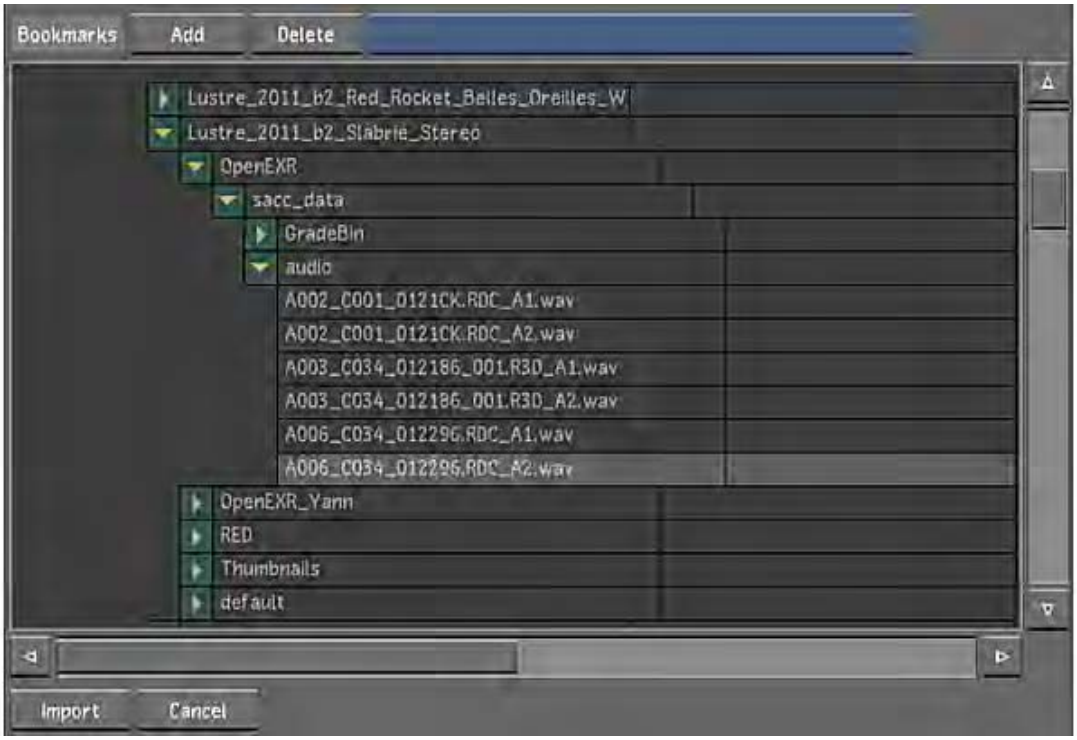
- Outside of Lustre, copy WAV or AIFF files manually to the scene's *Audio* folder.
- Within Lustre, import copies of audio files using the Import file browser.
- Within Lustre, import audio through the Wiretap server and Wiretap Gateway server.

To import an audio WAV or AIFF file while in Lustre:

1. Click Setup in the Main menu, and then click Audio.
2. In the audio file list, click the Browse button.



The Import file browser appears.



3 Navigate to the appropriate source folder and select one or more audio files (You can also use bookmarks (page 1862) to retain a list of folder locations). Traditional methods of multi-selecting (`Shift`-click, `Ctrl`-click) are enabled.

4 Click Import.

A copy of the audio file is imported into the *Audio* folder, and appears in the audio file list.

NOTE If you add audio files to the scene's *Audio* folder after launching Lustre, press `Ctrl+R` to refresh the audio file list.

Importing Audio From Wiretap

You have the option of including audio tracks when you import content containing audio from the Wiretap server. Since the content contains raw audio, you can import the audio tracks as a WAV (including multi-channel WAVE Extensible files) or AIFF file (the bit depth of the file is preserved and the Single File Capture option is not applicable). To set audio file type, see Capture Menu Options (page 2258).

You can also import audio embedded in compressed media files from the Wiretap Gateway server as a WAV or AIFF file. When importing multiple files, only the audio track(s) for one of these files can be used within Lustre since Lustre is limited to a single audio file (or group of files) per grade.

NOTE You cannot import audio only clips.

To import audio files using Wiretap:

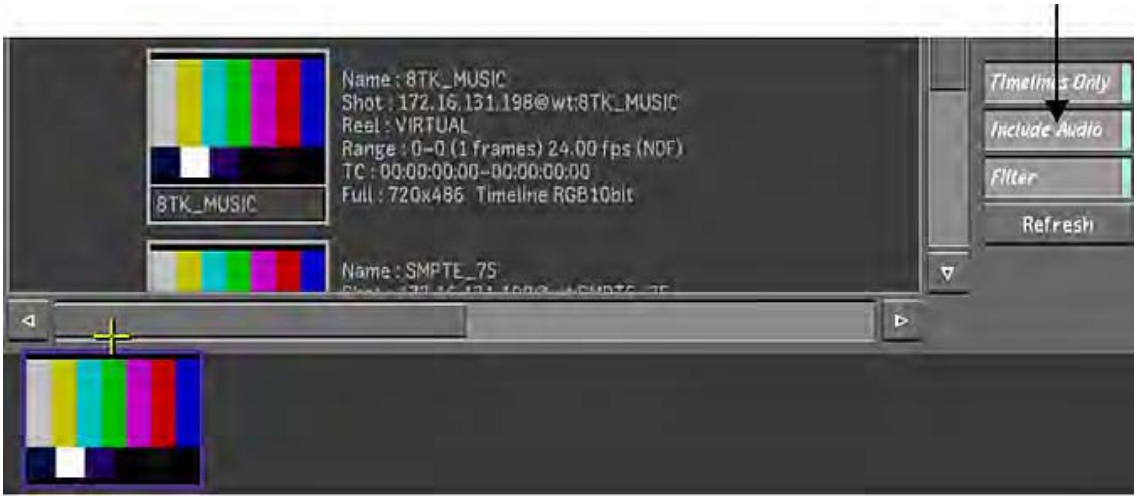
TIP All audio soft effects and dissolves must be rendered on a Visual Effects and Finishing application prior to being imported into Lustre.

1 Click Editing in the Main menu, and then click Browse.

2 Complete one of the following:

- In the file browser, use the Wiretap server to navigate to a library containing a timeline with audio tracks.
- In the file browser, use the Wiretap Gateway server to navigate to a media file that contains embedded audio tracks.

3 Make sure Include Audio is enabled, then select the file and drag it to the Storyboard.



4 Click Setup in the Main menu, and then click Audio.

The audio file from the Wiretap media has been imported and is automatically loaded and ready to play. If multiple tracks were imported, the tracks are grouped together under the same file name. For example, if the name of the media is **8TK_MUSIC**, Lustre displays the file as **8TK_MUSIC_A[1-8].wav** (where [1-8] represents the number of tracks within the file).



NOTE You can only import audio tracks from the Wiretap server and Wiretap Gateway server. The audio tracks cannot be rendered back.

Capturing an Audio File

You can capture an audio signal of up to 16 tracks as a single 48 kHz (16-bit or 24-bit), interleaved WAV or AIFF file. The audio is saved in the scene's *Audio* folder. If using the Linux version of Lustre, the filename uses forward slashes.

NOTE The audio folder is automatically created when you create a scene.

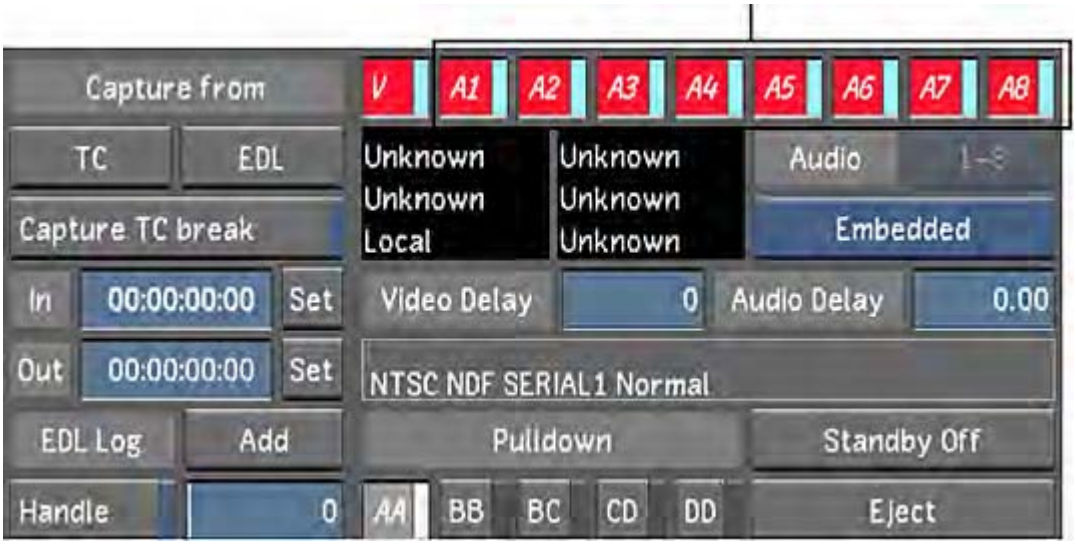
To capture an audio WAV or AIFF file:

1. Click Editing in the main menu, and then click Capture.
2. Select the appropriate raster. See Selecting a Raster for Capture (page 2264).
3. Select the audio file type, bit depth, and determine if you want to capture the audio tracks as a single or multiple files. Single File Capture is enabled by default.



4. Enable the audio track(s) you want to capture.

   NOTE You have the option of capturing the video along with the audio. See Capturing from Timecode (page 2267).



5 Click the TC button to start the capture.

If the audio you are capturing is on numerous tracks and the Single File Capture option is enabled, Lustre groups all the tracks with the same name and displays them as a single file (e.g., **file_name.wav**). If Single File Capture is disabled, each track is saved separately but grouped together in the audio file list. For example, the audio file name will be **file_name_A[#-#].wav**, where the numbers within the brackets represent the number of tracks within the file.

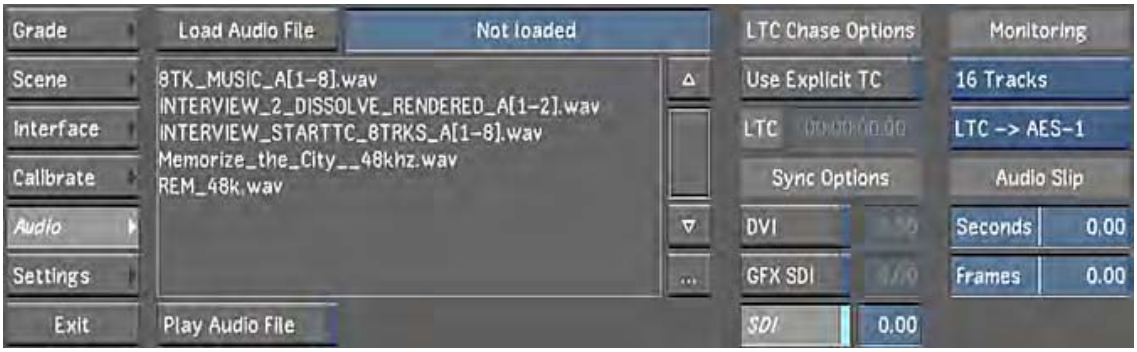
NOTE When you capture a broken selection (e.g., tracks A1, A3, and A5) and Single File Capture is enabled, the resulting single audio file contains three tracks. Therefore, in the Playout menu, tracks A1, A2, and A3 are enabled. If Single File Capture is disabled, the three tracks are saved as three separate files and in the Playout menu, tracks A1, A3, and A5 are enabled.

Playing Back Audio with the Timeline

Once the audio file is in the audio folder, you can play it back at the same time as your timeline.

To play back an imported audio file with the timeline:

1 Click Setup in the Main menu, and then click Audio.
The Audio menu is displayed.



2 (Optional) Import an audio file into the audio file list. See Importing an Audio File (page 2306).

3 Select a file from the audio file list.

NOTE Only one audio file can be associated with the timeline.

4 Click Load Audio File.

The selected audio file is displayed in the audio file field and the Play Audio File button is automatically enabled.



NOTE When you capture an audio file, it is automatically loaded.

5 Select a Monitoring option. See Audio Monitoring Options (page 2311).

6 (Optional) Set an audio offset if the audio is to begin at a certain time before, or after, the video has begun. See Offsetting the Audio File (page 2312).

7 (Optional) Select a Sync option. See Sync Options (page 2312).

Now when you play the timeline, the audio file is played as well.

TIP Disable the Play Audio File button if you do not want to hear the audio during playback.

While the audio file is playing back, you can scrub the audio.

Press:	To:
`Alt`+click location in the shot/scene timebar	Scrub the audio to the location you selected.
`Ctrl+Alt`+drag in the shot/scene timebar	Scrub the audio forward or backward quickly.

Audio Monitoring Options

When you are playing back audio with your video, you need to make sure your playback settings correspond to your audio configuration. Typically, each track is played through a corresponding port (e.g., track 1 passes through port 1, track 2 passes through port 2, etc.), but your system might be configured to have multiple tracks going through a single port (e.g., tracks 1, 3, and 5 pass through port 1). The Monitoring option allows you to set how the tracks are played back.

NOTE If you disable any audio tracks in the Playout menu, it will not be heard during playback.

To select a monitoring option, click the Monitoring option box to select the monitoring option you want.



Select:	Audio Device:	When:
Stereo Downmix	DVS or AJA	There are only two audio tracks that are supported on the audio monitoring device. All the odd numbered tracks (e.g., 1, 3, 5, etc.) go through port 1 and all the even numbered tracks go through port 2.

Select:	Audio Device:	When:
4-Track Downmix	DVS or AJA	There are only four audio tracks that are supported on the audio monitoring device.
8 Tracks	DVS only	There are up to eight tracks and each track is supported on the audio monitoring device.
8-Track Downmix	AJA only	There are only eight tracks that are supported on the audio monitoring device.
16 Tracks	AJA only	There are up to 16 tracks and each track is supported on the audio monitoring device.

NOTE The audio monitoring option can also be set within the Engineering settings tab of the project configuration settings. See Engineering Settings (page 1749).

Offsetting the Audio File

When you playback the audio with the timeline, both the audio and video begin at the same time. If you want your audio to play before, or after, the video has begun, you need to enter an offset for the audio file. Use the Audio Slip option to offset the audio file. For example, if the audio file starts two seconds after the start of the timeline, create a two second offset.

To offset the audio:

1. Do one of the following:
   - Click and drag the Audio Slip sliders to offset the audio. If the audio starts before the video, drag right to set a positive offset. If the audio starts after the video, drag left to set a negative offset. You can use one or both sliders to set the duration of the offset.
   - Right-click within the slider to display the calculator and enter the frames and/or seconds offset to align your audio and video.



Sync Options

Depending upon the output device you are using to view the timeline, the audio and video might not be in sync. Use the Sync Options feature to sync the audio and video. When you enable a sync option, a pre-calculated default is already applied. You only need to enter a value if the audio and video are still not in sync.



DVI Syncs the audio to the video that is displayed on the monitor (DVI).

GFX SDI Syncs the audio to the video that is displayed on an external device (e.g., projector, broadcast monitor, etc.).

SDI Syncs the audio to the video that is displayed on an external device connected to the DVS/AJA BOB.

To sync the audio and video:

1. In the Audio menu enable the Sync Options button corresponding to your output viewing device.

   NOTE Your system can be connected to multiple viewing outputs, but the sync is only applied to the option that is enabled.

2. (Optional) Click within the field and drag the mouse left or right to set the frame value for the sync. Enter a negative number if the audio plays after the video has begun and enter a positive number if the audio plays before the video has begun.

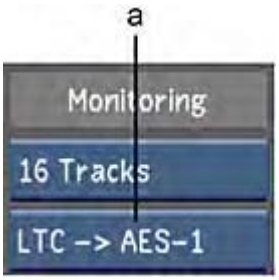
Playback Using the LTC Chase Option

The LTC Chase Options allow you to send the timeline's timecode through the DVS or AJA BOB to an audio device that can chase the LTC timecode. By default, the LTC timecode is the same as the record time (REC TC). For example, if the start timecode for the timeline is 10:00:00:00, the LTC will be 10:00:00:00. When you move the shot positioner to a further position in the timeline and press play, the LTC value is also updated so it is in sync with the record timecode. If the Telecine option is enabled in the Editing>Playout menu, the LTC timecode is the same as the source timecode.

NOTE Your LTC Chase Option settings are saved in the *context.config* file.

To play back the audio track using the LTC Chase Options:

1. Click Editing in the Main menu, and then click Capture.
2. Select the appropriate raster to observe the audio and video files together. See Selecting a Raster for Playout (page 2281). Refer to “Selecting a Raster for Playout” in the “Video Capture and Video Playout” chapter of the Luster user guide.
3. (Optional) Click Setup in the Main menu, and then Audio to display the Audio menu. If you are using the AJA BOB, select an audio channel from the LTC Track Selector that is to represent the LTC channel.



(a) LTC Track Selector option box

NOTE Since the AJA BOB does not have a designated LTC channel, you must assign one of the audio channels to be the LTC channel. Refer to the Hardware Guide for information on how to setup the AJA BOB for the LTC Chase Option.

4 Make sure the audio device is set to chase.

Now when you play back the timeline, the audio track is played at the same time.

Use Explicit Timecode

If the audio device has a different timecode than the timeline, you can set a different timecode by enabling the Use Explicit TC button. For example, if your source timecode starts at 10:00:00:00 and your audio timecode begins at 11:00:00:00, you can enable the Use Explicit TC option and create an offset of 1:00:00:00.

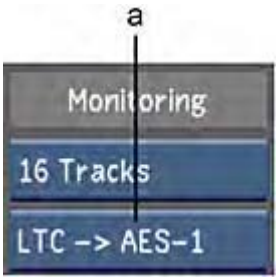
To play back an audio track with a different timecode:

1 Click Editing in the Main menu, and then click Capture.

2 Select the appropriate raster. See Selecting a Raster for Playout (page 2281). Refer to "Selecting a Raster for Playout" in the "Video Capture and Video Playout" chapter of the Lustre User Guide.

3 Click Setup in the Main menu, and then Audio to display the Audio menu.

4 (Optional) If you are using the AJA BOB, select an audio channel from the LTC Track Selector that is to represent the LTC channel.

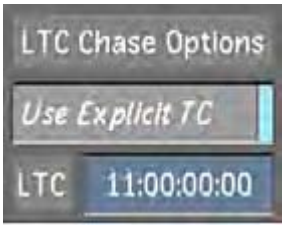


(a) LTC Track Selector option box

NOTE Since the AJA BOB does not have a designated LTC channel, you must assign one of the audio channels to be the LTC channel. Refer to the Hardware Guide for information on how to setup the AJA BOB for the LTC Chase Option.

5 Enable Use Explicit TC.

6 In the timecode field, enter the timecode of when you want the audio track to begin.



Now when you move the shot positioner, the audio and video are in sync.

Playout of Audio and Video

Once you have verified the playback of your audio file/track(s) and timeline, you can play out the timeline to tape. See Playing Out to a VTR (page 2280).

Working with Wiretap

Wiretap is Autodesk's cross-platform client-server interoperability framework, providing Lustre with high-performance access to remote media and metadata. With Wiretap, you can work with material from Autodesk Visual Effects and Finishing applications, without duplicating media. Import timelines directly from the host application, and read/write clips and frames on remote FS and Standard FS storage devices with the same ease as you would local media. In addition, Wiretap allows you work with multi-layer and stereoscopic timelines.

Lustre gains access to the remote media and metadata via Wiretap servers running on the host workstations. A Wiretap server is installed by default with most Visual Effects and Finishing applications, and runs independently of the application. By default, Lustre automatically detects all the Wiretap servers running on the Wiretap network. You can then browse for footage on the remote filesystem and load it into the Shot bin or timeline, just as if it were locally attached storage.

Working with Wiretap Timelines

Lustre can interpret a Visual Effects and Finishing timeline as a single one-sequence clip, or as an imported timeline. When you reference a Wiretap timeline in Lustre, the dissolve and splice transitions are maintained. The Wiretap timeline appears on the Lustre Storyboard as a series of shots and transitions. Each shot comprising the timeline is added separately to the Library. Working with remote timelines containing soft-imported clips is equally possible.

Lustre can import a multi-layer timeline from a Visual Effects and Finishing application and export it to the Wiretap server (only available with Wiretap 2010 Extension 1 or higher). When gaps are imported from a multi-layer timeline, they are seen as black media for Layer 1 and appear transparent for the layers above Layer 1.

Wiretap Timelines and Soft-Imported Media

Recall that in Autodesk Visual Effects and Finishing products, clips using soft-imported media contain only links to the media at its original location, so that no media is actually imported and duplicated in the Autodesk managed storage.

When importing a Wiretap timeline containing soft-imported media into Lustre, there is an additional advantage. Lustre recognizes the clips as soft-imports, and reads the associated media by way of the Wiretap

Gateway server, allowing you to modify the associated input parameters. That is, you are not bound by the resolution used in the originating application, and can work in Lustre at the resolution of your choice.

For example, suppose a Smoke timeline contains soft-imported RED media at HD resolution. When importing the timeline into Lustre via Wiretap, the soft-imported media is accessed directly, via the Wiretap Gateway. You thus have access to all the RED transcoding options for the *source* material, and can work with it at any available resolution, such as SD, HD, 2K or higher.

NOTE This discussion applies to media soft-imported to a Smoke or Flame project created in a Standard FS only. If the media has been soft-imported to Stone FS, Lustre will interpret this as media as "Stone" instead of RED.

Wiretap Timelines and Wiretap Gateway Media

Autodesk Visual Effects and Finishing applications can import a wide variety of media formats from the Wiretap Gateway. It is important to note that Lustre can read all these formats, when you import a Wiretap timeline where they are used.

In particular, when a Visual Effects and Finishing application imports RED media files via the Wiretap Gateway server, the artist sets the debayering, colour, gain and curve settings to adjust the look and size of the imported media. When Lustre imports a timeline that includes RED media, it automatically makes use of the Wiretap Gateway server to gain direct access to the original footage. You thus have access to all the RED transcoding options for the *source* material, and can work with it at any available resolution, such as SD, HD, 2K or higher.

Wiretap Playback Speed

Regarding clip playback speed, network bandwidth and traffic play the expected role. On an optimized GigE network, for example, you can achieve up to 80 MB/sec throughput. This is enough for real-time playback of SD clips, or about 1/4 real-time for 2K material. Better results can be achieved using InfiniBand or 10 GigE. Note that the Visual Effects and Finishing application on the Wiretap server host workstation can also affect performance. Visual Effects and Finishing applications have guaranteed access to the 'local' framestores—remote access by Lustre over Wiretap is at a lower priority. If you notice playback is suffering, consider generating local proxies for the remote material.

NOTE To improve real-time playback of RED media, the Wiretap Gateway can spawn multiple slave processes that increase performance without requiring additional licenses. See the *Wiretap Gateway Installation Guide*.

Rendering to Wiretap

Once you are finished grading the material from the Stone FS or Standard FS, there are different rendering possibilities. As with any Lustre grading project, you can render to the local storage array. Under Wiretap it is also possible to render back to the Stone FS or Standard FS, and to publish soft-import links to rendered media.

Before You Begin

This section presents important points to consider before beginning to work with Wiretap.

Timeline Considerations and Restrictions

When creating a Visual Effects and Finishing timeline for use in Lustre, please note the following.

- `Drop Frame and Non-Drop Frame`: Lustre now supports a timeline with a mix of media that has drop frame and non-drop frame. Instead of using only the timeline frame code mode, it now also uses the source clips' frame code mode.
- `BatchFX` must be rendered prior to importing your timeline in Lustre. They will be seen as regular clips in the Lustre timeline.
- `Supported Timeline Elements`: Colour sources, soft-effects, timeline gaps, dissolve transitions, and fade-ins and fade-outs are supported in Lustre.
- `Dissolves`: These are regenerated in Lustre, along with the animation curve. This is only supported by Wiretap server 2010 and higher.
- `Unsupported Transition Types`: Wipe and Axis transitions are unsupported. These are displayed as dissolves in Lustre and can be recognized by a special icon.
- `Focus Point and Saving`: In a Visual Effects and Finishing application timeline, the positioner's focus point indicates the layer of interest for edits and other operations. It also determines what appears when you import the timeline into Lustre. When saving the timeline for use in Lustre, ensure the focus point is on the top-most layer you want included in the Lustre timeline. For example, if the Visual Effects and Finishing timeline contains four layers and the focus point is on the third layer when it is saved, only three layers are included in the Lustre timeline.

Wiretap Limitations

It is important to note the following limitations:

- `Fade-Ins`: Avoid having a timeline starting with a fade-in at record timecode 00:00:00:00.
- `Tape Names`: Avoid using different shots that have the same timecode and tape name.
- `Hard-Committing`: Hard-committing a selection of shots creates a new clip with a tape name of *COMMIT*, and a source timecode obtained from the record timecode. There is no need to add this information to the shot manually. Hard-committing a single shot preserves the original tape name and timecode.
- `Start Timecode`: The Start timecode of the timeline should not be negative. Also, avoid using a timeline that begins with a fade-in and has a start timecode of 00:00:00:00.
- `Saving to a Clip Library`: Once you have added a clip to a clip library, make sure to switch to a different clip library. This both saves the clip library, and allows Wiretap to broadcast the updated information about the timeline and clips.
- `Dual Library View`: In Visual Effects and Finishing applications, the Dual Library View mode displays the contents of two clip libraries simultaneously, each in its own panel. When you switch from Dual Library View back to Single View, the newly-hidden library remains selected by the application, and is read-only to Wiretap. Lustre will be unable to render media to the clip library until it is deselected.

Wiretap Gateway Limitations

When ingesting media using the Wiretap Gateway, please note the following:

- `OpenEXR`: Lustre supports ingest of OpenEXR in 16-bit float RGB format only. Files with other channels cannot be read in Lustre.
- `RED Rocket Board`: The RED Rocket board allows real-time decoding and debayering of RED media. For Lustre, it is supported in one configuration only. This is with Lustre installed on an HP Z800 workstation running Linux, and the Wiretap Gateway installed on a Mac OS workstation containing the

RED Rocket board. The board must be installed in a PCI-E x16 slot. The network connection must be 10 GigE.

Incinerator Limitations

Incinerator provides in-line processing by several computers for Lustre, dramatically increasing the power of real-time playback. The following Wiretap and Wiretap Gateway features are not supported when an Incinerator cluster is connected:

- Stereoscopy
- Accessing media located on a Stone FS or Standard FS through Wiretap. Note: Timelines consisting of soft-imported media that has been "published" are supported.
- Real-time playback of media accessed through Wiretap Gateway.

Wiretap Backwards Compatibility

You can view and write back to projects created in any current Visual Effects and Finishing application. However, if a project was created using an earlier version of the software, Lustre has read-only access to it. Similar considerations apply to the Wiretap server. That is, Lustre has read/write access to media via any current Wiretap server. Earlier Wiretap servers may provide read-only access. The version of the Wiretap server is shown in the Lustre browser.

Networking

Lustre and Visual Effects and Finishing applications communicate through Wiretap over a network connection. The network infrastructure can be GigE or InfiniBand. A GigE connection is sufficient for SD and HD 8-bit workflows. An InfiniBand connection is recommended for high-resolution projects like HD 10-bit, 2K, and beyond.

See Configuring Wiretap and Lustre (page 2323) and the *Autodesk Visual Effects and Finishing Installation and Configuration Guide*.

Supported Filesystems and Media Types

From the perspective of the Lustre user, there are no operational differences between accessing media on a Stone FS and a Standard FS for colour grading. However, there may be differences in performance. Performance varies when using Standard FS, depending on the configuration. Refer to "Standard FS Performance Expectations" in the *Autodesk Visual Effects and Finishing Installation and Configuration Guide*.

Supported Image File Types

Lustre can access the following compressed and uncompressed image files from a Stone FS or Standard FS.

Server	Media and Metadata Supported
Wiretap Server	■ Single layer, multiple layer, and stereoscopic timelines ■ 8-bit RGB, 10-bit RGB, 16-bit RGB (for 12-bit uncompressed or 16-bit floating point)

Server	Media and Metadata Supported
	■ Soft-imported clips in their original format and bit-depth (DPX, TIFF, SGI, and TGA)
Wiretap Gateway (Imported Clips)	■ OpenEXR (including common compression schemes) ■ RED (including SDK 2.x, 3.x and 4.x transcoding options) ■ ARRIRAW (including 4.3 and 4.4 SDK updates) ■ Sony F65RAW (RH5 and RH6 only) ■ Quicktime ■ MXF

NOTE The Wiretap Gateway server also supports real-time decoding and debayering of RED media via the RED Rocket board.

For details on compressed media formats and format subtypes, see About Wiretap Gateway Supported Ingest File Formats (page 1865).

Although Autodesk's Visual Effects and Finishing products support the 16-bit RGB compressed format, Lustre does not. If directories containing this format are browsed, no media will appear.

Soft-Imported Media

In a Visual Effects and Finishing application, soft-importing creates a clip that references unmanaged media. Unmanaged media typically resides on shared network storage, such as a SAN or NAS, where numerous applications can have equal access to it, avoiding data duplication. If soft-imported media is overwritten, the clip referencing it is updated accordingly. If soft-imported media is deleted, the clip referencing it displays a checkerboard pattern indicating that the media files can't be found. Inversely if, in the Effects or Finishing application, all clips referencing media are deleted, the source media remains unaffected. Basically, soft-imported media is unmanaged, whereas imported media is managed.

Published Media

Publishing consists of exposing managed or unmanaged media to a location accessible by other applications. Once published, the media is still being referenced in a clip library, just like any other clip, but automatically becomes soft-imported and unmanaged media. If the publish destination resides on the same volume and if the format of the published files is the same as the original file format, hard links to the managed media can be created as an option. This media is accessible to Lustre and unmanaged by the Effects and Finishing application.

If you are accessing media residing on a direct attached storage over a slow network connection via Wiretap, you may choose to publish your media to a shared network storage, which allows Lustre to access the media through a fast network storage connection. See Sample Workflows (page 2321).

About QuickTime ProRes Support

On Smoke, which runs on Mac OS X, Apple QuickTime ProRes imports and exports are supported natively.

For all Linux-based application, exports to the Quicktime ProRes format requires connecting to a Wiretap Gateway running on a Mac OS X workstation.

Native ProRes import is supported in the following Linux applications:

- Flame
- Flame Premium
- Flame Premium - Grading
- Flare (when working with a Flame or Flame Premium project)

The following services can also decode clips using ProRes media imported by one of the above products:

- Burn
- Burn Lustre and Lustre Slave Renderer
- Wiretap Central

Other Linux-based products can import ProRes by connecting to a Wiretap Gateway running on a Mac OS X workstation.

Soft-Import Streaming Media Support

With Wiretap and the Wiretap Gateway, Lustre can read compressed media file types that are otherwise not natively supported. For example, users can access soft-imported formats, such as Quicktime®, over the network because the Wiretap Server converts the media on-the-fly without Lustre having to decode the files.

Lustre supports compressed media through Wiretap. The following compressed file formats are converted on-the-fly by the Wiretap Server:

Server	Soft-Import Streaming Media Support
Wiretap Server	■ Quicktime (no extension) ■ Quicktime (.MOV/.mov) ■ Media Exchange Format (.MXF/.mxf) ■ Alias® (.als) ■ Jpeg (.jpeg) ■ Tdi/Maya® (.iff) ■ Macintosh®Pict (.pict) ■ Pixar (.picio) ■ Softimage® (.pic) ■ Wavefront® (.rla)
Wiretap Gateway Server	■ OpenEXR (including common compression schemes) ■ RED (including both SDK 2.x and 3.x transcoding options) ■ Quicktime ■ MXF

NOTE It is also possible to work with compressed media outside of a Wiretap workflow. See Compressed Media Support (page 1864).

For a list of Lustre-supported native file formats, see Accessing the File Browser (page 1856).

Mixed Media Support

A 'mixed' clip or timeline contains media of different storage formats. Typically, this kind of timeline contains both media from a Stone FS framestore and media soft-imported from a Standard FS framestore. In earlier versions, these clips could not easily be used by Wiretap clients without an explicit *stonifize* or publish from within the Visual Effects and Finishing application. As a result of server-side video media conversion, this is no longer necessary.

NOTE One limitation is that the frames within a given shot must be the same format throughout the shot. For example, a shot must have all Stone FS media or all .DPX media.

Proxy Media

A Lustre user can see proxy media on a Wiretap server by enabling the Half resolution button in the player. It is also possible to generate local proxies for Wiretap content. See Project Menu Settings (page 2327) and Viewing Local Proxies for Wiretap Media (page 2328). Lustre can create proxies as you are rendering the hires content to the Wiretap server by enabling the Proxy Rendering button within Network Rendering of the user configuration settings. See Network Rendering Menu Settings (page 2327).

Sample Workflows

The ability to access remote footage makes it possible for Lustre to share footage with other Autodesk products. The example workflows provided below offer high-level descriptions of how Lustre could be integrated into different post-production pipelines.

NOTE While Smoke is used in the examples, other applications such as Flame are equally applicable.

Workflow #1: Importing One-Sequence Clips

In the one-sequence workflow, you work with a single clip representing the timeline coming from Wiretap. In this case, files are first imported into Smoke from the Stone FS and assembled into a timeline created especially for Lustre. The timeline does not contain any splices or soft transitions. Lustre then accesses the timeline as a one-sequence clip by way of Wiretap. This workflow is particularly useful as a guide for a confidence check or scene detection.

NOTE A one-sequence clip appears as a single shot on the Lustre timeline and in the Library. You can perform a scene detection on the clip in order to break it down to its component shots. See Using Scene Detection (page 1966).

In Lustre, you create local proxies, grade them, and then render the source clips back to the Stone FS using any of the standard Render mode controls in the Render Local menu. See Specifying the Destination for Local Render Files (page 2220). The graded material is imported into Smoke to create an updated timeline to which finishing touches are applied prior to rendering the final clip.

Workflow #2: Importing Timeline

Importing a Timeline allows you to work with an editorial sequence that contains shots and soft transitions (such as dissolves). In this case, timelines are created in a Visual Effects and Finishing application and Lustre gains access to the timeline by way of Wiretap. This is a useful workflow for shot-to-shot grading.

NOTE The timeline includes information for all the segments in a clip, including the head and tail information for dissolve and splice events. When you load a Timeline clip, it auto-assembles, allowing you to bypass the conform step.

Lustre renders the final clip to local storage using one of the four rendering options in the Render Local menu. See Specifying the Destination for Local Render Files (page 2220).

Workflow #3: Smoke Renders Final Version

Smoke and Lustre users each access the same files from the external SAN or NAS, and work on them simultaneously. The Smoke user assembles the files and creates a multi-layer timeline while grading is performed on the one-sequence clip in Lustre. At any time during the process, Smoke can be made to publish metadata to reflect editorial changes. Lustre can then be used to re-conform and create a new version of the timeline based on updates performed in Smoke. When grading is complete, the source clips are rendered by Lustre to the external storage using any of the standard Render Place controls in the Render Local menu. See Specifying the Destination for Local Render Files (page 2220). The timeline can then be updated in Smoke by soft-importing the graded sources. Finishing touches are applied to the timeline in Smoke prior to rendering the final clip.

NOTE As an alternative to rendering to the SAN or NAS, Lustre could render to the Stone FS in order to allow Smoke users to bypass the soft-import step.

Workflow #4: Lustre Renders Final Version

As in the previous example, files are soft-imported into Smoke and Lustre in order to be worked on at the same time. The only differences are: instead of referencing the timeline as a one-sequence clip, it is referenced as an timeline; Lustre renders the final version of the timeline, instead of Smoke.

Workflow #5: Simple Source Grading

The following procedure illustrates a possible workflow for source grading for complex projects that include multi-layer timelines with numerous effects, transitions, etc. Since Wiretap flattens the timelines it imports from a Visual Effects and Finishing application, for complex projects, it is worthwhile creating a new single-layer timeline in the Visual Effects and Finishing application. In this way, all the material needed by the Effects and Finishing artists can be made available.

Workflow for simple source grading

1. In Smoke, capture and import media from the off-line editor as a timeline.

   For greatest flexibility in the colour grading process, capture the material in A-mode or C-mode—that is, in ascending timecode sequence.
2. Create a timeline in Smoke that is especially for grading in Lustre, containing all the clips in a single video layer.

This can be accomplished most easily by selecting all the source clips needed in the clip library reel and performing a Load and Join operation.

3 In Lustre, use Wiretap to import the timeline created in the above steps.

4 Grade the sources in the imported timeline, then render it back, via Wiretap.

The rendered clips contain the updated grading but the same tape name and timecode information as the original clips.

5 In Smoke, create a copy of the multi-layer timeline—that is, the *destination* timeline.

6 Save the unlinked timeline to the clip library where Lustre has rendered the media, and relink.

The multi-layer timeline now contains the graded clips.

Preparing the Wiretap Timeline for Lustre

Before you can read a Visual Effects and Finishing timeline using Wiretap, the Visual Effects and Finishing artist must prepare the timeline.

To save a Wiretap timeline to a Visual Effects and Finishing clip library:

1 Process soft effects and transitions (see your Visual Effects and Finishing application's User Guide).

2 Save the timeline to the Clip Library.

NOTE When saving a timeline to the Clip Library for use in Lustre, the position of the focus point is important. See Timeline Considerations and Restrictions (page 2317).

3 Exit the library by changing to a different clip library.

Exiting the library updates the information in the Visual Effects and Finishing database, making it available to Wiretap, and frees up the directory for read/write activity in Lustre.

Configuring Wiretap and Lustre

Before you begin using Wiretap with Lustre, some one-time configuration is required. Depending on how you will be using Wiretap, the following items might require configuration:

- Dual network support
- Wiretap server auto-detection
- Access to a shared storage mount point
- Soft-imported timelines

Configuring Dual Network Support

If your facility has both a GigE network and an InfiniBand network, you can configure Wiretap to use both. The Wiretap server will then make use of the high-bandwidth network for moving media, greatly improving network performance and increasing reliability. The behaviour is transparent to end-users.

To configure the Wiretap server to use InfiniBand for media transfers:

1 On the workstation hosting the Wiretap server, open the Wiretap configuration file, *wiretapd.cfg*, using a text editor.

The file is located in the following directory:

/usr/discreet/sw/cfg

2 Scroll down to the **[Server]** section.

This section lists the IP addresses of the network interfaces used by Wiretap, in the order in which the connections are attempted.

3 In the case of a facility with both a GigE and an InfiniBand network, the IP address of the InfiniBand network must be first in the list, at **IP0**.

In the following example, the InfiniBand network IP address is **10.10.11.213**, and the GigE network IP address is **192.168.129.3**:

[Server]

IP0=10.10.11.213

IP1=192.168.129.3

4 Make the appropriate entries for your own networks, then save and close the file.

5 You must restart the Wiretap server for your changes to take effect:

/usr/discreet/sw/sw_restart

Configuring Lustre to Work with Wiretap

When starting Lustre, Visual Effects and Finishing systems running Wiretap servers on the same network are detected and displayed in the Browser by default. The Wiretap Server can be identified by its host name or IP address, depending on how the workstation's network settings have been configured. You can edit the *init.config* file to configure Lustre to restrict what is displayed to a specified selection of Wiretap servers if you want to limit access.

The Configuration File

All Wiretap-related items are located in the *init.config* file, located in the following directories:

- *C:\Program Files\Autodesk\Lustre <version>* (Windows)
- */usr/autodesk/lustre_<version>* (Linux)

To display a specified selection of Wiretap servers:

1 Open the *init.config* file in a text editor.

2 In the **<Wiretap>** section, change the Wiretap server auto-detection state to 'Off'.

<ServerAuto state="Off">

NOTE Settings in the *init.config* file are case-sensitive. You must use a capital 'O' when setting the auto-detection state to Off.

3 In the **<WiretapServerGroup>** subsection of the **<Wiretap>** section, add each Wiretap server's IP address or host ID to a separate **<WiretapServer String>** line.

NOTE The Wiretap server string lines must be inside the open and close comment tags of the **<WiretapServerGroup>** example.

For example, if your network uses DNS:

<WiretapServer string="berlin"/>

Otherwise:

<WiretapServer string="192.168.0.1"/>

You can add as many workstations as you need in this list. Just add one WiretapServer line for each workstation.

4 Save the *init.config* file and start Lustre.

Configuring the Wiretap Server to use a Shared Storage Mount Point

By default, Wiretap enables access to Visual Effects and Finishing media by way of the Wiretap server. However, if the applications share a storage mount point, Lustre can gain access to the media directly, while reading the metadata off the Wiretap server. You configure this behaviour by adding entries to the Wiretap *path translation file*.

Recall that the Linux and Windows operating systems present file path names using different syntaxes. For example, the path to network storage on a SAN that appears to a Windows workstation as **F:\SAN** might appear on Linux as **/SAN**. The path translation file ensures the path names are translated correctly as information moves between each application.

There are two main steps. First, determine how the shared mount point is seen by each system. Next, create a new translation rule-pair for each mount point in the Wiretap *path translation file*. In this way, you ensure that the Standard FS media used by a Visual Effects and Finishing timeline is directly read/write accessible in Lustre.

Note the following:

- The mount point/mapped network drives must already be set up in their respective operating systems before you refer to them in the path translation file.
- You can create translation rules on a host-by-host basis (most common), or on a per-platform basis. In situations where several hosts see their mount points in the same way, you can simplify the file by creating groups of hosts.
- Place each rule in the rule-pair on a separate line.
- Wiretap applies the rules in the order in which they appear in the file.

For complete information, see the *Autodesk Visual Effects and Finishing Installation and Configuration Guide*.

To configure the Wiretap path translation file:

1 On the workstation hosting the Wiretap server, open the Wiretap path translation configuration file, *wiretap_path_translation_db.xml*, using a text editor.

The file is located in the following directory:

/usr/discreet/wiretap/cfg

2 Scroll down to the section of interest, for example, the **Translation between two operating systems** section.

3 Copy and modify an existing entry, or enter your own rule-pair.

For example, the following rule-pair enables Lustre to read and write a shared mount point seen by the Linux workstation as */SAN*, and by the Windows workstation as *F:\SAN*:

<map src_os="WindowsNT" src_path="F:\SAN" dst_os="Linux" dst_path="/SAN" />

<map src_os="Linux " src_path="/SAN" dst_os=" WindowsNT "dst_path= "F:\SAN" />

4 Save the *wiretap_path_translation_db.xml* file and restart the Wiretap server:

/usr/discreet/sw/sw_restart

Restarting the server is not essential, since the Wiretap server regularly polls the path translation file for changes. However, by restarting the server, you force it to be parsed. This is a good way to test the file for any XML syntax errors. Access to the mount points themselves is not tested during this phase.

Configuring for Soft-Imported Timelines

If you will be working with timelines containing soft-imported media, there are two configuration details to take care of. First, you must configure the Wiretap path translation table to permit the use of a shared storage mount point (as indicated in Configuring the Wiretap Server to use a Shared Storage Mount Point (page 2325)). Next, you must correctly set up the directory structure on the shared storage mount point, as indicated in the following procedure.

To configure projects to access soft-imported timelines over Wiretap:

1. On the shared network storage system, such as a SAN or NAS, ensure that files to be soft-imported into the Visual Effects and Finishing application are all located in the same directory structure.

   For example, the directory on the shared network storage system could be configured as follows:

   /SAN/lustre/scans_full/2k/images/2048x1556/

   Such a configuration is necessary in order for you to be able to render locally with Lustre.

2. On the Visual Effects and Finishing workstation where the Wiretap server is running, ensure the Wiretap path translation file contains entries to translate the paths to the shared mount points from one operating system to another.

   This is necessary so that Lustre can have access to soft-imported media in Wiretap timelines.

   See Configuring the Wiretap Server to use a Shared Storage Mount Point (page 2325).

3. In the Lustre Project Management menu, ensure that the Scans Full Home path points to the Windows drive configured in the Wiretap path translation file (*S:*), for example:

   S:\lustre\scans_full\2k\images\2048x1556

   All other project setup fields can be configured as usual.

4. The shared network storage system, such as the SAN or NAS, must be mounted and accessible (writeable) from both the Wiretap server host and Lustre workstation.

   You are now ready to import soft-imported timelines by way of Wiretap.

   TIP You can tell whether or not the directory has been properly translated by checking to see if the <Home> prefix appears as part of the shot information in the Library. See Viewing Media File Information (page 1860).

Setting Up the Lustre Wiretap Project

Lustre projects that make use of Wiretap must be configured differently than regular Lustre projects. The choices you make depend upon several factors. These factors include whether you will render locally or to a SAN or NAS (as can be the case in non-Wiretap setups), or use Wiretap to render to a Stone FS or Standard FS. In addition, you need to decide whether you will generate local proxies for the clips, or make use of proxies that might already exist on the Stone FS or Standard FS. If your project will make use of soft-imported media, additional considerations apply.

This section describes the settings for Lustre projects that make use of Wiretap, for all configurations other than those that use soft-imported media. For Wiretap projects that also make use of soft-imported media, see Working with Soft-Imported Media (page 2333). For general considerations on setting up projects, see Project Management (page 1719).

Project Menu Settings

The following table explains the criteria to consider for each of the fields in the Project menu related to Wiretap projects.

Field	Criteria
Scans Full Home	■ To access a Stone FS or Standard FS, this field should have the same path as *Project Home*. ■ To access 'published', unmanaged media (also called soft-imported media), populate this field with the path to the media residing on the shared network storage (for example, a SAN or NAS). For example, if the media is located in *F:/SAN/Project/001*, the *Scans Full Home* path should be *F:/SAN*.
Scans Half Home	■ To use proxies coming from a Stone FS or Standard FS, this field should have the same path as *Project Home*.
Renders Full Home	■ To render media to a Stone FS or Standard FS, browse to a clip library destination. This automatically sets the Wiretap render path. ■ To render media to a shared network storage device (for example, a SAN or NAS), browse to a shared network destination. See Rendering Shots (page 2335).
Renders Half Home	■ To render proxy media to a Stone FS or Standard FS, browse to a clip library destination. This automatically sets the Wiretap render path. ■ To generate local proxies from media coming from a Stone FS or Standard FS, populate this field with the path to your local storage array. ■ To generate local proxies from Published content, populate this field with the path to the local storage array or shared storage network.

Rendering Menu Settings

If you are rendering to a Stone FS or Standard FS, enable the *Same Format As Scans* button. If you plan to render to shared storage, select the file format to be used (DPX, TIFF, etc.). Make sure the format you select is supported by the Wiretap server. See Supported Image File Types (page 2318).

Network Rendering Menu Settings

To improve interoperability of projects used by Lustre and a Visual Effects and Finishing application, enable the *Proxy Rendering* button. Proxies are automatically generated when you render the timeline. This makes it efficient to load clips in a Visual Effects and Finishing application as you no longer have to render proxies first.

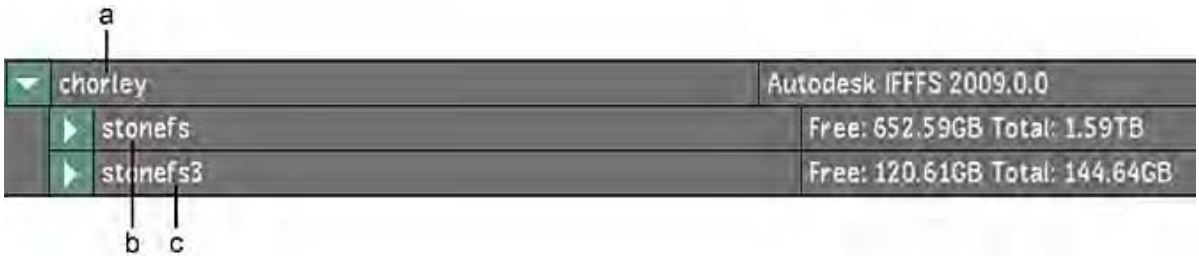
Viewing Local Proxies for Wiretap Media

As with non-Wiretap Lustre projects, proxy generation reduces processing time and can be helpful when working with large image formats. Wiretap projects will make use of Visual Effects and Finishing proxies, if available. This, nevertheless, involves transferring the proxies across the network. It can be advantageous to generate *local* proxies, regardless of proxy availability on the Visual Effects and Finishing side. Note that you can generate the local proxies at 100% of the original resolution. This allows you to work with high-resolution material on the local Lustre storage, avoiding network traffic without sacrificing image quality.

To view local proxies generated from the Wiretap media, enable the *Local Proxy* button in the Network Rendering menu (see Network Rendering Settings (page 1754)). When this button is disabled, you will be able to see the original Visual Effects and Finishing Wiretap proxies using the network connection, upon import. Note that this will not work for clips already imported when the button was disabled.

Browsing for Remote Footage

Wiretap servers are displayed in the Lustre file browser as an IP address or server workstation name, followed by the Wiretap server version. When you manually open the Wiretap server entry in the browser, both Stone FS and Standard FS filesystems are visible.



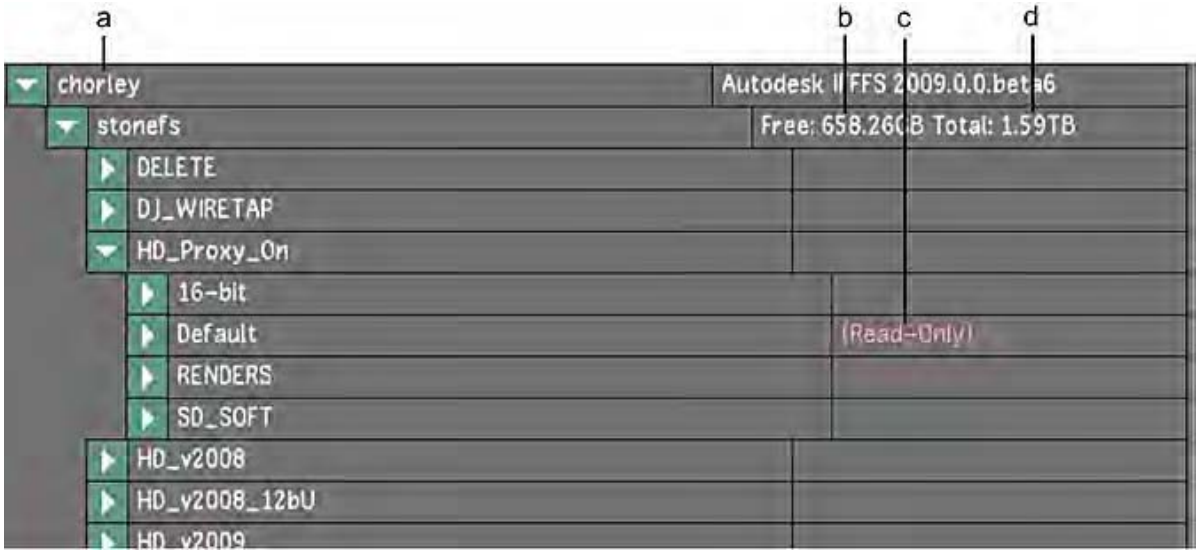
(a) Wiretap server workstation name (b) Stone FS volume (c) Standard FS volume

For details on using the file browser, see Accessing the File Browser (page 1856).

NOTE If you are unable to browse the remote filesystem, ensure the Wiretap service is functioning. See Verifying Connectivity between Lustre and the Wiretap Server (page 2339).

Viewing the Volume Hierarchy

Expanding a Wiretap volume presents its contents as a hierarchy.



(a) Wiretap server (b) Free space on the file system (c) Read-Only indicator (d) Total space on the file system

The following information is presented:

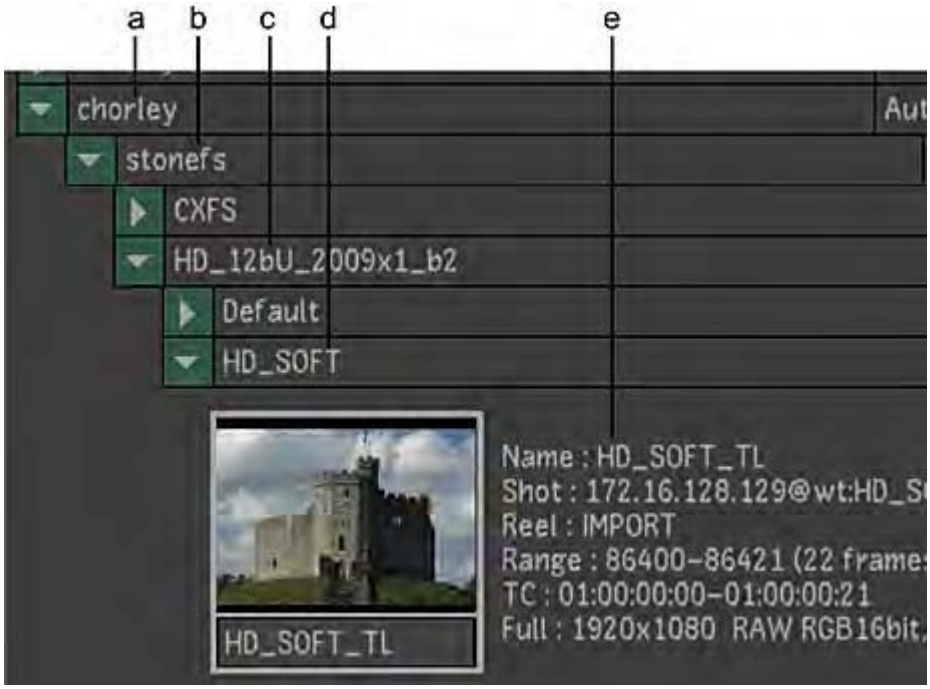
- Wiretap server
- Volume
- Project
- Library
- Reel (optional)
- Clip (in libraries and reels)

Note the following additional points:

- When you access a Wiretap server's volume in the file browser, the remaining space and the total space of the file server on the storage device is also displayed.
- *Read-Only* indicates the library is locked and new clips cannot be written to it. This is the case when a Visual Effects and Finishing artist is browsing the library, for example.
- In the browser, click Refresh to refresh the view of your filesystem.

Viewing Volume and Clip Details

Expanding the volume presents additional details.



(a) Wiretap server (b) Volume (c) Project (d) Library (e) Clip and Clip Metadata

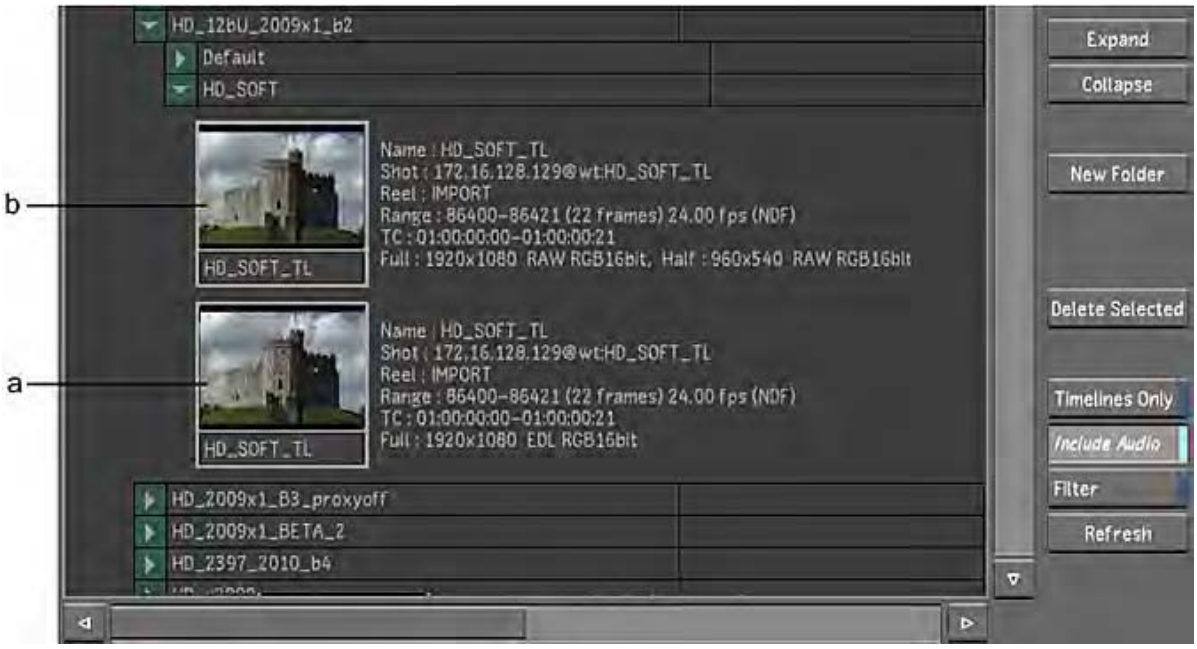
A detailed view of a Wiretap clip is represented in the Library and in the file browser with the following information:

- Clip name
- Shot location
- Frame range and timecode start/end
- Full and proxy resolutions
- Media type

For more information about viewing clip metadata, see Viewing Media File Information (page 1860).

One-Sequence and Timeline Views

By default, the Timelines Only button in the Browser is enabled, which allows Wiretap content to be presented in a timeline form. You can also choose to display the one-sequence representation of the Wiretap timeline by disabling the Timelines Only button, in which case both the one-sequence and timeline versions are displayed at the same time.



(a) Timeline version (b) One-sequence version

The difference between the two is as follows:

- One-sequence: a sequence of files is presented to Lustre with clip metadata as opposed to timeline metadata and you can still see transitions as 'read-only'. This is useful, for example, when manually comparing the contents of the timeline against a digital cut captured directly from an offline video tape. See Performing a Confidence Check (page 1965).
- Timeline: a sequence of shots that contains source clip metadata (for example, shot name, source timecode), dissolve, and cut transitions. This allows you to recreate the timeline that you had in your Visual Effects and Finishing application (in a single-layer format) when you import the Wiretap timeline. You can then grade each shot and render back either a single-sequence clip or render with the timeline structure intact.

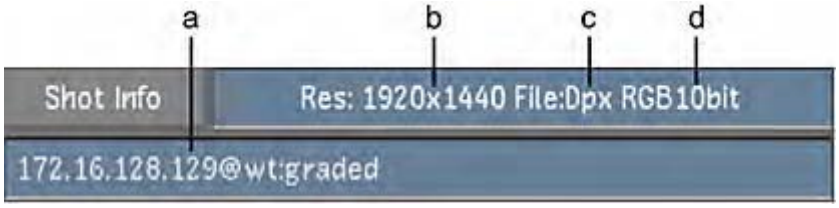
Additionally, you can choose to display the clip thumbnails with or without details about the clips by enabling Details for a detailed view or Proxies for thumbnail-only view. For more information about the Browser, see Accessing the File Browser (page 1856).

Importing the Wiretap Timeline

Please note the following points about importing footage from the browser:

- To maintain the editorial structure and transitions of the Wiretap timeline, it is recommended that you import the Wiretap timeline by dragging and dropping from the browser directly to the empty Storyboard area.
- If the Storyboard already contains a timeline, the new timeline is added as a layer above the existing layer(s). It is added at its record timecode position (not at the drop point), which contains the new timeline content.
- If the Storyboard or timeline is empty, importing a timeline from Wiretap automatically creates a cut. The name of the cut is the same as the name of the timeline.
- Source clips containing long tape names (more than seven characters) are supported.
- For long timelines, consider creating a new cut before dragging and dropping a clip from Wiretap. See Creating a New Cut (page 1774).

- It is possible to load multiple timelines from Wiretap into different Lustre timeline layers (this is only applicable if you are using a Wiretap server version older than 2010 Extension 1). This is useful to create a timeline as it appears in the Visual Effects and Finishing application. In this case, the Visual Effects and Finishing artist must create multiple timelines, one for each layer. In Lustre, you manually load these into the current timeline. Lustre adds each layer based on the record timecode of the Wiretap timeline.
- Once the Wiretap timeline is regenerated as a Lustre timeline in the Storyboard, the source clips are automatically added to the Library.
- If you drag and drop the Wiretap timeline directly into the Library (as you would for local footage), only the source clips are imported into the Library. The timeline is not created in Lustre.
- By default, audio tracks are imported along with the timeline it is associated with. If you do not wish to import any audio over Wiretap, disable the Include Audio button. For more information about importing and playing back audio from Wiretap, see Importing Audio From Wiretap (page 2308).
- Once shots are loaded into the Storyboard, information about the selected shot is displayed in the Shot Info field.



(a) Location of the Wiretap clip (b) Resolution of the clip (c) File format (d) Bit depth per channel (RGB)

Dust Bust and Marry Grade

It is possible to perform dust removal on media imported via Wiretap. However, since media stored on a Stone FS or Standard FS filesystem cannot be replaced by Lustre, you can create dust metadata for use when rendering media only.

NOTE The render options within the Image>Dust menu are inactive when media is imported from the Wiretap server or Wiretap Gateway. You can only render by using the Dust option within the Render>Local menu.

Media imported by Wiretap can also be used to generate Marry Grade data. However, since it is not possible to write Marry Grade files to Stone FS or Standard FS filesystems, the Marry Grade files are stored in the project's Full or Half Scan location. In the Marry Grade folder, a WT (Wiretap) folder is automatically created. Within it, a hierarchy of folders contain the Marry Grade data.

Working with Half Float Media

OpenEXR is a high-dynamic range (HDR) media format supporting multi-resolutions and an arbitrary number of channels and channel types, such as specular, diffuse, alpha, RGB, normals, in a single file. Lustre supports OpenEXR 16-bit RGB only (files with other channels cannot be read). This format is sometimes referred to as "16-bit half float".

Lustre can obtain 16-bit floating point media in two distinct ways. First, it can acquire the media by way of a Wiretap server. That is to say, by browsing the clip libraries and associated framestores of Visual Effects and Editing applications. Second, it can acquire the media directly, by importing OpenEXR media using a Wiretap Gateway server. In both cases, you need to tell Lustre how to work with the half float media.

As indicated, OpenEXR represents pixels as floating-point numbers. Internally, Lustre uses a 16-bit integer format. Thus, before you can grade OpenEXR media, it must be converted from floating point to integer.

This is done by setting a floating point conversion LUT for the project. See Applying an Existing Input LUT (page 2029).

Two special floating point conversion LUTs and one output LUT have been provided for this purpose. Before working with OpenEXR media, select the conversion LUT corresponding to the colour space defined for the project.

LUT	Description
lin_default	The default linear colour space input LUT, generally reserved for video images.
log_default	The default logarithmic colour space input LUT, generally reserved for film images.
identityExtended	An output LUT designed to preserve the floating point media.

When rendering to Wiretap as part of an interoperability workflow with an Autodesk Visual Effects and Finishing application, apply the inverse LUT, *identityExtended*, during the render, to restore the original colour space. If interoperability is not the goal, specify an output format and render as usual, without applying an output LUT. See Rendering Shots (page 2216).

NOTE It is not possible to output in the OpenEXR format.

Working with Soft-Imported Media

This section presents the considerations that apply when working with timelines or clips containing soft-imported media. When using soft-imported media, be aware that you have no control if another user modifies the original media files. All soft-imported clips that refer to the modified media files are updated to reflect the change. Ensure that all users accessing the same media are aware of these implications.

Creating Soft-Imported Links to Rendered Media

You can create soft-imported clips from rendered material and write the generated timeline back to the Wiretap Clip Library. The rendered results reside in the destination specified in the project settings, and a soft-import link is created on the Stone FS or Standard FS.

To create a soft-imported link to your graded renders in the Wiretap Clip Library:

1. Click Setup, and then click Project to display the Project menu.
2. In the Renders Full Home field, enter the path for the graded files to be rendered to.

   For example, if your Windows drive is mapped to the network storage as *S:*, then the renders full home path should be *S:/SAN/lustre_renders*.

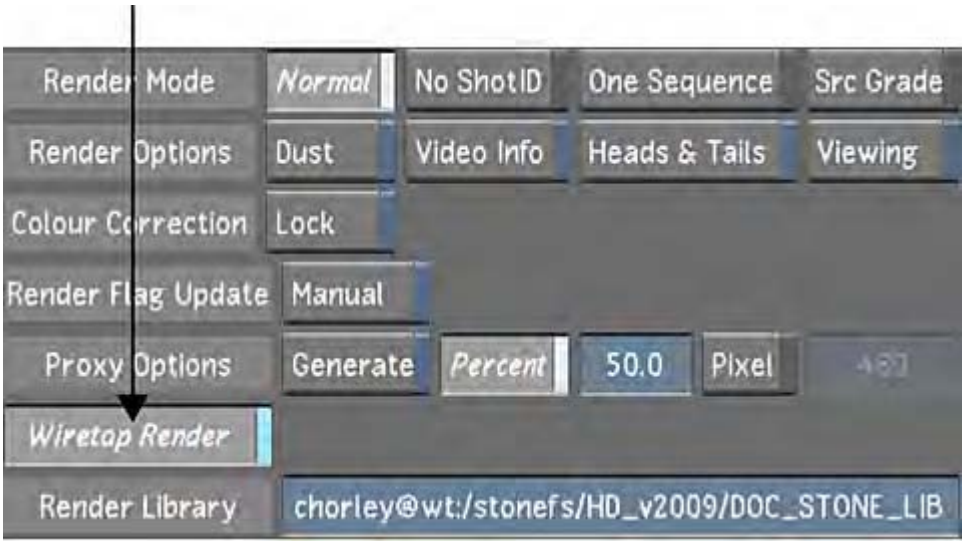
   NOTE The shared storage network (i.e., SAN or NAS) must be mounted and accessible from both the Wiretap Linux host and the Lustre workstation.
3. Click Render, and then click Local to display the Local menu.
4. Select one of the following render options:
   - Normal
   - No ShotID
   - One Sequence
   - Src Grade

5 Enter the path for the location of the soft-imported clip in the Render Library field.

For example, *10.145.43.31@wt:/stonefs/project/library*.

Note that you can set a default value for the Render Library field when configuring the project, as described in the next procedure.

6 Enable the Wiretap Render button.



7 Click Start.

This renders the files to the specific path of the shared network storage, and publishes the soft-import links to your rendered files in the destination Wiretap clip library. A new scene grade reel containing the soft-imported clips and/or timeline is created in the location specified by the Render Library field.

To set the Render Library path as a project setting:

1 From the Main menu, click Setup, and then Settings. Select the project to edit, and then click Edit in the Project group.

2 On the Network Rendering menu, enter the desired path in the Render Library field in either of the following ways:

- Navigate to the directory using the navigation aid button and browser.
- Enter the desired path directly in the Render Library field, in the following format:

  `<Wiretap_server>@wt:/<volume>/<project_name>/<library>`

For example:

rishikesh@wt:/stonefs/HD_project/lustresmoke

3 Save the settings for the project by clicking Save Project.

4 Click Exit Project.

5 To verify the path shows up as expected, from the Main menu, click Render, and then Local.

The path appears in the Render Library field.

Working with Stereoscopic Wiretap Timelines

Importing a stereoscopic Wiretap timeline follows all the principles and restrictions outlined for non-stereo Wiretap timelines (see Importing the Wiretap Timeline (page 2331)). Similarly, once in Lustre, it is indistinguishable from a “local” stereoscopic timeline. Under specific circumstances, however, when rendered

back to the Wiretap server, the timeline will lose its stereo status in the originating application. The status can easily be regained.

Recall that in an Autodesk Visual Effects and Finishing application, such as Smoke or Flame, the metadata must always be identical between the left and right eye layers. The layers must share the same resolution, bit-depth and framerate. They must also be of the same duration, and have the same number of edits, in the same places. If the metadata falls out of sync, the timeline loses its stereo status.

However, in Lustre it is sometimes necessary to apply processing to just one layer of a stereo timeline. For example, applying a cut and animating the brightness channel can be a convenient technique for correcting a shot that becomes increasingly bright as it progresses. A cut applied to one layer but not the other puts the metadata out of sync, and the shot is no longer recognized as a stereo timeline, once rendered back to the Autodesk Visual Effects and Finishing application. It appears as a timeline with two layers, but without any of the stereo features or interface components.

When this arises, simply select the shot that caused the metadata to fall out of sync, and adjust it. Often this can be done in the originating application. In the case of an added cut, for example, you can "hard commit" the two shots involved. This renders the frames and eliminates the cut, "re-synchronizing" the metadata.

For other information on the Stereoscopy feature, see Stereoscopy (page 2206).

Rendering Shots

When you have finished grading the imagery from the filesystem, you can render to the local storage array, or write back to the Stone FS via Wiretap.

NOTE Audio tracks that were imported through the Wiretap server cannot be rendered back to Wiretap.

If working with soft-imported clips, you must also set up the directory structure on the SAN or the NAS in such a way that Lustre can recognize it. See Setting Up the Lustre Wiretap Project (page 2326). You render clips from a local storage the same way as you would for any other clip. See Rendering (page 2215).

Rendering clips back to Wiretap requires that the render destination be specified in the project setup. See Setting Up the Lustre Wiretap Project (page 2326). The render destination you specify in the project setup will determine the location of the reel in which the rendered clip will be placed. See Rendering with Wiretap (page 2336).

NOTE The clip's long tape name metadata is also rendered back through Wiretap.

You can use the Render menus to render your work. You can render selected shots, all shots in a cut, a specific layer in the Timeline, or the flattened result of a multi-layer timeline. To prepare for rendering, do the following.

Rendering workflow:

1. Specify the layer to be rendered.
2. Set the resolution of the render files.
3. Set resize options, as needed.
4. Specify the destination for the render files.
5. Specify the output file format (if different than the original footage).
6. (Optional) Change the output colour space or specify an output LUT.
7. Set other options such as timecode burn-in and rendering with a viewing LUT.

Rendering with Wiretap

When rendering with Wiretap, a new reel (based on the scene name, grade, and timeline name) will be created in the destination Clip Library that was defined when the project was created. See Project Management (page 1719).

This is the syntax to use when rendering to Wiretap, or publishing soft-import links to your graded sources:

`<IP address or hostname>`*@wt:/*`<volume>/<project_name>/<library>`

In the above sample, `<IP address or hostname>` is the IP address or host name of the Wiretap server. For a Stone FS, the volume is specified as *stonefs*. For a Standard FS, it is specified as *stonefs<#>*—for example, *stonefs2*.

If, for example, *10.145.43.31@wt:/stonefs/HD_project/rendered_grades* was entered in the Renders Full Home field, the rendered clip will be placed in:

/stonefs/HD_project/rendered_grades/`<scene>_<grade>_<timeline_name>`

You can specify how you want render files to be organized in the Wiretap Clip Library using the Render Place controls in the Render Local menu.

Normal Rendering Mode

A Normal render creates source clips (with or without head and tail frames) and a timeline. The extension of the timeline is the grade that the timeline is associated with (e.g., `<cut/timeline_name>_grd<number>`. In this mode, dissolves are rendered, and cannot be further modified in the Visual Effects and Finishing application.

For shots rendered to a Wiretap Clip Library, a new reel (based on the scene name, grade, and timeline name) is generated in the Clip Library that was defined when the project was created. This is where the rendered material is placed.

For example, if *10.145.43.31@wt:/stonefs/tutor/lustregrades* was entered in the Renders Full Home field, the graded rendered shots are placed in *10.145.43.31@wt:/stonefs/tutor/lustregrades*. These shots contain the clip name, grade name, and the shot's unique ID.

`<clip_name>_<grade>_<unique_id>` (example:*shot1_grd14_AB3E9770000053*)

An additional clip with a *_grd*`<number>` suffix is also generated containing the cut or timeline name. For example, if the grade name is *grd14* from the timeline called "interior_003", the additional file is:

interior_003_grd14

The graded timeline is generated based on the new rendered source files.

No ShotID Rendering Mode

As outlined in the Normal mode description above, when rendering clips to Wiretap, a new reel (based on the scene name, grade, and timeline name) is created in the Wiretap Clip Library and all rendered material is placed there. The additional clip with the *_grd*`<number>` suffix is also generated.

One-Sequence Rendering Mode

As with Normal and No ShotID mode, when rendering clips to Wiretap, a new reel (based on the scene name, grade, and timeline name) is created in the Wiretap Clip Library and all rendered material is placed there.

One-Sequence mode generates one clip of the graded sequence containing the Lustre cut or timeline name and the grade number. If, in Lustre, your scene is named *interior_003*, the timeline is named *FINAL_CUT*, and the grade file is named *grd14*, the new reel in the Wiretap Clip Library will be called:

/stonefs/HD_project/rendered_grades/interior_003_FINAL_CUT_grd14/

The rendered, one-sequence clip will contain the grade name: *FINAL_CUT_grd14*

Source Grade Rendering Mode

Src Grade mode renders the complete timeline back to Wiretap with Lustre grading applied, but dissolves and retimes are rendered as soft effects with modifiable values. This is the most convenient rendering mode for regenerating source clips for relinking in the Visual Effects and Finishing application, in a "simple source grading" workflow.

For shots rendered to Wiretap, a new reel (based on the scene name, grade, and timeline name) is generated in the library that was defined when the project was created. This is where the rendered material is placed.

For example, if *10.145.43.31@wt:/stonefs/tutor/lustregrades* was entered in the Renders Full Home field, the graded rendered shots are placed in *10.145.43.31@wt:/stonefs/tutor/lustregrades/scene_grade*. These shots contain the clip name, grade name, and the shot's unique ID.

`<clip_name>_<grade>_<unique_id>` (example:*shot1_grd14_AB3E9770000053*)

An additional clip with a *_grd*`<number>` suffix is also generated containing the cut or timeline name. For example, if the grade name is *grd14* from the timeline called "interior_003", the additional file is:

interior_003_grd14

The graded timeline is generated based on the new rendered source files.

Rendered graded source files are renamed according to the grade number and the clip's unique ID. For example: *grd01_4323B1F4*.

Rendering Wiretap Clips to Local or Network Storage

The render directory structure for Wiretap clips is different from how local scans are rendered to the local or networked storage. The location and structure of the *scene* or *grd* directory depends on whether your shots are written on the Stone FS, or soft-imported into the Stone FS, and is based on your project directory structure. See Setting Up the Lustre Wiretap Project (page 2326).

You can specify how you want render files to be organized under the *grd* or *scene* directory using the Render Place controls in the Render Local menu. These controls affect the location where they are saved and the naming scheme of the render files.

NOTE There is currently no difference between Normal and No ShotID renders.

To set how render files are organized:

1. Enable Normal, No ShotID, One Sequence, or Src Grade.

NOTE Soloing and Muting rules for Wiretap rendering are the same as those for local rendering. See Specifying the Destination for Local Render Files (page 2220).

Normal Rendering Mode

Clips are rendered to sub-directories of a directory based on the scene name, which Lustre creates automatically when you first render shots. From there, they are rendered with each shot in its own *grd_UID* directory, creating the final name by combining the grade name with the scene name.

For example:

Stone-based clips are rendered as:

<Home>/<scene>/grd/<grdXX_UID>/<resolution>/<scene_grdXX.xxxx>.dpx

Soft-imported clips are rendered to a directory called *grd*. With this *grd* directory, all the shots are rendered to a single directory based on the source location of the soft-imports (let's call it the source reel) and the grade name. From there, the final name is created based on the source filename and Lustre grade name.

For example:

Soft-imported clips are rendered as:

<HOME>/grd/<sourcereel_grdXX_UID>/<resolution>/<origfilename_grdXX_UID.xxxx>.dpx

If a source clip called *shot13* has been soft-imported into the Effects or Finishing application from this location:

/SAN/lustre/scans_full/2k/reel23/2046x1556/

Lustre then imports this soft-imported clip via Wiretap, saves the grade as *grd01*, and renders it to the local or networked storage array. Lustre will take the name of the shot directory preceding the resolution directory, in this case, *reel23*.

For example:

Soft-imported clips are rendered as:

<HOME>/ grd/reel23_grd02_AB3E9770000053/2046x1556/shot13_grd01_4323B1F4.0001.dpx

No ShotID Rendering Mode

As with Normal renders, Stone-based clips are rendered to sub-directories based on the scene name, which Lustre creates automatically when you first render shots. From there, they are rendered with each shot in its own *grd_UID* directory, creating the final name by combining the grade name with the scene name.

For example:

Stone-based clips are rendered as:

<HOME>/<scene>/grd/<grdXX_UID>/<resolution>/<scene_grdXX.xxxx>.dpx

As with Normal renders, soft-imported clips are rendered to a directory called *grd*. With this *grd* directory, all the shots are rendered to a single directory based on the source location of the soft-imports (let's call it the source reel) and the grade name. From there, the final name is created based on the source filename and Lustre grade name.

For example:

Soft-imported clips are rendered as:

<HOME>/ grd/<sourcereel_grdXX_UID>/<resolution>/<origfilename_grdXX_UID.xxxx>.dpx

One Sequence Rendering Mode

As with Normal and No ShotID renders, clips are rendered to sub-directories of a directory based on the scene name, which Lustre creates automatically when you first render shots. From there, under the sub-directory called *grd*, the shots are rendered as a single sequence to a directory based on the *grd* name. The filenames consist of the scene name, the grade name, and a sequential number. There are no UIDs in the directory naming scheme.

Soft-imported clips are rendered to sub-directories based on the scene name, which Lustre creates automatically when you first render shots. From there, under the sub-directory called *grd*, the shots are rendered as a single sequence to the sub-directory based on the *grd* name. The filenames consist of the scene name, the grade name, and a sequential number. There are no UIDs in the directory naming scheme.

For example:

Stone-based clips are rendered as:

<HOME>/<scene>/grd/<grdXX>/<resolution>/<scene_grdXX.xxxx>.dpx

Soft-imported clips are rendered as:

<HOME>/<scene>/grd/<grdXX>/<resolution>/<scene_grdXX.xxxx>.dpx

Source Grade Rendering Mode

As with Normal and No ShotID renders, clips are rendered to sub-directories based on the scene name, which Lustre creates automatically when you first render shots. From there, they are rendered with each shot in its own *grd_UID* directory, creating the final name by combining the grade name with the scene name. In addition, Src Grade excludes dissolves and retimes and renders the files with their pre-dissolve and pre-retime handles.

Stone-based clips are rendered as:

<HOME>/<scene>/grd/<grdXX_UID>/<resolution>/<scene_grdXX.xxxx>.dpx

As with Normal and No ShotID renders, soft-imported clips are rendered to a directory called *grd*. With this *grd* directory, all the shots are rendered to a single directory based on the source location of the soft-imports (let's call it the source reel) and the grade name. From there, the final name is created based on the source filename and Lustre grade name. In addition, Src Grade excludes dissolves and retimes and renders the files with their pre-dissolve and pre-retime handles.

Soft-imported clips are rendered as:

<HOME>/ grd/<sourcereel_grdXX_UID>/<resolution>/<origfilename_grdXX_UID.xxxx>.dpx

Troubleshooting

This section presents a few helpful procedures relating to the Wiretap server and Lustre-server connectivity.

Verifying Connectivity between Lustre and the Wiretap Server

Verify that you have proper connectivity over the network.

To verify connectivity (Windows):

1. On your Lustre workstation, click the Start button, and then click Run.
2. In the Run dialog box, type:
   cmd
   A command console is displayed.
3. In the command console, type:
   ping `<IP address of Wiretap host or Wiretap host workstation name>`

   Example system response

   ping computer

   Reply from 172.16.128.129: bytes=32 time=3ms TTL=63

   Reply from 172.16.128.129: bytes=32 time=3ms TTL=63

   Reply from 172.16.128.129: bytes=32 time=3ms TTL=63

A system response with ping times similar to the example, above, indicates that your Lustre workstation can see the remote Wiretap host workstation.

To verify connectivity (Linux):

1 On your Lustre workstation, open a shell and type:

ping <IP address of Wiretap host or Wiretap host workstation name>

Example system response

ping computer

Reply from 172.16.128.129: bytes=32 time=3ms TTL=63

Reply from 172.16.128.129: bytes=32 time=3ms TTL=63

Reply from 172.16.128.129: bytes=32 time=3ms TTL=63

A system response with ping times similar to the example, above, indicates that your Lustre workstation can see the remote Wiretap host workstation.

Starting and Stopping the Wiretap Server

The following utilities have been provided for starting, stopping, and restarting the Wiretap server.

Utility	Description
sw_start	Starts the Wiretap server
sw_stop	Stops the server
sw_restart	Stops then restarts the server

The utilities are located in the following directory:

/usr/discreet/sw

Verifying that the Wiretap Server is Running

To verify a Wiretap server is running:

1 On the Wiretap server, open a shell and type:

ps -ef |grep wiretap

Example system response

root 19579 10 May14 ? 00:00:00 /bin/sh /usr/discreet/sw/sw_wiretapd

root 19588 195790 May14 ?00:01:51 /usr/discreet/wiretap/<version_number>/ifffsWiretapServer -c /usr/discreet/wiretap/cfg/wiretapd.cfg

Example system response

```
root27965 277280 10:02 pts/5 00:00:00 grep wiretap
```

In the above output, the line */usr/discreet/wiretap/<version_number>/ifffsWiretapServer -c /usr/discreet/wiretap/cfg/wiretapd.cfg* indicates the version number of Wiretap that is running. If you do not see information similar to this, restart the Wiretap server.

Hotkeys

Storyboard Navigation

Use these hotkeys to navigate in the Storyboard and shot timebars.

Press:	To:
`Up Arrow`	Play forward.
`Down Arrow`	Play backward.
`Left Arrow`	Go to the first frame of the previous shot.
`Right Arrow`	Go to the first frame of the next shot.
`Spacebar`	Toggle Play/Stop.
`Shift+Spacebar`	Switch to Render View mode, switch off the user interface, and start playback.
`Right Ctrl`	Step one frame forward.
`Right Alt`	Step one frame backward.
`Shift+I`	Mark an in point for loop play on the Scene timebar.
`Shift+O`	Mark an out point for loop play on the Scene timebar.
`Shift+L`	Clear in and out points from the Scene timebar.
`.` (decimal point on the numeric keypad)	Cycles through the play modes (loop sequence, loop shot, and loop in and out points).
`Tab`	Toggle the Storyboard on/off. In the Editing menus, the Storyboard is always on.
`Q`	Switch between Storyboard viewing modes (regular and large Storyboard view). Not applicable to the Editing menu.

Press:	To:
Shift+1	Enable or disable the Update Storyboard feature. When enabled, the Storyboard thumbnails are automatically updated as you make changes to your shot.
. (period)	Update the shot's thumbnail according to the colour correction applied on the current frame.
Shift+. (period)	Regenerates multiple Storyboard thumbnails consecutively, starting from the current shot onwards.
Spacebar	Stop the multiple Storyboard thumbnail regenerating procedure.
Middle-drag	Pans the Storyboard.
F	Toggle between the collapsed Storyboard and the original.

View Modes

Use these hotkeys to switch between the different view modes.

Press:	To:
P	Switch to Render View mode.
Shift+P	Switch to Render View mode and have Lustre re-detect and reparse the rendered files. This is useful when the files are not found automatically, for example, when they are not rendered from the application itself, but using the command line render.
Shift+ Spacebar	Switch to Render View mode, switch off the user interface, and start play. (This hotkey cannot be used in any of the Editing menus.)
O	Switch to Result view (this is the default).
I	Switch to Primary view. In this mode, only primary correction appears (secondaries or plugins are turned off).
Middle-drag	Pans the image in any direction.
Middle-right-drag	Zoom in/out of the image.
L	Show the framestore image.
K	Save the current image to the framestore.
Shift+K	Load only one reference image. You must define this image in the Lustre configuration file (with its full path).

Press:	To:
Ctrl+F	Enable or disable the filtering option. When enabled, it softens the shot so when you zoom in, the pixels are not as sharp or obvious.
Enter	Switch on/off the user interface.
~	Switch between the Player and a graphical schematic view of the axis hierarchy.
Shift+~	Switch the background colour between the selected colour and full black.
B	Toggle the Library or file browser's viewing mode: Proxies, List, or Details.
V	Toggle the Ratio button in the Grade menu to set the display aspect ratio. When you enable the Ratio button, the aspect ratio that appears in the Ratio field is used.
W	Toggle on/off the shot name text on the image.
Alt+1	Toggle on/off the RGB waveform monitor.
Alt+2	Toggle on/off the RGB histogram.
Alt+3	Toggle on/off the luminance waveform monitor.
Alt+4	Toggle on/off the vecterscope.
' (apostrophe)	Switch on/off monitor sub-region calculating.
F1	Display Playhead A.
F2	Display Playhead B.
Ctrl+F1	Cycle through the different colour view selections (i.e., colour, red, green, blue, and luminance channels).
Shift+Ctrl+F1	Return to the previous colour view selection.
F3	Cycle through the different viewers (i.e., Single, Dual, or Multi view).
Shift+F3	Return to the previous viewer option.
F4	Cycle through the Playhead A and B Dual view modes (i.e., 2-up, horizontal wipe, and vertical wipe).
Shift+F4	Return to the previous Playhead A and B Dual view mode.

Press:	To:
Ctrl+F4	Cycle through the Stereo Dual view modes (i.e., 2-up, horizontal wipe, vertical wipe, and blend).
Shift+Alt+F4	Return to the previous Stereo Dual view mode.
F5	Toggle between the current view options and the Single view (default).
Ctrl+F5	Cycle through the Multi view modes (i.e., 2, 4, 9, and 16).
Shift+Ctrl+F5	Return to the previous Multi view mode.
F6	Refresh the images in Multi view.
F7	Displays the timeline, Player, grading tools, and keyer (HLS or Diamond) and puts the mouse cursor and secondaries handling in the GFX SDI output.
F8	Toggle on/off the user interface update during playback.
F9	Switch between full and half resolution.
F10	Switch between the 1x and 2x zoom.
F11	Show the matte on the current secondary. Press F11 once to view the matte in Secondary view if Mask Type is set to Overlay. Press F11 twice to view the matte in Matte view if Mask Type is set to Greyscale.
F12	Show the result.
E	Show the selected colour's numbering type (code value, F-stop, or HDR).
Z	Show or hide the dust removal boxes.
H	Show or hide the selected overlay.
J	Switch on/off video preview.

Timeline Sort Modes

Use the following hotkeys to sort the shots in your timeline.

Press:	To:
Alt+F9	Sort the shots using A-mode sort and enable heads and tails.
Alt+F10	Sort the shots using A-mode sort and disable heads and tails.

Press:	To:
Alt+F11	C-mode sort by EDL reel name and enable heads and tails.
Alt+F12	C-mode sort by EDL reel name and disable heads and tails.
Ctrl+Alt+F11	C-mode sort by folder name and enable heads and tails.
Ctrl+Alt+F12	C-mode sort by folder name and disable heads and tails.
Ctrl+Alt+Shift+F11	C-mode sort by DPX reel name and enable heads and tails.
Ctrl+Alt+Shift+F12	C-mode sort by DPX reel name and disable heads and tails.
Alt+F8	Disable sort mode.

Notes

You can use any of the following standard text editor functions when drafting notes in the Notes window.

Press:	To:
cursor arrows	Move up, down, left or right.
Backspace, Delete	Delete text.
Home	Jump to the start of the current line.
Ctrl+Home	Jump to the start of the text.
End	Jump to the end of the current line.
Ctrl+End	Jump to the end of the text.
Page Up	Jump one text screen up.
Page Down	Jump one text screen down.
Shift+cursor arrow or left mouse button-drag	Select text.
Ctrl+A	Select all text.
Enter	Add an empty line.
Esc or click outside the text editor	Disable the text editor.

Calculators

The following hotkeys can be used with the standard calculator and the timecode calculator.

Press:	To:
Backspace	Erase the last digit in the calculator's numeric field.
Delete	Reset the calculator's numeric field to zero.
Page Down	Change the sign of the value in the calculator's numeric field.
Up cursor arrow	Add the current calculator value to the field or slider value.
Down cursor arrow	Subtract the current calculator value from the field or slider value.
Enter or click the calculator's active numeric field	For the standard calculator: calculate the value, apply it to the field or slider, and close the calculator. For the timecode calculator: jump to the frame number that corresponds with the value enteredin the calculator's numeric field.
Enter or click the calculator's = (equals) button	Calculate the value and display the result without applying it to the field or slider. Applicable to the standard calculator only.
Ctrl+Enter or right-click the calculator's = (equals) button	Calculate the value, display the result, and apply it to the field or slider. Applicable to the standard calculator only.
Esc or click outside the calculator	Close the calculator and leave the field or slider value unchanged. Applicable to the standard calculator only.

Dust Removal

Use these hotkeys when working in the Dust menu.

Press:	To:
Shift-click	Place a special “red pixel” repair box, which can repair a one-point scanned error.
Z	Show or hide the dust removal boxes.
, (comma)	Show or hide the dust removal boxes.
Backspace	Delete the active dust frame.
Shift-click (Shot button)	Render dust removal results in all shots in the timeline that have been modified (rather than the current shot only).

Grain Removal

Use these hotkeys when working in the Degrain menu.

Press:	To:
Z	Show or hide the degrain analysis sub-region boxes.
, (comma)	Show or hide the degrain analysis sub-region boxes.
Backspace	Delete the active degrain analysis sub-region boxes.
Right-click	Select or de-select multiple degrain analysis sub-region boxes.

Editing

Certain editing hotkeys are available from any menu, whereas the rest are available exclusively from the Edit menu.

Global Editing Hotkeys

When you enable the Editing While Grading button in the System & Menu page of the user configuration, you can access the following editing hotkeys from any Lustre menu (except the Colourist Timeline).

Press:	To:
Shift+C	Cut the shot in two at the current frame.
Shift+J	Join the current shot with the previous shot if they are from the same source.
Shift+T	Start interactive visual trimming.
Shift+R	Start interactive visual sliding between the current and the previous Shot.
Shift+S	Start a scene detection.

Edit Menu Hotkeys

Use these hotkeys in the Edit menu.

For numeric keypad hotkeys that pertain to editing, see Editing Menu—Edit (page 2358).

Press:	To:
Shift+D	Delete the current shot.

Press:	To:
Shift+Q	Locate and select the shot in the Library that corresponds to the selected shot in the Storyboard.
Alt-click	Locates and places a red border around the shot in the Storyboard and moves the positioner to the element in the Timeline that corresponds to the shot you Alt-click in the Library.
Esc	Exit visual interactive trimming, interactive slipping, and scene detection.
Backspace	Delete the last digit of the typed timecode number.
Shift+click-drag	Replace the shot.
Shift+hold-drag	Replace the shot with an offset.
Enter (on numeric keypad)	Confirm and exit Shot Replace mode.

Timeline Menu Hotkeys

Use these hotkeys in the Timeline menu when your cursor is on the Timeline canvas.

Press:	To:
Up Arrow	Move focus up.
Down Arrow	Move focus down.
Left Arrow	Go to the first frame of the previous shot.
Right Arrow	Go to the first frame of the next shot.
Spacebar	Toggles play and stop.
Right Ctrl	Step one frame forward.
Right Alt	Step one frame backward.
Shift+Q	Locate and select the shot in the Library that the positioner is placed on.
Shift+I	Mark an in point on the Scene timebar for loop play.
Shift+O	Mark an out point on the Scene timebar for loop play.
Shift+L	Clear all in and out points.

Press:	To:
. (period) on numeric keypad	Toggles through the play modes (loop timeline, loop shot, and loop in and out points).
F1	Go to Playhead A.
F2	Go to Playhead B.
Shift+M	To mute or unmute the track on the focus layer.
T	To enable or disable Trim mode.
Enter	Toggles between the Player, file browser, and full-screen player.
Alt+S	Enable or disable Solo mode.
Alt+C	To turn on or turn off copy element(s) and grade.
Shift-drag	Copy the selected shot and constrain.
Shift+C	To make a cut in the element.
Shift+J	To join/remove splice on selected element.
Shift+D	Delete element(s) from the Timeline.
, (comma)	Enable or disable the Slip mode.
Ctrl-right-click	Timeline home view.
Alt+H	Zooms into the selected shot(s).
Middle-click-drag	Pans the Timeline left or right.
Right-click-drag	Zooms in and out of the Timeline.
Shift-click	To select multiple elements in a continuous order.
Ctrl-click	To select multiple elements that are not in a continuous order.
Shift+N	Copies the focus element to a new layer.
S	Save a cut file and the associated grade files. If a grade file has not been created, a default grade file is saved.
Shift+R	Toggles the Ripple mode (Off/Start/End).

Press:	To:
[	Zooms out of the Timeline. Zoom is centred on the playhead.
]	Zooms in on the Timeline. Zoom is centered on the playhead.
D	Adds a dissolve according to the focus and the positioner.
Shift+\	Applies/removes priority to/from the shot with focus.
Shift+right-click	Applies/removes priority to/from the selected shot.

Capture Menu Hotkeys

Use these hotkeys during the capture process.

For numeric keypad hotkeys that pertain to editing, see Editing Menus—Capture and Playout (page 2359).

Press:	To:
Esc	Pause the capture. Press again to abort the capture.
E	Restart the capture.
S	Skip the current tape.
Backspace	Delete the last digit of the current timecode value.

Playout Menu Hotkeys

Use these hotkeys during the playout process.

For numeric keypad hotkeys that pertain to editing, see Editing Menus—Capture and Playout (page 2359).

Press:	To:
Shift+click Insert button	Record a single shot to a tape that already contains material. This mode plays out the video and/or audio (the existing timecode is not affected).
Shift+click Assemble button	Record a single shot to a tape that already contains material. This mode plays out the video (and/or audio) and timecode to tape.

Colour Grading

Use colour grading hotkeys to perform colour grading operations carried out in the Grading, Curves, and Secondaries menus.

For numeric keypad hotkeys that pertain to colour grading, see Colour Menus — Grading and Secondaries (page 2356).

Global Colour Grading Hotkeys

Use these hotkeys in any of the Colour menus.

Press:	To:
C	Copy the parameters in the Selector tool from the current shot to the selected shot(s).
RR	Reset all the current shot's parameters to the default setting.
U	Undo the last action.
S	Save the current grade files and cut. If a grade file has not been created, a default grade file is saved.
G	Switch between Gang and Solo mode when in Group mode.
Y	Enable/disable GPU acceleration.
Ctrl (left)+H	To hide or show the selected geometry.
Page Up (Secondaries menu)	Increase the Hue Rotator value.
Page Down (Secondaries menu)	Decrease the Hue Rotator value.
Page Up (other menus)	Go to the Saturation curve in the Curves menu.
Insert	Go to the Hue curve in the Curves menu.
Home (Secondaries menu)	Increase the saturation of a secondary when a layer is enabled, or of the entire image when either the Input or Output button is enabled.
Home (other menus)	Go to the Lightness curve in the Curves menu.
End (Secondaries menu)	Decrease the saturation.
Ctrl+click Delete button	Delete a grade setup.
Print Scrn	Go to the Grading menu.
Scroll Lock	Go to the Secondaries menu. Press it again to go to the next secondary.
Right-click menu	Display the preset menu for grading curves or secondaries.

Press:	To:
/ (slash)	Auto-grade with the Balance Input Primaries for the selected colour. The master (brightness or lift/gain) is not modified. You can auto-grade the master as well by Ctrl-clicking the image.
Click	Pick the colour at the current position. **NOTE** The Show button must be disabled for this to work.
Shift-drag	Drag an area to calculate this area's mean colour, and show it in the colour picker. In monitoring view, this defines the sub-region as well. If you picked a colour, you can then perform colour matching. **NOTE** The Show button must be disabled for this to work.
Alt-click	Sample an individual pixel and set the Brightness Balance wheel to match the reference sample. You can drag through the image until you locate the pixel you want to sample, then Alt-click.
Ctrl-click	Sample an individual pixel and set the Brightness Balance wheel and Brightness slider to match the reference sample. You can drag through the image until you locate the pixel you want to sample, then Ctrl-click.

Grade Bin Hotkeys

Use these hotkeys for the Grade bin.

Press:	To:
Shift+– (hyphen)	Cycle between Grade bin locations (cursor must be outside file browser or expanded Grade bin).
Double-click	Load all the settings for the selected preset thumbnail to the current shot only.
Right-click	Save the stored image to the framestore.
Shift-double-click	Load all the settings for the selected preset thumbnail to selected shots.
Ctrl-double-click	Load current menu settings for the selected preset thumbnail to the current shot only.
Ctrl+Shift-double-click	Load current menu settings for the selected preset thumbnail to a selection of shots.
Alt-double-click	Load specified parameters in the Selector to the current shot only.
Alt+Shift-double-click	Load specified parameters in the Selector to a selection of shots.

Expanded Grade Bin Hotkeys

Use these hotkeys for the expanded Grade bin.

Press:	To:
Right-click	Edit a note in the List view.
Shift+– (hyphen)	Cycle between expanded Grade bin locations (cursor must be over file browser or expanded Grade bin).
Shift-click	Select contiguous grade files.
Ctrl-click	Select multiple grade files.
Ctrl-drag	Copy a grade file to a Grade bin storage container.
Shift-double-click	Load all the settings for the selected grade file to selected shots.
Ctrl-double-click	Load current menu settings for the selected grade file to the current shot only.
Ctrl+Shift-double-click	Load current menu settings for the selected grade file to a selection of shots.
Alt-double-click	Load specified parameters in the Selector to the current shot only.
Alt+Shift-double-click	Load specified parameters in the Selector to a selection of shots.

Secondary Layers Hotkeys

Use these hotkeys when working with the secondary layer panel.

Press:	To:
1, 2, 3, 4, 5, 6, 7, 8, 9, 0, -, =	Select a secondary layer (1-12, 13-24, 25-36, or 37-48).
\	Activate/deactivate the selected secondary layer.
Ctrl+(1, 2, 3, or 4)	Select the secondary layers page 1, 2, 3, or 4.
Scroll Lock	Cycle through all the activated secondary layers.
Ctrl-click secondary layer button	Copy the grading information from the current menu (e.g., Grading, Curves, or Secondaries) to the selected secondary layer.
Shift-click secondary layer button	Copy all the colour grading information to the selected secondary layer.

Curves Menu Hotkeys

Use these hotkeys in the Curves menu.

Press:	To:
A	Add a vertex to the mouse position in the current curve.
Shift+A	Lock the curve in place while adding a vertex.
D	Delete the selected vertex from the current curve.
R	Reset the current shot's curve parameters to the default setting.
Right-drag	Zoom in or out.
Middle-drag	Pan the curve.
Alt-drag	Zoom in on a curve using rectangle zoom.
Alt-right-drag	Zoom in on a curve using proportional zoom.
Ctrl (right)-left-click	Reset the curve zoom in/out properties to the default setting.
Ctrl (right)-right-click	Reset the curve viewer.
Shift-drag	Restrict movement of selected vertices to the Y axis.
Shift+Alt-drag	Restrict movement of selected vertices to the X axis.

Secondaries Menu Hotkeys

Use these hotkeys in the Secondaries menu.

Press:	To:
, (comma)	Show or hide a geometry.
R	Reset all secondary colour correction parameters for the current shot.
Shift+D	Enable Del Geom, and then delete a geometry.
A	Enable Add Pts, and then add vertices to a geometry.
D	Enable Del Pts, and then delete vertices from a geometry.

Press:	To:
Z	Enable the Rotate button, and then click on the geometry's axis to rotate.
Right-click-drag	Rotate the geometry when you right-click its axis and drag.
Click-drag (on Z-axis)	Proportional scale left.
T	Enable the Move button, and then move a geometry's axis, vertex, or tangent.
X	Enable the Scale button, and then scale a geometry. You can scale a geometry along the X-axis, Y-axis, or you can scale it proportionally using the Global Scaling axis.
Backspace (tracking)	Delete the latest tracked position.
Right-click (basic geometry)	Convert a basic geometry to a free-form geometry.
Ctrl-click (free-form geometry)	Break or repair a tangent handle when you click one of the geometry's vertices.
Alt-right-click	Offset the axis.

Timeline Menu Hotkeys

Use these hotkeys in the colour grading Timeline menu.

Press:	To:
Shift+C	To make a cut in an element.
Alt+S	Enable or disable Solo mode.
Up Arrow	Move the focus up.
Down Arrow	Move the focus down.
F7	Displays the timeline, Player, grading tools, and keyer (HLS or Diamond). Also activates the SDI out if the GFX SDI is enabled.
Ctrl+drag+drop	Drag and drop the grade from one shot to another.
Ctrl+Shift+drag+drop	Drag and drop the grade from one shot to a selection of shots.
Ctrl+Alt+drag+drop	Drag and drop the Selector parameters to one shot.
Ctrl+Alt+Shift+drag+drop	Drag and drop the Selector parameters to a selection of shots.

Numeric Keypad

The numeric keypad hotkeys vary depending on the menu. The following section includes hotkey tables for the following menus:

- Colour—Grading and Secondaries menus
- Editing—Edit menu
- Editing—Capture and Playout menus

Colour Menus — Grading and Secondaries

Refer to the following table for numeric keypad hotkey functions when working in the Grading and Secondaries menus.

Press:	To:
1	Decrease cyan brightness by 1 point.
Shift+1	Decrease cyan brightness by 1/2 point.
2	Decrease magenta brightness by 1 point.
Shift+2	Decrease magenta brightness by 1/2 point.
3	Decrease yellow brightness by 1 point.
Shift+3	Decrease yellow brightness by 1/2 point.
4	Increase cyan brightness by 1 point.
Shift+4	Increase cyan brightness by 1/2 point.
5	Increase magenta brightness by 1 point.
Shift+5	Increase magenta brightness by 1/2 point.
6	Increase yellow brightness by 1 point.
Shift+6	Increase yellow brightness by 1/2 point.
7	Decrease red brightness by 1 point.
Shift+7	Decrease red brightness by 1/2 point.
8	Decrease green brightness by 1 point.
Shift+8	Decrease green brightness by 1/2 point.

Press:	To:
9	Decrease blue brightness by 1 point.
Shift+9	Decrease blue brightness by 1/2 point.
0	Copy the same type setting from the previous shot.
Enter	Decrease master brightness by 1 point.
Shift+Enter	Decrease master brightness by 1/2 point.
+ (add)	Increase master brightness by 1 point.
Shift+'+' (add)	Increase master brightness by 1/2 point.
– (subtract)	Reset brightness.
/ (divide)	Increase green brightness by 1 point.
Shift+/ (divide)	Increase green brightness by 1/2 point.
* (multiply)	Increase blue brightness by 1 point.
Shift+* (multiply)	Increase blue brightness by 1/2 point.
NumLock	Increase red brightness by 1 point.
Shift+NumLock	Increase red brightness by 1/2 point.

Curves Menu — Hue, Lightness, Saturation and Luminance

Refer to the following table for numeric keypad hotkey functions when working on the Hue, Lightness, Saturation or Luminance values in the Curves menu.

Press:	To:
1	Decrease the cyan point's hue, lightness, saturation, or luminance value.
2	Decrease the magenta point's hue, lightness, saturation, or luminance value.
3	Decrease the yellow point's hue, lightness, saturation, or luminance value.
4	Increase the cyan point's hue, lightness, saturation, or luminance value.
5	Increase the magenta point's hue, lightness, saturation, or luminance value.

Press:	To:
6	Increase the yellow point's hue, lightness, saturation, or luminance value.
7	Decrease the red point's hue, lightness, saturation, or luminance value.
8	Decrease the green point's hue, lightness, saturation, or luminance value.
9	Decrease the blue point's hue, lightness, saturation, or luminance value.
Enter	Decrease all points' hue, lightness, saturation, or luminance value.
+ (add)	Increase all points' hue, lightness, saturation, or luminance value.
/ (divide)	Increase the green points' hue, lightness, saturation, or luminance value.
* (multiply)	Increase the blue points' hue, lightness, saturation, or luminance value.
– (subtract)	Reset the current curve.
NumLock	Increase the red point's hue, lightness, saturation, or luminance value.

Editing Menu—Edit

Refer to the following table for numeric keypad hotkey functions when working in the Edit menu.

Press:	To:
1	Enter 1 in the timecode field.
2	Enter 2 in the timecode field.
3	Enter 3 in the timecode field.
4	Enter 4 in the timecode field.
5	Enter 5 in the timecode field.
6	Enter 6 in the timecode field.
7	Enter 7 in the timecode field.
8	Enter 8 in the timecode field.
9	Enter 9 in the timecode field.

Press:	To:
0	Enter 0 in the timecode field.
. (decimal)	Enter 00 in the timecode field.
+ (add)	Set the edit operation to “plus.”
/ (divide)	Reset the current timecode.
* (multiply)	Switch between the trim in, trim out, slide and default editing modes.
– (subtract)	Set the edit operation to “minus.”

Editing Menus—Capture and Playout

Refer to the following table for numeric keypad hotkey functions when working in the Capture and Playout menus.

Press:	To:
1	Enter 1 in the timecode field.
2	Enter 2 in the timecode field.
3	Enter 3 in the timecode field.
4	Enter 4 in the timecode field.
5	Enter 5 in the timecode field.
6	Enter 6 in the timecode field.
7	Enter 7 in the timecode field.
8	Enter 8 in the timecode field.
9	Enter 9 in the timecode field.
0	Enter 0 in the timecode field.
. (decimal)	Enter 00 in the timecode field.
/ (divide)	Reset the current timecode.
* (multiply)	Switch the timecode typing focus to the in timecode.

Press:	To:
– (subtract)	Switch the timecode typing focus to the out timecode.

Stereoscopy

Use these hotkeys when the Stereoscopy feature is enabled.

Press:	To:
Alt+L	Assigns the layer the focus point is positioned on as the Left Eye.
Alt+R	Assigns the layer the focus point is positioned on as the Right Eye.
; (semicolon)	Toggles the Player to display the Left Eye or Right Eye footage.
Ctrl+F4	Cycle through the Stereo Dual View modes (i.e., 2-up, Blend, Horizontal Wipe, and Vertical Wipe).
Shift+Alt+F4	Return to the previous Stereo Dual View mode.

Animation

Use these hotkeys in the Animation menu.

Press:	To:
Alt+A	Enable/disable Autokey.
Delete	Delete the selected keyframe.
Drag	Draw a selection box.
Right-drag	Zoom in or out.
Middle-drag	Pan the view.
Backspace	Delete last tracked position.
Ctrl (left)+C	Copy keyframe(s).
Ctrl (left)+V	Paste keyframe(s).
Alt+C	Toggle between 'One Fr' and All Fr' copy modes.

Miscellaneous Functions

These hotkeys are for assorted tasks (e.g., text editing, toggling render settings, and manipulating geometries).

Press:	To:
Right-click	Scroll through the option box options in reverse order. For example, refer to the Filter option box within Image > Transcode > Format settings.
`Shift+Insert`	Exit Lustre without confirmation.
`Shift+Esc`	Cancel Burn buffer preallocation.
`Shift+F1`	Access Lustre help system.
`Shift`-drag	Quickly increase/decrease the value of colour wheels and certain sliders. Use this hotkey to allow you to surpass the minimum/maximum value restricted by using the user interface.
`Alt`-drag	Slowly increase/decrease the value of colour wheels and certain sliders. Use this hotkey to allow you to surpass the minimum/maximum value restricted by using the user interface.
`Tab`	Move the focus to the next text field in the current menu if a text field has focus.
`Ctrl+C`	Copy selected text.
`Ctrl+V`	Paste copied text into the current text field.
`Ctrl`-click	Reset a parameter when you click the parameter's user interface element.
`Shift+B`	Bypass all menu parameters.
`R`	Resets the current menu.
`Ctrl+R`	Refresh the displayed list to show any changes you have made. You can press `Ctrl+R` to refresh the Cutlist, EDL list, Audio list, overlay list, LUT list and Presets list.
`Ctrl+Backspace` (Presets menu)	Delete the active preset.
`Ctrl+Backspace` (Gang tool)	Delete the active gang.
`M`	Start result caching.
`N`	Switch off the result cache.
`[` (left bracket)	Switch off the render flag of the current shot.

Press:	To:
] (right bracket)	Switch on the render flag of the current shot.
{ (left brace)	Switch off the render flag of all the shots.
} (right brace)	Switch on the render flag of all the shots.
X	Switch on/off background rendering.
Shift+W	Switch on/off Incinerator.
Ctrl+Z	Undo the last operation.
Ctrl+Alt+Z	Reverse the last 'Undo' operation.

XML in Lustre

Lustre's XML-based files are stored as plain text on disk so that you may view, generate, and, in certain cases, modify them. The three types of XML-based files are:

- CUT (**.cut.xml*)
- CLIP (**.clipixml*)
- GRADENOTES (**.desc.xml*)

The XML files are generated by Lustre and should be modified sparingly and with caution, as errors can cause serious malfunctions. An example of a modification you could safely make would be to change the Note field in the DESC file. The Note field contains a shot's note text. See Assigning Notes to Shots (page 1807).

Limitations

Lustre's XML-based files have the following limitations:

- Certain functions have not been implemented.
- You can only include hard returns in the structure tags. You must include all other tags and their contents (opening tag, value, and closing tag) in a single line.
- You can use XML features that are included in the predefined tag list only. This is to ensure optimized performance.
- The Lustre XML parser ignores tags that you use to store additional information.

CUT File Tags

The CUT file is located in the Library directory and contains the following shot information:

- Physical location of files, trims, and additional metadata information (such as EDL and keycode data).
- Clip information (such as timecode and keycode data).

- Timeline information (such as framerate and video tracks).
- Transcode information, if relevant.
- Retime effect parameters.
- Dissolve parameters.

The CUT file is built from the sequence of shots, and its order determines the order of shots on the Lustre Timeline.

When modifying the CUT file, you are restricted to the following tags.

NOTE Items in the Children row are occasionally marked with a + (plus sign) or * (asterisk). The + indicates that the parent tag can contain more than one instance of this element. The * indicates that the element is mandatory.

Cut	
Description	The root tag that contains opening and closing tags. It is a structure tag and as such can include hard returns.
Parameters	For the current release the only parameter allowed is **Ver**="**3.0**"
Parents	N/A
Children	+*Shot

Shot	
Description	A single shot (uninterrupted sequence of images). It is a structure tag and as such can include hard returns. The parent cut tag can contain any number of shots.
Parameters	N/A
Parents	Cut
Children	+Seq, +AliasName, +EData

Seq	
Description	The image sequence location. For example: **[HOME]/scene1/1828x1332/%05d.dpx*7751**.
Parameters	N/A
Parents	Shot

Seq	
Children	N/A

AliasName	
Description	The shot's short name. An asterisk (*) indicates a previous wedge number. You can make the alias name appear in the Library and the browser by pressing `W` on the keyboard.
Parameters	N/A
Parents	Shot
Children	N/A

EData	
Description	The structure for storing editing metadata.
Parameters	N/A
Parents	Shot
Children	+UniqueId, +SrcFullStart, +SrcFullLength, +SrcTrimStart, +SrcTrimLength, +ResStart, +ResLength, FadeType, FadeStartFromEnd, FadeSize, FadeCurve, RetimeRefFrame, RetimeSpeed, RetimeRemain, EDLEvent, EDLFadeDur, EDLFrameRate, EDLReelName, EDLFadeType, EDLSrcTcIn, EDLSrcTcOut, EDLRecTcIn, EDLRecTcOut, EDLFadeDur, EDLRetimeSpeed, EDLRetimeRef, EDLFadeUID, EDLKeyCodeStart, EDLKeyCodeEnd, DLEDLRes, DLEDLStartTc, DLEDLFrameIdList, DLEDLClipName, MediaUnlinked, MediaPtr, MediaDef, MediaLinkData, DPXTimeCodeStart, DPXTimeCodeEnd, DPXKeyCodeStart, DPXKeyCodeEnd, DPXFrameRate, DPXCodeRes, DPXKeyCodeType

UniqueId	
Description	A shot's unique identification string. If generated internally, do not modify. This would result in grade loss since shots and grades are linked by the unique ID. If the CUT file was generated externally, the ID must respect the following conditions: it must be unique within the cutlist, and the data type must be a 64-bit integer in a hexadecimal format. See CUT File Data Types (page 2376).
Parameters	Type
Parents	EData

UniqueId	
Children	N/A

SrcFullStart	
Description	The first frame of the original image sequence. The frame number is converted and displayed as timecode.
Parameters	N/A
Parents	EData
Children	N/A

SrcFullLength	
Description	The shot's number of frames.
Parameters	N/A
Parents	EData
Children	N/A

SrcTrimStart	
Description	The trim start frame. This value should be greater than the SrcFullStart value and smaller than the sum of SrcFullStart plus SrcFullLength.
Parameters	N/A
Parents	EData
Children	N/A

SrcTrimLength	
Description	The trimmed shot length. The sum of SrcTrimStart plus SrcTrimLength should always be smaller than the sum of SrcFullStart plus SrcFullLength.
Parameters	N/A
Parents	EData

SrcTrimLength	
Children	N/A

ResStart	
Description	The start record frame. The frame number can be converted to timecode if required.
Parameters	N/A
Parents	EData
Children	N/A

ResLength	
Description	The record length in frames.
Parameters	N/A
Parents	EData
Children	N/A

FadeType	
Description	The transition type. Type 1 is a dissolve, and type 0 is a cut.
Parameters	N/A
Parents	EData
Children	N/A

FadeStartFromEnd	
Description	The starting frame of the transition, counting from the end of the current shot.
Parameters	N/A
Parents	EData

FadeStartFromEnd	
Children	N/A
FadeSize	
Description	Length of the transition in frames.
Parameters	N/A
Parents	EData
Children	N/A
FadeCurve	
Description	Container for transition curves. There are two types of transition curves: InChannel and OutChannel. The InChannel transition curve determines the transparency of the current shot, whereas the OutChannel transition curve determines the transparency of the overlapped shot.
Parameters	N/A
Parents	EData
Children	+InChannel, +OutChannel
InChannel	
Description	The transition curve that determines the transparency of the current shot.
Parameters	N/A
Parents	EData
Children	+*Point
OutChannel	
Description	The transition curve that determines the transparency of the overlapped shot.
Parameters	N/A
Parents	EData

OutChannel	
Children	+*Point

Point	
Description	One control point on a curve. For dissolves, the curve calculation is based on Hermite interpolation, and all values are normalized to a 0-65535 range.
Parameters	N/A
Parents	InChannel, OutChannel
Children	+X, +Y, +TangentX, +TangentY

X	
Description	X coordinate of a control point.
Parameters	N/A
Parents	Point
Children	N/A

Y	
Description	Y coordinate of a control point.
Parameters	N/A
Parents	Point
Children	N/A

Tangent X	
Description	The absolute X position of the tangent point. This is unrelated to the tangent angle.
Parameters	N/A
Parents	Point

Tangent X	
Children	N/A

Tangent Y	
Description	The absolute Y position of the tangent point. This is unrelated to the tangent angle.
Parameters	N/A
Parents	Point
Children	N/A

RetimeRefFrame	
Description	The retime's base point in frames. This is usually equivalent to the SrcTrimStart.
Parameters	N/A
Parents	EData
Children	N/A

RetimeSpeed	
Description	Speed change rate. Supported data type is long float (“lf”, aka double). For example, normal speed is 1.0, half speed is 0.5, double speed is 2.0, -1.0 is normal speed reverse play, -0.5 is half speed reverse play, and -2.0 is double speed reverse play.
Parameters	Type
Parents	EData
Children	N/A

RetimeRemain	
Description	Value for compensating rounding errors. This tag is used only if a retimed shot was retrimmed. Otherwise its value is zero. Supported data type is long float (“lf”, aka double).
Parameters	Type
Parents	EData

RetimeRemain	
Children	N/A
EDLEvent	
Description	EDL event number.
Parameters	N/A
Parents	EData
Children	N/A
EDLFadeDur	
Description	EDL fade duration in frames.
Parameters	N/A
Parents	EData
Children	N/A
EDLFrameRate	
Description	EDL framerate. This tag's parameter is the float data type. See CUT File Data Types (page 2376).
Parameters	Type
Parents	EData
Children	N/A
EDLReelName	
Description	EDL reel name. The tag's parameter is the string data type. See CUT File Data Types (page 2376).
Parameters	Type
Parents	EData

EDLReelName	
Children	N/A

EDLFadeType	
Description	The EDL fade type. Legal values include: -1= cut -2=dissolve >=SMPTE wipe type number
Parameters	N/A
Parents	EData
Children	N/A

EDLSrcTcIn	
Description	EDL source in timecode. This tag's parameter is the timecode data type. See CUT File Data Types (page 2376).
Parameters	Type
Parents	EData
Children	N/A

EDLSrcTcOut	
Description	EDL source out timecode. This tag's parameter is the timecode data type. See CUT File Data Types (page 2376).
Parameters	Type
Parents	EData
Children	N/A

EDLRecTcIn	
Description	EDL record in timecode. This tag's parameter is the timecode data type. See CUT File Data Types (page 2376).
Parameters	Type
Parents	EData

EDLRecTcIn	
Children	N/A

EDLRecTcOut	
Description	EDL record out timecode. This tag's parameter is the timecode data type. See CUT File Data Types (page 2376).
Parameters	Type
Parents	EData
Children	N/A

EDLFadeDur	
Description	EDL fade duration in frames.
Parameters	N/A
Parents	EData
Children	N/A

EDLRetimeSpeed	
Description	EDL retime as a percentage value.
Parameters	N/A
Parents	EData
Children	N/A

EDLRetimeRef	
Description	The retime's base point in frames. This is usually equivalent to the SrcTrimStart.
Parameters	N/A
Parents	EData

EDLRetimeRef	
Children	N/A

EDLFadeUID	
Description	Unique ID of the second shot in the dissolve.
Parameters	N/A
Parents	EData
Children	N/A

EDLKeyCodeStart	
Description	EDL keycode start. This tag's parameter is the keycode data type. See CUT File Data Types (page 2376).
Parameters	N/A
Parents	EData
Children	N/A

EDLKeyCodeEnd	
Description	EDL keycode end. This tag's parameter is the keycode data type. See CUT File Data Types (page 2376).
Parameters	N/A
Parents	EData
Children	N/A

DLEDLRes	
Description	String from the DLEDL that describes the image format.
Parameters	N/A
Parents	EData

DLEDLRes	
Children	N/A
DLEDLStartTc	
Description	Start timecode from the DLEDL.
Parameters	N/A
Parents	EData
Children	N/A
DLEDLFrameIdList	
Description	Pointer to the frame ID list, which is used by the system to locate frames on the Stone filesystem. Supported data type is integer. See CUT File Data Types (page 2376).
Parameters	N/A
Parents	EData
Children	N/A
DLEDLClipName	
Description	DLEDL clip name.
Parameters	N/A
Parents	EData
Children	N/A
DPXTimeCodeStart	
Description	DPX header timecode start.
Parameters	N/A
Parents	EData

DPXTimeCodeStart	
Children	N/A

DPXTimeCodeEnd	
Description	DPX header timecode end.
Parameters	N/A
Parents	EData
Children	N/A

DPXKeyCodeStart	
Description	DPX header keycode start.
Parameters	N/A
Parents	EData
Children	N/A

DPXKeyCodeEnd	
Description	DPX header keycode end.
Parameters	N/A
Parents	EData
Children	N/A

DPXFrameRate	
Description	DPX header frame rate.
Parameters	N/A
Parents	EData

DPXFrameRate	
Children	N/A

DPXCodeRes	
Description	The resolution of the DPX header origin (full or half).
Parameters	N/A
Parents	EData
Children	N/A

DPXKeyCodeType	
Description	The number of frames stored on one foot of film.
Parameters	N/A
Parents	EData
Children	N/A

CUT File Data Types

The following data types are supported in CUT file tags.

Data Type	Description
f	Floating point number. These can be always expressed as a fraction of two integers. For example: -147.244
s	Sequence of ASCII characters. These can include both letters and numbers. For example: abcdef_12345-ABCDEF
tc	Encoded timing format. Format: “hh:mm:ss:ff” where hh=hours, mm=minutes, ss=seconds, and ff=frames. For example: 01:23:45:23
kk	Film-based timing format. Format: “TT PPPP FFFF+OO” where TT=film type, PPPP=prefix, FFFF=feet frames, and OO=offset. For example: KI 120863 7624+07
u	Unsigned integer. These are the positive numbers, and are usually stored in 32-bit. For example: 1
i64	64-bit integer. These can be positive or negative numbers, and are usually stored in 64-bit. For example: -7846238974878347

If you want to use an integer, do not specify a data type.

CUT File Example

The following is an example of an XML-based CUT file.

```
<Timeline Ver="1.0">
    <TimelineName Type="s">PAL_sample_cut</TimelineName>
    <StereoEnabled>0</StereoEnabled>
    <TimelineFrameRate Type="f">25</TimelineFrameRate>
    <TimelineDropMode Type="s">0</TimelineDropMode>
    <TimelineFullRes>
        <Width>720</Width>
        <Height>576</Height>
        <BitsPerPixel>32</BitsPerPixel>
        <NumChannels>3</NumChannels>
        <IsFloat Type="s">false</IsFloat>
        <FrameBufferSize>1658880</FrameBufferSize>
        <FrameByteOrder Type="s">LE</FrameByteOrder>
        <AspectRatio Type="f">1.33333</AspectRatio>
        <ScanFormat Type="s">P</ScanFormat>
        <FrameRate Type="f">25</FrameRate>
        <DropFrameMode Type="s">0</DropFrameMode>
    </TimelineFullRes>
    <videotrack idx="0" trackType="Primary">
        <videolayer idx="0">
            <OutEye>Mono</OutEye>
            <Shot>
                <FullRes>
                    <Width>720</Width>
                    <Height>576</Height>
                    <BitsPerPixel>32</BitsPerPixel>
                    <NumChannels>3</NumChannels>
                    <IsFloat Type="s">false</IsFloat>
                    <FrameBufferSize>1658880</FrameBufferSize>
                    <FrameByteOrder Type="s">BE</FrameByteOrder>
                    <AspectRatio Type="f">1.33333</AspectRatio>
                    <ScanFormat Type="s">P</ScanFormat>
                    <FrameRate Type="f">23.976</FrameRate>
                    <DropFrameMode Type="s">0</DropFrameMode>
                </FullRes>
                <Seq>
                    <ClipName>3d_track_NIGHT__4E84A13F000047</ClipName>
                    <File>&lt;HOME&gt;/PAL/DPX/3d_track_NIGHT/720x576/3d_track_night_shot_grd01.%04d.dpx</File>
                </Seq>
                <EData>
                    <UniqueId Type="i64">4E84A13F000047</UniqueId>
                    <ShotType>Source</ShotType>
                    <LibraryId Type="i64">4E4C202C000228</LibraryId>
                    <SrcFullStart>141</SrcFullStart>
                    <SrcFullLength>78</SrcFullLength>
                    <SrcTrimStart>161</SrcTrimStart>
                    <SrcTelecineOffset>0</SrcTelecineOffset>
                    <SrcTrimLength>44</SrcTrimLength>
                    <ResStart>0</ResStart>
                    <ResLength>44</ResLength>
                    <Track>0</Track>
                    <Layer>0</Layer>
                    <FolderReelName Type="s">3d_track_NIGHT</FolderReelName>
                    <DPXDropFrameMode>0</DPXDropFrameMode>
                    <DPXFrameRate Type="f">23.976025</DPXFrameRate>
                    <DLEDLPixelRatio Type="f">1.066664</DLEDLPixelRatio>
                    <DPXTimeCodeStart Type="tc">00:00:05:21</DPXTimeCodeStart>
                    <DPXTimeCodeEnd Type="tc">00:00:09:02</DPXTimeCodeEnd>
                    <DPXCodeRes>0</DPXCodeRes>
                    <ThumbnailPath Type="s">/mnt/StorageMedia/projects/Bernice_2012x1/Thumbnails/4E84A13F0000470.rgba</ThumbnailPath>
                    <TL_DRAGED_TC>0</TL_DRAGED_TC>
                    <TL_ALIAS_ID Type="s">3d_track_NIGHT</TL_ALIAS_ID>
                    <TL_TRIM_RES_START_ID>0</TL_TRIM_RES_START_ID>
                    <TL_TRIM_RES_LEN_BAK_ID>52</TL_TRIM_RES_LEN_BAK_ID>
                    <TL_TRIM_SRC_START_ID>153</TL_TRIM_SRC_START_ID>
                    <TL_TRIM_SRC_LEN_BAK_ID>52</TL_TRIM_SRC_LEN_BAK_ID>
                    <EDLReelNameLong Type="s">3d_track_NIGHT</EDLReelNameLong>
                    <DLEDLStartTc Type="tc">00:00:05:21</DLEDLStartTc>
                    <DLEDLStartTcProxy Type="tc">00:00:05:21</DLEDLStartTcProxy>
                    <MediaUnlinked>7</MediaUnlinked>
                    <DLEDLPath Type="s">&lt;HOME&gt;/PAL/DPX/3d_track_NIGHT/720x576/</DLEDLPath>
                    <DLEDLFileName Type="s">3d_track_night_shot_grd01.%04d.dpx</DLEDLFileName>
                    <DLEDLRes Type="s">720:576:32:3:1.06666:1658880:BE:P:23.976:0</DLEDLRes>
                    <ClipFrameRate Type="f">23.976000</ClipFrameRate>
                    <ClipDropMode>0</ClipDropMode>
                    <TL_TMP_INT_ID>0</TL_TMP_INT_ID>
                    <ReelName Type="s">3d_track_NIGHT</ReelName>
                    <TL_SHOT_SELECTED>0</TL_SHOT_SELECTED>
                    <TL_DISOLVE_SELECTED>0</TL_DISOLVE_SELECTED>
                </EData>
                <ImgInfo Res="Full">
                    <Mallign>512</Mallign>
                    <Miniosz>512</Miniosz>
                    <Maxiosz>2147483136</Maxiosz>
                    <HeadSize>8192</HeadSize>
                    <Width>720</Width>
                    <Height>576</Height>
                    <DBFFlag>111</DBFFlag>
                    <FileType>0</FileType>
                    <ImageType>4</ImageType>
                    <WtSoft>0</WtSoft>
                </ImgInfo>
                <ImgInfo Res="Half">
```

Tangent Element Control Surface

The Tangent Element control surface is designed to improve interactivity when colour grading film and video footage. You can accomplish all of the main grading tasks you do in the Lustre user interface using the Tangent Element control surface.

The Modes

The basic Tangent Element workflow is mode based. This means that you set the appropriate mode for the grading tasks you wish to accomplish and the knob, button and trackball functions are dynamically updated. Each mode consists of multiple pages of parameters that can be cycled through.

To select a mode, press the appropriate button on the MF module, as indicated on the module's screen.

The following modes are supported by the Tangent Element Control Surface:

Grading Mode (Linear and Logarithmic)

The available functions in this mode correspond to those in the Colour/Grading menu in the application. By default, the control surface is in Grade Mode (Linear or Logarithmic, according to your project settings).

Curves Mode

The available functions in this mode correspond to those in the Colour/Grading menu in the application.

Geometry Mode

The available functions in this mode correspond to those in the Colour/Secondaries menu in the application.

Pan & Scan Mode

The available functions in this mode correspond to those in the Image/Reposition menu in the application.

HLS Keyer Mode

The available functions in this mode correspond to the HLS Keyer functions in the Colour/Secondaries menu in the application.

Diamond Keyer Mode

The available functions in this mode correspond to the Diamond Keyer functions in the Colour/Secondaries menu in the application.

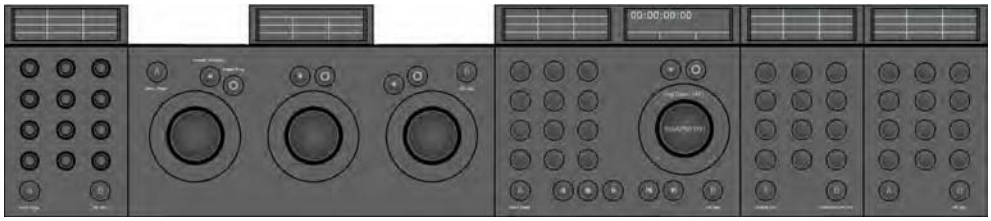
Editing Mode

The available functions in this mode correspond to those in the Editing/Timeline menu in the application.

Preferences Mode

The Preferences mode groups together various user set preferences.

Using the Modules



The Tangent Element Control Surface is composed of four modules.

The layout of the displays of each module mirrors the functions of the knobs/buttons of each module.

BEST PRACTICE It is recommended to add a second BT module to display and access commonly used functions at all times, otherwise only available the second and third pages of the MF module.

KB module (Knobs)

The KB module is a knob-based interface for user input parameters.

The values of each parameter assigned to the knobs can be reset by pressing the knob.

A button: Cycle through the different pages of functions mapped to the module.

B button: Enable either Precise Mode or Turbo Mode, based on your user settings.

TK module (Trackballs)

The TK module is also an interface for user input parameters. The trackballs and rings provide an intuitive way of precisely modifying parameters and are especially well suited for grading, among other things.

Moving the trackball from left to right / right to left has the same effect as moving them up and down / down and up.

Turning the ring from left to right increases the value of the assigned parameter, whereas turning the ring from right to left decreases the value.

The values of each parameter assigned to each trackball and ring can be reset by pressing the Reset button above them.

A button: Cycle through the different pages of functions mapped to the module.

B button: Enable either Precise Mode or Turbo Mode, according to your user settings, for all knobs, trackballs and rings.

MF module (Multi-Function)

The MF module enables you to select the mode you want to work in and provides a transport section, which allows navigation via the trackball and ring as well.

The values of each parameter assigned to each trackball and ring can be reset by pressing the Reset button above them.

A button: Cycle through the different pages of functions mapped to the module.

B button: Enable alternate functionality of the buttons, trackball and ring.

Trackball: The trackball's default behavior is scrub. By holding the B button, the behavior changes to pan.

Ring: The ring's default behavior is jog. By holding the B button, the behavior changes to zoom.

NOTE You can press the B button twice and lock Alt mode.

Button sequences for alternate functions on the MF module

- **Forward Play button + B Button:**Cycle forward through the grades applied to succeeding shots and temporarily apply them to the current shot. Pressing Enter permanently applies the selected grade to the current shot.
- **Backward Play button + B Button:**Cycle through the grades applied to previous shots and temporarily apply them to the current shot. Pressing Enter permanently applies the selected grade to the current shot.

- **Next Clip Button + B Button:** Move the positioner forward to the next keyframe in your timeline .
- **Previous Clip Button + B Button:** Move the positioner backward to the previous keyframe in your timeline.
- **Stop Button + Forward Play button:** Move the positioner forward by one frame in your timeline.
- **Stop Button + Backward Play button:** Move the positioner backward by one frame in your timeline.

BT (1) module (Buttons)

The BT module is dedicated to providing quick access to the secondaries and the Grade Bin. It is available in all modes, except Preferences Mode. The buttons mirror the position of the secondaries and of saved grades in the Grade Bin.

A button: allows you to enable/disable Grade Bin access.

B button: allows you to cycle through the four pages of secondaries, giving you access to all 48 secondaries.

BT (2) module (Buttons)

The second BT module is recommended. It enables you access at all times to View Modes and commonly used functions such as setting keyframes, copying and saving, among others.

A button: The A button on this module is not mapped. Pressing it will have no effect.

B button: Enable alternate functionality of the buttons, trackball and ring.

Installing the Control Surface

The first time you connect the Tangent Element Control Surface to your Lustre station, it automatically connects to your software and populates all the menus.

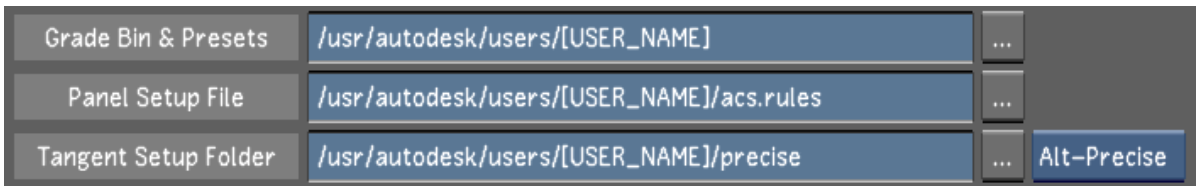
If the Control Surface does not automatically connect, see the Troubleshooting (page 2419) section.

Once it is connected, you must access the User Preferences and select whether you want the behavior of the B button on the KB and TK panels to be Precise Mode, which increases and decreases the parameter values in smaller increments, giving you more precision. Or Turbo Mode, which increases and decreases the parameter values in larger increments, enabling you to set your values quickly.

TIP You can create a user for each mode and select one or the other based on whether you want to work with Precise Mode or Turbo Mode.

To set the behavior of the B button

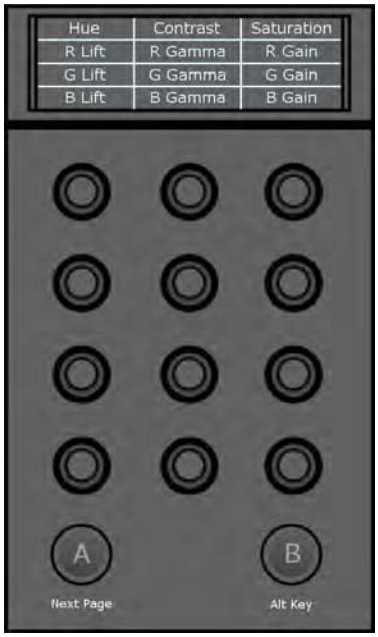
1. Access the Setup/Settings menu.
2. Press Edit, next to the User Settings.
3. Select the System & Menu tab.
4. Press the Alt-Precise/Alt-Turbo button to the right of the Tangent Setup Folder field to select either Precise or Turbo Mode.



Control Surface Layouts

Linear Grading Layout

Module 1 (KB) - Page 1 of 2

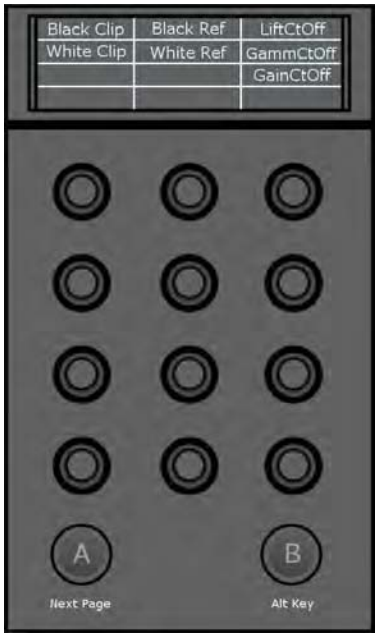


- **Hue:** Modify the hue value.

NOTE The hue can only be modified on secondaries.

- **R Lift:** Modify the lift of the red channel.
- **G Lift:** Modify the lift of the green channel.
- **B Lift:** Modify the lift of the blue channel.
- **Contrast:** Modify contrast value.
- **R Gamma:** Modify the gamma of the red channel.
- **G Gamma:** Modify the gamma of the green channel.
- **B Gamma:** Modify the gamma of the blue channel.
- **Saturation:** Modify the saturation value.
- **R Gain:** Modify the gain of the red channel.
- **G Gain:** Modify the gain of the green channel.
- **B Gain:** Modify the gain of the blue channel.

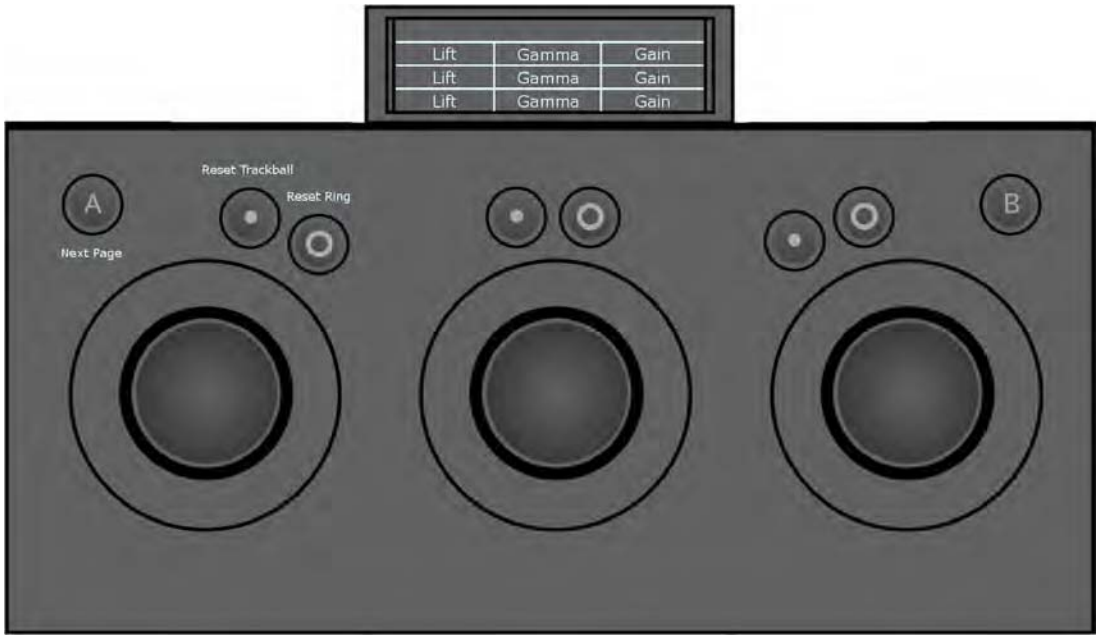
Module 1 (KB) - Page 2 of 2



- **Black Clip:** Modify the black clip value.
- **White Clip:** Modify the white clip value.
- **Black Ref:** Modify the black reference value.
- **White Ref:** Modify the white reference value.
- **LiftCtOff:** Modify the lift cutoff value.
- **GammaCtOff:** Modify the gamma cutoff value.
- **GainCtOff:** Modify the gain cutoff value.

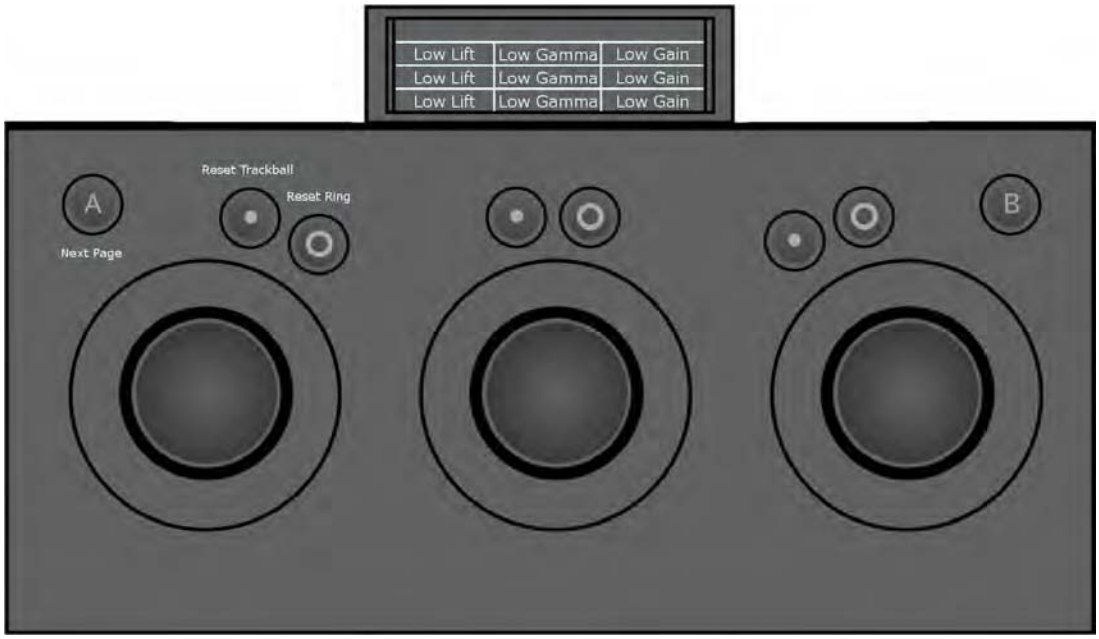
Module 2 (TK) - Page 1 of 4

NOTE The Grade mode functions are always available, even when not in Grading mode. To access the Grading functions while in other modes, press the A button to cycle through the Grading pages. Pressing the A button twice brings you back to the first page of the current mode.



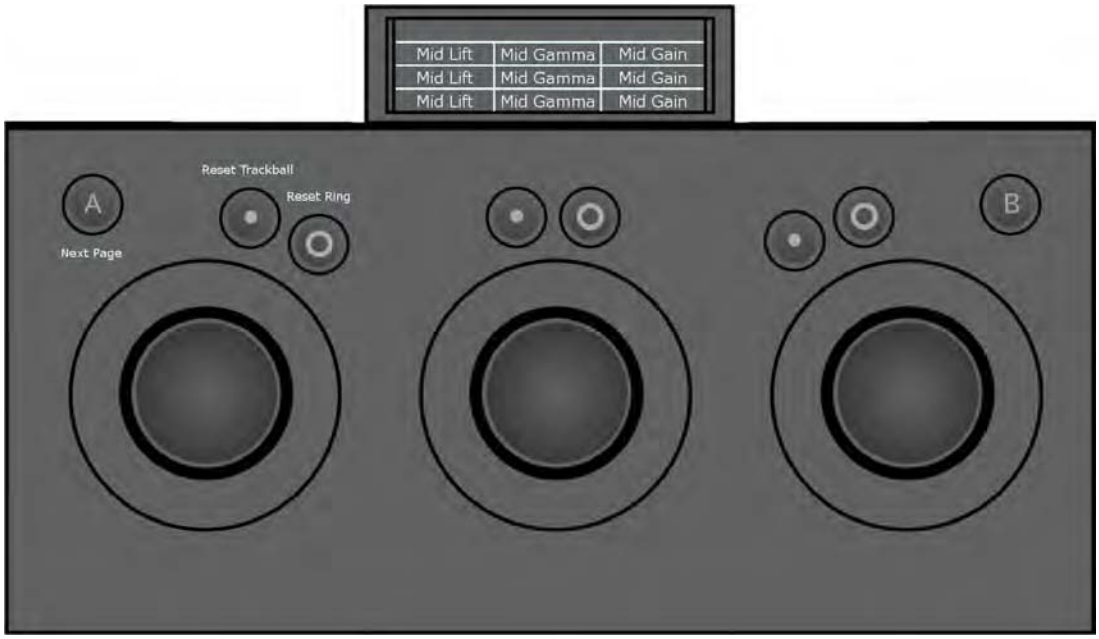
- **Lift:** Move the trackball right or left (x axis) to modify the lift in the low, mid and high range colour values (Master).
- **Lift:** Move the trackball up or down (y axis) to modify the lift in the low, mid and high range colour values (Master).
- **Lift:** Move the ring left or right to modify the lift of the luminance channel.
- **Gamma:** Move the trackball right or left (x axis) to modify the gamma in the low, mid and high range colour values (Master).
- **Gamma:** Move the trackball up or down (y axis) to modify the gamma in the low, mid and high range colour values (Master).
- **Gamma:** Move the ring left or right to modify the gamma of the luminance channel.
- **Gain:** Move the trackball right or left (x axis) to modify the gain in the low, mid and high range colour values (Master).
- **Gain:** Move the trackball up or down (y axis) to modify the gain in the low, mid and high range colour values (Master).
- **Gain:** Move the ring left or right to modify the gain of the luminance channel.

Module 2 (TK) - Page 2 of 4



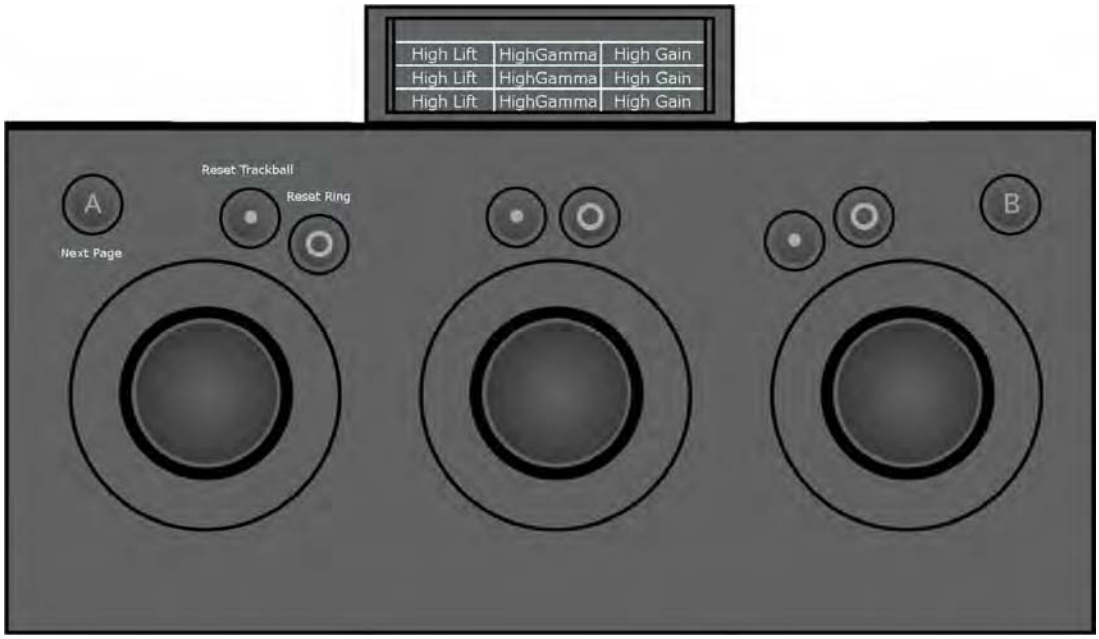
- **Low Lift:** Move the trackball right or left (x axis) to modify the lift in the low range colour values.
- **Low Lift:** Move the trackball up or down (y axis) to modify the lift in the low range colour values.
- **Low Lift:** Move the ring left or right to modify the lift of the luminance channel in the low range colour values.
- **Low Gamma:** Move the trackball right or left (x axis) to modify the gamma in the low range colour values.
- **Low Gamma:** Move the trackball up or down (y axis) to modify the gamma in the low range colour values.
- **Low Gamma:** Move the ring left or right to modify the gamma of the luminance channel in the low range colour values.
- **Low Gain:** Move the trackball right or left (x axis) to modify the gain in the low range colour values.
- **Low Gain:** Move the trackball up or down (y axis) to modify the gain in the low range colour values.
- **Low Gain:** Move the ring left or right to modify the gain of the luminance channel in the low range colour values.

Module 2 (TK) - Page 3 of 4



- **Mid Lift:** Move the trackball right or left (x axis) to modify the lift in the mid range colour values.
- **Mid Lift:** Move the trackball up or down (y axis) to modify the lift in the mid range colour values.
- **Mid Lift:** Move the ring left or right to modify the lift of the luminance channel in the mid range colour values.
- **Mid Gamma:** Move the trackball right or left (x axis) to modify the gamma in the mid range colour values.
- **Mid Gamma:** Move the trackball up or down (y axis) to modify the gamma in the mid range colour values.
- **Mid Gamma:** Move the ring left or right to modify the gamma of the luminance channel in the mid range colour values.
- **Mid Gain:** Move the trackball right or left (x axis) to modify the gain in the mid range colour values.
- **Mid Gain:** Move the trackball up or down (y axis) to modify the gain in the mid range colour values.
- **Mid Gain:** Move the ring left or right to modify the gain of the luminance channel in the mid range colour values.

Module 2 (TK) - Page 4 of 4



- **High Lift:** Move the trackball right or left (x axis) to modify the lift in the high range colour values.
- **High Lift:** Move the trackball up or down (y axis) to modify the lift in the high range colour values.
- **High Lift:** Move the ring left or right to modify the lift of the luminance channel in the high range colour values.
- **High Gamma:** Move the trackball right or left (x axis) to modify the gamma in the high range colour values.
- **High Gamma:** Move the trackball up or down (y axis) to modify the gamma in the high range colour values.
- **High Gamma:** Move the ring left or right to modify the gamma of the luminance channel in the high range colour values.
- **High Gain:** Move the trackball right or left (x axis) to modify the gain in the high range colour values.
- **High Gain:** Move the trackball up or down (y axis) to modify the gain in the high range colour values.
- **High Gain:** Move the ring left or right to modify the gain of the luminance channel in the high range colour values.

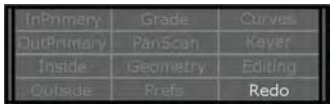
Module 3 (MF) - Page 1 of 3



- **InPrimary:** Enable input primary grading mode.
- **OutPrimary:**Enable output primary grading mode.
- **Inside:** Double-press to enable inside grading in secondary mode (grading affects the inside of your shape) for the currently selected shape. Inside grading is enabled by default. Single-press to select inside grading and edit your grade.

  NOTE Inside grading is enabled by default.

- **Outside:** Double-press to enable outside grading in secondary mode (grading affects the outside of your shape) for the currently selected shape. Once enabled, single-press to select outside grading, to edit your grade.
- **Grade:** Access the Colour/Grade menu.
- **PanScan:** Access the Image/Reposition menu.
- **Geometry:** Enable the draw geometry mode in the Secondaries menu. The fourth row of functions on the module is dynamically updated to display the rectangle, circle and wipe options. Pressing the corresponding button enables you to add the desired geometry.
- **Prefs:** Access the preferences functions.
- **Curves:** Access the Colour/Curves menu.
- **Keyer:** Enable the keyer in the secondaries menu. The default behavior is to enable the HLS keyer. Pressing Keyer again enables the diamond keyer.
- **Editing:** Access the Editing/Timeline menu.
- **Undo:** Undo the last operation performed. Pressing the B button changes the function of the Undo button to Redo. The panel display is be updated. Pressing the B button and Redo will redo the last operation (if you previously undid it).



- **Confirm:** When the Lustre UI prompts you to confirm an operation, the Undo button is automatically updated to display Confirm. Press to confirm the operation.



Module 3 (MF) - Page 2 of 3



- **InPrimary:** Enable input primary grading mode.
- **OutPrimary:**Enable output primary grading mode.
- **Inside:** Double-press to enable inside grading in secondary mode (grading affects the inside of your shape) for the currently selected shape. Inside grading is enabled by default. Single-press to select inside grading and edit your grade.

NOTE Inside grading is enabled by default.

- **Outside:** Double-press to enable outside grading in secondary mode (grading affects the outside of your shape) for the currently selected shape. Once enabled, single-press to select outside grading, to edit your grade.
- **O/P/M:** Toggle between output, print and matte views.
- **Autokey:** Enable/disable Autokey.
- **Reset:** Press once to reset the currently selected grading tool. Press twice to reset all grading tools.
- **SetKey:** Set a keyframe at the current position on the currently selected channel. Pressing the B button changes the function of the SetKey button to SetKey All. The panel display is updated.



- **SetKeyAll:** Set a keyframe at the current position on all channels.
- **A/B:**Toggle A/B playheads.
- **SingleView:** Toggle between single, dual and multi views, as defined under the Preferences.
- **Bypass:**Press once to bypass the currently selected grading tool. Press twice to bypass all grading tools.
- **Undo:** Undo the last operation performed. Pressing the B bnutton will change the function of the Undo button to Redo. The panel display is updated. Pressing the B button and Redo will redo the last operation (if you previously undid it).



Module 3 (MF) - Page 3 of 3



- **InPrimary:** Enable input primary grading mode.
- **OutPrimary:**Enable output primary grading mode.
- **Inside:** Double-press to enable inside grading in secondary mode (grading affects the inside of your shape) for the currently selected shape. Inside grading is enabled by default. Single-press to select inside grading and edit your grade.

NOTE Inside grading is enabled by default.

- **Outside:** Double-press to enable outside grading in secondary mode (grading affects the outside of your shape) for the currently selected shape. Once enabled, single-press to select outside grading, to edit your grade.
- **Stereo:** Enable stereoscopy.
- **L/R:** Toggle between tight eye view and left eye view modes (when working with stereoscopic footage).

- **MultiCopy:** Enter multi copy mode. When in multi copy mode, turn the ring to cycle your shots forward or backwards to copy the selected grade to your current shot. When you release the ring, the Undo button is automatically updated to display Apply. Press Apply to confirm and repeat the process for other shots as needed. To exit multi copy mode, press the MultiCopy button again.



- **SaveAll:** Save your current cut and grade.
- **Sync:** Sync your left and right eyes when working with stereoscopic footage. When in sync, any operation performed on the left eye is also applied to the right eye.
- **Cursor:** Enable cursor mode. When cursor mode is enabled, the trackball is used to move the cursor in the UI. In cursor mode, the Reset Trackball button acts as the left mouse button and the Reset Ring button acts as the right mouse button.
- **Copy:** Enter copy mode. When in copy mode, turn the ring to cycle your shots forward or backwards to copy the selected grade to your current shot. When you release the ring, the Undo button is automatically updated to display Apply. Press Apply to confirm the copy operation. Once the copy operation confirmed, you automatically exit copy mode.

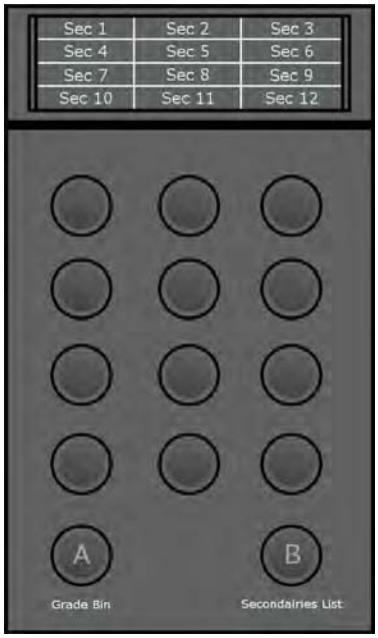


- **Undo:** Undo the last operation performed. Pressing the B button will change the function of the Undo button to Redo. The panel display will be updated. Pressing the B button and Redo will redo the last operation (if you previously undid it).



Module 4 (BT) - Page 1 of 6

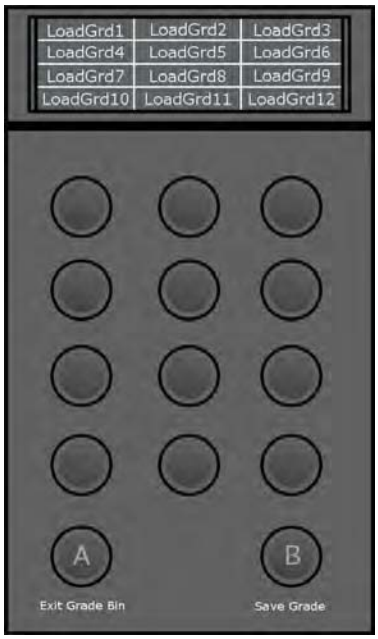
Secondaries List (4 pages, 12 secondaries per page).



Module 4 (BT) - Page 5 of 6

Grade Bin - Load

The BT Panel also controls your Grade Bin for saving and loading grades. The buttons mirror the position of saved grades in the Grade Bin.

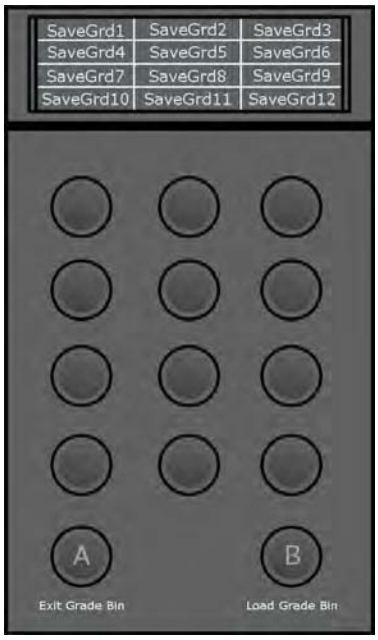


- **A Button:** Exit the Grade Bin.
- **B Button:** Toggle between load mode and save mode in the Grade Bin.

NOTE The trackball of the MF module navigates the Grade Bin when in Grade Bin mode.

Module 4 (BT) - Page 6 of 6

Grade Bin - Save

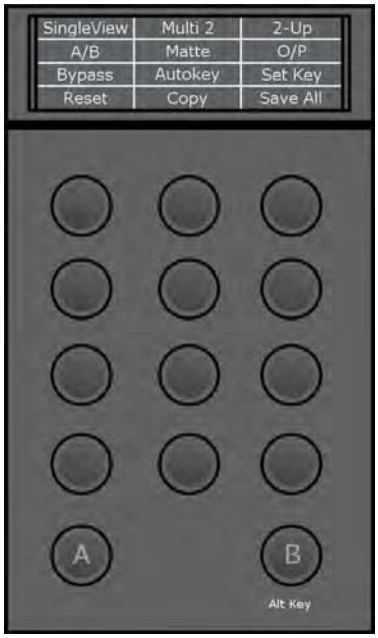


- **A Button:** Exit the Grade Bin.
- **B Button:** Toggle between load mode and save mode in the Grade Bin.

NOTE The trackball of the MF module navigates the Grade Bin when in Grade Bin mode.

Optional 2nd BT Module - Page 1 of 1

It is recommended to attach a second BT module, which is mapped to give you access to many of the most used functions at all times. These functions correspond to the second and third page of the MF module.



- **SingleView:** Toggle between single, dual and multi views, as defined under the Preferences.
- **A/B:** Toggle A/B playheads. Pressing Alt (B key) will change the function of the A/B button to L/R. The panel display is updated.



- **L/R:** Toggle between right eye view and left eye view modes (when working with stereoscopic footage).
- **Bypass:**Press once to bypass the currently selected grading tool. Press twice to bypass all grading tools.
- **Reset:**Press once to reset the currently selected grading tool. Press twice to reset all grading tools.
- **Multi 2:** Toggle the number of active viewports when in multi view. Options are:
  - 2
  - 4
  - 9
  - 16
- **Matte:** Enable/disable matte view.
- **Autokey:** Enable/disable Autokey.
- **Copy:** Enter copy mode. When in copy mode, turn the ring on the MF module to cycle your shots forward or backwards to copy the selected grade to your current shot. When you release the ring, the Undo button on the MF module is automatically updated to display Apply. Press Apply to confirm. Once the copy operation confirmed, you automatically exit copy mode. Pressing the B button changes the function of the Copy button to MultiCopy. The panel display is updated.



- **MultiCopy:** When in muilti copy mode, turn the ring on the MF module to cycle your shots forward or backwards to copy the selected grade to your current shot. When you release the ring, the Undo button on the MF module is automatically updated to display Apply. Press Apply to confirm and repeat the process for other shots as needed. To exit multi copy mode, press the MultiCopy button again.
- **2-up:** Toggle the display mode when in dual view. Options are:

  For A/B

  - 2-up
  - HorizWipe
  - VertWipe

  For L/R

  - 2-up
  - HorizWipe
  - VertWipe
  - Blend

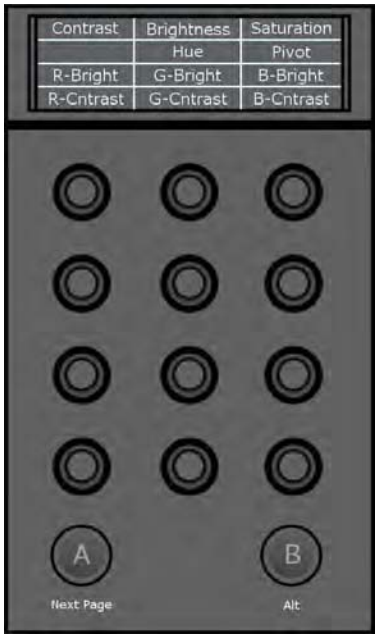
- **O/P:** Toggle between output and print views.
- **SetKey:** Set a keyframe at the current position on the currently selected channel. Pressing the B button will change the function of the SetKey button to SetKeyAll. The panel display is updated.



- **SetKeyAll:** Set a keyframe at the current position on all channels.
- **Save All:** Save your current cut and grade.

Logarithmic Grading Layout

Module 1 (KB) - Page 1 of 2

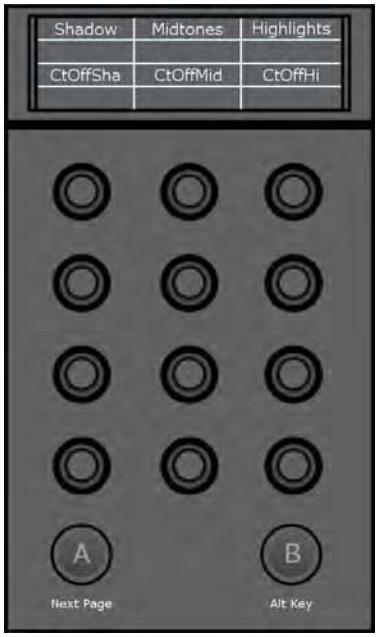


- **Contrast:** Modify the global contrast value.
- **R Bright:** Modify the brightness of the red channel.
- **R Cntrast:** Modify the contrast of the red channel.
- **Brightness:** Modify the global brightness value.
- **Hue:** Modify global hue value.

NOTE The hue can only be modified on secondaries.

- **G Bright:** Modify the brightness of the green channel.
- **G Cntrast:** Modify the contrast of the green channel.
- **Saturation:** Modify the global saturation value.
- **Pivot:** Modify the pivot value.
- **B Bright:** Modify the brightness of the blue channel.
- **B Cntrast:** Modify the contrast of the blue channel.

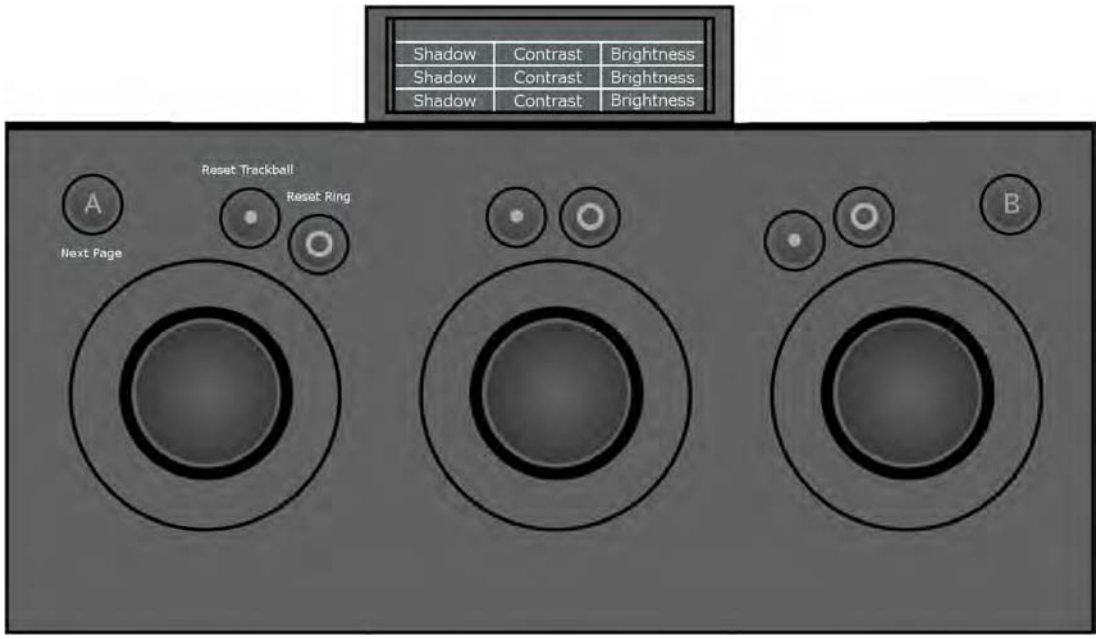
Module 1 (KB) - Page 2 of 2



- **Shadow:** Modify the luminance of the low range colour values.
- **CtOffSha:** Set the cutoff value for the shadows.
- **Midtones:** Modify the luminance of the mid range colour values.
- **CtOffMid:** Set the cutoff value for the midtones.
- **Highlights:** Modify the luminance of the high range colour values.
- **CtOffHi:** Set the cutoff value for the highlights.

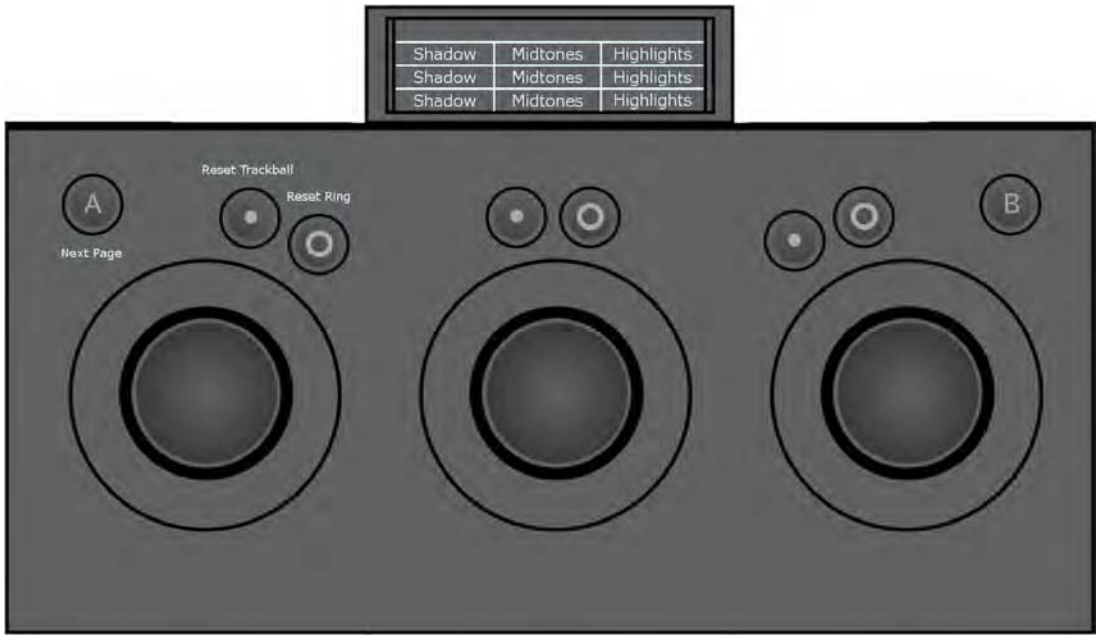
Module 2 (TK) - Page 1 of 2

NOTE The Grade mode functions are always available, even when not in Grade mode. To access the grading functions while in other modes, press the A button to cycle through the grading pages. Pressing the A button twice brings you back to the first page of the current mode.



- **Shadow:** Move the trackball right or left (x axis) to add colour to the low range colour values.
- **Shadow:** Move the trackball up or down (y axis) to add colour to the low range colour values.
- **Shadow:** Move the ring left or right to modify the luminance of the low range colour values.
- **Contrast:** Move the trackball right or left (x axis) to add colour to the contrast ratio.
- **Contrast:** Move the trackball up or down (y axis) to add colour to the contrast ratio.
- **Contrast:** Move the ring left or right to modify the luminance of the contrast ratio.
- **Brightness:** Move the trackball right or left (x axis) to add colour to the brightness channel.
- **Brightness:** Move the trackball up or down (y axis) to add colour to the brightness channel.
- **Brightness:** Move the ring left or right to modify the luminance of the brightness channel.

Module 2 (TK) - Page 2 of 2



- **Shadow:** Move the trackball right or left (x axis) to add colour to the low range colour values.
- **Shadow:** Move the trackball up or down (y axis) to add colour to the low range colour values.
- **Shadow:** Move the ring left or right to modify the luminance of the low range colour values.
- **Midtones:** Move the trackball right or left (x axis) to add colour to the mid range colour values.
- **Midtones:** Move the trackball up or down (y axis) to add colour to the mid range colour values.
- **Midtones:** Move the ring left or right to modify the luminance of the mid range colour values.
- **Highlights:** Move the trackball right or left (x axis) to add colour to the high range colour values.
- **Highlights:** Move the trackball up or down (y axis) to add colour to the high range colour values.
- **Highlights:** Move the ring left or right to modify the luminance of the mid range colour values.

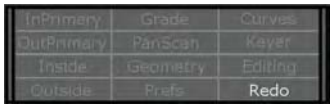
Module 3 (MF) - Page 1 of 3



- **InPrimary:** Toggle input primary grading mode.
- **OutPrimary:** Enable output primary grading mode.
- **Inside:** Double-press to enable inside grading in secondary mode (grading affects the inside of your shape) for the currently selected shape. Inside grading is enabled by default. Single-press to select inside grading and edit your grade.

  NOTE Inside grading is enabled by default.

- **Outside:** Double-press to enable outside grading in secondary mode (grading affects the outside of your shape) for the currently selected shape. Once enabled, single-press to select outside grading, to edit your grade.
- **Grade:** Access the Colour/Grade menu.
- **PanScan:** Access the Image/Reposition menu.
- **Geometry:** Enable the draw geometry mode in the secondaries menu. The fourth row of functions on the module is dynamically updated to display the rectangle, circle and wipe options. Pressing the corresponding button enables you to add the desired geometry.
- **Prefs:** Access the preferences functions.
- **Curves:** Access the Colour/Curves menu.
- **Keyer:** Enable the keyer in the secondaries menu. The default behavior is to enable the HLS keyer. Pressing Keyer again enables the diamond keyer.
- **Editing:** Access the Editing/Timeline menu.
- **Undo:** Undo the last operation performed. Pressing the B button changes the function of the Undo button to Redo. The panel display is be updated. Pressing the B button and Redo will redo the last operation (if you previously undid it).



- **Confirm:** When the Lustre UI prompts you to confirm an operation, the Undo button is automatically updated to display Confirm. Press to confirm the operation.



Module 3 (MF) - Page 2 of 3



- **InPrimary:** Enable input primary grading mode.
- **OutPrimary:** Enable output primary grading mode.
- **Inside:** Double-press to enable inside grading in secondary mode (grading affects the inside of your shape) for the currently selected shape. Inside grading is enabled by default. Single-press to select Inside Grading and edit your grade.

NOTE Inside grading is enabled by default.

- **Outside:** Double-press to enable outside grading in secondary mode (grading affects the outside of your shape) for the currently selected shape. Inside grading is enabled by default. Single-press to select inside grading and edit your grade.
- **O/P/M:** Toggle between output, print and matte views.
- **Autokey:** Enable/disable Autokey.
- **Reset:** Press once to reset the currently selected grading tool. Press twice to reset all grading tools.
- **SetKey:** Set a keyframe at the current position on the currently selected channel. Pressing the B button changes the function of the SetKey button to SetKey All. The panel display is updated.



- **SetKeyAll:** Set a keyframe at the current position on all channels.
- **A/B:** Toggle A/B playheads.
- **SingleView:** Toggle between single, dual and multi views, as defined under the preferences.
- **Bypass:** Press once to bypass the currently selected grading tool. Press twice to bypass all grading tools.
- **Undo:** Undo the last operation performed. Pressing the B button will change the function of the Undo button to Redo. The panel display is updated. Pressing the B button and Redo will redo the last operation (if you previously undid it).



Module 3 (MF) - Page 3 of 3



- **InPrimary:** Enable input primary grading mode.
- **OutPrimary:** Enable output primary grading mode.
- **Inside:** Double-press to enable Inside grading in secondary mode (grading affects the inside of your shape) for the currently selected shape. Inside grading is enabled by default. Single-press to select inside grading and edit your grade.

NOTE Inside grading is enabled by default.

- **Outside:** Double-press to enable outside grading in secondary mode (grading affects the outside of your shape) for the currently selected shape. Once enabled, single-press to select outside grading, to edit your grade.
- **Stereo:** Enable stereoscopy.
- **L/R:** Toggle between right eye view and left eye view modes (when working with stereoscopic footage).

- **MultiCopy:** Enter multi copy mode. When in multi copy mode, turn the ring to cycle your shots forward or backwards to copy the selected grade to your current shot. When you release the ring, the Undo button is automatically updated to display Apply. Press Apply to confirm and repeat the process for other shots as needed. To exit multi copy mode, press the MultiCopy button again.



- **SaveAll:** Save your current cut and grade.
- **Sync:** Sync your left and right eyes when working with stereoscopic footage. When in sync, any operation performed on the left eye is also applied to the right eye.
- **Cursor:** Enable cursor mode. When cursor mode is enabled, the trackball is used to move the cursor in the UI. In cursor mode, the Reset Trackball button acts as the left mouse button and the Reset Ring button acts as the right mouse button.
- **Copy:** Enter copy mode. When in copy mode, turn the ring to cycle your shots forward or backwards to copy the selected grade to your current shot. When you release the ring, the Undo button is automatically updated to display Apply. Press Apply to confirm the copy operation. Once the copy operation confirmed, you automatically exit copy mode.

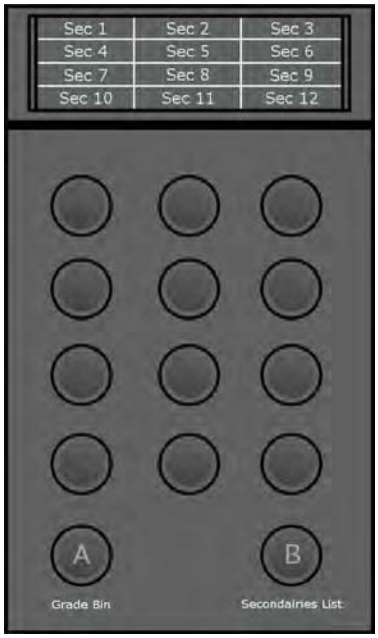


- **Undo:** Undo the last operation performed. Pressing the B button will change the function of the Undo button to Redo. The panel display will be updated. Pressing the B button and Redo will redo the last operation (if you previously undid it).



Module 4 (BT) - Page 1 of 6

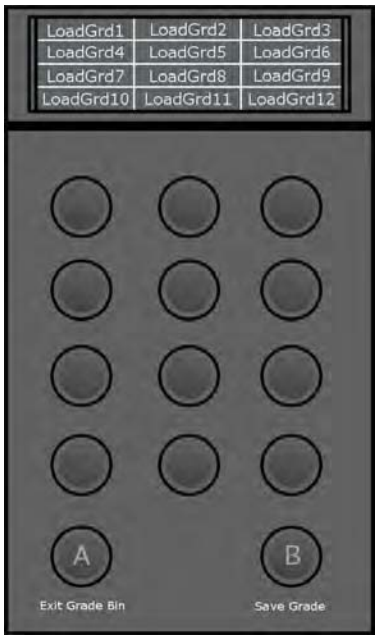
Secondaries List (4 pages, 12 secondaries per page).



Module 4 (BT) - Page 5 of 6

Grade Bin - Load

The BT Panel also controls your Grade Bin for saving and loading grades. The buttons mirror the position of saved grades in the Grade Bin.

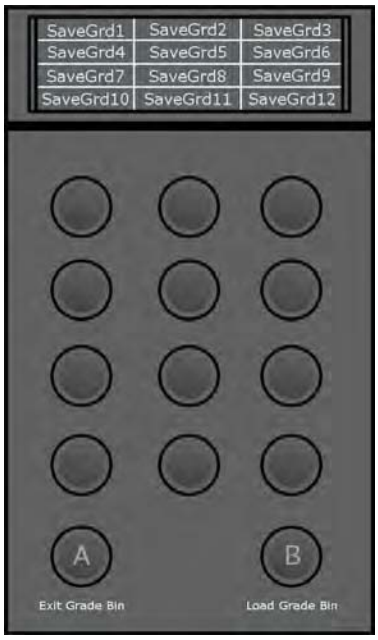


- **A Button:** Exit the Grade Bin.
- **B Button:** Toggle between load mode and save mode in the Grade Bin.

NOTE The trackball of the MF module navigates the Grade Bin when in Grade Bin mode.

Module 4 (BT) - Page 6 of 6

Grade Bin - Save

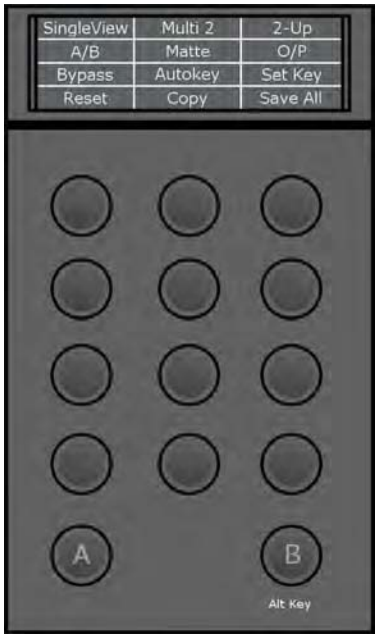


- **A Button:** Exit the Grade Bin.
- **B Button:** Toggle between load mode and save mode in the Grade Bin.

NOTE The trackball of the MF Module navigates the Grade Bin when in Grade Bin mode.

Optional 2nd BT Module - Page 1 of 1

It is recommended to attach a second BT module, which is mapped to give you access to many of the most used functions at all times. These functions correspond to the second and third page of the MF module.



- **SingleView:** Toggle between single, dual and multi views, as defined under the preferences.
- **A/B:** Toggle A/B playheads. Pressing the B button will change the function of the A/B button to L/R. The panel display is updated.



- **L/R:** Toggle between right eye view and left eye view modes (when working with stereoscopic footage).
- **Bypass:** Press once to bypass the currently selected grading tool. Press twice to bypass all grading tools.
- **Reset:** Press once to reset the currently selected grading tool. Press twice to reset all grading tools.
- **Multi 2:** Toggle the number of active viewports when in Multi View. Options are:
  - 2
  - 4
  - 9
  - 16
- **Matte:** Enable/disable matte view.
- **Autokey:** Enable/disable Autokey.
- **Copy:** Enter copy mode. When in copy mode, turn the ring on the MF module to cycle your shots forward or backwards to copy the selected grade to your current shot. When you release the ring, the Undo button on the MF module is automatically updated to display Apply. Press Apply to confirm. Once the copy operation confirmed, you automatically exit copy mode. Pressing the B button changes the function of the Copy button to MultiCopy. The panel display is updated.



- **MultiCopy:** When in multi copy mode, turn the ring on the MF module to cycle your shots forward or backwards to copy the selected grade to your current shot. When you release the ring, the Undo button on the MF module is automatically updated to display Apply. Press Apply to confirm and repeat the process for other shots as needed. To exit multi copy mode, press the MultiCopy button again.
- **2-up:** Toggle the display mode when in dual view. Options are:

  For A/B

  - 2-up
  - HorizWipe
  - VertWipe

  For L/R

  - 2-up
  - HorizWipe
  - VertWipe
  - Blend

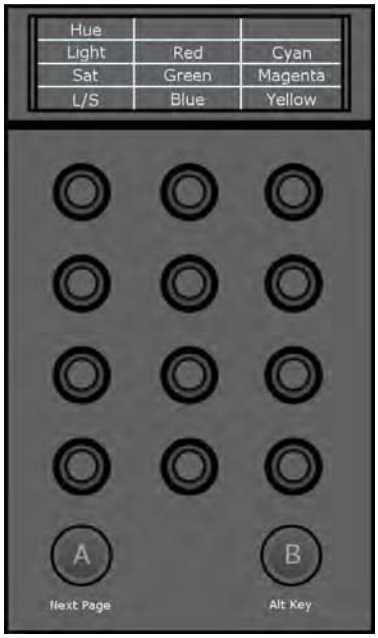
- **O/P:** Toggle between output and print views.
- **SetKey:** Set a keyframe at the current position on the currently selected channel. Pressing the B button will change the function of the SetKey button to SetKeyAll. The panel display is updated.



- **SetKeyAll:** Set a keyframe at the current position on all channels.
- **Save All:** Save your current cut and grade.
- **B Button:** Enable alternate functionality for specified buttons.

Curves Layout

Module 1 (KB) - Page 1 of 2

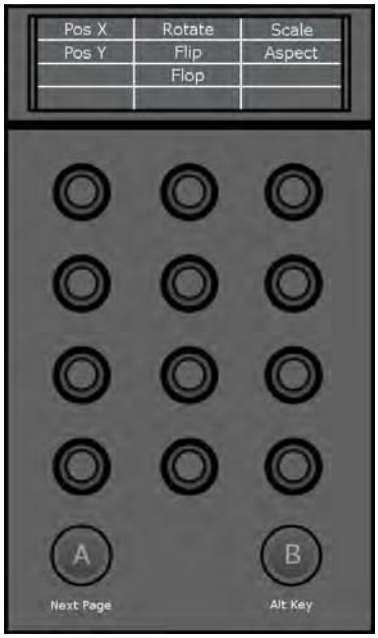


- **Hue:** Press on the knob to enable the hue curve. Turning the knob left or right will raise or lower the Hue across the entire spectrum.
- **Lightness:** Press on the knob to enable the lightness curve. Turning the knob left or right will raise or lower the lightness across the entire spectrum.
- **Saturation:** Press on the knob to enable the saturation curve. Turning the knob will raise or lower the saturation across the entire spectrum.
- **L/S:** Press on the knob to enable the lightness - saturation curve. Turning the knob left or right will raise or lower the lightness - saturation across the entire spectrum.
- **Red:** Adjust the red point on the hue, lightness or saturation curve, according to the selected curve.
- **Green:** Adjust the green point on the hue, lightness or saturation curve, according to the selected curve.
- **Blue:** Adjust the blue point on the hue, lightness or saturation curve, according to the selected curve.
- **Cyan:** Adjust the cyan point on the hue, lightness or saturation curve, according to the selected curve.
- **Magenta:** Adjust the magenta point on the hue, lightness or saturation curve, according to the selected curve.
- **Yellow:** Adjust the yellow point on the hue, lightness or saturation curve, according to the selected curve.

Pan & Scan Layout

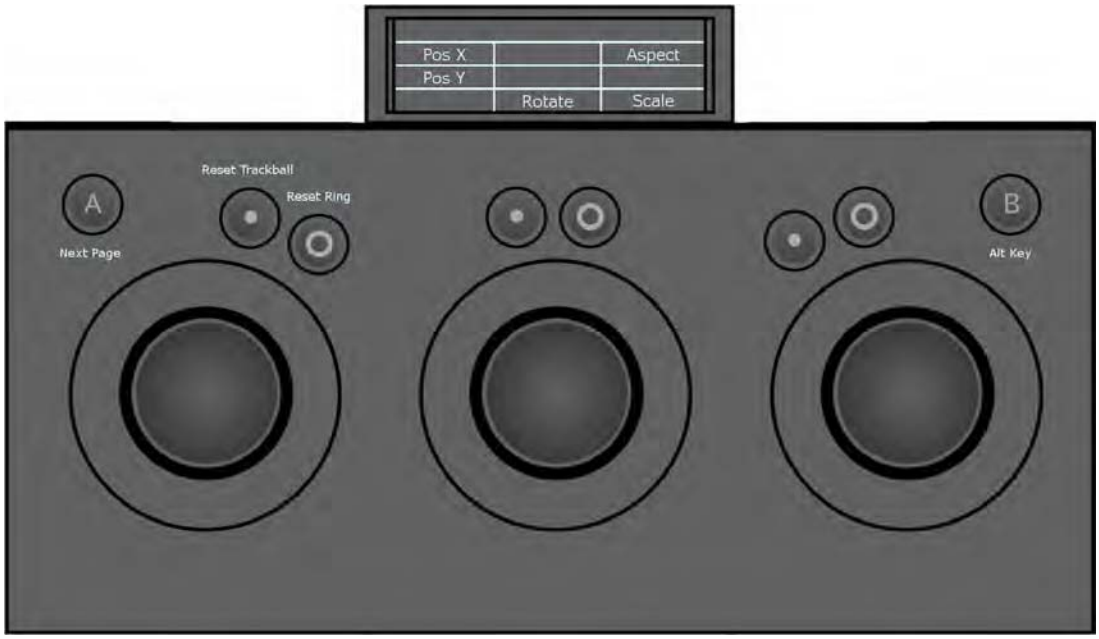
Module 1 (KB) - Page 1 of 2

Pan & Scan mode accesses the Image/Reposition menu.



- **Pos X:** Move your image on the X axis.
- **Pos Y:** Move your image on the Y axis.
- **Rotate :** Rotate your image.
- **Flip:** Press the knob to enable/disable Flip.
- **Flop:** Press the knob to enable/disable Flop.
- **Scale:** Scale your image proportionally.
- **Aspect:** Scale your image in X.

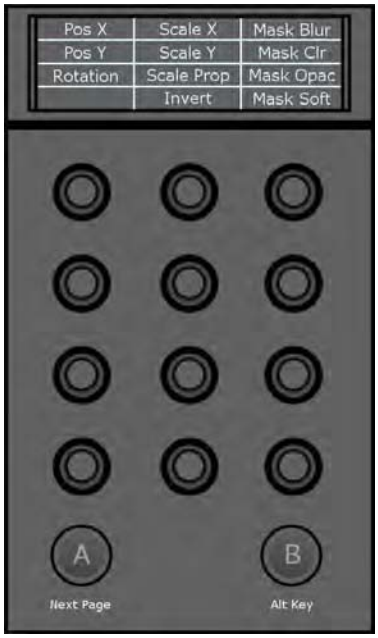
Module 2 (TK) - Page 1 of 4



- **Pos X:** Move the trackball left or right to move your image on the X axis.
- **Pos Y:** Move the trackball left or right to move your image on the Y axis.
- **Rotate:** Turn the ring to rotate your image.
- **Aspect:** Move the trackball left or right to scale your image in X.
- **Scale:** Turn the ring to scale your image proportionally.

Geometry Layout

Module 1 (KB) - Page 1 of 2



- **Pos X:** Move your geometry on the X axis.
- **Pos Y:** Move your geometry on the Y axis.
- **Rotation:** Rotate your geometry.
- **Scale X:** Scale your geometry in X.
- **Scale Y:** Scale your geometry in Y.
- **Scale Prop:** Scale your geometry proportionally.
- **Invert:** Press the knob to invert your geometry.
- **Mask Blur:** Set the level of mask blur.
- **Mask Clr:** Set the level of mask colour.
- **Mask Opac:** Set the level of mask opacity.
- **Mask Soft:** Set the level of mask softness.

Module 2 (TK) - Page 1 of 4



- **Pos X:** Move the trackball left or right to move your geometry on the X axis.
- **Pos Y:** Move the trackball left or right to move your geometry on the Y axis.
- **Mask Soft:** Turn the ring to add/remove softness to your mask.
- **Rotate:** Turn the ring to rotate your geometry.
- **Scale X:** Move the trackball left or right to scale your geometry in X.
- **Scale Y:** Move the trackball left or right to scale your geometry in Y.
- **ScaleProp:** Turn the ring to scale your geometry proportionally.

Module 3 (MF) - Page 1 of 4

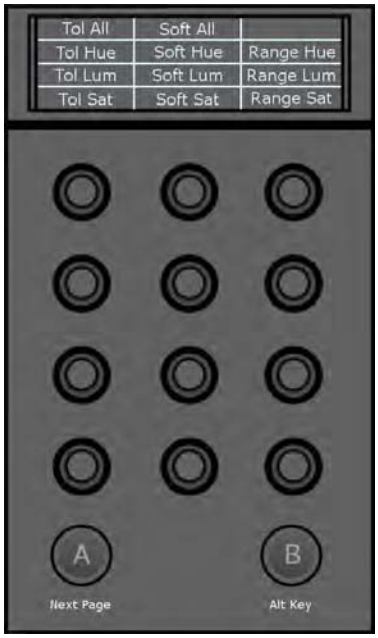


NOTE The bottom row functions of the MF module in Geometry mode (Rectangle, Circle, Wipe) appear when you press the Geometry button. Once you press either Rectangle, Circle or Wipe to add the desired shape or start editing an existing shape, the bottom row reverts back to its default functions (Prefs, Undo, Enter). To add another shape, press Geometry again and press the button corresponding to the shape you want to add.

- **InPrimary:** Toggle input primary and output primary grading modes.
- **Grade:** Access the Colour/Grade menu.
- **Geometry:** Enable the draw geometry mode in the Secondaries menu. The fourth row of functions on the module is dynamically updated to display the rectangle, circle and wipe options. Pressing the corresponding button enables you to add the desired geometry.
- **Rectangle:** Add a default rectangle.
- **Inside:** Double-press to enable inside grading in secondary mode (grading affects the inside of your shape) for the currently selected shape. Inside grading is enabled by default. Single-press to select inside grading and edit your grade.
- **Curves:** Access the Colour/Curves menu.
- **Keyer:** Enable the keyer in the secondaries menu. The default behavior is to enable the HLS keyer. Pressing Keyer again enables the diamond keyer.
- **Circle:** Add a default circle.
- **Outside:** Double-press to enable outside grading in secondary mode (grading affects the outside of your shape) for the currently selected shape. Single-press to select outside grading, to edit your grade.
- **PanScan:** Access the Image/Reposition menu.
- **Editing:** Access the Editing/Timeline menu.
- **Wipe:** Add a default wipe.

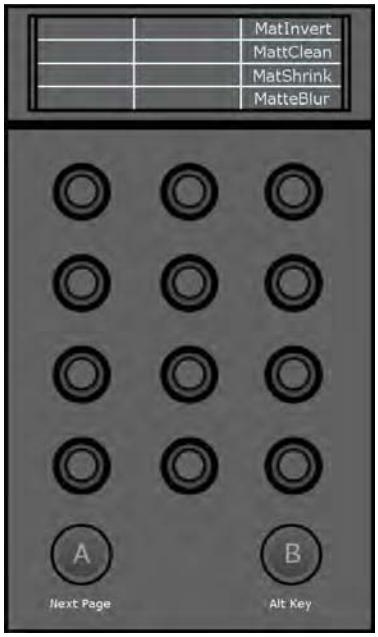
HLS Keyer Layout

Module 1 (KB) - Page 1 of 2



- **TolAll:** Set the global tolerance for your key (hue, luminance and saturation channels).
- **TolHue:** Set the tolerance for the hue channel.
- **TolLum:** Set the tolerance for the luminance channel.
- **TolSat:** Set the tolerance for the saturation channel.
- **SoftAll:** Set the global softness for your key (hue, luminance and saturation channels).
- **SoftHue:** Set the softness for the hue channel.
- **SoftLum:** Set the softness for the luminance channel.
- **SoftSat:** Set the softness for the saturation channel.
- **Range Hue:** Offset the tolerance range for the hue channel.
- **Range Lum:** Offset the tolerance range for the luminance channel.
- **Range Sat:** Offset the tolerance range for the saturation channel.

Module 1 (KB) - Page 2 of 2



- **MatInvert:** Press the knob to invert your matte.
- **MatClean:** Set the matte cleanup value.
- **MatShrink:** Set the matte shrink value.
- **MatteBlur:** Set the matte blur value.

Module 3 (MF) - Page 1 of 4

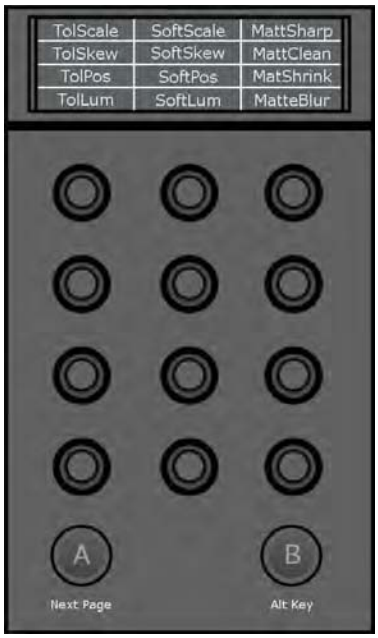


- **InPrimary:**Toggle input primary and output Primary grading modes.
- **Grade:** Access the Colour/Grade menu.

- **Geometry:**Enable the draw geometry mode in the Secondaries menu. The fourth row of functions on the module is dynamically updated to display the rectangle, circle and wipe options. Pressing the corresponding button enables you to add the desired geometry.
- **Prefs:** Access the preferences functions.
- **Inside:** Double press to enable inside grading in secondary mode (grading affects the inside of your shape) for the currently selected shape. Inside grading is enabled by default. Single press to select inside grading and edit your grade.
- **Curves:** Access the Colour/Curves menu.
- **Keyer:** Enable the keyer in the Secondaries menu. The default behavior is to enable the HLS keyer. Pressing Keyer again enables the diamond keyer.
- **Undo:** Undo the last operation performed. Pressing the B button will change the function of the Undo button to Redo. The panel display will be updated. Pressing the B button and Redo will redo the last operation (if you previously undid it).
- **Outside:** Double press to enable outside grading in secondary mode (grading affects the outside of your shape) for the currently selected shape. Single press to select outside grading, to edit your grade.
- **PanScan:** Access the Image/Reposition menu.
- **Editing:** Access the Editing/Timeline menu.
- **Enter:** Confirm operations.

Diamond Keyer Layout

Module 1 (KB) - Page 1 of 2



- **TolScale:** Set the tolerance scale value.
- **TolSkew:** Set the horizontal tolerance skew value.
- **TolPos:** Set the tolerance position value.
- **TolLum:** Set the tolerance luminance value.
- **SoftScale:** Set the softness scale value.

- **SoftSkew:** Set the horizontal softness skew value.
- **SoftPos:** Set the softness position value.
- **SoftLum:** Set the softness luminance value.
- **MattSharp:** Set the matte sharpness value.
- **MattClean:** Set the matte cleanup value.
- **MatShrink:** Set the matte shrink value.
- **MatteBlur:** Set the matte blur value.

Module 3 (MF) - Page 1 of 4

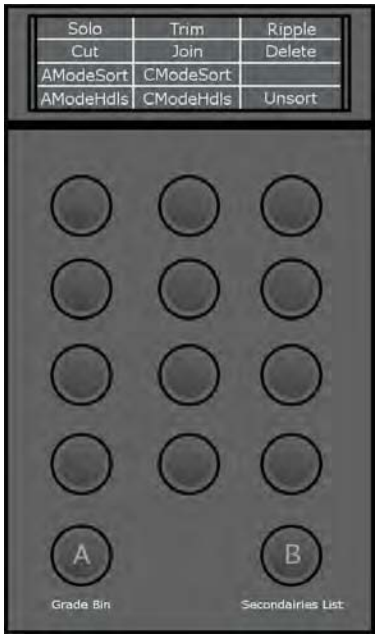


- **InPrimary:** Toggle input primary and output primary grading modes.
- **Grade:** Access the Colour/Grade menu.
- **Geometry:** Enable the draw geometry mode in the secondaries menu. The fourth row of functions on the module is dynamically updated to display the rectangle, circle and wipe options. Pressing the corresponding button enables you to add the desired geometry.
- **Prefs:** Access the preferences functions.
- **Inside:** Double-press to enable inside grading in secondary mode (grading affects the inside of your shape) for the currently selected shape. Inside grading is enabled by default. Single-press to select inside grading and edit your grade.
- **Curves:** Access the Colour/Curves menu.
- **Keyer:** Enable the keyer in the secondaries menu. The default behavior is to enable the HLS keyer. Pressing Keyer again enables the HLS keyer.
- **Undo:** Undo the last operation performed. Pressing the B button will change the function of the Undo button to Redo. The panel display will be updated. Pressing the B button and Redo will redo the last operation (if you previously undid it).
- **Outside:** Double-press to enable outside grading in secondary mode (grading affects the outside of your shape) for the currently selected shape. Single-press to select outside grading, and edit your grade.
- **PanScan:** Access the Image/Reposition menu.
- **Editing:** Access the Editing/Timeline menu.

- **Enter:** Confirm operations.

Editing Layout

Module 4 (BT) - Page 1 of 1



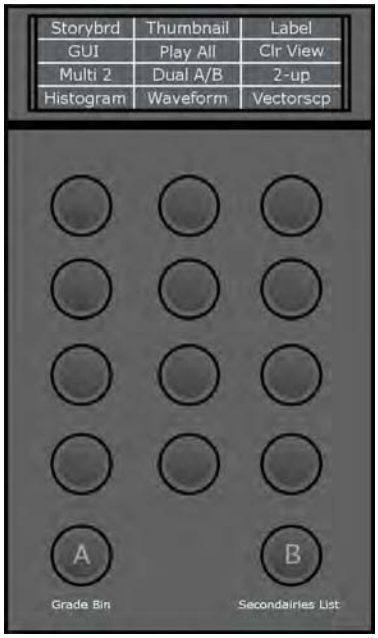
- **Solo:** Solo the currently selected track/layer.
- **Cut:** Insert a cut at the current location of the positioner.
- **AModeSort:** Enable A-mode timeline Sort.
- **AModeHdls:** Enable A-mode timeline Sort with handles.
- **Trim:** Enable trim mode.
- **Join:** Join the timeline segment on which the positioner is parked with the segment immediately succeeding it.
- **CModeSort:** Enable C-mode timeline Sort.
- **CModeHdls:** Enable C-mode timeline Sort with handles.
- **Ripple:** Toggle between ripple off, ripple start and ripple end modes.
- **Delete:** Delete the timeline segment on which the positioner is parked.

NOTE

- When in Editing mode, pressing the B button and turning the ring on the MF panel enables timeline zoom in and out.
- Pressing the B button and moving the trackball enables you to move the timeline positioner's focus point up or down.

Preferences Layout

Module 4 (BT) - Page 1



- **Storybrd:** Enable/disable the storyboard.
- **GUI:** Enable/disable onscreen GUI hiding.
- **Multi 2:** Toggle the number of active viewports when in multi view. Options are:
  - 2
  - 4
  - 9
  - 16
- **Histogram:** View/hide the histogram.
- **Thumbnail:** Enable/disable auto thumbnail refresh.
- **Play All:** Toggle play modes. Options are:
  - Play All (plays full timeline).
  - Play Shot (plays current shot).
  - PlayInOut (plays the interval between the In and Out marks).
- **Dual A/B:** Toggle the active sources displayed when in dual view. Options are: A/B or L/R.
- **Waveform:** Display/hide the waveforms in the audio tracks.
- **Label:** Enable/disable the image label.
- **Clr View:** Toggle single channel views. Options are:
  - Colour (all channels)
  - Red
  - Green
  - Blue
  - Luminance

- **2-up:** Toggle the display mode when in dual view. Options are:

  For A/B

  - 2-up
  - HorizWipe
  - VertWipe

  For L/R

  - 2-up
  - HorizWipe
  - VertWipe
  - Blend

- **Vectorscp:** Display/hide the vectorscope.

Troubleshooting

In the event that one or more of your panel modules are not recognized by Lustre, restarting the background service that manages the interactions between Lustre and the Tangent Element Control Surface can resolve the issue.

To restart the background service:

1. Exit Lustre.
2. Open a shell on your Linux machine.
3. Login as root.
4. Type: */etc/init.d/tangenthub restart*
5. Launch Lustre.

All of your module panels should now be recognized.

If the above procedure does not resolve the issue, you can manually connect the Control Surface to your software. To do this, you must define each module's unique ID number in the panel-list.xml file within the Lustre filesystem for the panel to be recognized.

To manually install the panel:

1. Plug and connect the four or five modules (depending on your setup) of your Tangent Element Control Panel to a power outlet and to your USB hub.
2. Once the modules are booted up and are displaying the Tangent logo, hit the A button on each module to display its unique ID number.

3. On your Linux machine, log in as root and use kEdit to edit the panel-list.xml file:
   kedit /var/opt/Tangent/Hub/panel-list.xml

```
<?xml version="1.0" encoding="utf-8" standalone="yes"?>
<TangentWave fileType="PanelList" fileVersion="3.0">
    <Panel type="Element-Mf" number="1" serial="30"/>
    <Panel type="Element-Bt" number="1" serial="68"/>
    <Panel type="Element-Bt" number="2" serial="77"/>
    <Panel type="Element-Kb" number="1" serial="53"/>
    <Panel type="Element-Tk" number="1" serial="13"/>
    <Panel type="Wave"/>
    <Options>
        <DHCP enabled="true"/>
    </Options>
</TangentWave>
```

4 Add the corresponding ID numbers to the panel-list.xml file.

NOTE In a four module setup, the line <Panel type= "Element-BT" number="2" serial="00"/> should be left blank.

5 Save the file.

The Tangent Element Control Surface should be now recognized by Lustre.

The Autodesk Control Surface

The Autodesk® control surface provides improved interactivity when colour grading film and video footage. You can perform many of the same tasks you do in the Autodesk Lustre® user interface using the control surface. The modular design of the control surface also allows you to work on different grading stages simultaneously. For example, you can perform curves grading while modifying the image lift.

The Autodesk control surface consists of three panels. Although licensed separately, they are designed to work together.

Connecting the Autodesk Control Surface

For information on connecting the control surface, refer to the *Autodesk Lustre Installation and Configuration Guide* for your operating system.

Autodesk Control Surface Technical Information

For technical information about the Autodesk control surface, see*http://www.tangentdevices.co.uk/downloads_documents.asp*. For those who wish to mount the Autodesk control surface in a desk, this link provides the precise dimensions of the panels in millimeters.

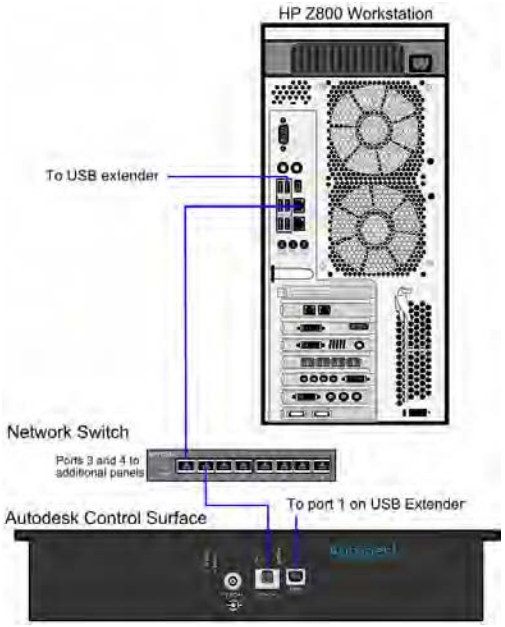
Consult the following table for the power consumption requirements of the Autodesk control surface.

Component	Quantity	Amps (120v)	Amps (240v)	Watts
Autodesk control surface panel	1	1.0	.5	120

Connecting the Autodesk Control Surface to a Lustre Workstation

The Autodesk Control Surface consists of three panels connected to the workstation's on-board GigE port 1 by way of the supplied network switch. The central panel requires an additional connection to the remote unit of the USB extender kit.

This section provides information on making the physical connections needed to operate the Autodesk Control Surface. For information on using the Control Surface, see the *Autodesk Control Surface User Guide*.



NOTE

The illustration shows the central panel only (the Colour Grading panel). This panel is easily identified, since it has both an Ethernet network port and a USB connector. It is also the panel with the built-in tablet.

To connect the Autodesk Control Surface:

1. Connect each panel to a power supply using the AC power adapter cables.
2. Use network cables to connect each panel to the available ports in the network switch.
3. Connect the Colour Grading panel to the remote unit of the USB extender kit.
4. Use a network cable to connect an available port on the Netgear network switch to the workstation, as illustrated.

Assigning an IP Address to the Autodesk Control Surface

You must assign an IP address and subnet mask to the Autodesk Control Surface.

Assign the network port connected to the control surface with a static IP address that does not interfere with any of the IP addresses on your network.

To configure the Autodesk Control Surface:

1. Type:
   nedit /etc/sysconfig/network-scripts/ifcfg-eth`<port#>`
2. Modify the IPADDR and NETMASK values. For example:

```
IPADDR=192.168.125.10
NETMASK=255.255.255.0
```

Configuring Lustre to Connect to the Autodesk Control Surface

To configure Lustre to use the Autodesk control surface:

1. Turn the power on to each of the modules and look at the top display panel on the module. It should display the panel name and ID.
2. After you install Lustre, open the init.config file located in /usr/autodesk/<lustre_version> with a text editor.
3. In the ControlSurface section of the init.config file of your project, enter the panel IDs for each panel.

   ```
   ControlSurface
   AutodeskPanels state="On"
   PanelIDs function="<panel_ID>" grading="<panel_ID>" navigation="<panel_ID>"
   ```

4. Start Lustre. The following message should appear in each Console:

   ```
   Panel #<panel_ID> is detected
   ```

   This confirms that the Autodesk Control Surface is enabled.

Control Surface Panel Descriptions

Each of the three control surface panels are designed to perform different functions.

The following image shows the Navigation panel.



Use the Navigation panel to:

- Select menus. Pressing one of the menu buttons along the top of the panel activates a menu. For example, pressing the Grade button activates grading functionality.
- Reset or bypass the currently selected menu.
- Cache memory.
- Navigate the Storyboard.
- Pan and zoom the image.
- Copy grades.
- Activate mouse functionality.
- Add keyframes.
- Revert and compare grades.
- Toggle playback mode between Storyboard and clip loop.
- Switch playheads.
- Scroll shots.
- Undo and redo actions.
- Perform actions with the Grade bin.

- Adjust printer lights.
- Jump to specified frame number or timecode on the Storyboard.
- Show or hide user interface elements.
- Adjust the appearance of the panels.

The following image shows the Function panel.



Use the Function panel to work in the menu you activated in the Navigation panel. For example, if you pressed the Key menu button in the Navigation panel, the functions for pulling a key are activated in the Function panel. The features available on the Function panel depend entirely on the menu selected in the Navigation panel. You can use the digital display at the top of the panel as a guide for understanding how functionality is distributed when a given menu is active.

You can also use the Function panel to set viewing modes. Some of the viewing functions included in the Function panel include Multi view, Dual view in 2-up mode, Dual view in Wipe mode, proxy view, output view, and print view.

The following image shows the Colour Grading panel.



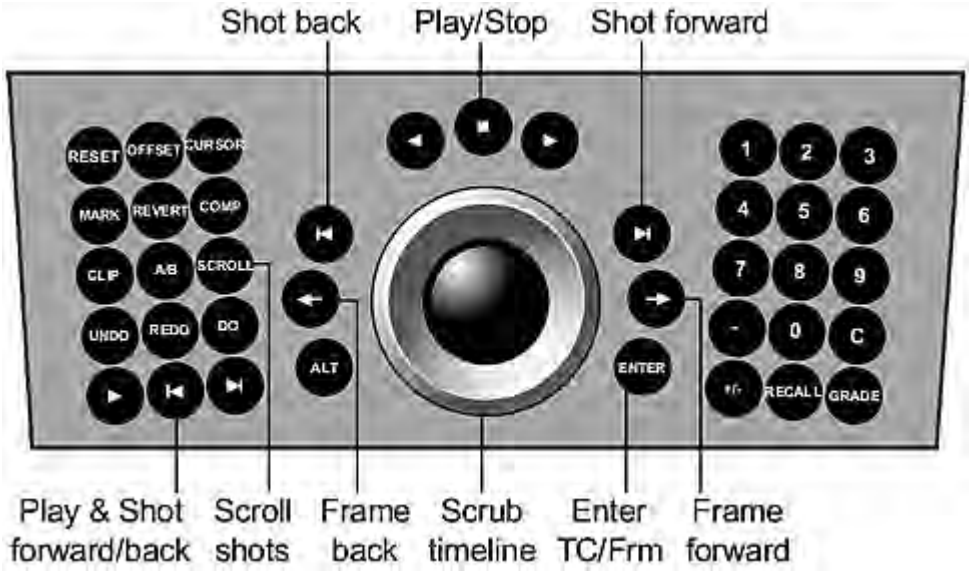
Use the Colour Grading panel for most grading functions. Colour grading functionality is split between the Colour Grading and the Function panels. The Colour Grading panel is always in grading mode. Depending on the colour space selected in the Setup Project menu, either Linear or Logarithmic functionality is available.

The Colour Grading panel also contains the tablet.

Navigating the Storyboard

You can use the Navigation panel to:

- Scrub the Storyboard.
- Scroll through the shots in the Storyboard.
- Toggle between the regular and large Storyboard viewing modes.
- Jump to a specified frame or timecode on the Storyboard.
- Jump ahead or back a frame at a time, or a shot at a time.
- Play forwards or backwards.



To scrub the Storyboard:

1 Use the ring portion of the trackball.

To scroll through shots on the Storyboard:

1 Press the Scroll button.
2 Use the ring portion of the trackball.
3 Press the Scroll or Do button to confirm the move.

To switch between regular and large Storyboard viewing modes:

1 Press the top More button on the Navigation panel.



The Label, Thumbnails, and GUI options appear in the digital display at the top of the panel. The location of each option in the digital display reflects their button position on the panel.

2 If the Storyboard option does not appear, press the button for Thumbs (the P&S button) to display Storyboard thumbnails. This makes the Storyboard option appear in the digital display.



3 Press the button for Storyboard (the Key button). This switches Storyboard thumbnails to the large Storyboard option.

NOTE This is equivalent to using the Q hotkey.

To jump to a specified frame on the Storyboard:

1 Press the Enter button.
2 Do one of the following:
- To jump ahead, enter the frame number using the numeric pad on the right side of the panel.
- To jump back, press the +/- button on the numeric pad and then enter the frame number.

3 Press the Enter button to jump to the specified frame.

To jump to a specified timecode on the Storyboard:

1 Press the Enter button.
2 Enter a timecode using the numeric pad on the right side of the panel.
3 Press the Enter button to jump to the specified timecode.

To jump back or ahead a frame at a time:

1 To jump back a frame, do one of the following:
- Press the Frame Back button (left arrow button to the left of the trackball).
- Simultaneously press the Shot Back button (below the Redo button) and the Play button (below the Do button).

2 To jump ahead a frame, do one of the following:
- Press the Frame Forward button (right arrow button to the right of the trackball).
- Simultaneously press the Shot Forward button (below the Undo button) and the Play button (below the Do button).

To jump to the beginning or end of a shot:

1 To jump to the first frame of the shot, press either of the two Shot Back buttons. One button is located to the left of the trackball, and the other is below the Redo button.
2 To jump to the last frame of the shot, press either of the two Shot Forward buttons. One button is located to the right of the trackball, and the other is below the Do button.

To play forwards or backwards:

1. To play the Storyboard forward, do one of the following:
   - Press the Play Forward button (above the trackball).
   - Press the Spacebar-style Play button (below the Undo button). This button operates like the Spacebar on the keyboard, in that you use it both to start and stop playback.
2. To play the Storyboard backwards, press the Play Backwards button (above the trackball).
3. To stop playback, do one of the following:
   - Press the Stop button (above the trackball).
   - Press the Spacebar-style Play button.

Toggling Play Modes

You can toggle the playback mode between looping a shot or looping the entire Storyboard.

To toggle play modes:

1. Press the Clip button on the Navigation panel.



This button corresponds to the Play Mode button in the user interface.

Activating Mouse Functionality

You can enable mouse functionality on the Navigation panel.

To activate mouse functionality:

1. Press the Cursor button on the Navigation panel.

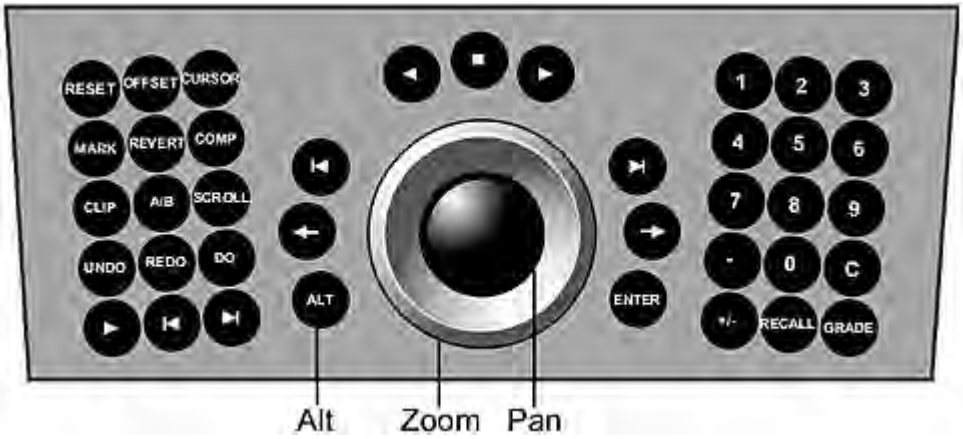


2 Use the panel's features as follows.

Use the:	As the:
Trackball	Mouse.
Backward play button	Left mouse button.
Stop button	Middle mouse button.
Forward play button	Right mouse button.

Panning and Zooming the Image

You can use the Navigation panel to pan and zoom the image.



To zoom the image:

1 Hold down the ALT button on the Navigation panel and use the ring on the trackball to zoom in and out.

To pan the image:

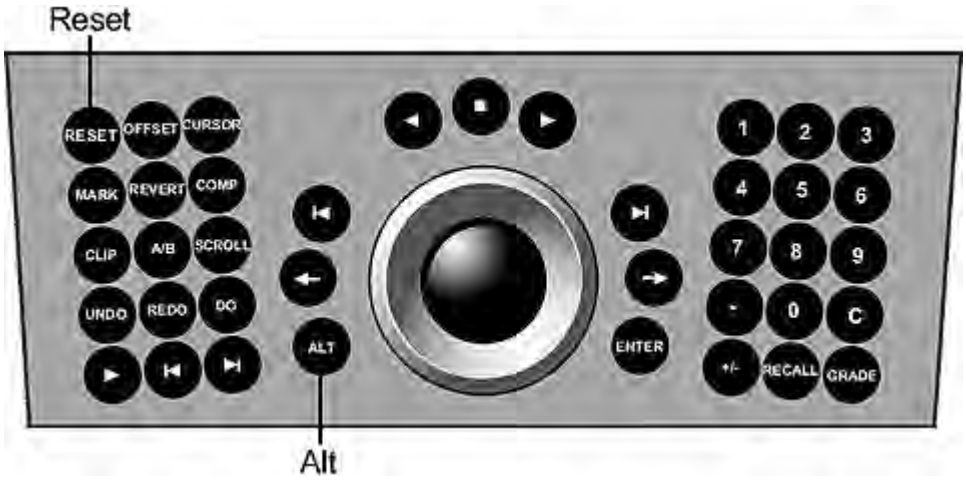
1 Hold down the ALT button on the Navigation panel and use the jog ball on the trackball to pan the image.

Resetting Zoom and Pan

You can now reset the zoom and pan.

To reset the zoom and pan:

1 Hold down the ALT button on the Navigation panel and press the Reset button.



NOTE The zoom and pan are always reset together.

Customizing the View

There are options for customizing the view on both the Navigation panel and the Function panel.

When working with secondaries, users can use the Function panel to view them in Saturated Mask view or Matte view. Saturated Mask view displays the range of colours you define through a greyscale image. The unselected colours remain grey. Matte view displays a black and white template indicating which parts of the image are transparent (black), and which are opaque and selected (white).

To access view options available on the Navigation panel:

1 Press the top MORE button on the Navigation panel.



The View, Panel, Storyboard, Label, Thumbs, and GUI options appear in the digital display at the top of the panel. The location of each option in the digital display reflects their button position on the panel.

2 Select one of the following options.

Press:	To:
GRADE	Select the viewing options. Player Viewing Options (page 2437).
CURVES	Adjust the panel sensitivity. See Customizing the Panel Sensitivity (page 2434).
KEY	Toggle the Storyboard on or off.
GEOM	Toggle the shot name on or off the image.
P&S	Toggle the Storyboard thumbnails on or off.
MISC	Toggle the full-screen Player on or off.

To access view options available on the Function panel:

1 Use the viewport options to change the view as you work on your grades. For example, press PROXY to view a proxy version of your shot in the Player.



Viewport option	Description
MULTI	Cycles through the Multi view modes. You can view 2, 4, 9, or 16 shots within the Player.
WIPE	Cycles through the horizontal wipe, vertical wipe, and blend (for a stereoscopic project only).
SPLIT	Toggles between the current view options and the Single view (default).
PROXY	Toggles between the full-resolution and half-resolution (proxy) version of your shot in the Player. If you have not rendered a proxy, an X appears in the Player.
MATTE	Toggles between secondary grading view and result view.
O/P	Toggles between a rendered version of your shot (P) and the pre-rendered version with grading applied (O). If you have not rendered the shot, an X appears in the

Viewport option	Description
	Player when in P mode. Press the ALT and O/P buttons simultaneously to toggle to I mode (this displays the image with colour corrections made in the Grading and Curves menus only).
STILL	Displays the stored reference image. Press the ALT and Still buttons simultaneously to store the current frame in the reference buffer.

NOTE You can also access the SHOW button in the Function panel. This button shows or hides the geometry wireframe for the current secondary layer.

NOTE There is no functionality assigned to the Function panel's ALT button for this release.

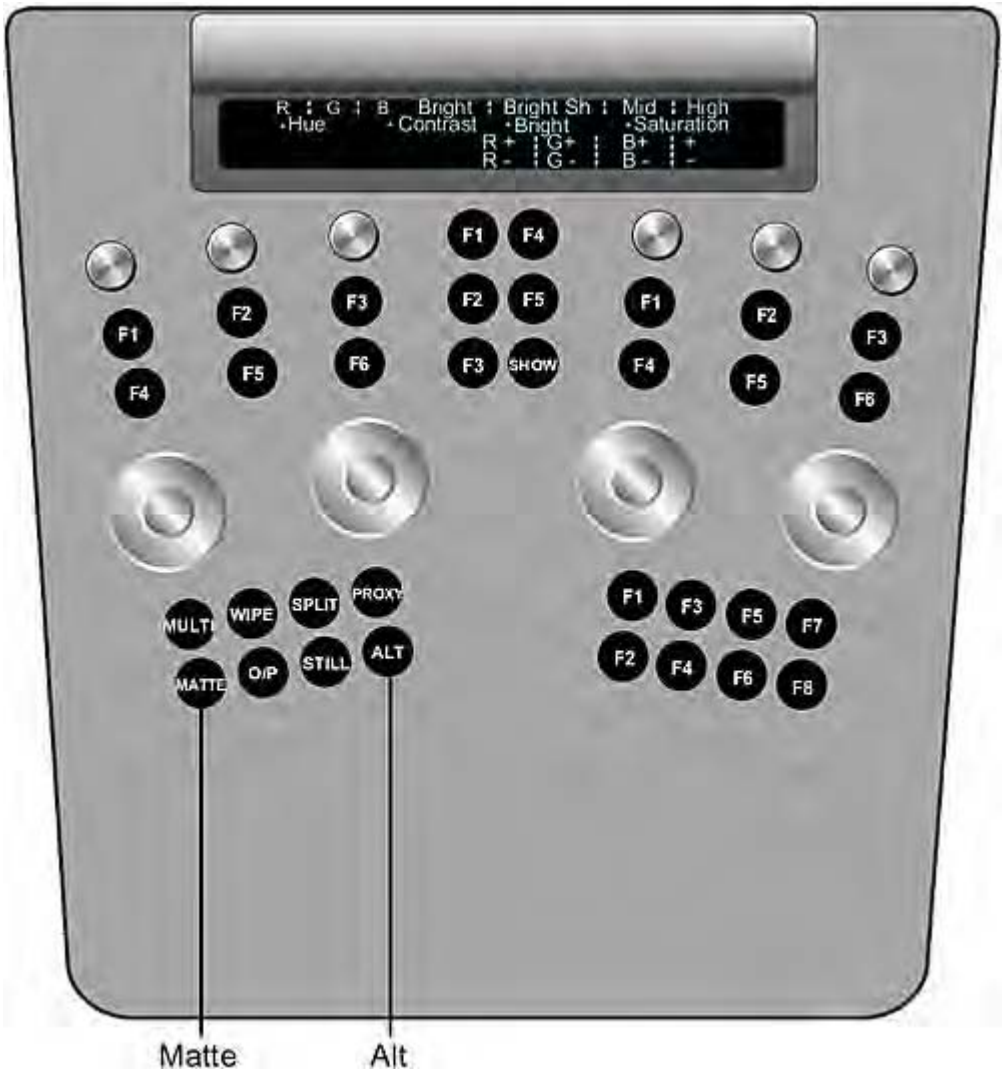
To change fonts in the digital display:

1 On the keyboard, press `Shift+F` until you obtain the desired font.

NOTE The change is only maintained for the current session.

To toggle between "O" view and Matte view:

1 Press the MATTE button on the Function panel.



2 Hold down the ALT button on the Function panel while pressing MATTE again to switch between Saturated Mask view and black and white Matte view.

TIP You can go directly to Saturated Mask view mode from another mode, e.g. Output, by holding down the ALT button and pressing MATTE.

Customizing the Panel Sensitivity

When you enable the Edit function, you can modify the sensitivity of any modifier (e.g., knobs, wheels, and rings). These items include the Grading, Curves, Key, Geom, P&S, and Misc menu.

To edit the sensitivity of the Autodesk Control Surface panels:

1 On the Navigation panel, press the top MORE button.

2 Press the CURVES button to enter the Panel menu.



In the Panel menu you can Edit (F4 button), Save (IN button), and Revert (OUT button) your panel sensitivity changes.

3 Press the F4 button to enable the Edit mode.

4 Touch any modifier you want to change its sensitivity on the panel. Turn the modifier clockwise to increase the sensitivity or counter-clockwise to decrease the sensitivity.

NOTE The Function panel displays the Grade menu by default. To change the sensitivity of the Curves, Geometry, Keyer, or Reposition menu, you must enable the menu you want to modify first before entering the panel sensitivity option.

When a control is selected, its name and sensitivity are displayed on all the LCD panels. The last line of the LCD displays the control you are modifying, its value, and the enabled Edit mode.

NOTE Some control names are long and therefore are truncated on the Navigation panel LCD. The Function and Grading panels display the full name and value.

5 Press the F4 button again to disable the Edit mode. You can now test out the new sensitivity value.

NOTE Pressing F4 does not exit you from the Panel menu.

NOTE You need to save the rules file in order to commit your modification. If you do not save the modification, you will lose the modified value(s) when you quit Lustre.

Saving the Sensitivity Modifications

When you have completed making modifications to the sensitivity of your Autodesk control surface panel and tested the sensitivity, you must save your changes.

To save the modifications to your Autodesk Control Surface panel:

1 Press the IN button. The Save option is selected and the LCD displays “Confirm Save”.

NOTE If you do not want to confirm the save, press the F1 button to exit the Save menu. The display returns to the Panel menu.

2 Press the IN button again. The LCD displays “Saving Panel File”.

The changes are saved and the display returns to the top level menu.

Reverting the Panel Modifications

If you have made modifications to the panel control values and have not saved the changes, you can revert back to the previous saved values.

To revert the saved panel modifications:

1. Press the OUT button to select the Revert option.
   The LCD displays “Confirm Revert”.

   NOTE If you do not want to confirm the revert, press the F1 button to cancel the revert. The display returns to the Panel menu.

2. Press the OUT button again to confirm the revert. The LCD displays “Reverting Panel File” and then returns to the top level menu.

Selective Backlighting of Active and Available Buttons

In order to make it easier to determine which functions are currently available on the Autodesk control surface panels, you can select Mid from amongst the options accessible from the Navigation panel.

The Mid option highlights active buttons and available buttons.

An active button is one that has been enabled by the user. For example, if you enable secondary two for Inside grading, the 2 button on the numeric keypad and the Inside button are backlit on the Navigation panel.

An available button is one that has been enabled by the system based on a feature that you selected. For example, if you switch from the Geom menu to the Reposition menu, the backlighting on the Function panel changes to show which buttons are available for performing repositioning tasks.

To enable backlighting for active and available buttons:

1. Press the bottom MORE button on the Navigation panel.
   The Low, Mid, High, and Screen options appear in the digital display.



2. Press the F4 button.
   Active and available buttons are now backlit.

   NOTE The Low option is now accessible by pressing the F3 button.

Toggling Playheads

You can toggle between the images assigned to Playhead A and B.

To toggle playheads:

1. Press the A/B button on the Navigation panel.

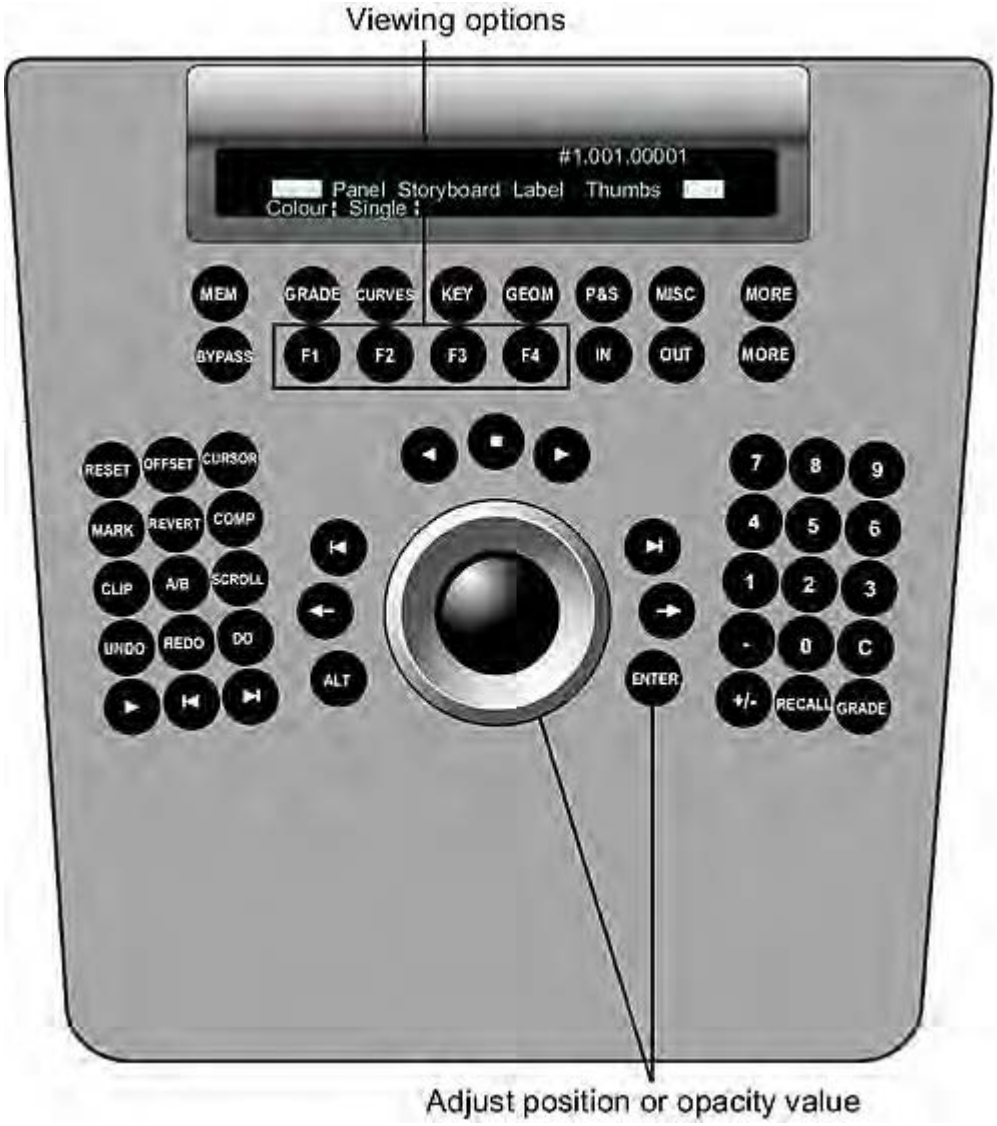


Player Viewing Options

There are several ways you can view the shots within the Player. You can choose to display the shot(s) in a Single, Dual, or Multi view. These viewing options are accessible through the Navigation panel.

To access the viewing options:

1. Press the top MORE button on the Navigation panel.
2. Press GRADE to access the viewing options.
3. Use the following controls.



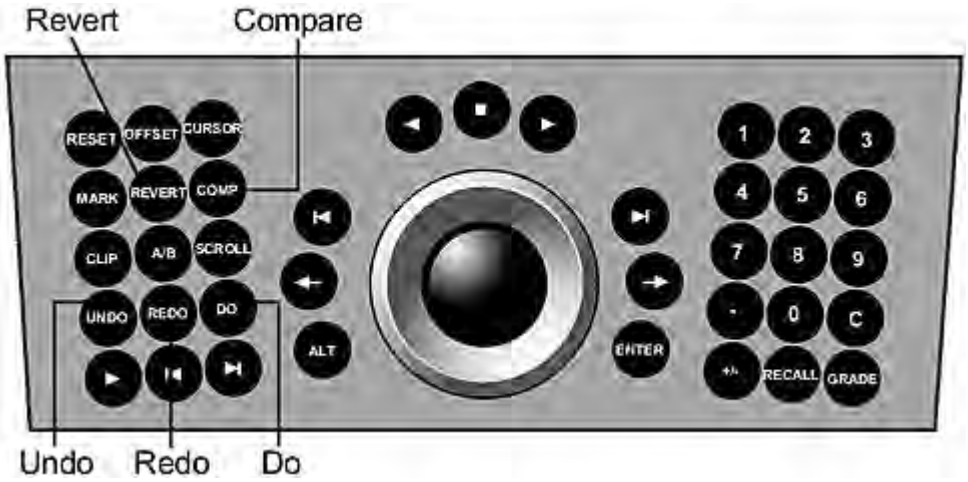
Press:	To:
F1	Cycle through the different colour view selections (i.e., colour, red, green, blue, and luminance channels).
F2	Cycle through the different viewers (i.e., Single, Dual, or Multi view).
F3	Toggle between Playheads A and B (A/B) or Left Eye and Right Eye (L/R). This is only applicable for Dual view.
F4	Cycle through the Dual view modes (i.e., 2-up, horizontal wipe, vertical wipe, or blend). Blend is only available if you have L/R selected. Cycle through the Multi view modes (i.e., 2-up, 4-up, 9-up, or 16-up).

NOTE To revert to the previous selection, press ALT on the Navigation panel and the corresponding F1, F2, F3, or F4 button.

4 (Optional) If you have selected a wipe or blend, you can adjust the position of the wipe or the opacity of the blend by holding ENTER on the Navigation panel and rotating the ring portion of the trackball.

Using the Undo List

You can access the undo list from the Navigation panel. The undo list works differently than it does in the user interface. When you use the panel, you must save your changes to the undo list using the Do button. Undo lists are saved on a shot-by-shot basis.



To save a change to the undo list:

1 Press the DO button.

To go through the undo list:

1 Press the UNDO or REDO button.

Reverting to Previous Grades

You can use the Navigation panel to toggle between the grade that was applied to a shot when you first selected it in the Storyboard and the current grade.

To revert to previous grades:

1 Press the REVERT button on the Navigation panel.

To save the reverted stage to the undo list:

1 Press the DO button on the Navigation panel.

Comparing Grades

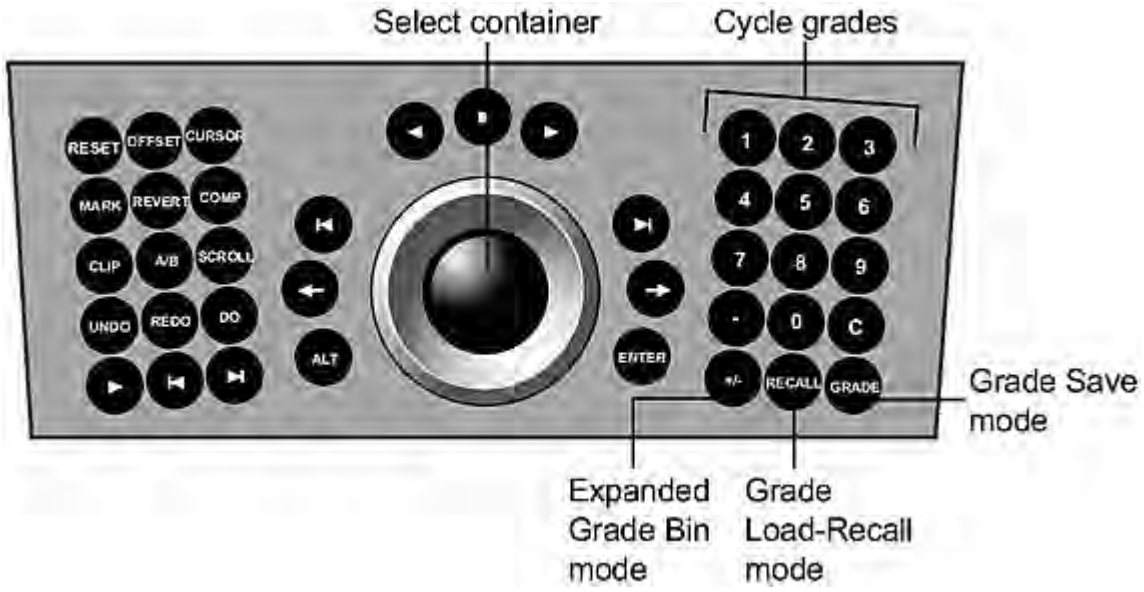
You can compare the current image with the grade you last committed to the undo list.

To toggle between the current image and the grade you last committed to the undo list:

1 Press the COMP button on the Navigation panel.

Working with the Grade Bin

You can use the Navigation panel to access Grade bin functionality.



Saving a Grade to the Grade Bin

Use Grade Save mode to save one or more grades to the Grade bin.

To save a grade to the Grade bin:

1 On the Storyboard, use the ring to navigate to the shot with the grade you want to save.

TIP This step can also be performed after you enter Grade Save mode.

2 Press the Grade button on the Navigation panel.

This activates Grade Save mode.

3 To select a storage container in the Grade bin, do one of the following:

- Use the jog ball on the trackball.
- Press the keypad number button that corresponds to the desired Grade bin container position.

4 To save the shot's grade to the storage container, double-press the keypad number button that corresponds to the desired Grade bin container.

After the grade is saved, you are still in Grade Save mode and can save other grades. Use the trackball ring to navigate to other shots.

5 To exit Grade Save mode, press Grade.

Loading a Grade From the Grade Bin

Use Grade Load-Recall mode to load grades from the Grade bin to one or more shots on the Storyboard.

To load a grade from the Grade bin:

1. On the Storyboard, navigate to the shot to which you want to load one or more grades.

   TIP This step can also be performed after you enter Grade Load-Recall mode.

2. Press the Recall button on the Navigation panel.

   This activates Grade Load-Recall mode.

3. To select a grade in the Grade bin, do one of the following:
   - Use the jog ball on the trackball to navigate the Grade bin.
   - Press the keypad number button corresponding to the desired Grade bin container position once to jump to it.

4. To load the grade to the current selected shot on the Storyboard, double-press the appropriate number button twice.

   After the grade is loaded to the shot, you are still in Grade Load-Recall mode and can load other grades from the Grade bin to the shot. Use the trackball ring to navigate to other shots.

5. To exit Grade Load-Recall mode, press Recall.

Saving the Current Grade

You can use the Navigation panel to save the current grade file and cut.

NOTE If a grade file has not been created, a default grade file is saved.

To save the current grade file and cut:

1. Press ALT and DO on the Navigation panel.

Adjusting Printer Lights

You can adjust printer lights on the Navigation panel. Printer light values are displayed in the Colour Grading panel's digital display.

NOTE You can also adjust printer lights on the Function panel when the Grade menu is enabled. See Grading in Logarithmic Mode Using the Function Panel (page 2454).

To adjust printer lights:

1. On the Navigation panel, double-press the ALT button.

   The words “Shift ON” appear in the digital display.



2 Use the following buttons on the keypad to make printer light adjustments to the image.

Press:	To:
7	Increase red brightness by one step.
4	Decrease red brightness by one step.
8	Increase green brightness by one step.
5	Decrease green brightness by one step.
9	Increase blue brightness by one step.
6	Decrease blue brightness by one step.
1	Increase cyan brightness by one step.
- (minus sign)	Decrease cyan brightness by one step.
2	Increase magenta brightness by one step.
0	Decrease magenta brightness by one step.
3	Increase yellow brightness by one step.
. (period)	Decrease yellow brightness by one step.

Copying Grades

You can temporarily or permanently copy a grade to the current shot from another shot in the Storyboard.

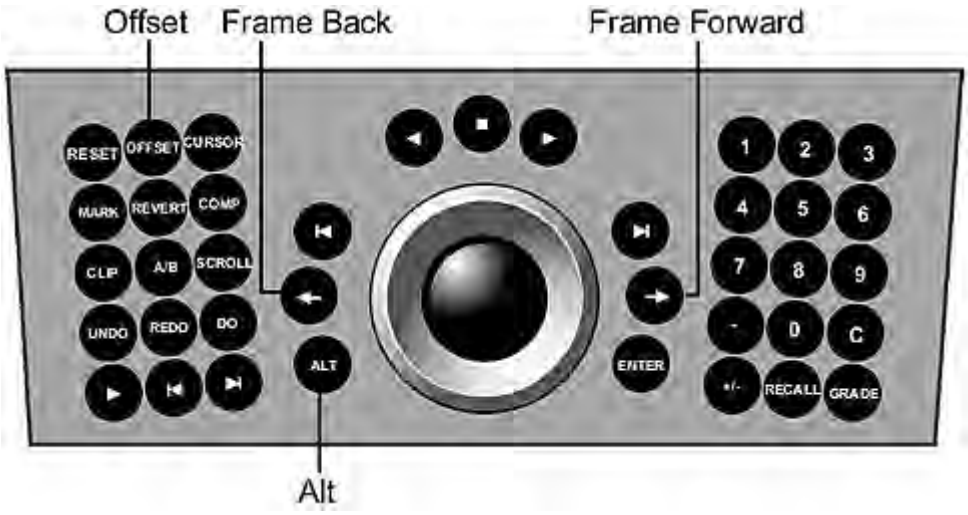
To copy a grade to the current shot from another shot in the Storyboard:

1 Scroll to the shot you would like to copy grades to by performing one of the following two actions:

- Hold down the ALT button on the Navigation panel and then press the Frame Forward or Frame Back button to temporarily apply the grade from the previous or next shot to the current shot.

NOTE Each time you press ALT and Frame Forward or Frame Back, you move one shot at a time through possible sources to copy back to the current Storyboard position. For example, if you hold down ALT and press Frame Forward three times, you see a preview of the current Storyboard position with the grade from three shots ahead applied.

- Press Offset on the Navigation panel and then scroll through the shots by pressing the Scroll button and using the ring portion of the trackball.



2. Press Do to permanently copy the grade from the shot outlined in blue to the current shot.
3. Press the ALT button to cancel the operation.

Adding Keyframes

When you animate parameters, you can set keyframes using the Mark button.

To add keyframes:

1. Go to the frame where you want to add the keyframe.
2. Press Mark.
3. Modify the parameters.
4. Repeat the above steps as required.

Copying and Pasting Keyframes

You can copy and paste keyframes using the Navigation and Function panels of the Autodesk control surface panel.

To copy and paste keyframes:

1. On the Navigation panel, press the bottom MORE button.
2. Press the F1 button to enable the P. C & P menu.
   The Permanent Copy/Paste (P. C & P) menu is displayed.



3 On the Function panel, press the appropriate button (F1-F8) to perform your choice of copy and paste operation.

NOTE When you press one of these buttons, the corresponding button is automatically pressed in the Channel hierarchy and the corresponding menu command is highlighted in the Function panel display. For example, when you press F1 to add a keyframe, the Add button is pressed and the Add command is highlighted.

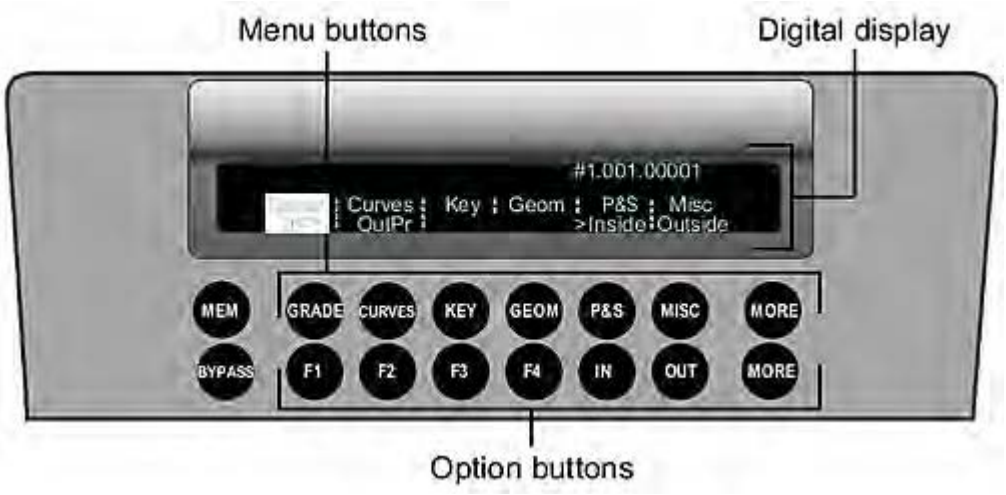


Copy/Paste Keyframe option	Function
F1	Add keyframe
F2	Copy
F3	Remove keyframe
F4	One frame
F5	Next keyframe
F6	All frames
F7	Previous keyframe

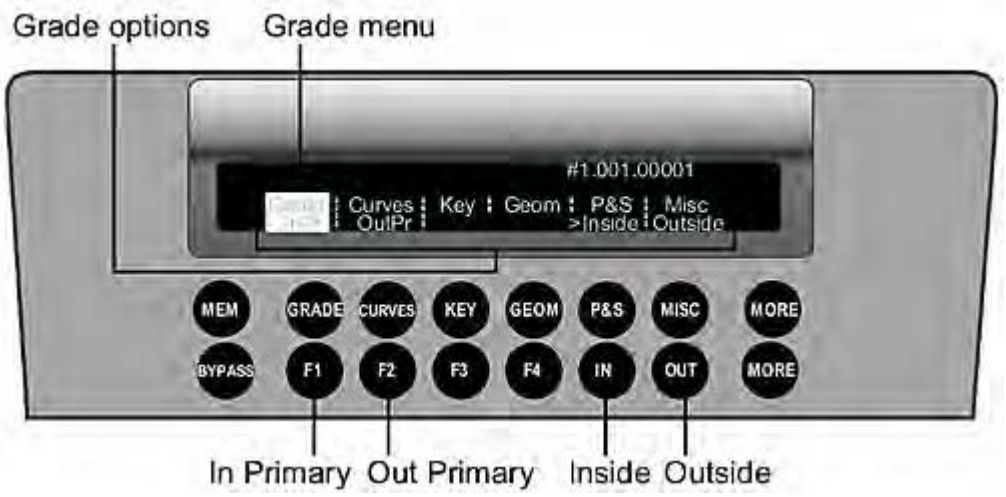
Copy/Paste Keyframe option	Function
F8	Paste

Selecting Menus

Selecting a menu from the first row of buttons on the Navigation panel determines which options are available in the second row of buttons. What you select in the Navigation panel also determines the features available on the Function panel.



You can see which options are available by checking the digital display. For example, in the following illustration the Grade menu has been selected and the InPr, OutPr, Inside, and Outside options appear in the digital display. The location of these options in the digital display is the same as their location amongst the option buttons.



To select a menu:

1 Choose one of the following.

Press:	To:
Grade	Enable the In Primary, Out Primary, Inside, and Outside options, and activate Logarithmic or Linear grading in the Function panel. See Performing Colour Grading (page 2451).
Curves	Enable the In Primary, Out Primary, Hue, and RGB options, and activate curves grading in the Function panel. See Performing Curves Grading (page 2462).
Key	Enable the Key menu, and activate keying functionality in the Function panel. See Extracting Keys using the HLS Keyer (page 2467) and Extracting Keys using the Diamond Keyer (page 2472).
Geom	Enable the Rectangle, Circle, Wipe (in Dual view), Bezier (free-form geometry), Inside, and Outside options, and activate geometry grading in the Function panel. See Creating and Grading Geometries (page 2477).
P&S	Activate repositioning functionality in the Function panel. When this function is selected, no additional options are activated in the Navigation panel. See Repositioning Shots (page 2449).
More (upper)	Access the following additional options: Label, Thumbnails, and GUI. These options pertain to elements in the user interface. See Customizing the View (page 2430).
More (lower)	Access the following additional options: Low, High, and Screen. These options pertain to the control surface appearance. See Modifying the Appearance of the Autodesk Control Surface (page 2483).

NOTE There is no functionality assigned to the Misc button for this release.

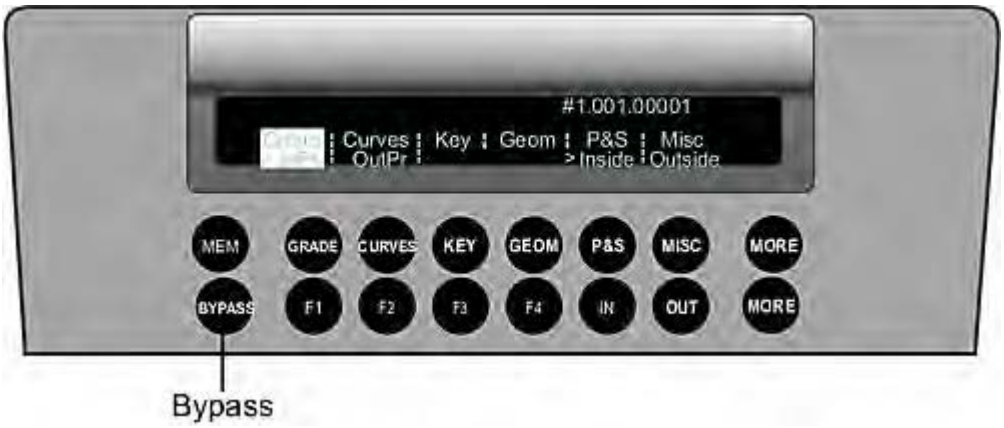
Caching Memory

When a shot does not play back in real time, you can cache the frames into memory using the Mem button.



Bypassing Functions

Use the Bypass button on the Navigation panel to bypass the currently selected menu.



Resetting Functions

Use the Reset button on the Navigation panel to reset parameters for the current menu only, or to reset all parameters on the currently selected shot.



To reset parameters for the current menu only:

1 Press the Reset button once.

To reset all parameters on the shot:

1 Double-press the Reset button.

TIP You can reset individual parameters by pressing its assigned Reset button. These buttons are indicated throughout this chapter.

Repositioning Shots

You can reposition a shot by:

- Scaling
- Panning vertically and horizontally
- Rotating
- Changing the aspect ratio
- Flipping and flopping

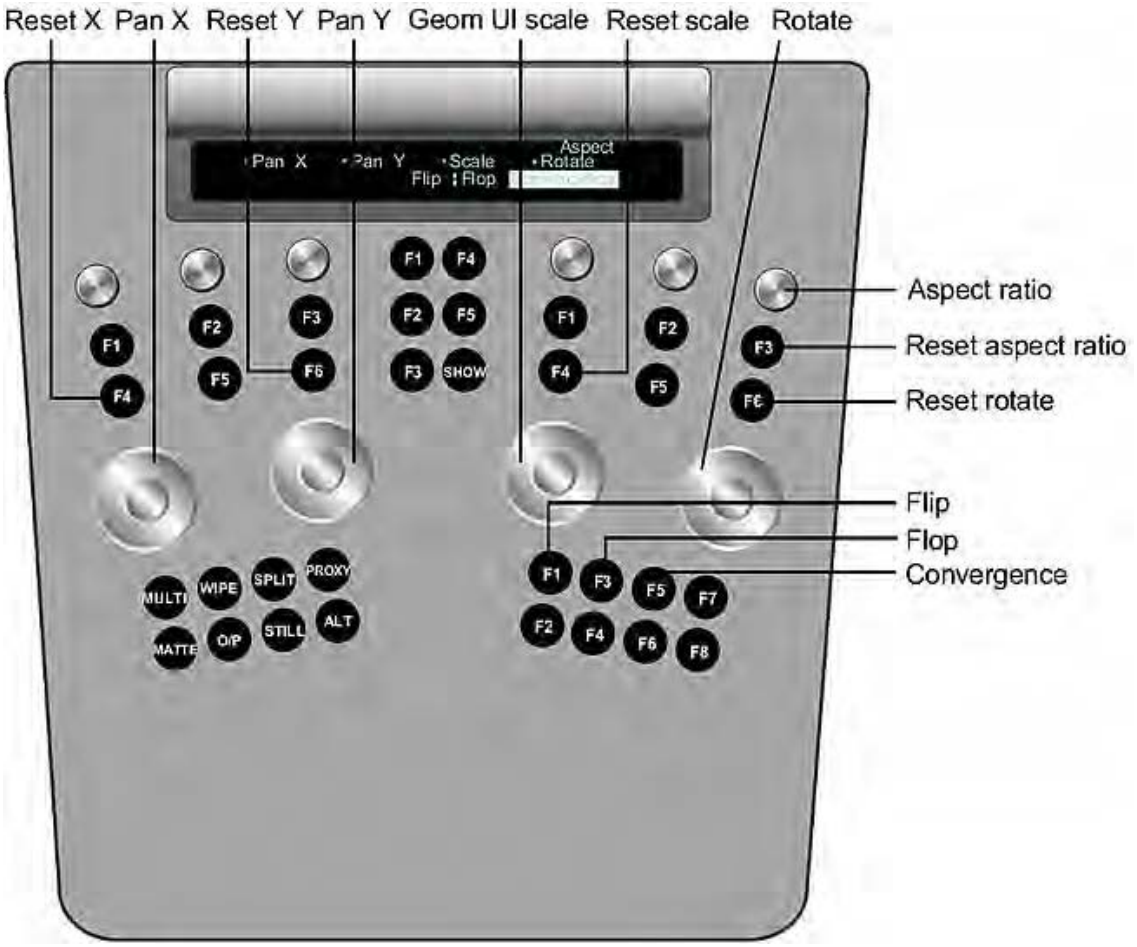
When Sync mode is enabled in a stereoscopic project, you can use the Convergence tool to pan and scale both eyes at the same time in opposite directions.

To reposition a shot:

1 Press the P&S button on the Navigation panel.



2 On the Function panel, use the following controls.



Use:	To:
Wheels	Reposition the image. Wheel 1=Pan X Wheel 2=Pan Y Wheel 3=Scale Wheel 4=Rotate To reset a wheel, press the button above it.
Knob at right	Change the image's aspect ratio. To reset, press the button below it.
First row of buttons at bottom right	Flip or flop the image. F1=Flip F3=Flop Toggle the state of the Convergence Tool. F5=Toggle Convergence
SHOW button	Toggle the geometry wireframe on or off.

Performing Colour Grading

You can use the Autodesk control surface to perform input and output grading in Logarithmic or Linear mode.

As with other menus, you select the Grade menu from the Navigation panel. This activates colour grading functionality on the Function panel. The Colour Grading panel is enabled regardless of which menu is active in the Function panel. The features available on these panels depend on whether you are in Logarithmic or Linear mode. You specify the colour architecture in the Setup Grade menu in the user interface.

Logarithmic Mode

When Log is enabled in the Setup Grade menu, the Colour Grading panel and the Function panel are activated with logarithmic colour grading functionality.

Grading in Logarithmic Mode Using the Colour Grading Panel

Use the Colour Grading panel to:

- Adjust brightness balance and contrast balance for the entire image or for a range of colours.
- Adjust brightness in the shadows, midtones, and highlights.

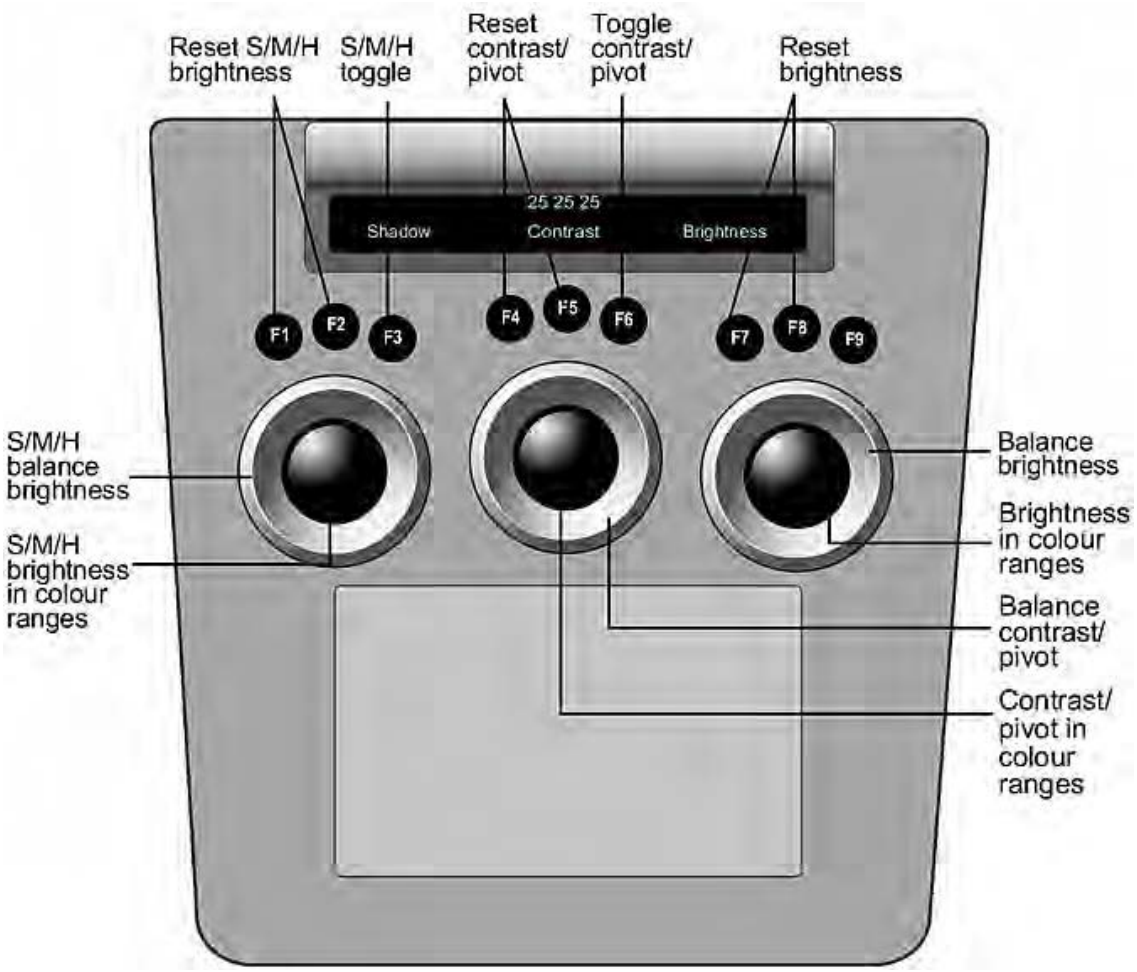
There are two available setups for the Colour Grading panel. You can switch between setups by pressing the F9 button.

The default setup allows you to modify contrast, overall brightness, and brightness in the shadows, midtones, and highlights. Adjusting brightness in the shadows, midtones, and highlights requires that you switch the functionality of the first trackball among the three. You use the second and third trackballs to adjust contrast and brightness in the overall image.

The alternate setup distributes brightness controls for the shadows, midtones, and highlights amongst the three trackballs. This allows you to balance the values without having to constantly switch the functionality of a single trackball.

To colour grade using the Colour Grading panel default setup:

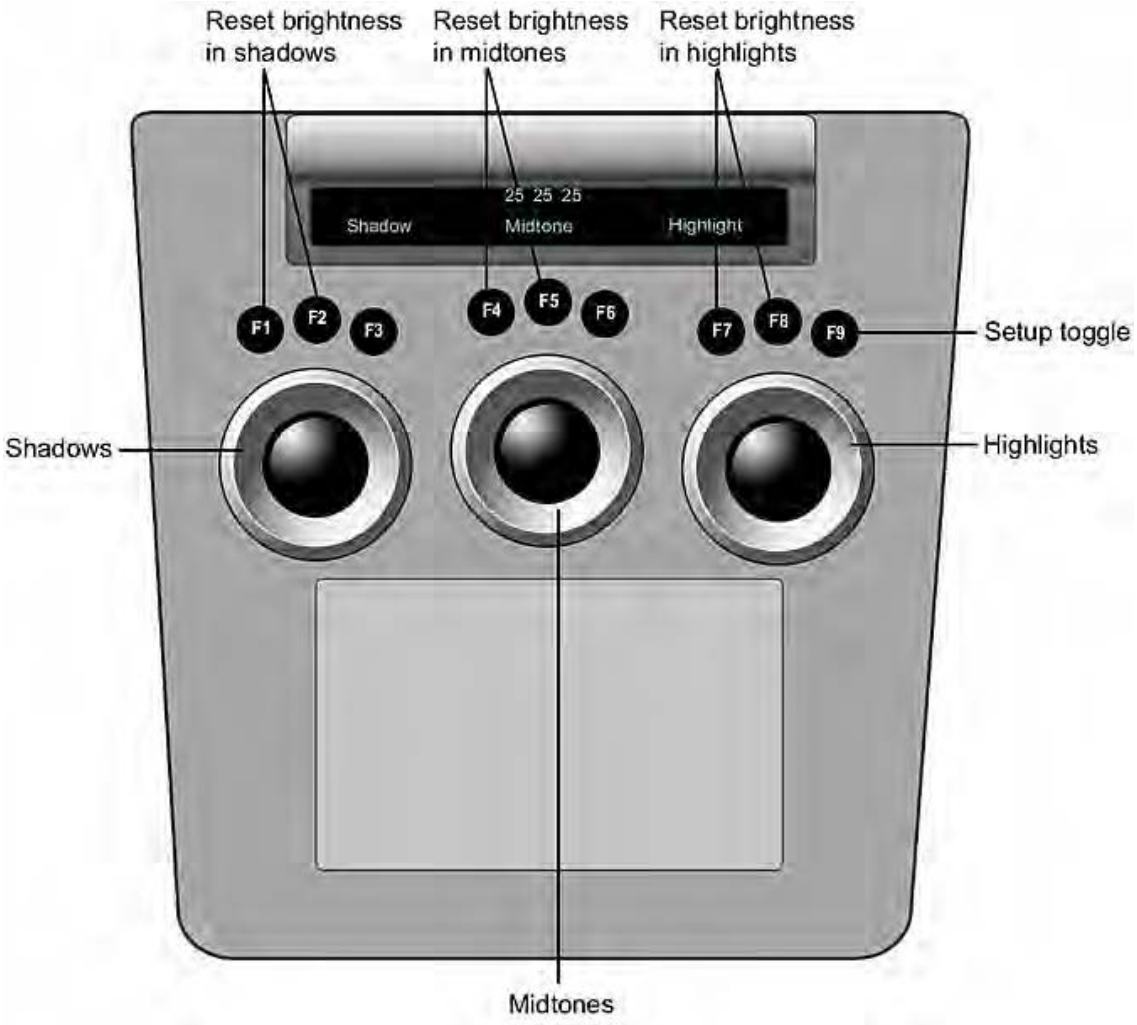
1 On the Colour Grading panel, use the following controls.



Use:	To:
First ring	Adjust brightness in the shadows, midtones, or highlights for the overall image. To reset, press F1.
First jog ball	Adjust brightness in the shadows, midtones, or highlights for a range of colours. To reset, press F2.
F3 button	Switch functionality for the first trackball between shadows, midtones, or highlights.
Second ring	Adjust contrast/pivot in the overall image. To reset, press F4.
Second jog ball	Adjust contrast/pivot in a range of colours. To reset, press F5.
F6 button	Toggle functionality for the second trackball between contrast and pivot.
Third ring	Adjust brightness in the overall image. To reset, press F7.
Third jog ball	Adjust brightness in a range of colours. To reset, press F8.

To colour grade using the Colour Grading panel alternate setup:

1. Press the F9 button to switch to the alternate setup.
2. On the Colour Grading panel, use the following controls.



Use:	To:
First ring	Adjust overall brightness in the shadows. To reset, press F1.
First jog ball	Adjust brightness in the shadows for a range of colours. To reset, press F2.
Second ring	Adjust overall brightness in the midtones. To reset, press F4.
Second jog ball	Adjust brightness in the midtones for a range of colours. To reset, press F5.
Third ring	Adjust overall brightness in the highlights. To reset, press F7.
Third jog ball	Adjust brightness in the highlights for a range of colours. To reset, press F8.

Use:	To:
F9 button	Toggle between the panel default and alternate setup.

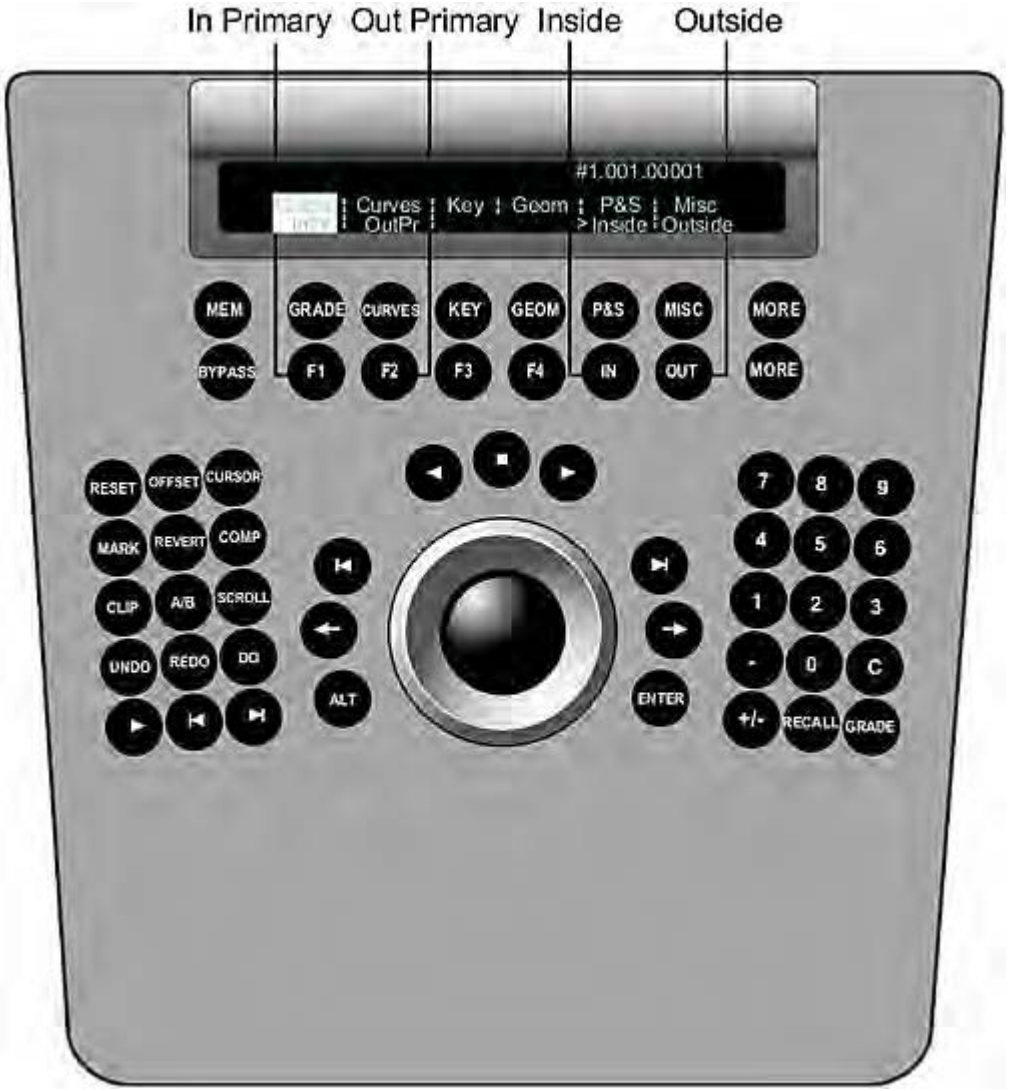
Grading in Logarithmic Mode Using the Function Panel

When the Grade menu is enabled, you can use the Function panel to:

- Adjust overall hue, brightness, and saturation.
- Balance brightness and contrast in the red, green, and blue channels.
- Balance brightness of colours in the shadows, midtones, and highlights.
- Adjust cutoff values.
- Increase or decrease printer light settings.

To colour grade in Logarithmic mode using the Function panel:

1. Press the Grade button on the Navigation panel.
   The In Primary, Out Primary, Inside, and Outside options are activated.



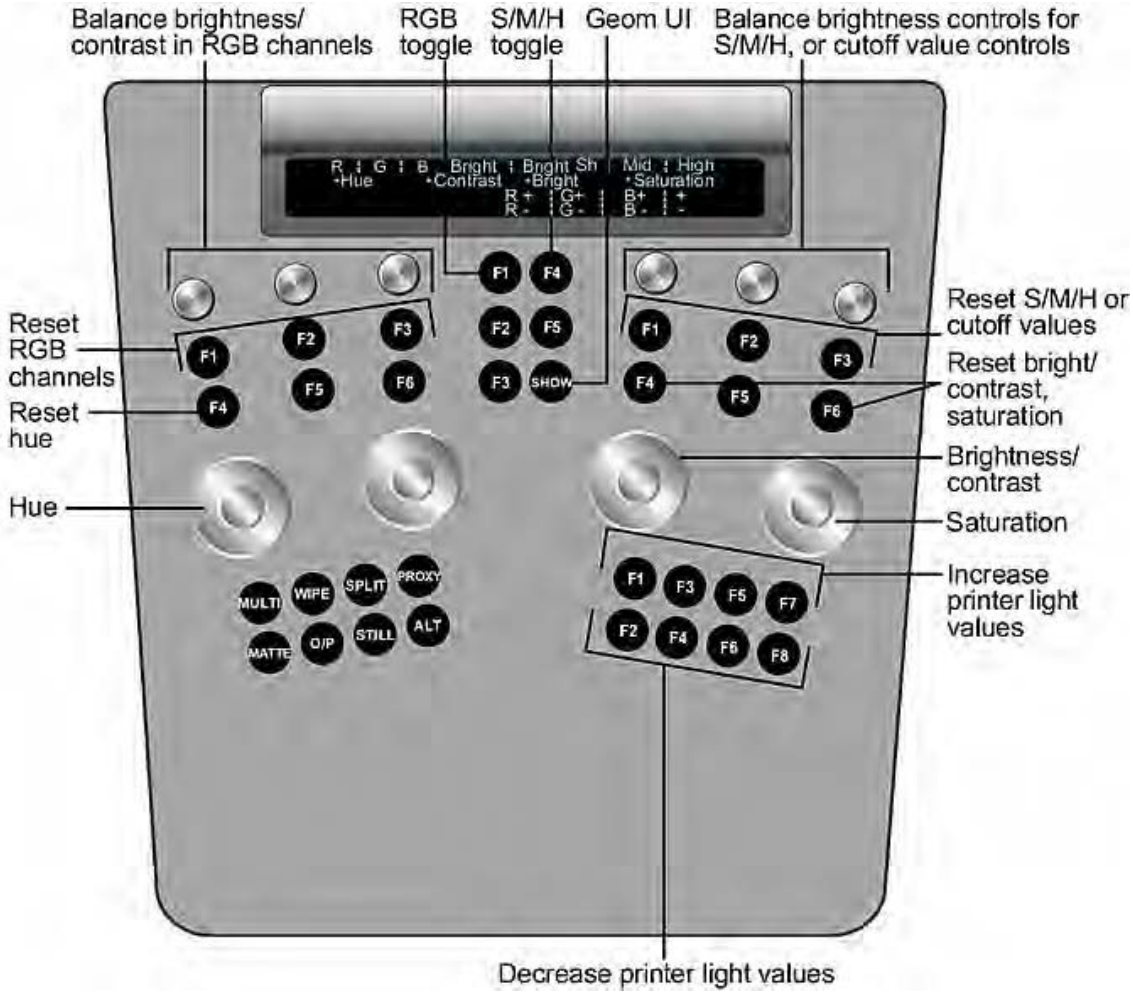
2 Do one of the following:

- Press the F1 option button to work on the initial primary grade.
- Press the F2 option button to work on final curve modifications.
- If you are colour grading a secondary, either a key or a geometry, use the numeric keys on the Navigation panel to select and activate a secondary layer. See Adding Secondary Layers (page 2465).

3 If you are accessing the Grade menu after having created a geometry or key, do one of the following:

- Press the IN option button to grade inside the secondary.
- Press the OUT option button to grade outside the secondary.

4 On the Function panel, use the following controls.



Use:	To:
Wheels	Modify overall hue, brightness/contrast, and saturation. Wheel 1=Hue Wheel 3=Brightness/contrast Wheel 4=Saturation To reset a wheel, press the button above it.
Knobs at top left	Balance brightness or contrast in the RGB channels. Knob 1=Red brightness/contrast Knob 2=Green brightness/contrast Knob 3=Blue brightness/contrast To toggle knob functionality, press the F1 button in the centre column. To reset a channel, press the button below it.
Knobs at top right	Balance S/M/H brightness of the colours, or adjust cutoff values. Knob 4=Brightness in shadows/cutoff values Knob 5=Brightness in midtones/cutoff values Knob 6=Brightness in highlights To toggle knob functionality, press the F4 button in the centre column. To reset a knob, press the button below it.

Use:	To:
First row of buttons at bottom right	Increase printer light values. F1=Red F3=Blue F5=Green F7=Overall
Second row of buttons at bottom right	Decrease printer light values. F2=Red F4=Blue F6=Green F8=Overall Press ALT for half point adjustments.
SHOW button	Toggle the geometry wireframe on or off.

Linear Mode

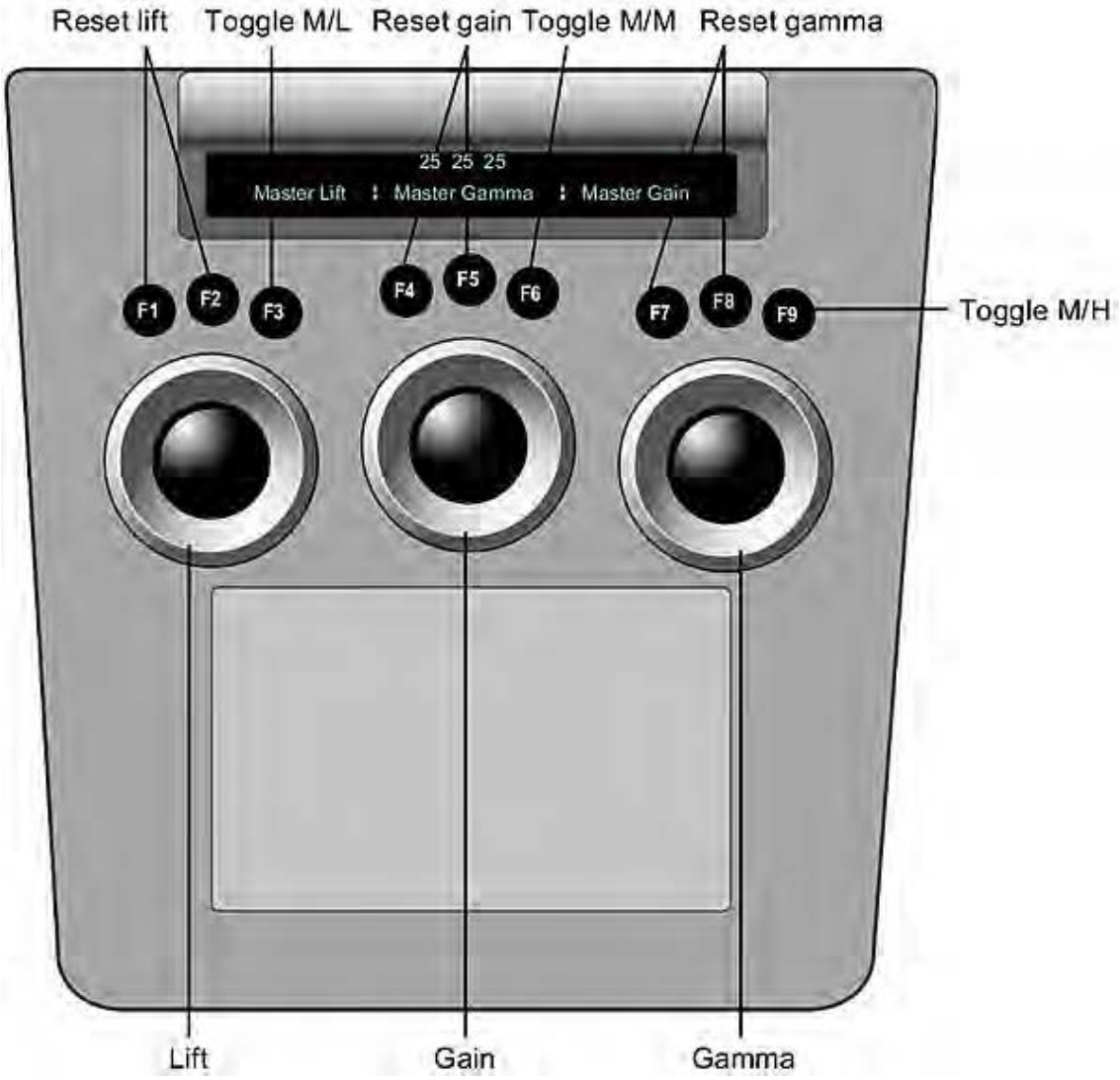
When Lin is enabled in the Setup Grade menu, the Colour Grading panel and the Function panel are activated with linear colour grading functionality.

Grading in Linear Mode Using the Colour Grading Panel

Use the Colour Grading panel to modify lift, gain, and gamma in the low, mid, and high levels of the image, or across the entire image.

To colour grade in Linear mode using the Colour Grading panel:

1 On the Colour Grading panel, use the following controls.



Use:	To:
First ring	Modify lift in the overall image. To reset, press the F1 button.
First jog ball	Modify lift in a range of colours. To reset, press the F2 button.
Second ring	Modify gain in the overall image. To reset, press the F4 button.
Second jog ball	Modify gain in a range of colours. To reset, press the F5 button.
Third ring	Modify gamma in the overall image. To reset, press the F7 button.
Third jog ball	Modify gamma in a range of colours. To reset, press the F8 button.
F3 button	Toggle between Master and Low levels.

Use:	To:
F6 button	Toggle between Master and Mid levels.
F9 button	Toggle between Master and High levels.

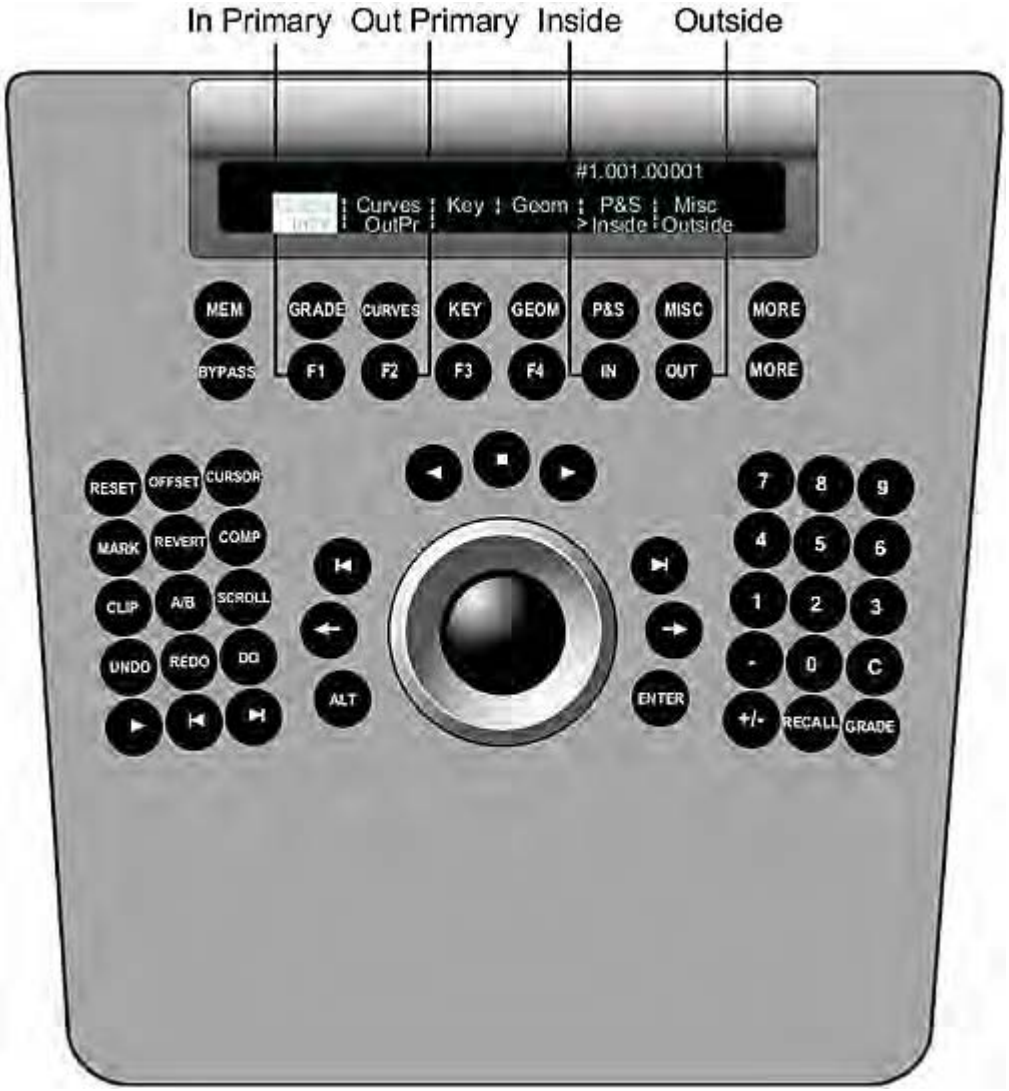
Grading in Linear Mode Using the Function Panel

When the Grade menu is enabled, you can use the Function panel to:

- Modify hue, contrast, gain, and saturation values.
- Balance lift, gamma, and gain values for the red, green, and blue channels.
- Set black and white clip and reference values.

To colour grade in Linear mode using the Function panel:

1. Press the Grade button on the Navigation panel.
   The In Primary, Out Primary, Inside, and Outside options are activated.



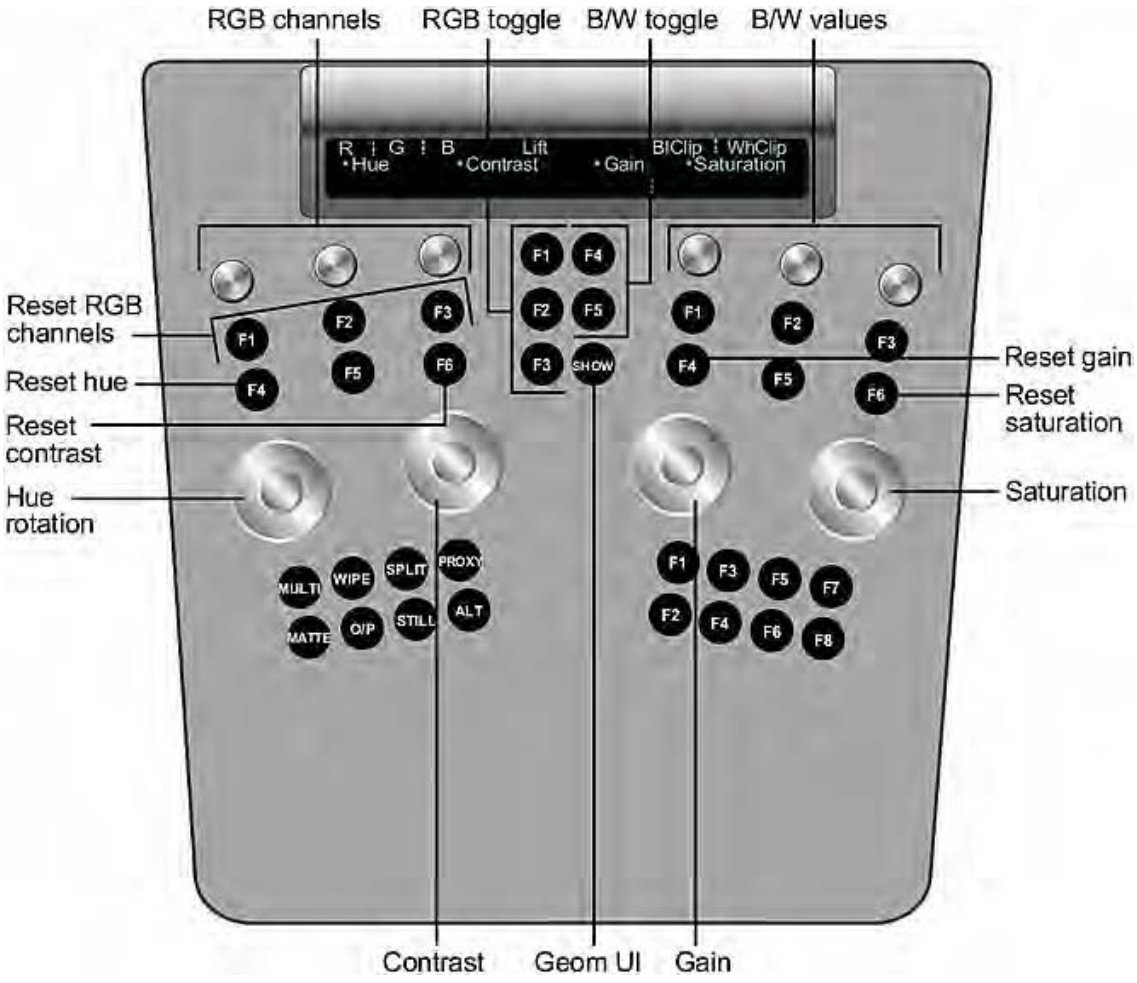
2 Do one of the following:

- Press the F1 option button to work on the initial primary grade.
- Press the F2 option button to work on final curve modifications.
- If you are colour grading a secondary, either a key or a geometry, use the numeric keys on the Navigation panel to select and enable a secondary layer. See Adding Secondary Layers (page 2465).

3 If you are accessing the Grade menu after having created a geometry or key, do one of the following:

- Press the IN option button to grade inside the secondary.
- Press the OUT option button to grade outside the secondary.

4 On the Function panel, use the following controls.



Use:	To:
Wheels	Modify overall hue, contrast, gain, and saturation. Wheel 1=Hue Wheel 2=Contrast Wheel 3=Gain Wheel 4=Saturation To reset a wheel, press the button above it.
Knobs at top left	Balance the RGB channels. Knob 1=Red Knob 2=Green Knob 3=Blue To reset a knob, press the button below it.
Buttons in left centre column	Toggle the type of value to be adjusted in the RGB channels (Knobs 1-3). F1=Lift F2=Gamma F3=Gain

Use:	To:
Knobs at top right	Define values to clamp minimum and maximum luminance. Knob 4=Black Knob 5=White
Buttons in right centre column	Toggle the type of black and white values to be adjusted (Knobs 4-5). F4=Clip F5=Reference
SHOW button	Toggle the geometry wireframe on or off.

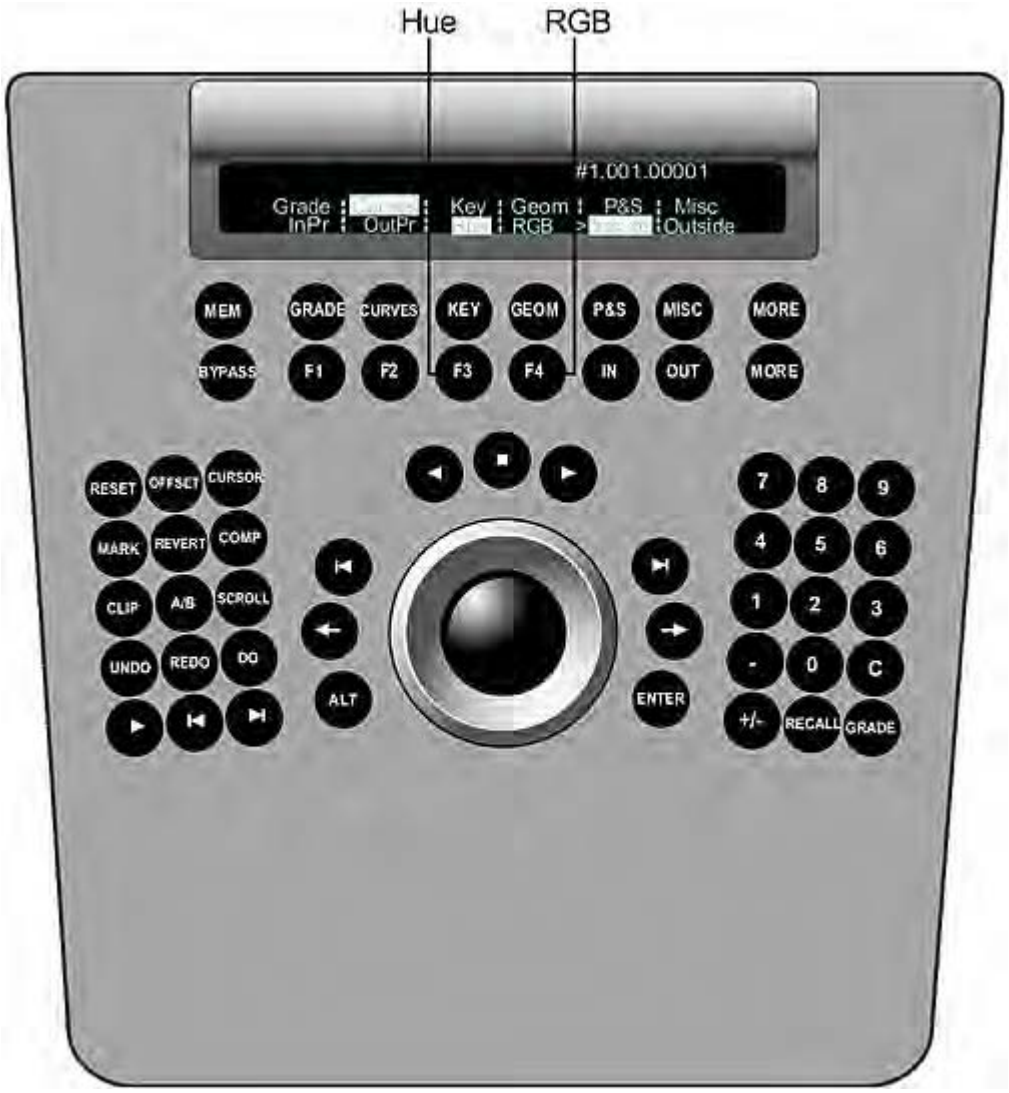
Performing Curves Grading

When the Curves menu is enabled, you can use the Function panel to:

- Modify Hue and RGB curves either after the initial primary colour grade, or after the addition of any Lustre Sparks® effects.
- Adjust the Hue, Lightness, Saturation, and Luminance curves separately.
- Modify the entire curve uniformly or modify individual vertices for RGBCMY channels (red, green, blue, cyan, magenta, and yellow).

To modify Hue and RGB curves:

1. Press Curves on the Navigation panel.
   The In Primary, Out Primary, Hue, and RGB options are activated.



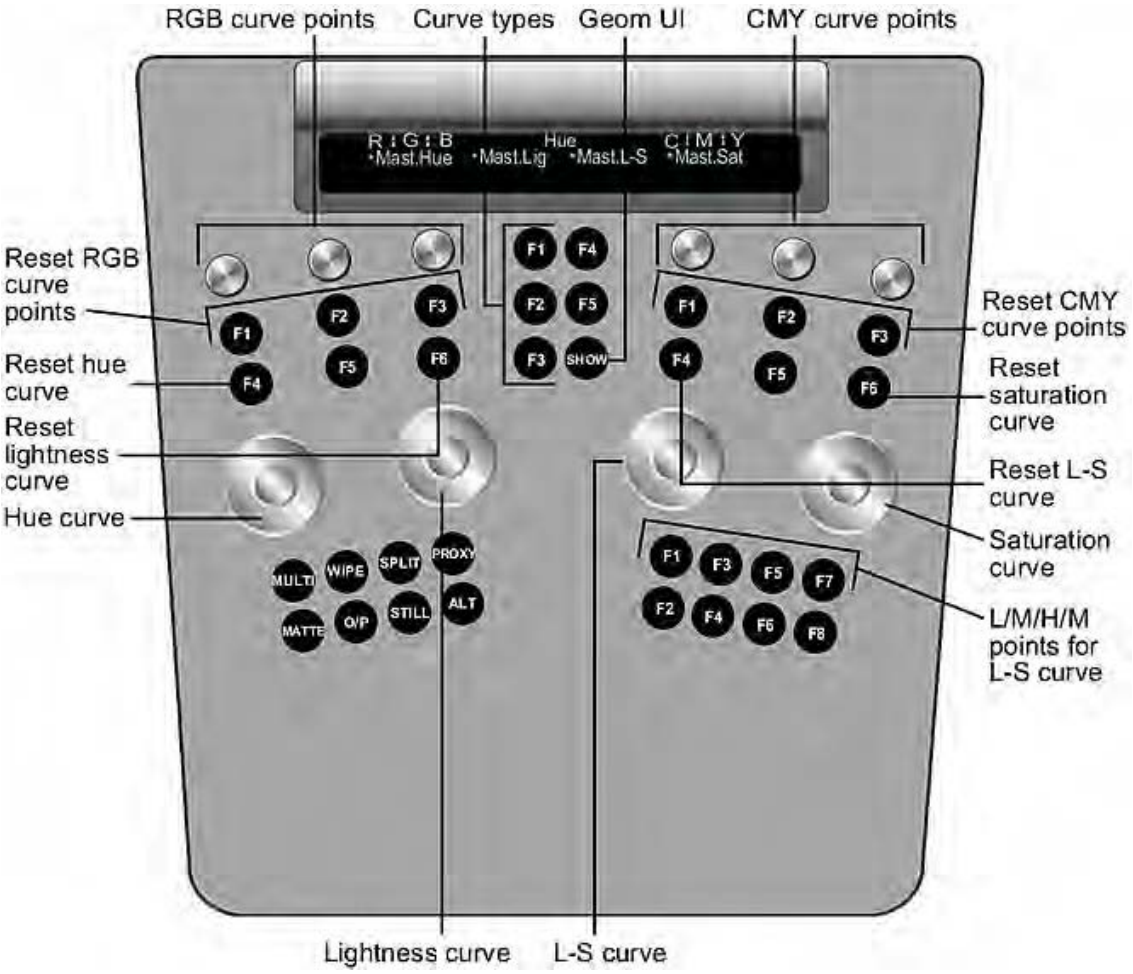
2 Do one of the following:

- Press the F1 option button to indicate that you are working on the initial primary grade.
- Press the F2 option button to indicate that you are working on final curve modifications.

3 Do one of the following:

- Press the F3 option button to work on Hue curves.
- Press the F4 option button to work on RGB curves.

4 On the Function panel, use the following controls.

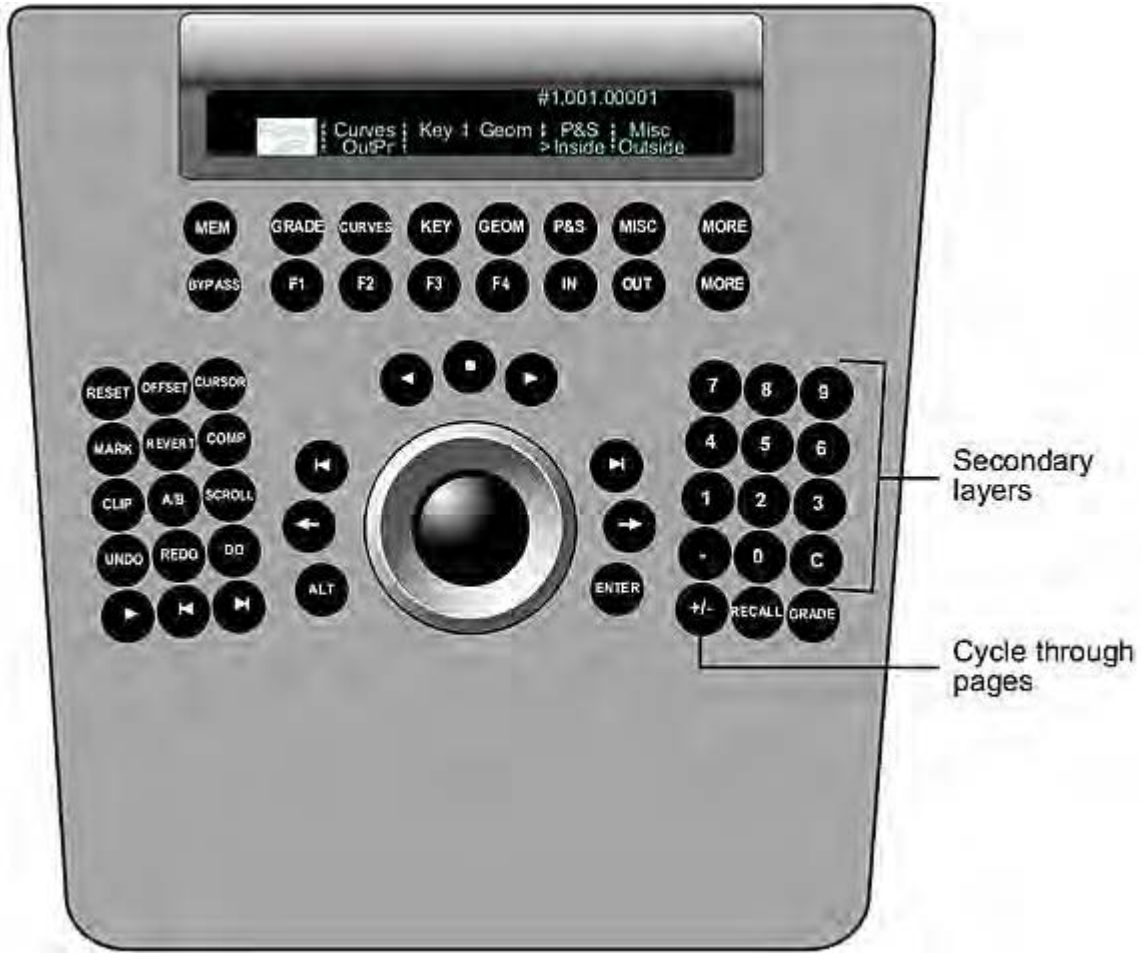


Use:	To:
Wheels	Make overall changes to curves. Wheel 1=Hue curves Wheel 2=Lightness curves Wheel 3=L-S curves Wheel 4=Saturation curves To reset a wheel, press the button above it.
Knobs at top left and top right	Individually modify RGBCMY curve points. Knob 1=Red Knob 2=Green Knob 3=Blue Knob 4=Cyan Knob 5=Magenta Knob 6=Yellow To reset a knob, press the button below it.
Buttons in centre column	Select the type of curve to be modified using the RGBCMY curve points. F1=Hue F2=Light

Use:	To:
	F3=Saturation If you press the F1 button when working with RGB curves, you are toggled to Hue curves.
First row of buttons at bottom right	Select the low point, mid point, high point, or all for the L-S curve. F1=Low F3=Mid F5=High F7=Master
SHOW button	Toggle the geometry wireframe on or off.

Adding Secondary Layers

Secondary layers are used to colour grade specific hues and areas within an image. You can create up to 48 secondary layers for each shot within your project. Secondary layers can be added to your shot by accessing the numerical buttons on the Navigation panel. You can add layers to your shot from the Grade, Curves, Key, or Geometry menu. Once a secondary layer has been created, it can be modified by generating keys and geometries, removing grain, or by adding Lustre Sparks Effects.



Press:	To:
0-9, -, or C	Select, activate, or deactivate a layer. The buttons on the Navigation panel correspond to the position of the secondary layer on the user interface, and not to the actual numbers. For example, press 7 for layer 1. Press the button once to select a layer. Press the button twice to activate/deactivate a layer.
+/-	Cycle through the four pages of secondary layers. Each page consists of 12 secondary layers (e.g., P1 displays layers 1-12, P2 displays layers 13-24, etc.).

Extracting Keys

Lustre provides two separate keyers for extracting keys. When the Key menu is enabled, you can use the Function panel to work with:

- The Hue, Luminance, and Saturation (HLS) Keyer. See Extracting Keys using the HLS Keyer (page 2467).
- The Diamond Keyer. See Extracting Keys using the Diamond Keyer (page 2472).

NOTE The keyer default is set in the Lustre user interface. See the Tools Settings section in the Project Management chapter of the *Autodesk Lustre User Guide*.

Extracting Keys using the HLS Keyer

Use the Navigation panel to:

- Enable the HLS Keyer.
- Select a secondary layer.
- Sample the image.
- Extract a key.

Use the Function panel to adjust the key by setting softness and tolerance ranges, modify key edges, invert the key, and link layers.

To enable the HLS Keyer in the Navigation panel:

1. Press the Key button once to activate the Key menu, and then press F1 to enable the HLS Keyer.
   The HLS Keyer and Inside are enabled.



To select a secondary layer in the Navigation panel:

1 Use the numeric keys to select and enable a secondary layer. Press the number key once to select the layer, and then double-press it to enable it. To enable layer 10, press the minus button (-). To enable layer 11, press 0. To enable layer 12, press the C button.

To sample the image in the Navigation panel:

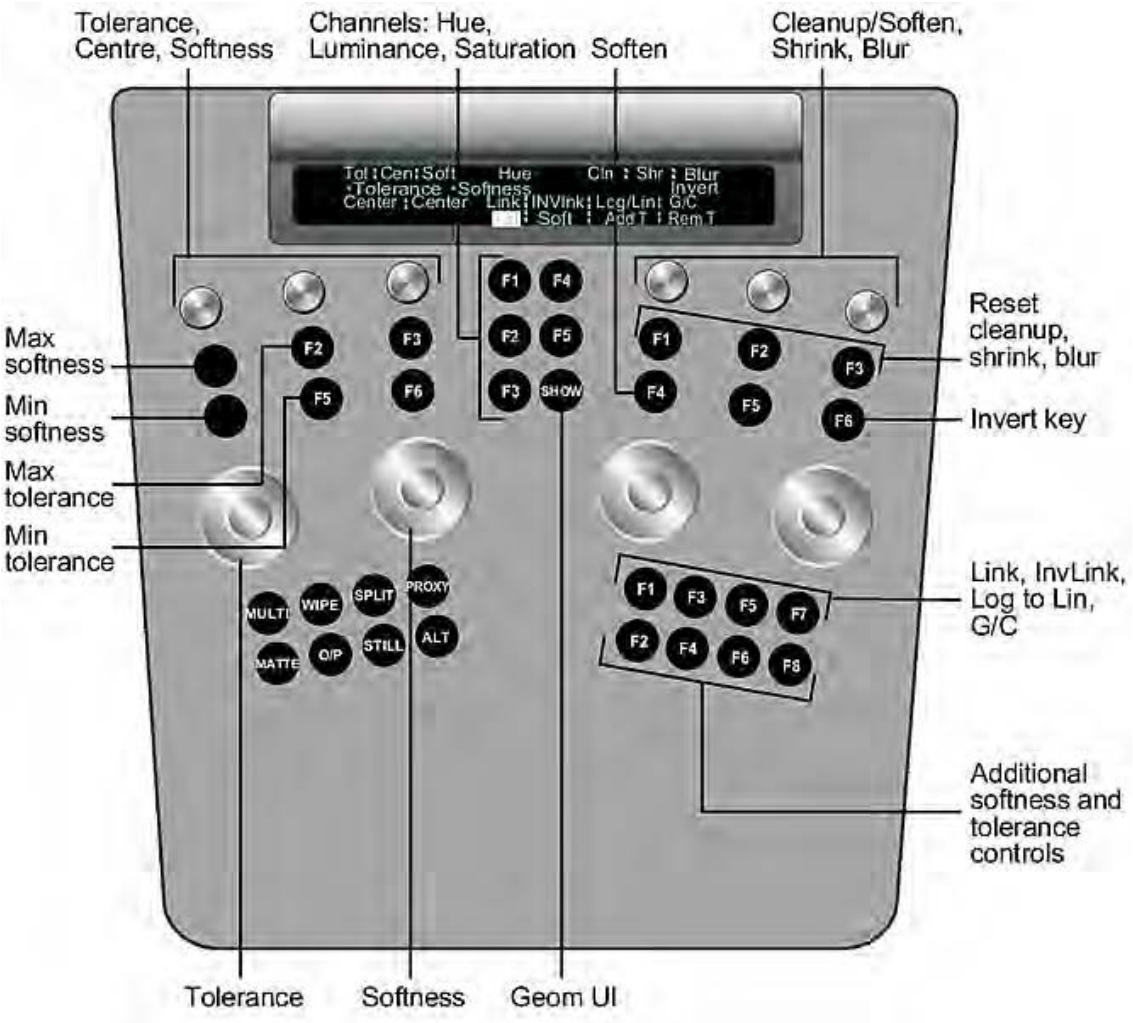
1 Press F3 (pick). The trackball on the Navigation panel is activated as a picker, allowing you to move through the image. Press F3 again to pick the current pixel for the sample. To sample an area of the image, hold down the ALT button while in picker mode and draw, with the trackball, a rectangular area of the image, and then release the ALT button.

To extract a key in the Navigation panel:

1 To extract a key after input primary grading has been performed, press F4. To switch back to the primary-graded result, press F4 again.

To adjust a key in the Function panel:

1 Use the following controls listed in the table below to adjust the key for the sampled colour.



Use:	To:
Wheels	Adjust the tolerance and softness around the picked colour. Wheel 1=Tolerance Wheel 2=Softness
Knobs at top left	Adjust minimum/maximum tolerance, centre tolerance, and minimum/maximum softness. Knob 1=Minimum/maximum tolerance Knob 2=Centre Knob 3=Minimum/maximum softness
Upper left F1 and F4 buttons	Enable either minimum or maximum tolerance (Knob 1) adjustments. F1=Minimum F4=Maximum
Upper left F3 and F6 buttons	Enable either minimum or maximum softness (Knob 3) adjustments. F3=Minimum

Use:	To:
	F6=Maximum
Buttons in centre column	Adjust tolerance, centre, and softness values separately for each of the channels forming the basis of the key. F1=Hue F2=Luminance F3=Saturation
Knobs at top right	Modify key edges and remove stray pixels from a key. Knob 4=Cleanup Knob 5=Shrink Knob 6=Blur
Buttons at top right	Press F1 to enable cleanup (Key-out), shrink, and blur. Press F4 to enable soften (Key-in filter). F1=Cleanup F2=Shrink F3=Blur F4=Soften Single press F2 and F3 to reset shrink and blur, respectively. Double-press F1 and F4 to reset cleanup and soften, respectively.
Upper right F6 button	Invert the key.
First row of buttons at bottom right	Press F1 to cycle through add, intersect, subtract, or link layers. F1=Add, intersect, subtract, or link F3=Invert link F5=Log to Lin F7=GPU compatibility
Second row of buttons at bottom right	Add or subtract tolerance and softness from the key. F2=Tolerance F4=Softness F6=Add softness/tolerance F8=Subtract softness/tolerance Press F3 on the Navigation panel to change the cursor into a picker prior to using any of the above buttons. When done defining tolerance and softness, press F3 again.
SHOW button	Toggle the geometry wireframe on or off.

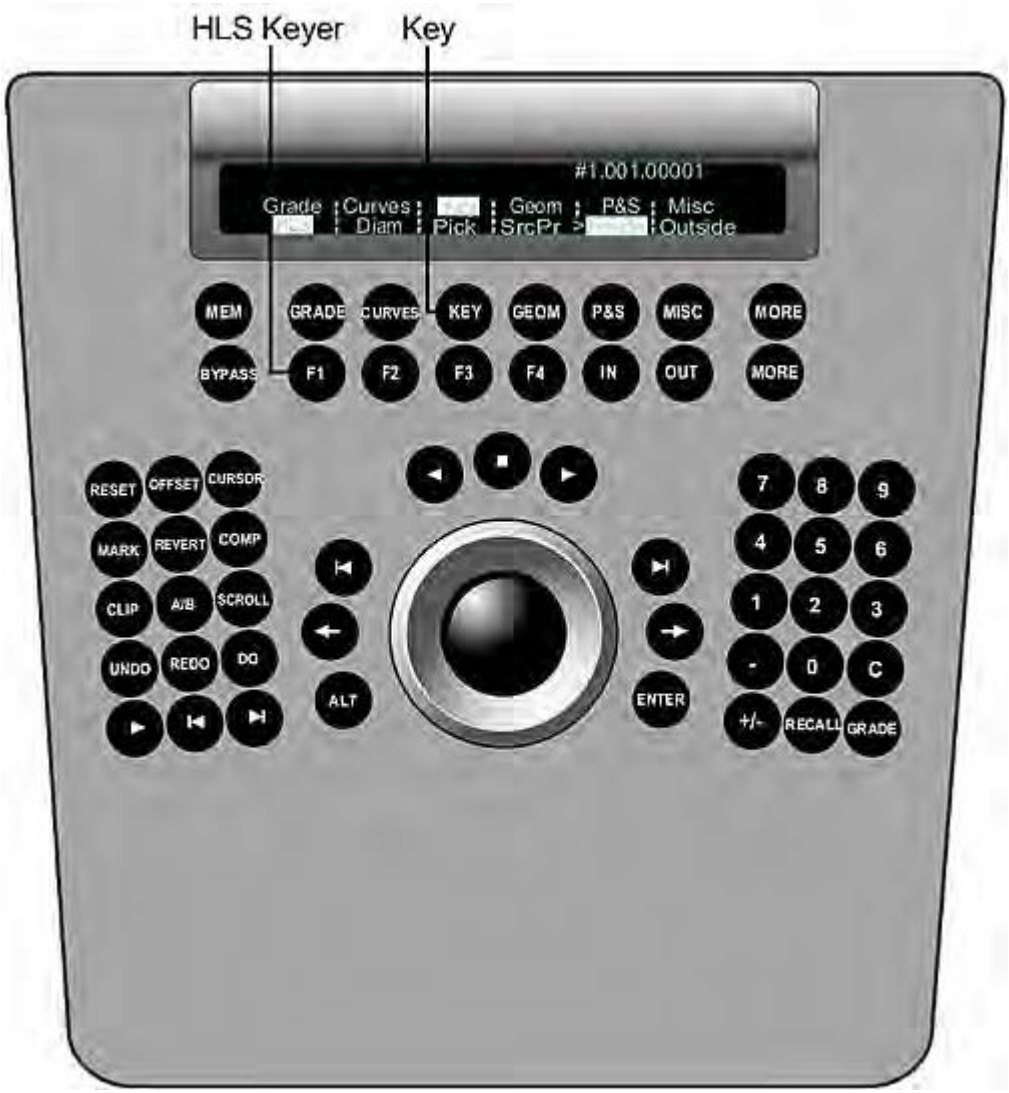
Toggling the Keyer Button and Keyer Channels On and Off

Users can toggle the on or off status of the Keyer button visible on the Navigation panel. Users can also enable or disable any or all of the Hue, Luminance, or Saturation channels on the Function panel.

To make the HLS Keyer status visible on the Navigation panel:

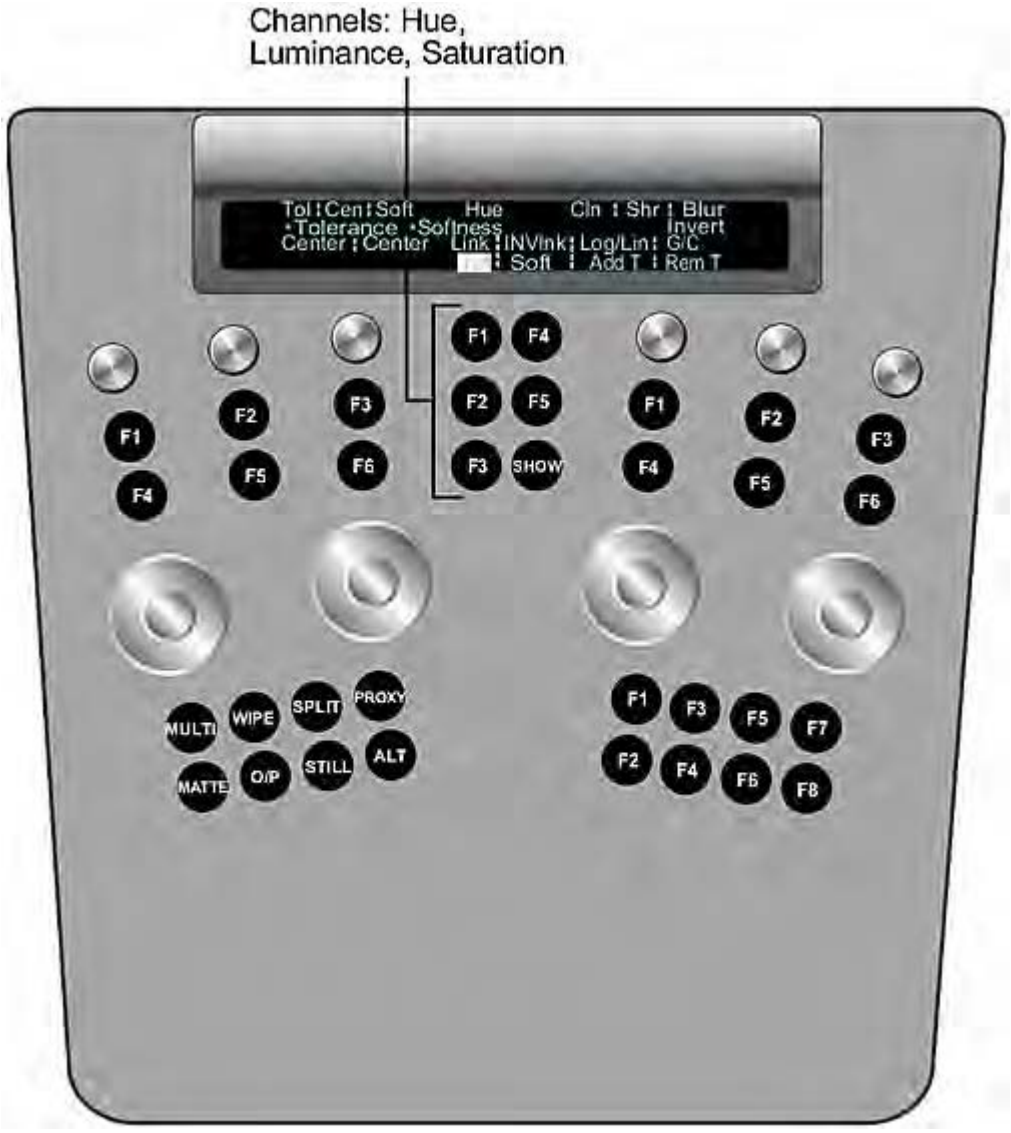
1. Press the Key button once to activate the Key menu, and then press F1 to enable the HLS Keyer channel and activate backlighting to show its enabled status.
2. Double-press the F1 button to disable the HLS Keyer channel.

   The HLS Keyer is disabled in the Lustre application.



To enable or disable the Hue, Luminance, or Saturation channels on the Function panel:

1. Double-press F1, F2, or F3.



Extracting Keys using the Diamond Keyer

Use the Navigation panel to:

- Enable the Diamond Keyer.
- Select a secondary layer.
- Sample the image.
- Extract a key.

Use the Function panel to adjust the key by setting softness and tolerance ranges, modify key edges, invert the key, and link layers.

To enable the Diamond Keyer in the Navigation panel:

1 Press the Key button once to activate the Key menu, and then press F2 to select the Diamond Keyer.

The Diamond Keyer is enabled.

NOTE To hide/show the hue cube, press Alt+Show.



To select a secondary layer in the Navigation panel:

1 Use the numeric keys to select and enable a secondary layer. Press the number key once to select the layer, and then double-press to enable it. To enable layer 10, press the minus button (-). To enable layer 11, press 0. To enable layer 12, press the C button.

To sample the image in the Navigation panel:

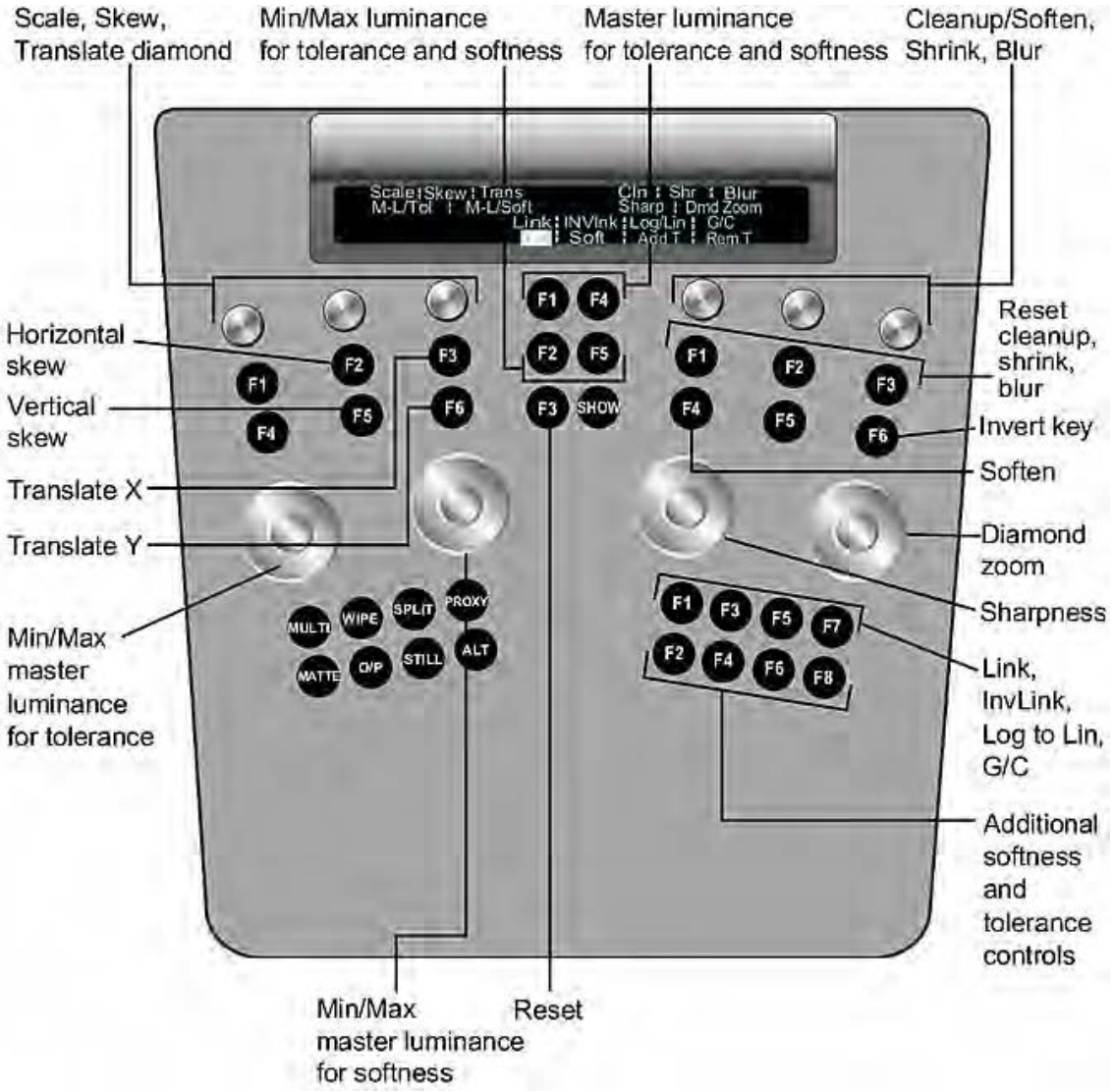
1 Press F3 (pick). The trackball on the Navigation panel is activated as a picker, allowing you to move through the image. Press F3 again to pick the current pixel for the sample. To sample an area of the image, hold down the ALT button while in picker mode and draw, with the trackball, a rectangular area of the image, and then release the ALT button.

To extract a key in the Navigation panel:

1 To extract a key after input primary grading has been performed and the input LUT has been applied, press F4. To switch back to the primary-graded result, press F4 again.

To adjust a key in the Function panel:

1 Use the following controls listed in the table below to adjust the key for the sampled colour.



Use:	To:
Wheels	Adjust tolerance, softness, and sharpness around the picked colour. Zoom in (left) and zoom out (right). Wheel 1=Tolerance Wheel 2=Softness Wheel 3=Sharpness Wheel 4=Zoom in/out

Use:	To:
Knobs at top left	Scale, skew, or translate (move) the chrominance Softness and chrominance Tolerance Diamonds. Knob 1=Scale Knob 2=Skew Knob 3=Translate
Upper left F2 and F5 buttons	Skew the chrominance Tolerance or chrominance Softness Diamonds horizontally and vertically. F2=Horizontal skew F5=Vertical skew
Upper left F3 and F6 buttons	Move the Softness Diamond or Tolerance Diamond left or right, up or down. F3=X-axis (left or right) F6=Y-axis (up or down)
Buttons in centre column	Enable master luminance for tolerance and softness and minimum/maximum luminance levels for tolerance and softness. F1=Master luminance for tolerance F2=Minimum/maximum for luminance for tolerance F4=Master luminance for softness F5=Minimum/maximum luminance for softness
Knobs at top right	Modify key edges and remove stray pixels from a key. Reduce image grain and noise, and keep edges intact prior to keyer input processing. Knob 4=Cleanup/Soften toggle Knob 5=Shrink Knob 6=Blur
Upper right buttons	Press F1 to enable cleanup (Key-out filter), shrink, and blur. Press F4 to enable soften (Key-in filter). F1=Cleanup F2=Shrink F3=Blur F4=Soften Single-press F2 and F3 to reset shrink and blur, respectively. Double-press F1 and F4 to reset cleanup and soften, respectively.
Upper right F6 button	F6=Invert the key
First row of buttons at bottom right	Press F1 to cycle through add, intersect, subtract, or link layers. F1=Add, intersect, subtract, or link F3=Invert link F5=Log to Lin F7=GPU Compatibility (G/C)
Second row of buttons at bottom right	Add softness/tolerance and remove softness/tolerance. F2=Tolerance F4=Softness

Use:	To:
	F6=Add softness/tolerance F8=Remove softness/tolerance Press F3 on the Navigation panel to change the cursor into a picker prior to using any of the above buttons. When done defining tolerance and softness, press F3 again.
ALT+SHOW buttons	Toggle the hue cube on or off.

Toggling the Diamond Keyer Button On and Off

Users can toggle the on or off status of the Diamond Keyer button visible on the Navigation panel.

To make the Diamond Keyer status visible on the Navigation panel:

1. Press the Key button once to activate the Key menu, and then press F1 to enable the Diamond Keyer and activate backlighting to show its enabled status.
2. Double-press the F1 button to disable the Diamond Keyer.
   The Diamond Keyer is disabled in the Lustre application.



Creating and Grading Geometries

When the Geom menu is enabled, you can create and colour grade up to 12 secondaries by using the Function panel to:

- Select a geometry type.
- Modify geometries.
- Link secondary layers.
- Delete geometries.

After you have created geometries, you can use the Grade menu to colour grade them.

To create geometries:

1. Press the Geom button on the Navigation panel.
   The Rectangle, Circle, Wipe (in Dual view), and Bezier (free-form geometry) options are activated.



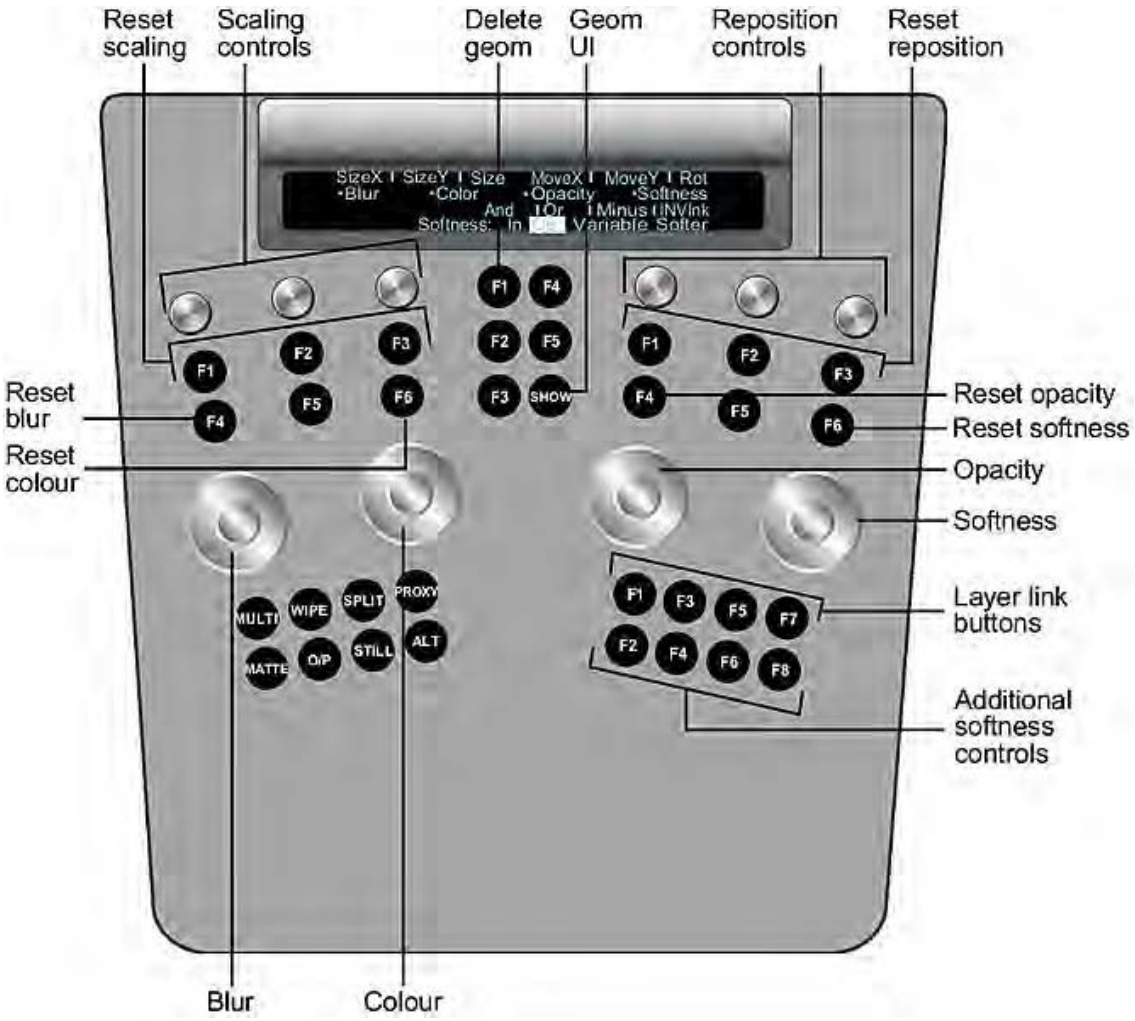
2 Use the numeric keys on the Navigation panel to select and enable a secondary layer. Press the number key once to select the layer, and then press it a second time to enable it. To enable layer 10, press the minus button (-). To enable layer 11, press 0. To enable layer 12, press the decimal button (.).

3 Do one of the following:

- Press F1 to add a rectangle.
- Press F2 to add a circle.
- Press F3 to add a wipe.
- Press F4 to add a Bezier shape using the pen or mouse.

4 Use the jog ball as a geometry mover and press the ALT and shot forward buttons to select the geometries on the current active secondary layer.

5 On the Function panel, use the following controls to adjust the geometry.



Use:	To:
Wheels	Modify the geometry's blur, colour, opacity, and softness. Wheel 1=Blur Wheel 2=Colour Wheel 3=Opacity Wheel 4=Softness The direction for edge softness is inwards or outwards depending on the direction that you rotate the wheel. To reset a wheel, press the button above it.
Knobs at top left	Scale the geometry. Knob 1=Size X Knob 2=Size Y Knob 3=Master (scales in locked proportions) To reset a knob, press the button below it.
Knobs at top right	Move the geometry. Knob 1=Move X Knob 2=Move Y

Use:	To:
	Knob 3=Rotate To reset a knob, press the button below it.
First row of buttons at bottom right	Link secondary layers. F1=Add F3=Intersect F5=Subtract F7=Invert link
Second row of buttons at bottom right	Further modify softness. F2=Apply softness inside mask border F4=Apply softness outside mask border F6=Apply variable softness F8=Change softness type
F1 button in centre column	Delete the currently active geometry.
SHOW button	Toggle the geometry wireframe on or off.

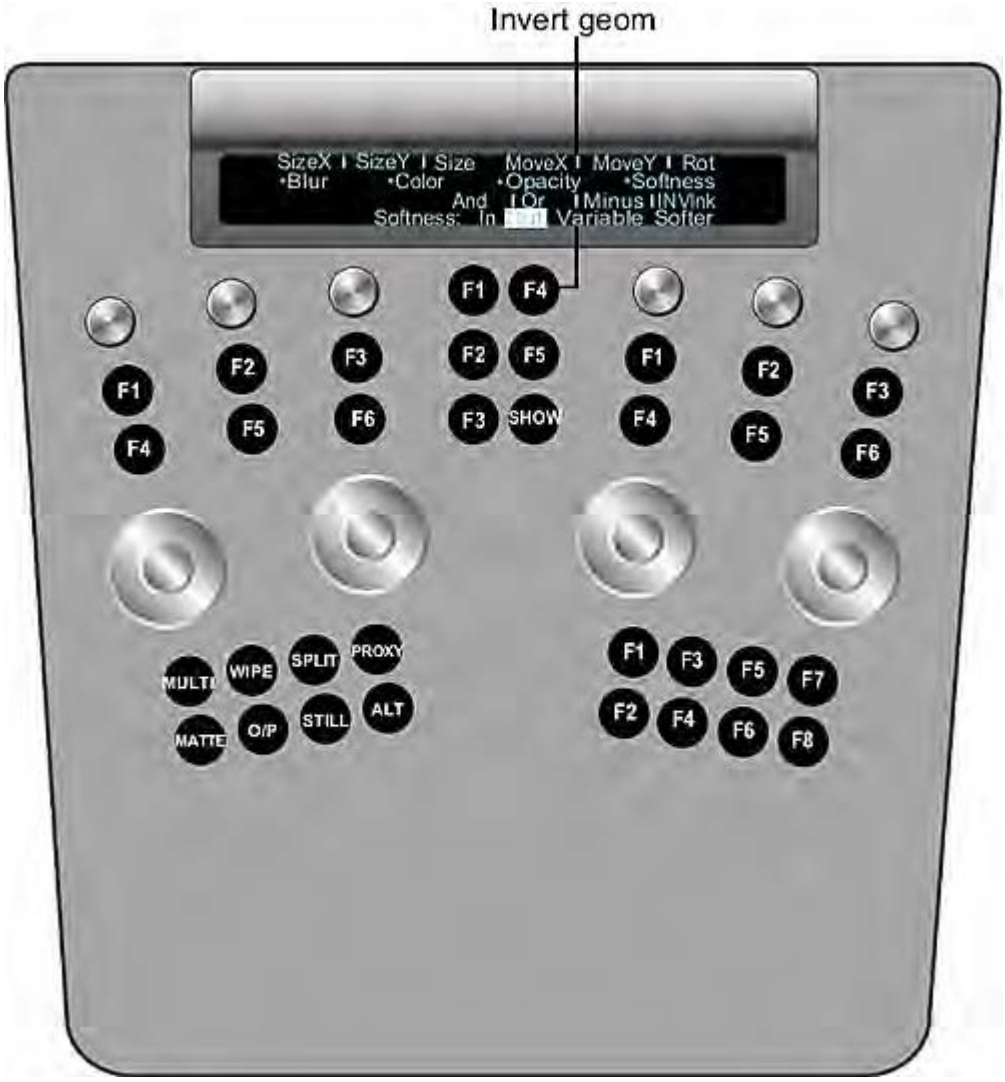
6 To colour grade the geometry, select either the Inside or Outside option button depending on whether colour grading should occur inside or outside the geometry. See Performing Colour Grading (page 2451).

Geometry Invert Functionality

Users can invert geometries on the current secondary layer using the Autodesk control surface.

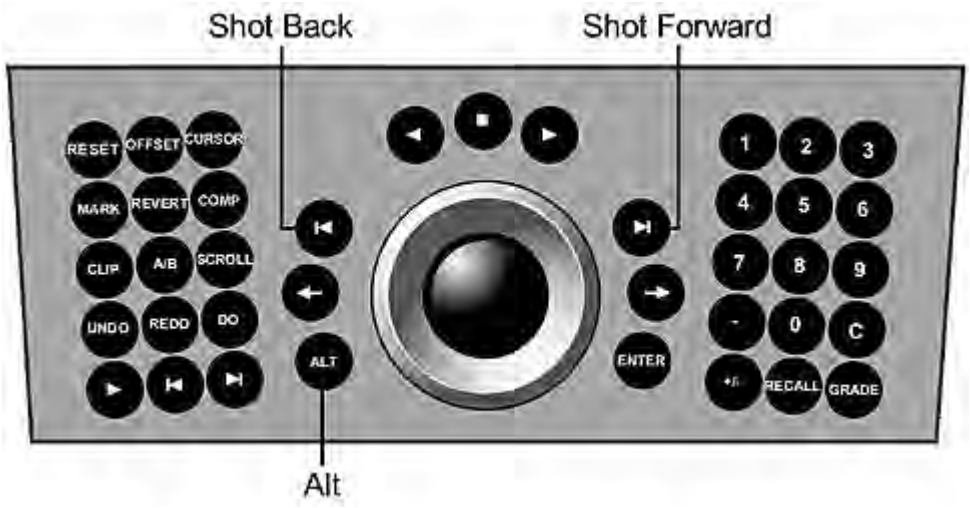
To invert geometries using the Autodesk control surface:

1 On the Function panel, press the F4 button in the centre column.



Cycling Among Geometries

You can now move from one geometry to the next in the order of their creation.



To cycle among geometries:

1 With the Geom menu enabled, hold down the ALT button on the Navigation panel and then press the Shot Forward or Shot Back button to cycle forward or backwards.

Working with Stereoscopic Projects

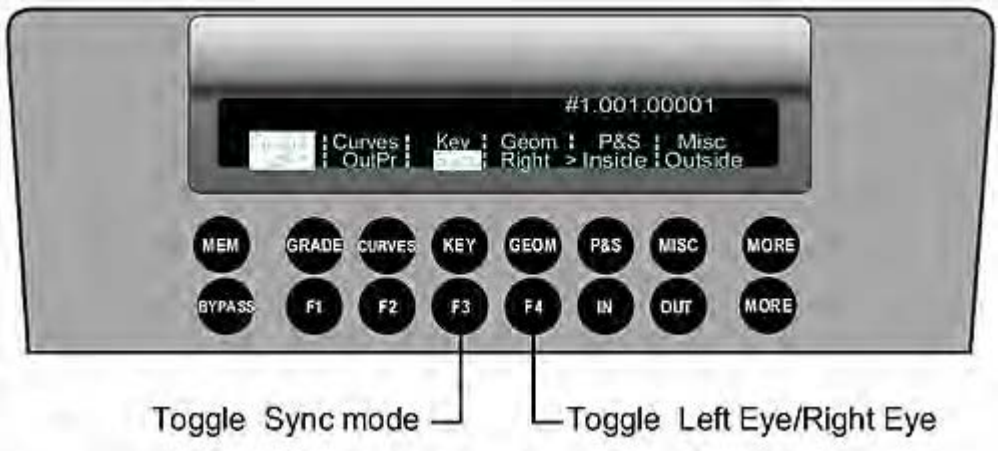
The Autodesk Control Surface panel accommodates common stereoscopic workflow operations. You must be in the Colour Grading or P&S menu to use these controls.

Using the Navigation panel, you can change the Sync mode state. You can also switch between the Left Eye and Right Eye.

When you are in Sync mode, you can use the Function panel to toggle the Convergence tool. See Repositioning Shots (page 2449).

To adjust stereoscopic features in the Navigation panel:

1 In the Navigation panel, enable the Grade or P&S menu.



2 You can use the following controls.

Press:	To:
F3	Toggle Sync mode.
F4	Toggle the assigned eye.

The Sync mode state and the assigned eye are displayed in the digital displays of the Navigation panel and Colour Grading panel. When Sync mode is enabled, the “L+R” appears in the digital display to indicate that grading affects both eyes.

Modifying the Appearance of the Autodesk Control Surface

You can customize the appearance of the control surface by accessing options from the bottom More button on the Navigation panel. Use these options to determine the brightness of:

- Backlighting for all buttons.
- Backlighting for a button being pressed.
- Backlighting for active and available buttons.
- The digital display.

To modify the appearance of the Autodesk control surface:

1 Press the bottom More button on the Navigation panel.



The Low, High, and Screen options appear in the digital display.

2 Press one of the following option buttons and use the ring portion of the trackball to increase or decrease brightness.

Press:	To change the brightness of:
F4	Backlighting for all buttons on the control surface. The change is visible as you modify with the ring.
IN	Backlighting for a button that is being pressed. The change is visible only when you press a button after having used the ring.
OUT	The digital display. The change is visible as you modify with the ring.

ACS Hotkeys

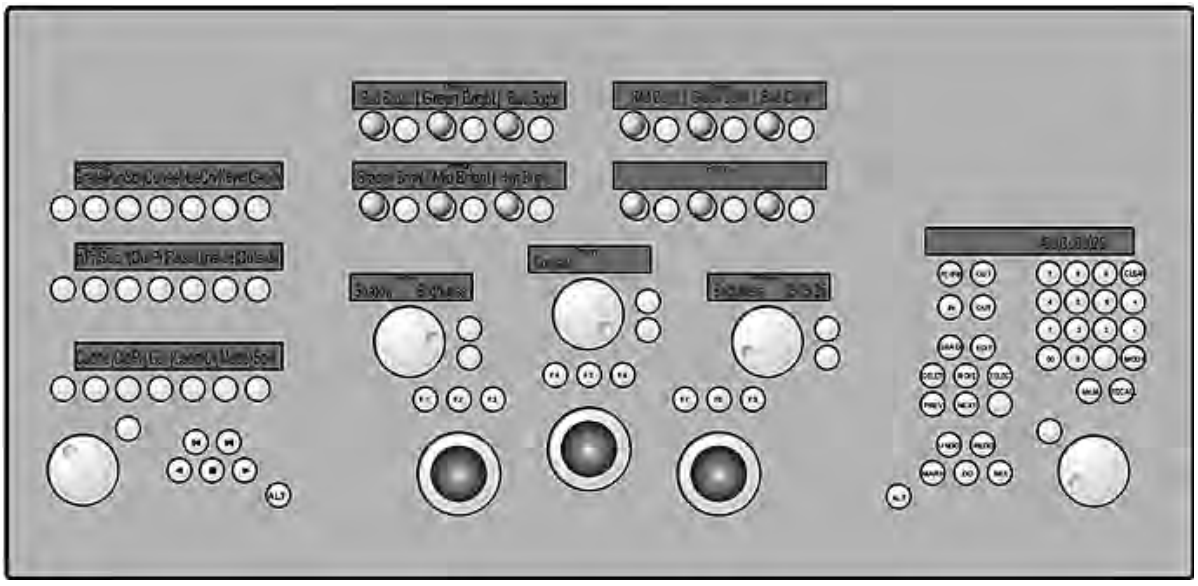
Use these hotkeys on the Navigation panel.

Press:	To:
	Enable the cache.
	Bypass all menu parameters.
	Reset the zoom and pan.
	Add keyframes to all animation channels in the current frame.
	Save the current grade file and cut.
	Temporarily apply the grade from the previous shot to the current shot.
	Temporarily apply the grade from the next shot to the current shot.
	Cycle backward among the geometries (this is only applicable when the geometry menu is enabled).
	Cycle forward among the geometries.

The Tangent CP100 Control Surface

The Tangent CP100 control surface is designed to improve interactivity when colour grading film and video footage. You can accomplish many of the tasks you do in the Lustre user interface using the Tangent CP100 control surface.

The Tangent CP100 control surface is only supported on the Master Station.



You can do the following tasks with the control surface:

- Customize the view
- Toggle play mode
- Cache memory
- Navigate the Storyboard
- Undo and redo operations
- Copy grades from one shot to another
- Create and modify keyframes
- Reposition shots
- Perform primary colour grading
- Modify Hue curves
- Perform secondary colour grading

TIP While you learn how to do each of these tasks, show the user interface so that you can view the results in the menus.

Most buttons and dials on the control surface are labelled. For information on connecting the Tangent CP100 control surface, refer to the *Autodesk Lustre Software Installation Guide* for your platform.

To operate the control surface:

1. Do any of the following:
   - Press a button to select the displayed option.
   - Turn a dial to increase or decrease the value of the displayed channel or parameter. Turn clockwise to increase the value. Turn counter-clockwise to decrease it.
   - Use the Player controls to navigate the Storyboard.
   - Use the trackballs to modify the displayed parameter.

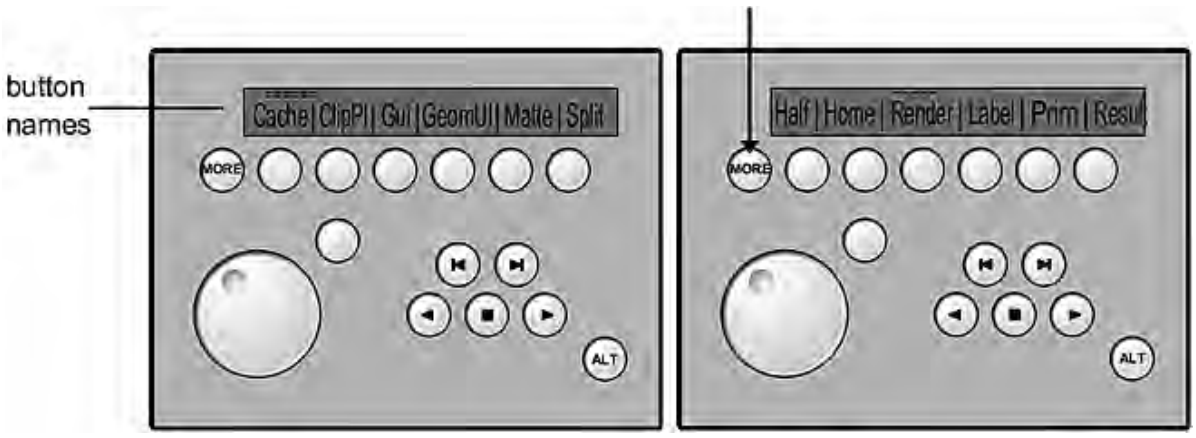
Customizing the View

Use the controls on the lower right of the control surface to change the view as you work on your grades. For example, enable Half to view a half-resolution version of your shot in the Player.

You can select any of the following options.

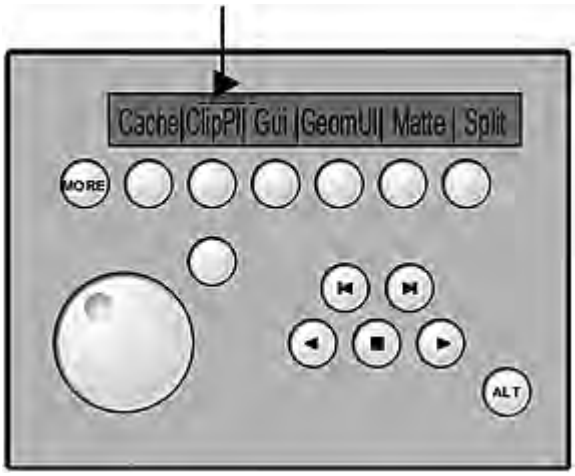
Control Surface	Description	Hotkey or UI Option
GUI	Toggles the full-screen Player.	`Enter`
Geom UI	Shows or hides a geometry for the current secondary.	Show button in Secondaries menu
Matte	Switches between secondary view and result view.	`F11` and `F12`
Split	Switches on and off split view in the Player. When you enable Split, you can press the PERFM button to enable multi-view and view up to 16 shots, or press the CUE button to switch between horizontal and vertical split view. These buttons are located on the upper right of the control surface.	`F5`, `F3`, and `F4`
Half	Switches between the full-resolution and half-resolution (proxy) version of your shot in the Player. If you have not rendered a proxy, an X appears in the Player.	`F9`
Zoom/Home	Switches between the 1x and 2x zoom.	`F10`
Render	Switches between a rendered version of your shot and output view. If you have not rendered the shot, an X appears in the Player.	`P`
Label	Toggles the shot name text on the image.	`W`
PRIM	Toggles the display of your shot. Displays your shot as it appears after modifications are made from the Grading or Curves menu, or as it currently appears (with the entire colour grade applied). You can use this option only after you have added secondaries.	`I`
Result	Displays the shot as it currently appears—with the grade applied.	`O`

NOTE Press the MORE button to view all the available options.



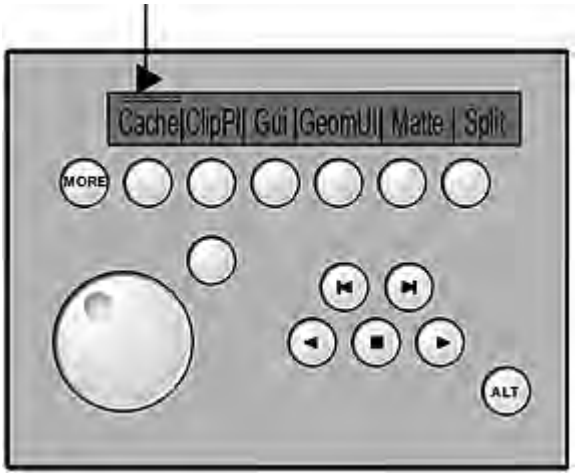
Toggling Play Mode

You can loop play a shot or the entire Storyboard using the ClipPl button. This button corresponds to the Play Mode button in the user interface.



Caching Memory

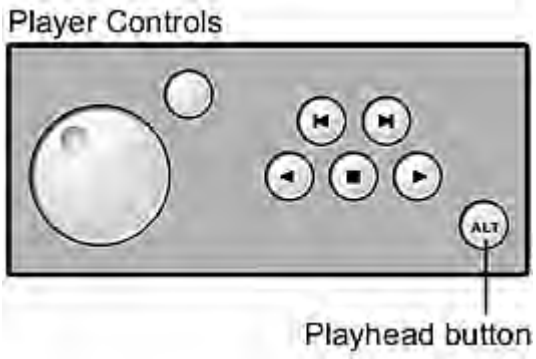
When a shot does not play back in real time, you can cache the frames into memory using the Cache button or by pressing M.



Navigating the Storyboard

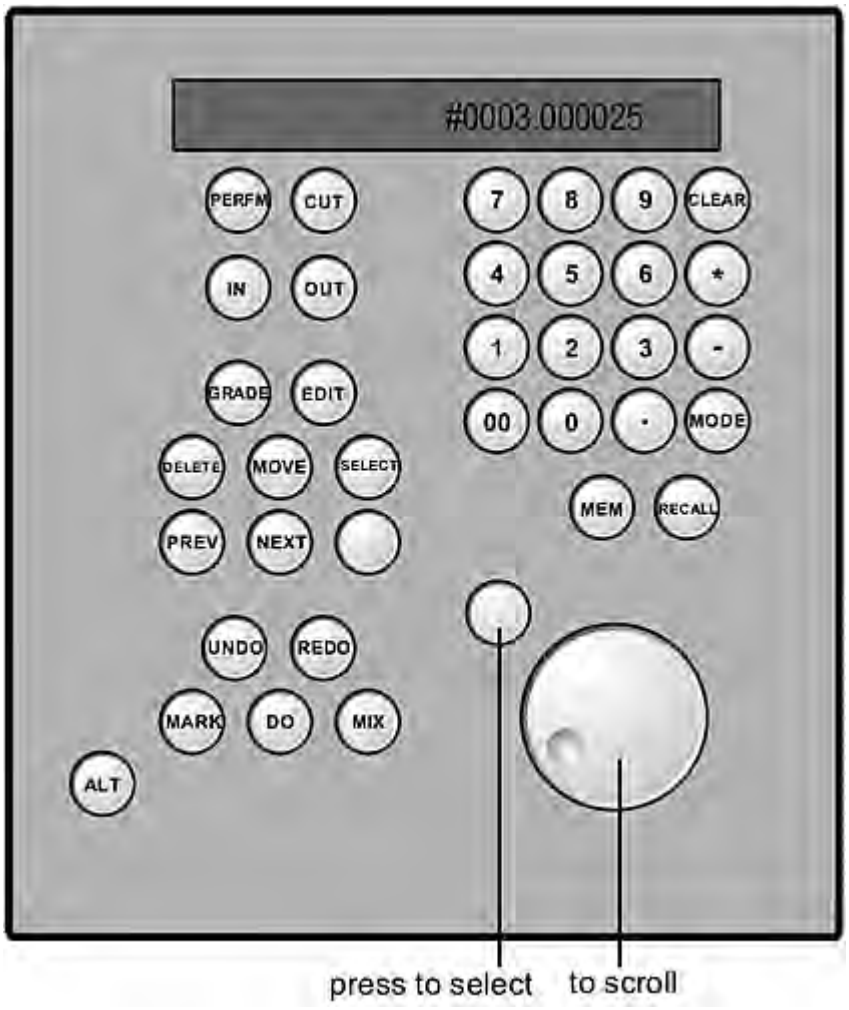
Use the Player controls on the lower right and the scrolling controls on the left to navigate the Storyboard.

Use the left dial (jog knob) and transport controls to play and scrub the Storyboard. You can also use the Alt button to switch between shots assigned to Playhead A and Playhead B.



Use the right dial to scroll through the shots in the Storyboard. When you scroll, a yellow arrow in the Storyboard marks your position. You can then select the shot by pressing the button next to the dial.

NOTE If the shot is outlined in blue, you are in copy mode. See Copying Grades (page 2492).



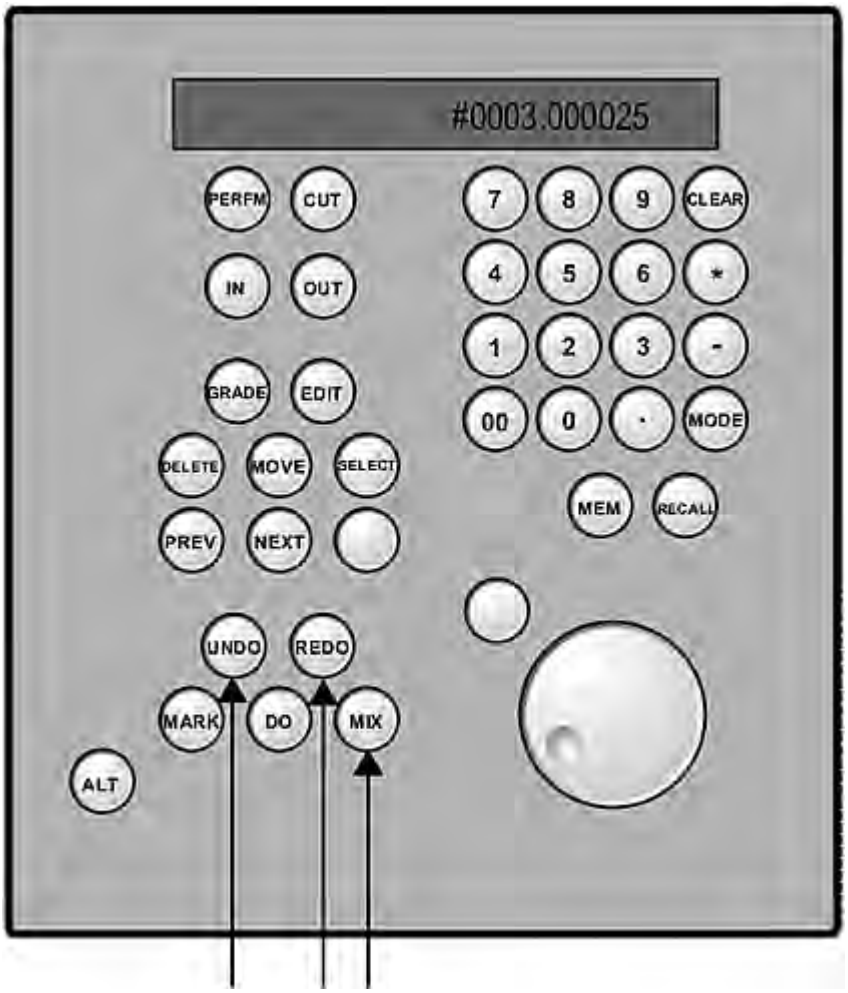
You can show the Storyboard all the time or only when you scroll. To show the Storyboard all the time, press `Tab` on the keyboard. To show the Storyboard only when you scroll, first hide it by pressing `Tab`, and then scroll the Storyboard. In this mode, you hide the Storyboard again by pressing the button next to the scroll dial.

Undo List

The undo list works differently on the control surface than it does in the user interface. Use the buttons on the lower right of the control surface to work with the undo list.

Saving to the Undo List

When you use the control surface, you must save your changes to the undo list. Undo lists are saved on a shot-by-shot basis. To save a change to the undo list, press Do. You can then press Undo and Redo to go through the undo list.



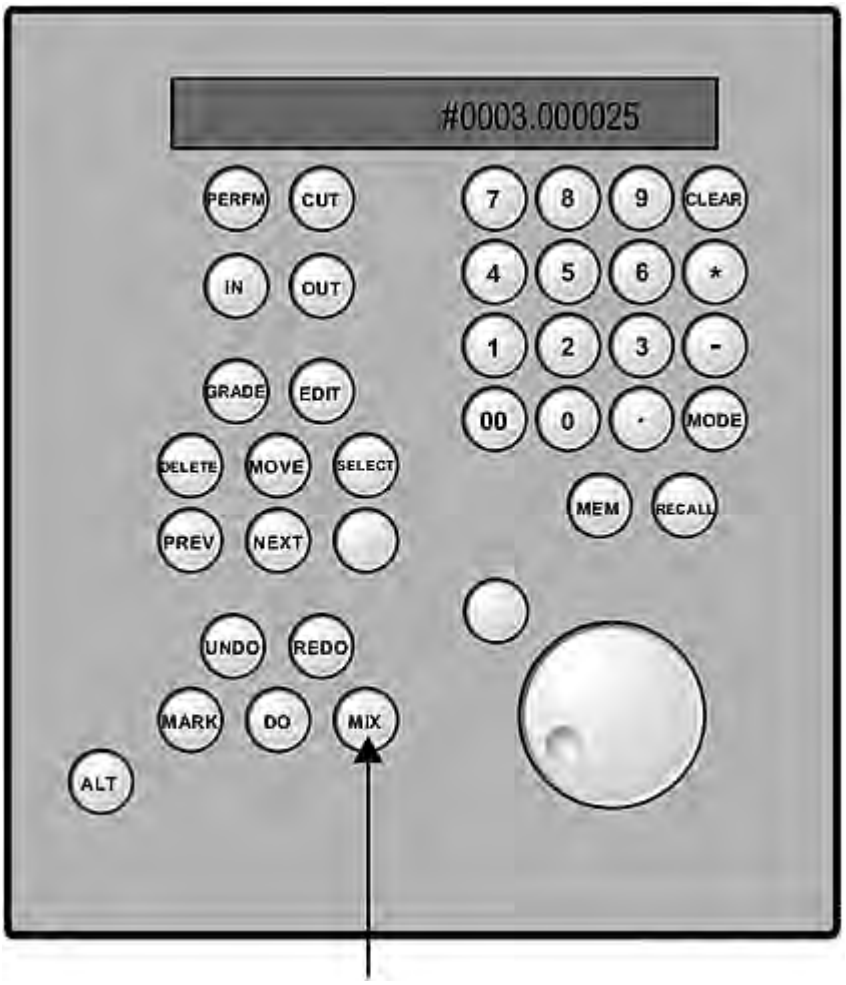
Reverting to Previous Grades

Use the Alt button on the lower right to toggle between the grade that was applied to a shot when you first selected it in the Storyboard and the current grade.

TIP To save the reverted stage to the undo list, press Do.

Comparing Grades

You can compare the current image with the grade you last committed to the undo list. To toggle between these settings, use the Mix button.

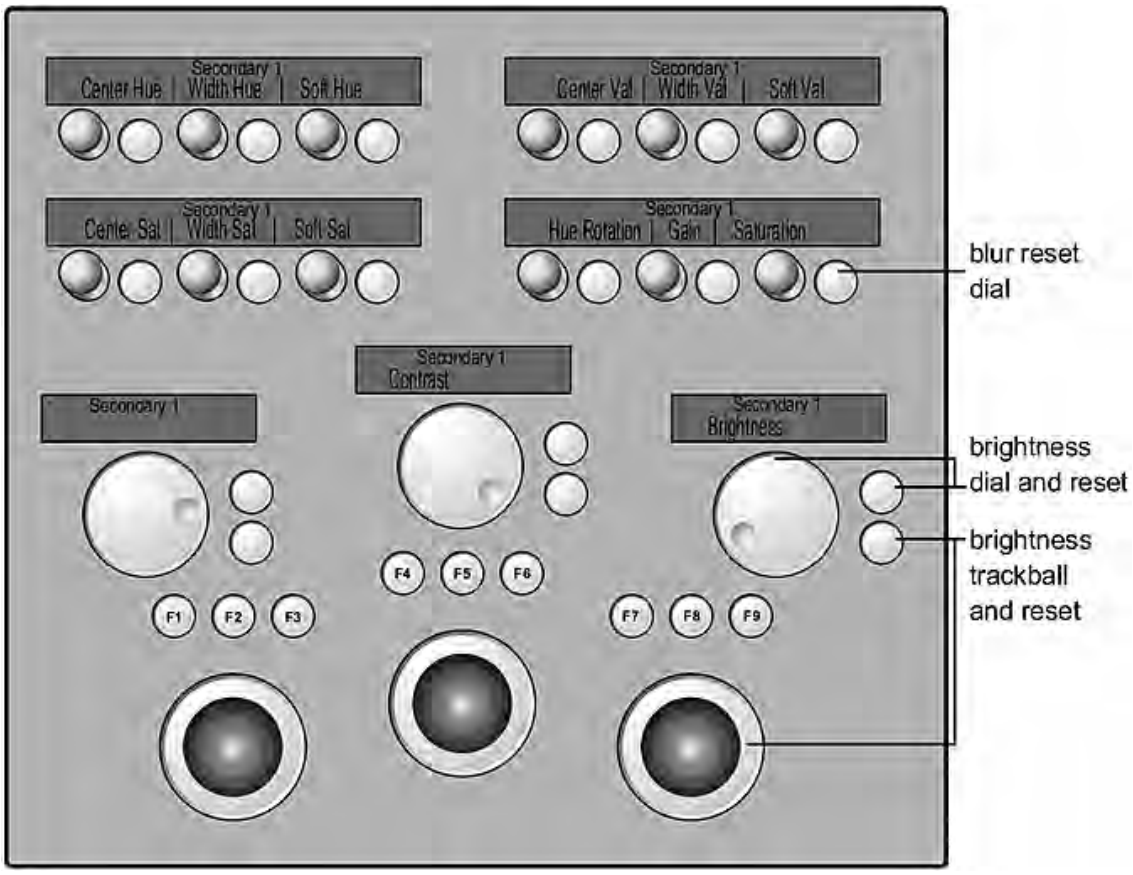


Resetting Parameters

On a shot-by-shot basis, you can reset the parameters set in some menus. You can also reset a single parameter.

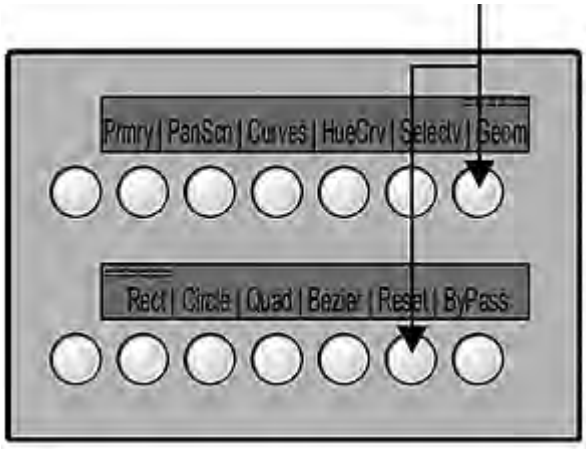
To reset a parameter:

1 Press the Reset button associated with the dial or trackball.



To reset a menu:

1 Select the menu and then enable Reset. For example, select Geom, and then press Reset to clear all secondaries.



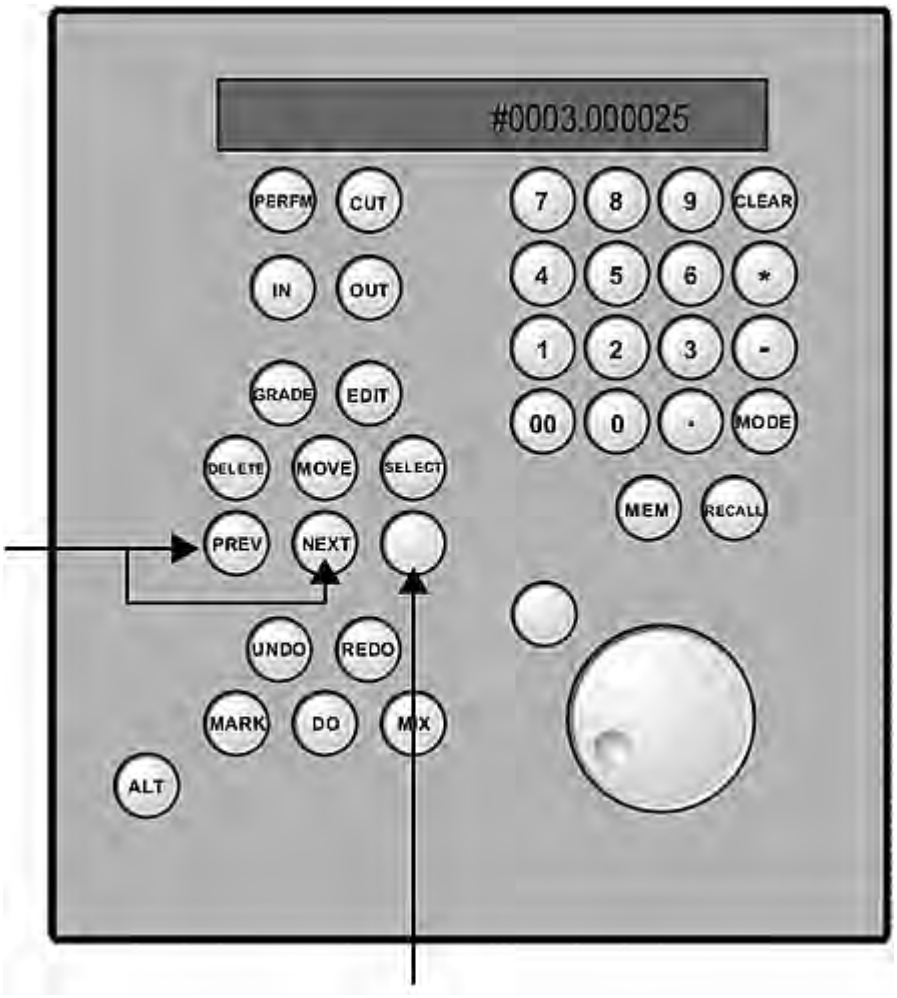
Copying Grades

You can temporarily or permanently copy a grade from another shot in the Storyboard to the current shot.

To copy grades:

1. Do one of the following:
   - Press Prev or Next to temporarily apply the grade from the previous or next shot to the current shot (outlined in red or magenta). Press Prev or Next multiple times to move through the shots in the Storyboard. The selected shot is outlined in blue.
   - Press Offset and then scroll through the shots. As you scroll, the grades are temporarily applied to the current shot (outlined in red or magenta). The selected shot is outlined in blue.

     TIP Press the Offset button again to switch to scroll mode.



2. Press Do to permanently copy the grade from the shot outlined in blue to the current shot.

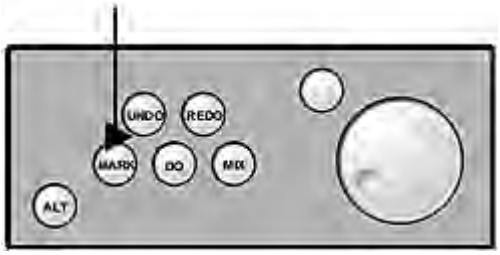
Adding and Moving Keyframes

When you animate parameters, you can set keyframes using the Mark button. You can also adjust the timing of the first and last keyframes in the shot.

To add a keyframe:

1. Go to the frame where you want to add the keyframe.

2 Press Mark.



To adjust the timing of an existing keyframe:

1 Do any of the following:
- Press In and then use the right dial to move the first keyframe.
- Press Out and then use the right dial to move the last keyframe.

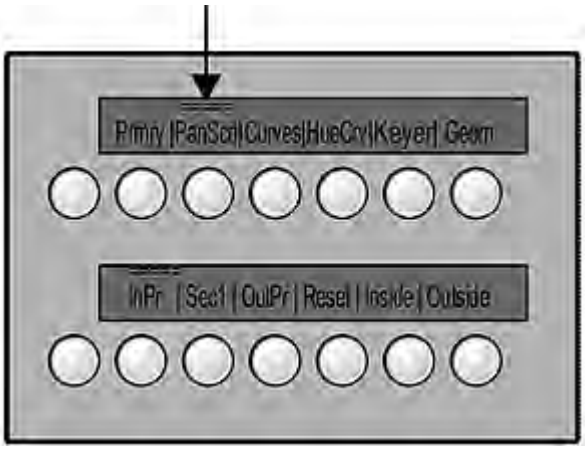


Repositioning Shots

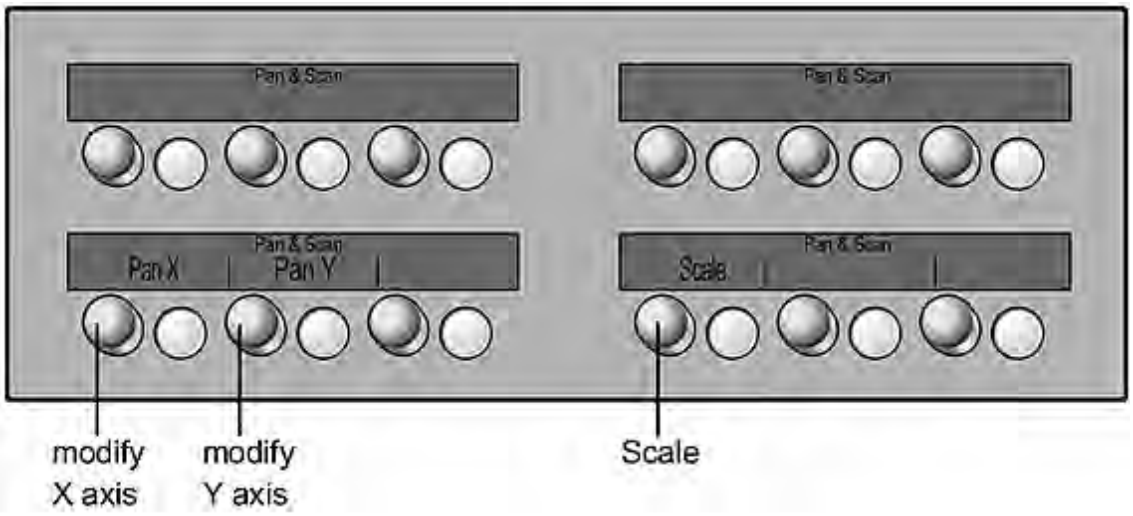
You can reposition a shot by scaling it, or by panning it vertically and horizontally.

To reposition a shot:

1 Enable PanScn.



2 Use the following controls to reposition the shot.



Primary Colour Grading

You can use the dials and trackballs in the centre of the control surface to perform primary colour grading. Primary colour grading can be done from the Grade menu.

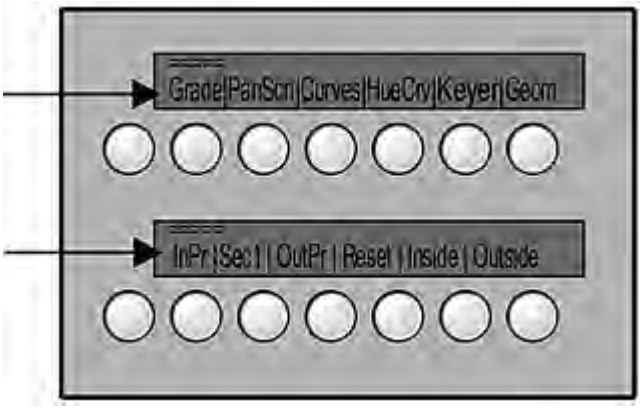
Logarithmic Mode: Modifying Brightness and Contrast

You can modify brightness and contrast in several ways from the Grade menu. You can modify:

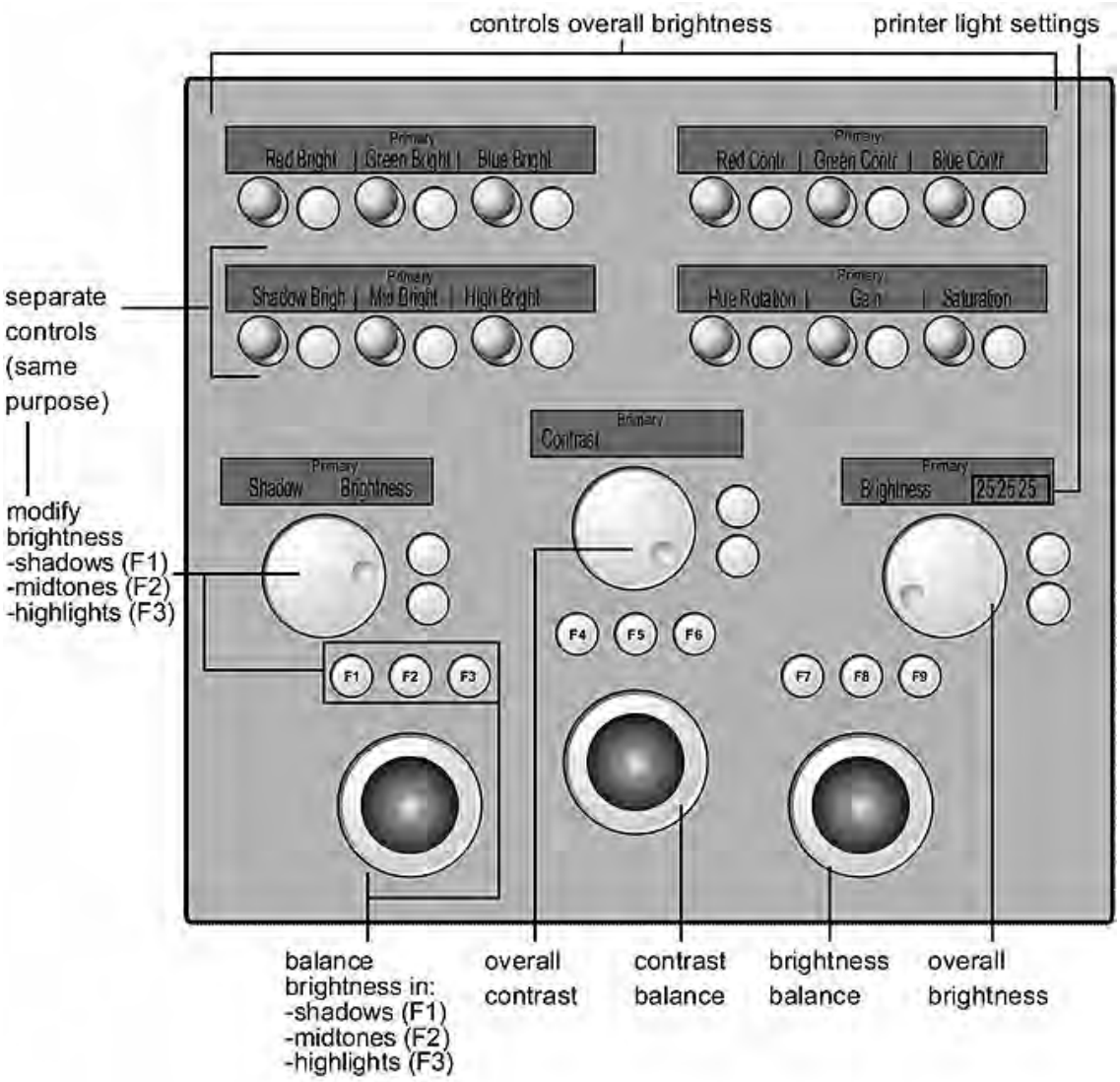
- Overall brightness and contrast of the red, green, and blue channels together.
- Overall brightness and contrast of the red, green, and blue channels separately.
- Brightness balance and contrast balance.
- Brightness in the shadows, midtones, and highlights.

To modify brightness and contrast:

1. Enable Grade and then do one of the following:
   - Enable InPr to modify the input primary values.
   - Enable OutPr to modify the output primary values.



2 Use the following controls to perform primary colour grading.



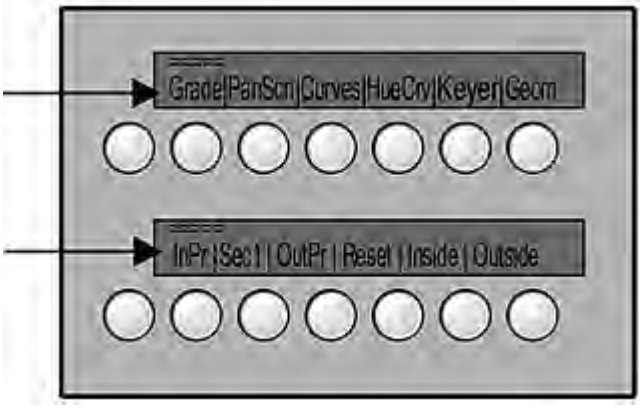
Linear Mode: Modifying Gamma, Gain, and Lift

You can modify gamma, gain, and lift in several ways from the Grading menu. You can modify:

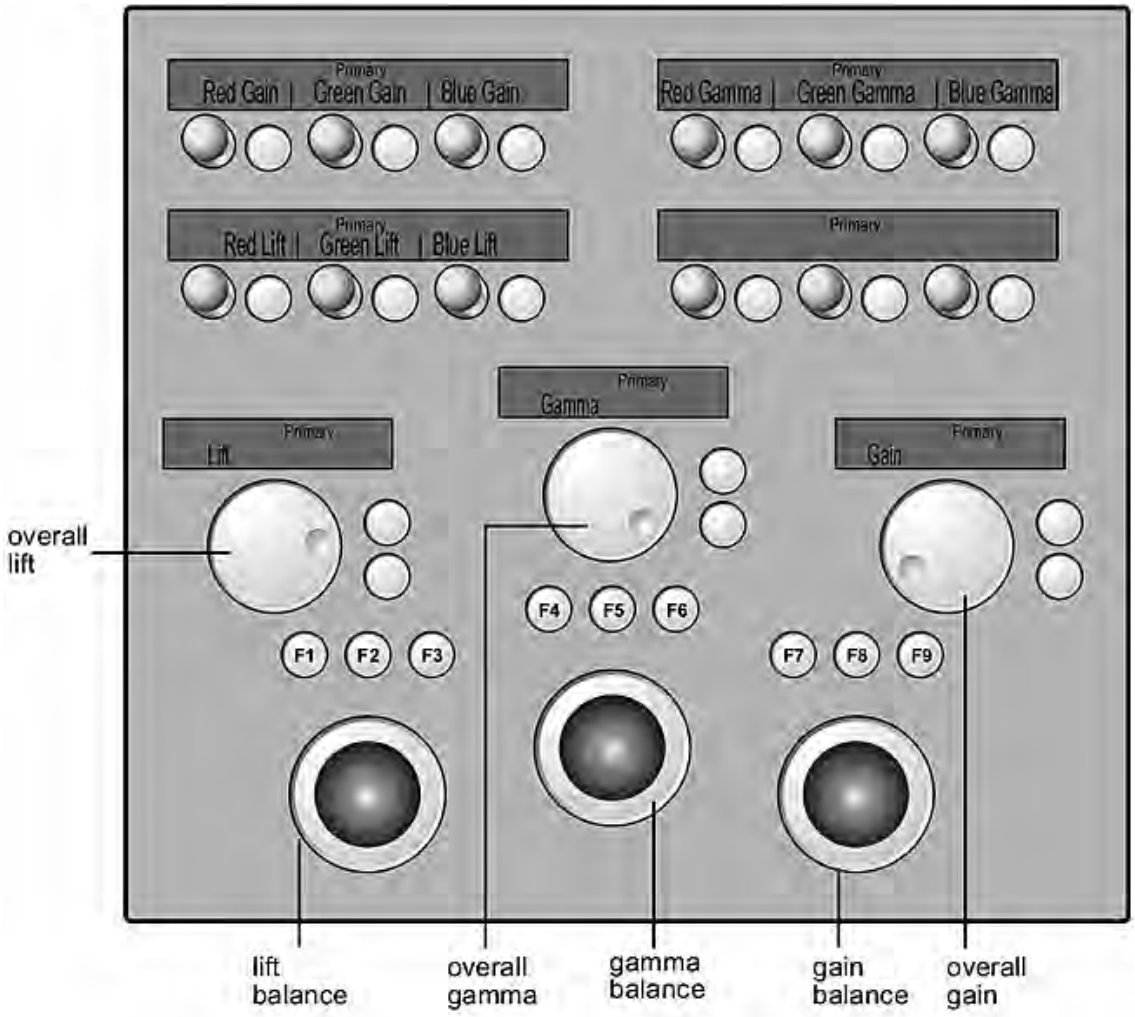
- Overall gamma, gain, and lift settings of the red, green, and blue channels together.
- Overall gamma, gain, and lift settings of the red, green, and blue channels separately.
- Gamma, gain, and lift balance settings.

To modify gamma, gain, and lift:

1 Enable Grade and then do one of the following:
   - Enable InPr to modify the input primary values.
   - Enable OutPr to modify the output primary values.



2 Use the following controls to perform primary colour grading.



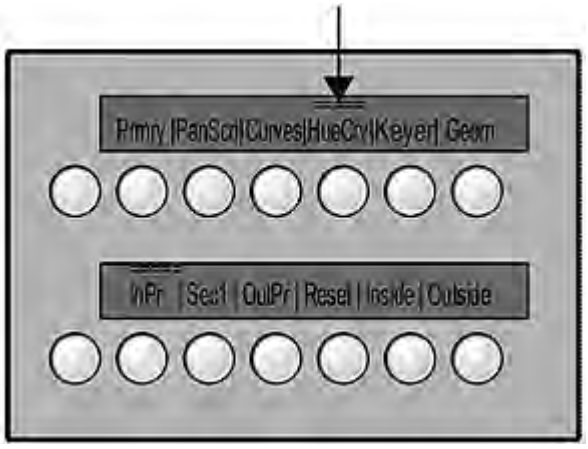
Modifying the Hue, Saturation, and Lightness Curves

You can modify the hue, saturation, and lightness curves that appear in the Curves menu. You can modify the entire curve uniformly or modify individual vertices for RGBCMY channels (red, green, blue, cyan,

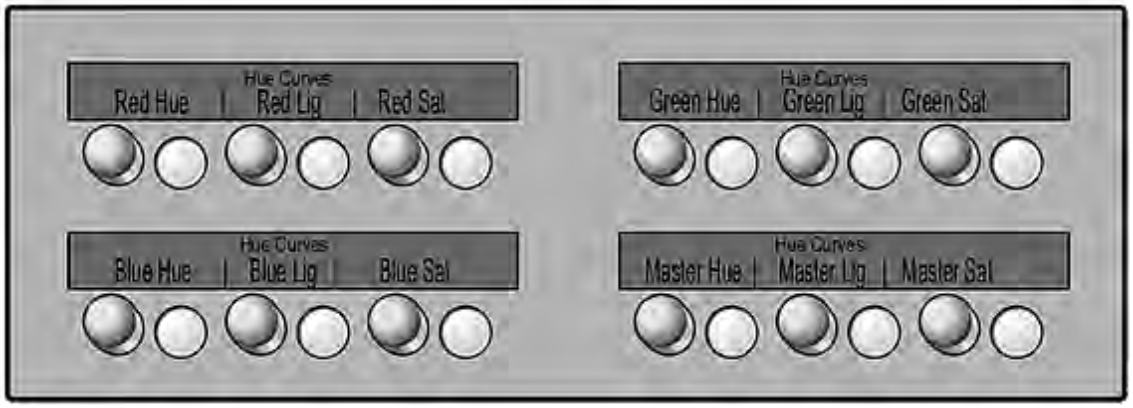
magenta, and yellow). You must toggle between controls that modify the position of the red, green, and blue vertices and those that modify the position of the cyan, magenta, and yellow vertices.

To modify hue, saturation, and lightness curves:

1 Enable HueCrv to access controls for the red, green, and blue vertices.



2 Use the following controls to modify the hue, saturation, and lightness curve at each displayed vertex.



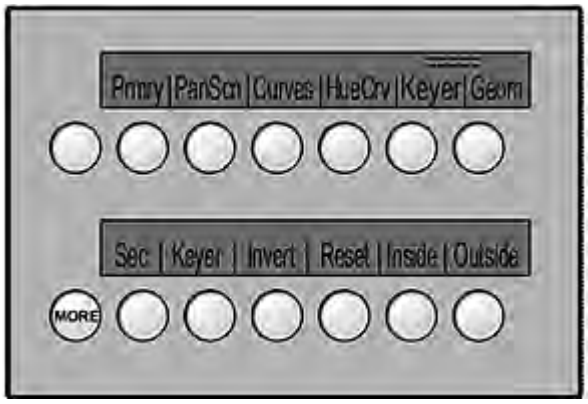
3 Press HueCrv again to access controls for the cyan, magenta, and yellow vertices, and then modify the curves.

Secondary Colour Grading

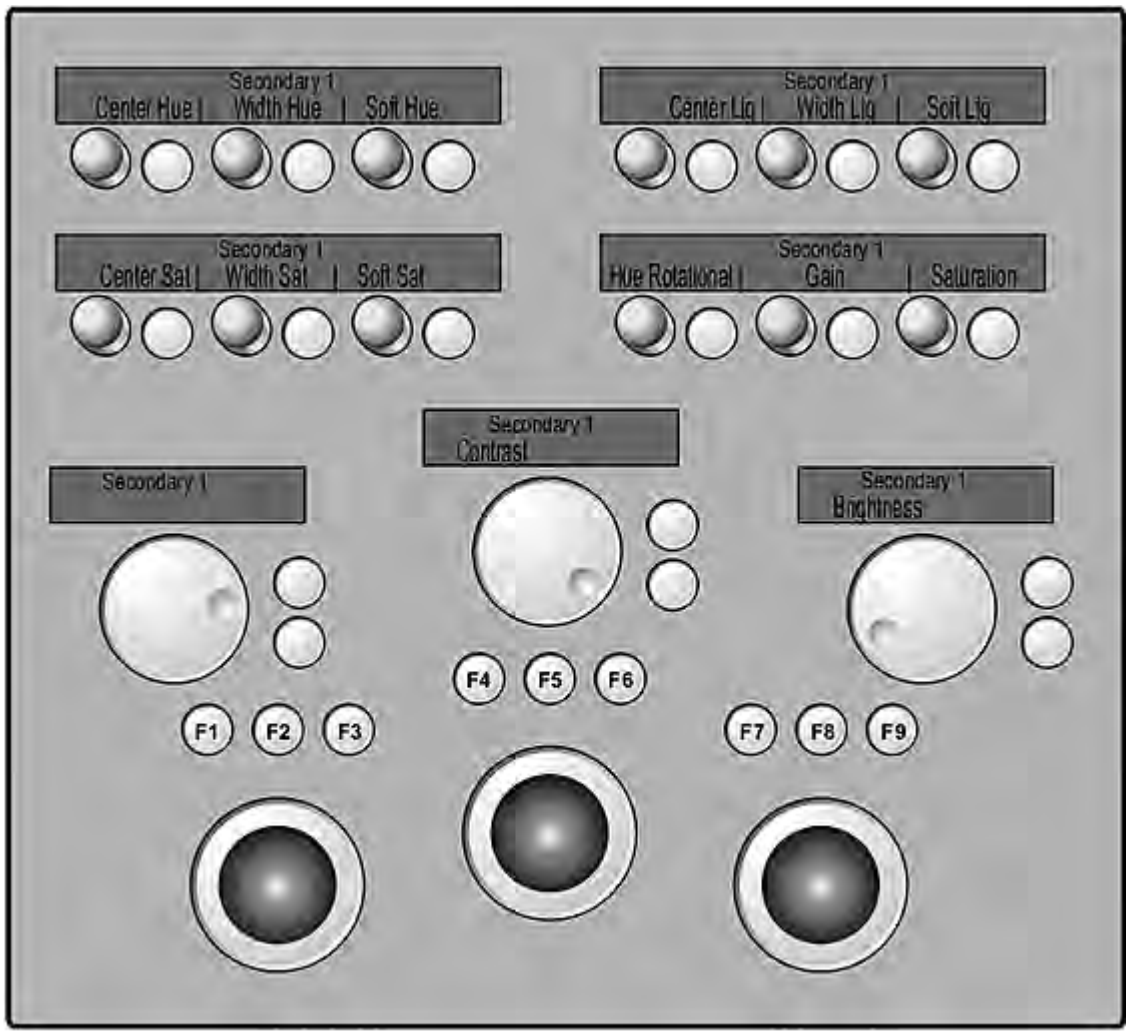
You can colour grade up to eight secondaries from the control surface.

To access a secondary:

1 Enable Keyer.



The secondary controls for the first secondary appear. Once you enable and key a secondary, you can modify hue, gain, and saturation. You can also modify the brightness and contrast of the current secondary.



2 On the right side of the control surface, press Recall and then press a number between one and eight to go to that secondary. You can also press Selectv to toggle through the eight secondaries.

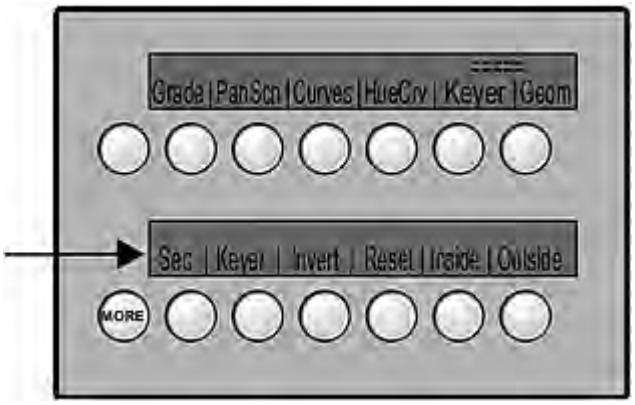


Applying Secondaries to the Shot

Use the second row of buttons on the left side of the control surface to set basic parameters for secondaries.

Press:	To:
Sec	Switch on and off the current secondary.
Keyer	Switch on and off the Keyer for the current secondary.
Invert	Invert the key for the current secondary.
Reset	Reset the key for the current secondary.

Enable the Sec button to apply the displayed secondary to the shot.



Creating Keys

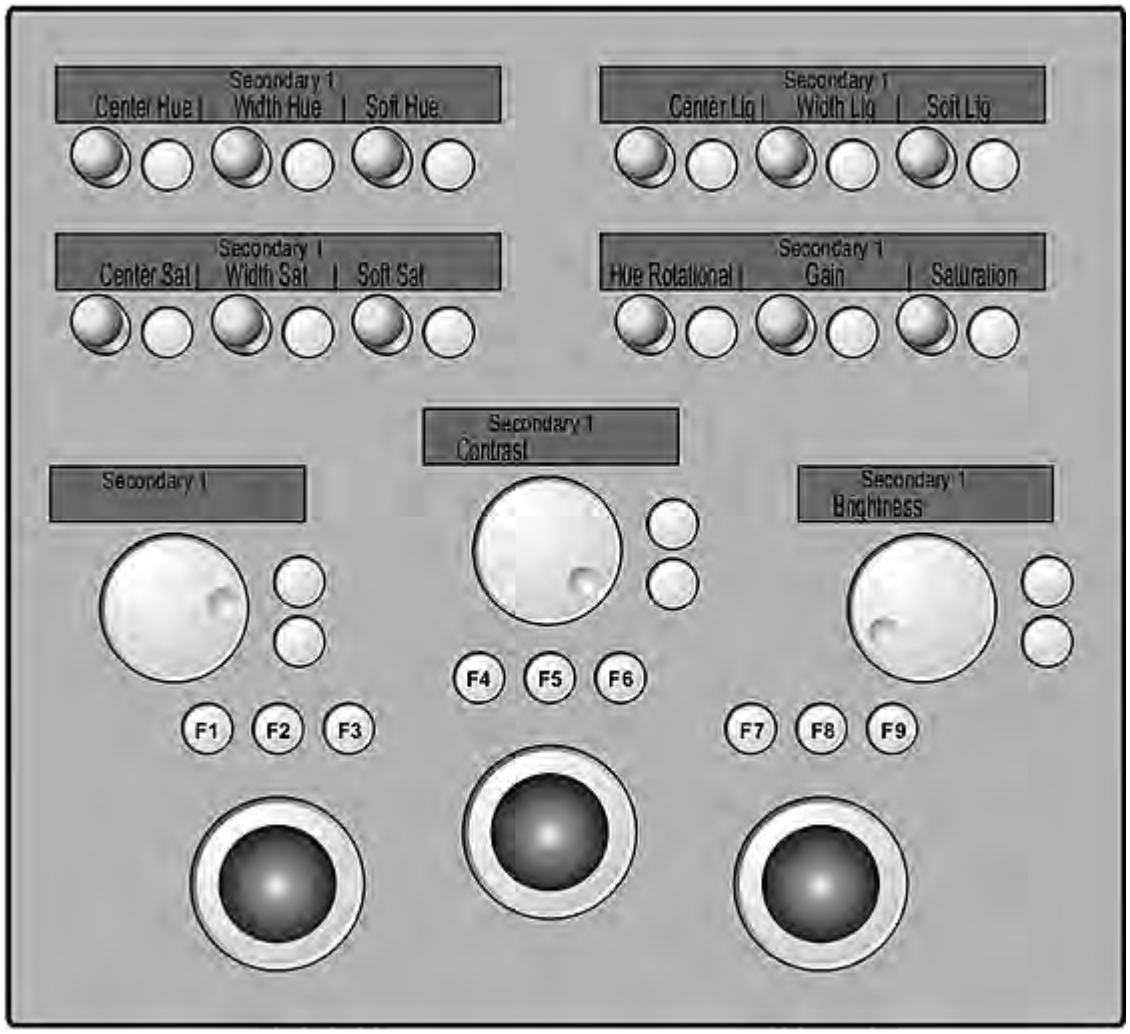
Enable the Keyer button and then use the controls in the centre of the control surface to create a key.

To create a key:

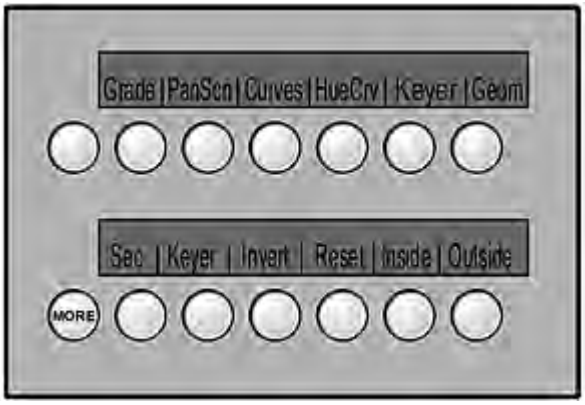
1. Enable Keyer in the upper control surface menu, and then enable Keyer in the lower menu.
2. To sample initial softness and tolerance, press F1 on the control surface and then use the left trackball to position the colour picker over the image. Press F1 again to sample the image.

   NOTE Sampling from the control surface works slightly differently than sampling from the Lustre user interface. When you sample from the control surface, the value and saturation channels are set to the maximum softness. When you sample from the user interface, the value and saturation channels are not set to maximum values.

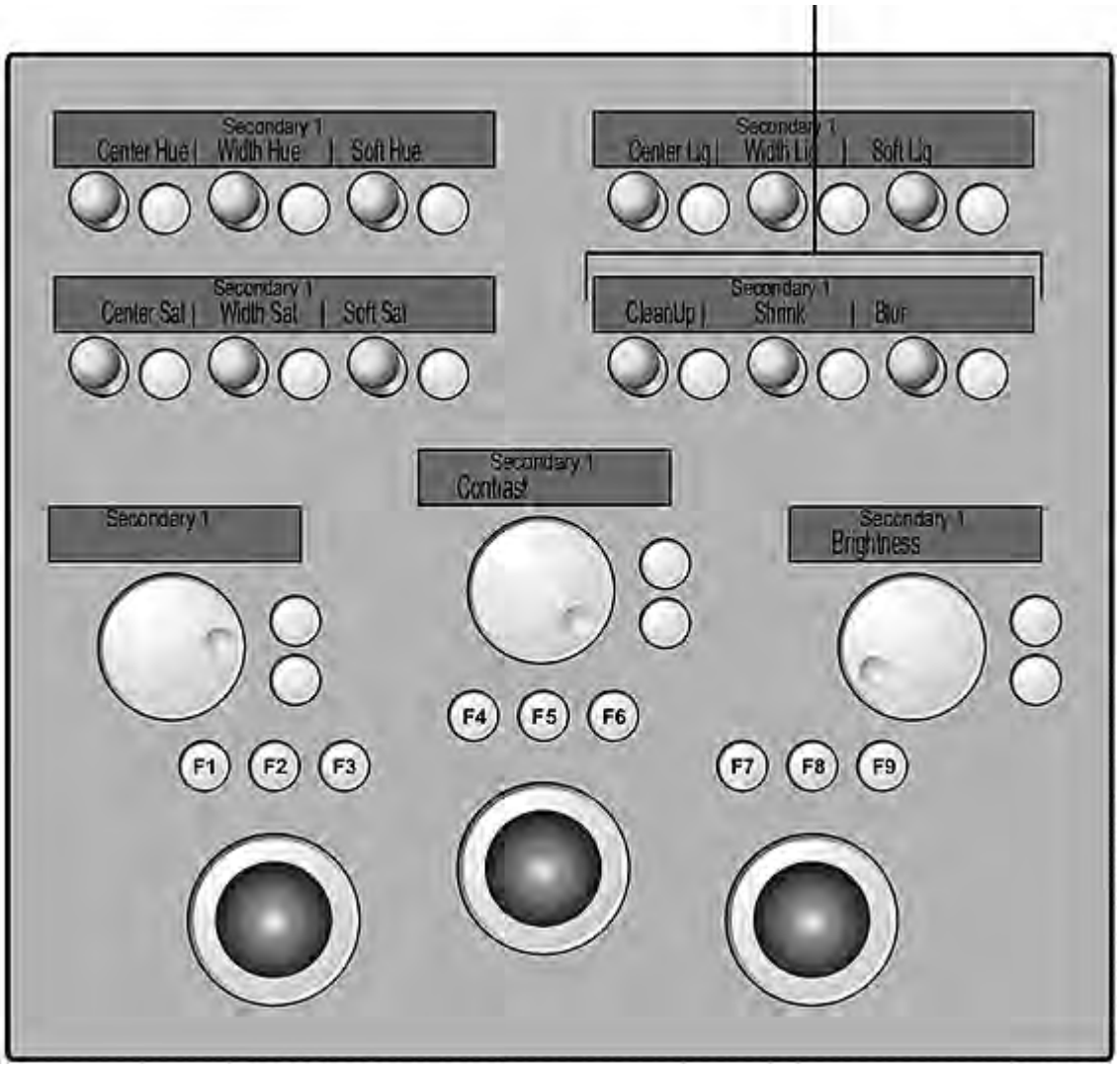
3. Set the hue, brightness/gain, and saturation levels for the key. Each value is modified according to the original sample—you can adjust the centre tolerance value, as well as the softness and tolerance ranges.



4. Press More (lower left) to shrink, blur, or clean up the edges of the key.



The Cleanup, Shrink, and Blur controls appear.

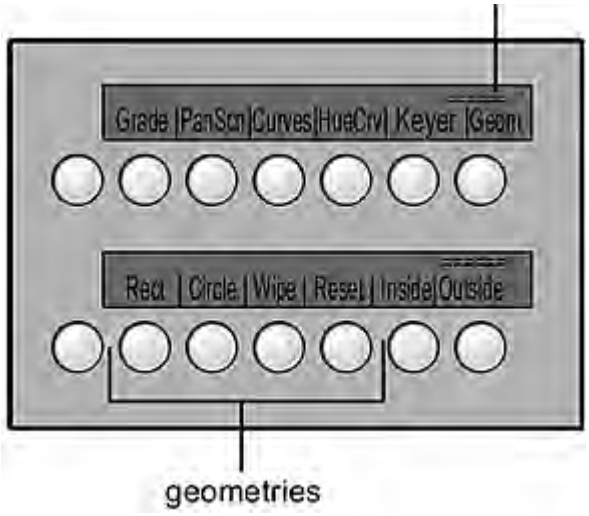


Creating Geometries

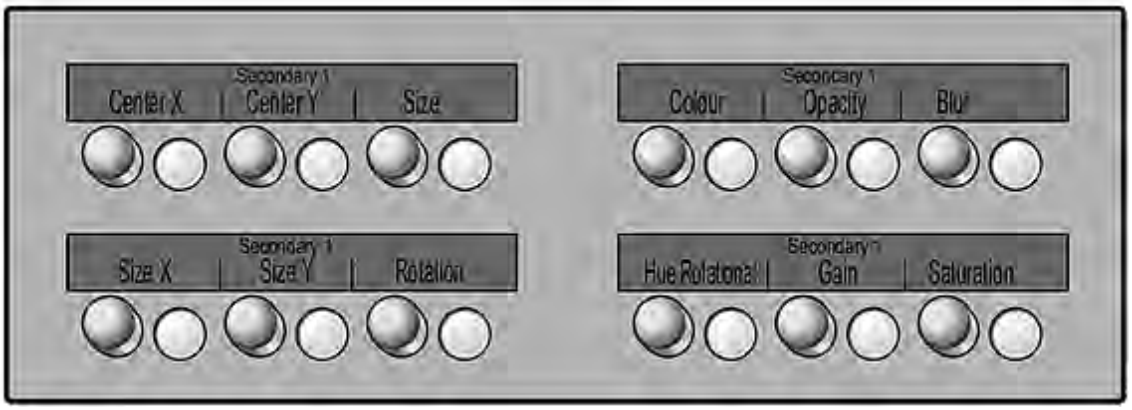
You can create and apply basic and free-form geometries to each secondary.

To create a geometry:

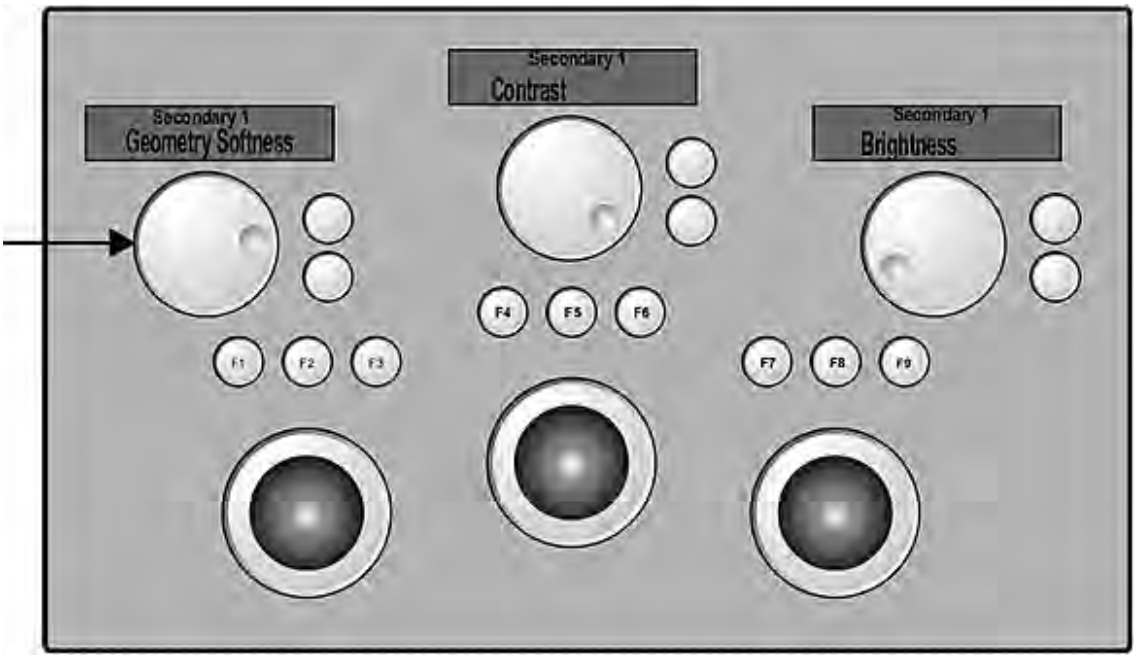
1. Enable Geom and then select the shape from the second row.



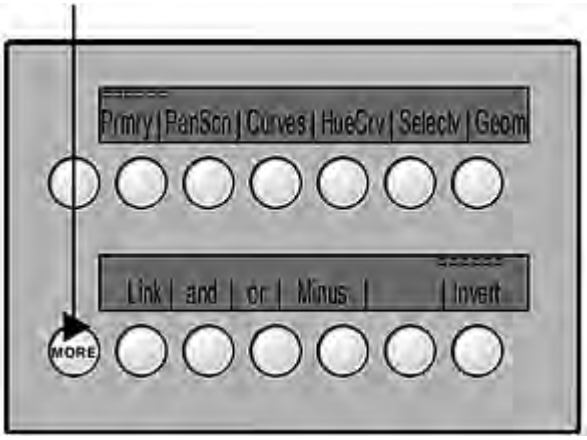
2. Use the controls in the centre of the control surface to modify the shape of the geometry, as well as its colour, opacity, and position on the image. You can also apply a blur.



3. To set softness, use the Geometry Softness dial.



4 To apply logical operations to the geometry, or invert the geometry, press the More button associated with the second row on the left.



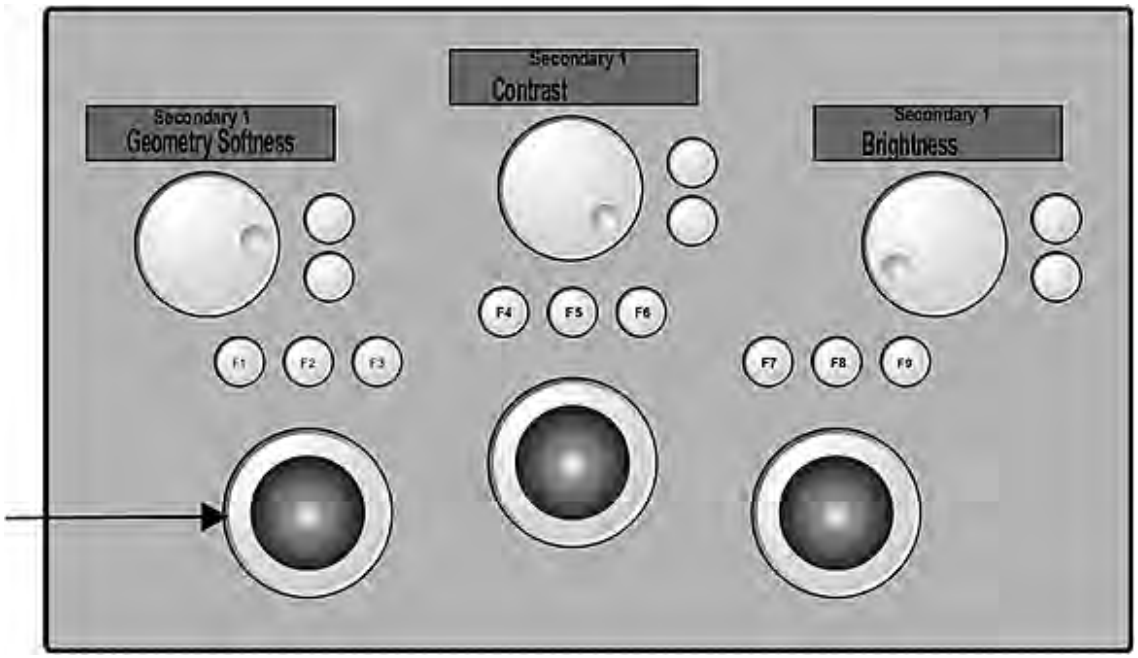
Moving Geometries

You can move geometries from the control surface.

To move a geometry:

1 To select a geometry, press F1 on the control surface and then use the left trackball to position the cursor over the geometry. Press F1 again to select it. Alternatively, press F2 and F3 to move back and forth through the geometries (listed in the Axes list).

2 Use the left trackball to move the geometry.


Secondary 1
Geometry Softness
Secondary 1
Contrast
Secondary 1
Brightness
F1
F2
F3
F4
F5
F6
F7
F8
F9

Index

.R3D in Final Cut Pro 90

10-bit
  recording without rendering 2291
2D Histogram 1223
2D Histogram menu settings 1223
2D Histogram node 1223
2D luma histogram, in Colour Warper 970
2D Transform 1224
  2D Transform menu settings 1226
2D Transform node 1224
2D vectorscope, in Colour Warper 966
3D Blur 1231
3D Blur menu settings 1231
3D Blur node 1231
3D cameras, in Action
  importing 669
3D models
  3ds Max import, in Action 531
  Alembic import, in Action 531
  deforming models 544
  FBX, link 537
  importing, in Action 531
  modifying, in Action 538
  particle generation, in Action 751
  projected textures 627
  specular highlights, in Action 543
  transparency, in Action 542
3D path
  creating, in Action 634
  look-at, in Action 638
3D text
  adding presets 550
  bevelling, in Action 571
  creating, in Action 550
3D tracker
  exporting FBX 646
3D vectorscope, in Colour Warper 966
3ds Max
  cameras, in Action 646
  models, importing 531

A

A-mode sort 1991
aaf 94, 178
About Sparks 1427
Accessing the Axis Keyer 338
Accessing the Wipe Schematic 318
Action
  See also 3D models, 3D text, 3D tracker, media, source nodes, surfaces
  adaptive degradation options 423
  analyzer 674
  Atomize 633
  axes 451
  axis planes 453
  bicubic surfaces 465
  bilinear surfaces 465
  blending curves per surface 470
  blending surfaces 467
  blooming 523
  blurring clips 445
  camera clipping planes 645, 666
  camera, stereo 658
  cameras, adding 640
  cast shadows 506
  creating branches in schematic 420
  cropping clips 446
  deformation meshes, adding 544
  deforming models 544
  diffuse mapping 615
  displacement mapping 595
  drop shadows 478
  emission mapping 611
  flipping normals 503
  functions, using with particles 775
  garbage masks as geometry 549
  grouping media 482
  image-based lighting 527
  in Batch or Batch FX 371
  len flares 510
  lights 495
  Magnet Curve Editor 475
  material node 587
  Media list 441
  menus, accessing 415
  models, importing 531
  Modular Keyer, accessing 450
  motion blur 653
  multiversion clips, loading 407
  node bin 409
  normal mapping 604
  OpenEXR support 407
  outputs 435
  parallax mapping 600
  particle presets 717
  particles 715
  Perspective Grid 491

Photoshop files, importing 434
position mapping 621
Priority Editor 480
processing options 423
projector blend modes 631
rays 521
reflection mapping 619
replacing front or matte clips 701
schematic, using 413
selective lighting 497
setup options 423
setups, loading 433
shader node 588
slipping clips 448
source nodes 699
specular highlights 501, 543
specular mapping 608
stabilizing 455
stereo camera 658
stereo footage 485
stereo objects 486
stereoscopy 485
Substance textures 579
surfaces 457
textures 578
tracking 455
UV mapping 624
wireframe 502
Z-buffer for shadows 480
Z-buffer setting 482
Adding Axis Timeline FX 337
Adding Axis Timeline Transition 326
Adding Colour Correct Timeline FX 335
Adding Compression Timeline FX 339
Adding Delay Send Timeline FX 341
Adding Dissolve Timeline Transition 316
Adding EQ Timeline FX 339
Adding Fade Timeline Transition 327
Adding Gain Timeline FX 340
Adding Gate Timeline FX 338
Adding Matte Containers 338
Adding Modulation Send Timeline FX 341
Adding Resize Timeline FX 334
Adding Reverb Send Timeline FX 340
Adding Sparks Timeline FX 336, 337
Adding Text Timeline FX 334
Adding Timeline FX to Gaps 331
Adding Timeline transitions 316
Adding Timewarp Timeline FX 333
Adding Wipe Timeline FX 337
Adding Wipe Timeline Transitions 317
Adjusitng Preset SMPTE and Custom Wipes 318
adjusting aspect ratio, viewing aspect ratio, aspect ratio, broadcast monitor 256
Adjustment Segment
in Batch FX 346
adobe 179
advanced gradient
defined 891
pickers, using with 919
removing, in Tracer 920
softness borders 916
Tracer 916
ALE files
assembling 1892
auto-conforming 2273
copying into Lustre 1897
marking shots for later capture 1905
Alembic
models, importing 531
aligned edit 270
Analyse tools
using to view colour distributions 2049
analysing
colours 2047
grain profiles 2008
analyzer
accessing in Action 678
converting results 692
ground plane, defining 689
object tracking 693
real world measures 689
animating
colour grades 2053
dissolves 2188
garbage masks 906
garbage masks with tracking 911
geometries with the point tracker 2144
geometries with the shape tracker 2144
in Keyer 855
keyframes 1471
Paint strokes 1068
reposition values 2024
shape of geometries with the point tracker 2147
shapes 1096
text 1144
Tracer 923
with the Tangent CP100 2493
animation
Animation Channel Editor 1472
Channel Editor 1472
creating filters 1487
expressions, creating 1496
in Warper 1218
keyframe indicators 1472
linking audio 1488
linking channels with expressions 1498
previewing in Paint 1032
using selection sets 1487
Animation Channel Editor
adding keyframes 1474
display preferences 1485
panning, zooming 1473
using 1472
animation curves
copying,keyframes
copying 1475
flipping channels 1482
mattes, deleting in Colour Warper 982
moving 1479

tangents 1483
timing 1476
animations
adding from outside the Animation menu 2204
adding keyframes 2177
adjusting duration 2203
adjusting timing 2201
breaking tangent handles 2188
editing keyframes 2184
hiding inactive tangents 2190
hot keys 2360
modifying curves 2186
overview 2171
panning and zooming 2174
resetting tangents 2189
reversing keyframe order 2190
selecting channels 2177
undoing and redoing 2206
using the channel hierarchy 2174
workflow 2176
Anisotropic shading, in Action 592
anti-aliasing
in Warper 1222
append edit 270
applying grades 1824
archaic archive 235
archive 228, 229, 237
archive configuration 236
archive selection 230
archive, best practice 240
archive, clip 234
archive, delete 246
archive, DLT 232, 233, 234, 235
archive, erase 246
archive, HTML 241
archive, LTO 232, 233, 234, 235
archive, other locations 240
archive, projects 232
archive, SAIT 232, 233, 234, 235
archive, setups 232
archive, supplementing 240
archive, to restore clips 235
archive, to restore projects 233
archive, TOC 241
archive, troubleshooting 236
archives 227
browsing 241
deleting TOCs 243
Flame Reactor 244
legacy 244
old 244
recovering 243
restoring previous versions 244
archiving 237
archiving folders 229
Array element
contained by LUT1D 1576
contained by LUT3D 1577
contained by Matrix 1580
ARRI 1866
ARRIRAW 1866
ASC_CDL element 1594
ASCII table of contents, recovering 243
Asian character sets 1133
aspect ratio, setting for the output frame 2018
assemble mode
blacking tapes 198
outputting using 198
assembling
EDL files 1898
EDLs, ALE and Cutlist files 1892
assembling in stereoscopic projects 2209
assembling stereoscopic EDL files 2209
Atomize node, in Action 633
audio
capturing a file 2309
generating keyframes 1488
importing and offsetting 1844
importing file 2306
importing from Wiretap 2308
LTC chase option 2313
monitoring options 2311
offsetting the file 2312
patching, in EDLs 171
playing back 2310
playing out with video 2315
preferences 1671
sync options 2312
use explicit timecode 2314
workflow 2306
Audio Mixer 311
Audio Preferences 1671
Auto Edit after Add 1076
Auto Matte 1238, 1239
Auto Matte menu settings 1239
Auto Matte node 1238
Auto Point 1093
Auto Stabilize 1239
Auto Stabilize menu settings 1241
Auto Stablize node 1239, 1241
AutoConfigure element 1604
Autodesk control surface
individual surface descriptions 2422
overview 2420
Autodesk Control Surface
connecting 2421
autograding, shots based on sampled pixels 2073
automated dust removal
overview 1999
performing 2001
AutoPaint 1063
playing recorded strokes 1068
Random mode 1065
recording brush strokes 1066
tracking 1067
User mode 1066
using graphics 1095
Wipe mode 1064
Average 1244, 1245
Average menu settings 1245

Average node 1244
avid 94, 178
axes
adding and deleting from geometries 2109
moving within geometries 2098
viewing in schematic view 2110
axes list
creating a parent axis 2109
deleting an axis 2109
view of geometries and their axes 2109
axes, in Action 451

B

back up system, to 240
back up, to 227
Backburner
caching degrained frames 2014
Backburner Manager, starting 2234
Backburner Monitor
Linux workstation, accessing from 2237
monitoring render tasks 2235
background rendering
overview 2231
backup 227
baking keyframes 1481
banking, 3D path in Action 639
Batch
about 345
node bins 399
snapshots 384
Batch FX
about 345
adding to sequence 346
node bins 399
batch, digitizing 2271
Bezier curves, for polygons 1093
bicubic surfaces, in Action 465
bilinear surfaces, in Action 465
black clip, adding to the Storyboard 1931
black level
adjusting 1847
modifying for input LUTs 2034
black media, adding to the Timeline 1940
blacking VTR tapes 198
Blend & Comp 1245, 1247
Blend & Comp Menu settings 1247
Blend & Comp Node 1245
blending 1020
surfaces, in Action 467
blending curves
per surface, in Action 470
blooming
adding, in Action 523
Blur 1249, 1250
Blur medium 1048
Blur menu settings 1250
Blur node 1249
blurring
edges of keys 2134
geometries 2104
in Colour Warper 982
with Sparks 2157, 2161, 2166
boxes
adjusting for dust removal 2004
adjusting for grain removal 2010
brightness
adjusting printer lights 2061
modifying for low, mid, or high levels 2062
modifying for the entire image 2060
resetting 2063
brush attributes 1040
Direction 1009, 1042, 1045
Jitter 1009, 1041
modes 1044
Opacity 1008, 1040
Pressure 1008, 1041, 1045
Rate 1008, 1040
Roll 1010, 1043
Size 1008, 1040
using reference images 1044
Brush Spacing preference 1045
brushes
See also special effects media
attributes, setting in Paint node 1007
custom 1114
selecting 1035
setups, saving and loading 1113
strokes, applying 1013
types 1035, 1076
Bump Displace 1255
Bump Displace menu settings 1255
Bump Displace node 1255
Burn for Lustre
overview 2233
rendering locally 2233
rendering remotely 2234
Burn-In Letterbox 1257, 1258
Burn-In Letterbox menu settings 1258
Burn-In Letterbox node 1257
Burn-In Timecode 1258, 1259
Burn-In Timecode menu settings 1259
Burn-In Timecode node 1258
bypassing
menu parameters 1805
menu parameters, in a stereoscopic project 2215
menus with the Autodesk control surface 2448

C

C-mode sort 1993
caching
degrained frames 2013
for real-time playback 1996
memory with the Tangent CP100 2487
removal of degrained frames 2015

calculator
applying values to UI elements 1819
hot keys 2346
using to calculate timecode 1977
calibrating. *See* monitor calibration
cameras, in Action
adding 640
clipping planes 645, 666
depth of field 646
exporting 646
importing 646
modifying 652, 673
stereoscopic 658
cancelling, actions 1806
canvas
clearing strokes 1022
rotating 1024
wiping 1023
capturing
audio file 2309
frame grab 181
from a batch list 2271
from a VTR 2264
from an EDL, ALE or Cutlist file 2273
from timecode 2267
hot keys 2359
live video signals 200
performing live from non-controlled source 2276
removing pulldown 2265
shots after the initial conform 1905
single frames 181
supported video formats 2298
video, overview 2257
with timecode breaks 2270
YUV material 209
Car Paint shading, in Action 594
cascading 3D text, in Action 573
cast shadows
adding, in Action 506
catalogues, keyboard shortcuts 1665
catastrophic system failure 240
CDL 1907
adding data to EDL file 1895
example workflow 1895
loading data in EDL panel 1910
Offset 1908
panel 1907
parameter in Selector 1909
Power 1908
Slope 1908
CDL files 1892
change lists
creating 1776
deleting 1776
loading 1776
using to track changes 1775
Channel Editor
accessing 1472
analysing audio 1488
linking channels with expressions 1498
particle parameters 759
selection sets 1487
tangents 1483
channel hierarchy, using to animate channels 2174
channels
selecting 2177
selecting for a key 2116
viewing animated 2174
viewing selected animated 2174
checkerboard 51
chroma
modifying, in Colour Warper 995
chroma key, green screen, Modular Keyer 791
Clamp 1260
Clamp node 1260
Clamp menu settings 1260
clamping
colours, in Colour Warper 963, 1266
cleaning up, render files 2230
clearing
canvas 1022
strokes, sources 1005
clip history 1533
clip I/O
See also clips, capturing, outputting
configuring hardware 180
generating proxies at capture 181
HD formats, supported 201
input using VTRs 181
monitoring video 199
naming clips and tapes for 185
outputting to a VTR 187
software initialization configuration file 180
using VTR, general workflow 180
clip, sharing 140
clip, transfer 141
clip, wire 141
clips
See also clip I/O
ganging in Source Area 39
inserting on timeline 268
locking duration 1027
overwriting on timeline 269
replacing on timeline 270
splicing, on Desktop 303
unlocking duration 1027
Clone medium 1049
cloning, in Paint node 1017
CMX 152
collaboration 139, 141
collapsing, Storyboard 1983
color transform file format 1564
coloucolocolour transforms
r transforms
exposure and contrast 1560
colour balance
adjusting, in Colour Corrector 955
colour casts, in Colour Warper 995
Colour Correct 1261
Colour Correct menu settings 1261

colour correcting
clamping, in Colour Warper 963, 1266
colour balance, adjusting 955
colour values, remapping 961
contrast, adjusting 953
contrast, decreasing 959
contrast, in Colour Warper 987
contrast, increasing 958
deleting, in Colour Warper 982
desaturation, in Colour Warper 988
gamma, adjusting 952
highlights 948
hue, adjusting 949
imbalances, in Colour Warper 983
in Action 448
in Colour Warper 971
in Paint 1120
inverting mattes, in Colour Warper 973
luma ranges, defining 959
matching colours 946
matching, in Colour Warper 993
midtones 948
offsets, adjusting 952
pivot, adjusting 950
removing colours, in Colour Warper 984
resetting in Colour Warper 974
sampling colours 944
sampling, in Colour Warper 975
saturation, adjusting 949
shadows 948
spills, removing in Colour Warper 997
suppressing colours 954
suppressing colours, in Colour Warper 986
Colour Corrector
accessing from Action 448
accessing from other modules, tools 934
background brightness, changing 939
clips, viewing 938
colour curves 961
colour patches 944
crop box 940
cutouts 1109
gain 952
histogram 956
input levels 958
menu, hiding 935
output levels 959
overlay user interface 935
processing 962
resetting 942
RGB channels, rewiring 950
setup options 939
setups and preferences, saving 944
updating images interactively 941
Colour Curves 1264, 1265
Colour Curves menu settings 1265
Colour Curves node 1264
colour grading 2038
accessing Colour menus 2041
accessing secondary grading menus 2084
animating grades 2053
concepts 2039
creating continuity grades 2040
curves grading 2040, 2074, 2462, 2497
hot keys 2350, 2356
in a stereoscopic project 2214
keying ranges of colours to create secondaries 2116
loading presets 2052
primaries with the Autodesk control surface 2451
primary grading 2040, 2058
saving presets 2051
secondaries with the Autodesk control surface 2477
secondaries with the Tangent CP100 2498
secondary grading 2040, 2083, 2088
selecting Logarithmic or Linear mode 2040
using control surfaces 2046
using printer lights 2041
workflows 2039
colour management 1539
about 1539
applying to broadcast monitor or projector 1559
applying to clips 1539
applying to display 1556
supported file formats 1563
Colour Matching on the Timeline 335
colour models
selecting to key with in Keyer 839
colour palette
colour gradient 1032
in Paint 1030
mixing colours 1031
colour sampling
HLS Keyer 2118
colour space
rendering 2226
setting for projects 1786, 1849
UI differences between Logarithmic and Linear mode 1786
colour space conversion, enabling on clip I/O 211
colour spill
removing, in Colour Warper 997
colour transforms
applying 1541
applying on import (Lustre) 1884
applying to display 1556
applying to display (Lustre) 1853
applying to export (Lustre) 2250
applying to render (Lustre) 2226
building custom 1543
building custom (Lustre) 1885
controlling the look 1560
exporting 1555
exporting (Lustre) 1886
file locations 1562
graphicsMonitor alias 1556
transcode options (Lustre) 1883
types 1561
Colour Warper
about 962, 1265
accessing 964, 1266

colour correcting 971
reference clips, viewing 966
resetting colour corrections 974
subsetups 999
undo levels 975
viewports, displaying 966
working environment 965
Coloured Frame 1271
Coloured Frame menu settings 1271
Coloured Frame node 1271
colourimeter, calibrating a monitor 1850
colourist timeline 2054
colours
adjusting for geometries 2105
analysing 2047
comparing when creating conversion LUTs 2033
sampling values 2047
setting Colour view in the Player 1984
viewing distributions with the Analyse tools 2049
Combine 1272, 1273
Combine menu settings 1272
Combine node 1272, 1273
Command Line Renderer
example 2239
rendering with 2237
syntax 2237
comments 1602
compare offline, compare online 252
comparing
grades with the Autodesk control surface 2439
grades with the Tangent CP100 2490
shots using a reference 2046
shots using multi-view 2046
compositing
with Action 405
Compound 1272
Compound node 1272
compressed media
browsing for footage 1865
file formats 1864
confidence check, performing 1965
configuration file
SynColor 1602
confirming, actions 1806
conform 72, 94, 150, 151, 152, 179
Constant shape, for garbage masks 894
containers
creating on timeline 285
continuity grades, overview 2040
continuity, matching colours between shots 2072
contrast
adjusting in Colour Corrector 953
adjusting in Colour Warper 987
adjusting, pivot in Colour Corrector 950
decreasing, in Colour Corrector 959
displayed images 255
gamma, in Colour Warper 989
increasing, in Colour Corrector 958
modifying in Linear mode 2071
modifying in Logarithmic mode 2064
saturation, in Colour Warper 988
setting the pivot point 2065
with color transforms 1560
control surface
Autodesk Control Surface, connecting 2421
control surfaces
Autodesk control surface 2420
Tangent CP100 2484
using for colour grading 2046
controlling
VTR 2262
Controlling a Mask's Effect 322
converting
data with input LUTs 2029
from logarithmic to linear 2030
Cook-Torrance shading, in Action 593
copying
audio files into Lustre 2306
EDL, ALE, and Cutlist files into Lustre 1897
elements 1939
grades in the colourist timeline 2055
grades with the Autodesk control surface 2442
grades with the Tangent CP100 2492
keyframes 2194, 2196, 2197, 2198
objects in Paint 1084
parameters with the Selector 1816
shapes and grades 2090
Copying Moving Removing Timeline Transitons 316
Copying Timeline FX 332
Copyright element 1569
crawls, text 1141
Create BFX clip 385
creating
change lists 1776
cuts 1774
projects 1730
users 1755
Creating a Mask 320
Creating a Text Matte Key 335
Creating Customized Wipes with Masks 320
Creating Your Own Matchbox Effects 1388
credits, adding to the timeline 1844
crop box
creating in Colour Corrector 940
using, in Colour Warper 972
cropped frames, repositioning 2017
cropping
in Action 446
CTF file
example 1564
CTF file format 1564
cursor
activating on the Autodesk control surface 2428
curves
adjusting degrain 2013
adjusting saturation 2079
colour grading 2074
measuring for monitor calibration 1851
modifying lightness 2082
modifying luminance 2081

modifying RGB channels 2076
plotting colour values 2075
shifting hue 2078
curves colour grading
hot keys 2357
overview 2040
view mode 1973
with the Autodesk control surface 2462
with the Tangent CP100 2497
custom brushes, in Paint node 1016
customizing 105
the view with the Autodesk control surface 2430
the view with the Tangent CP100 2486
CUT files
data types 2376
example 2377
modifying 2362
cut point, sliding between shots 1923
Cutlist files
assembling 1892
auto-conforming 2273
copying into Lustre 1897
marking shots for later capture 1905
cutouts
adding drop shadows 1106
animating 1110
blending 1108
colour correcting 1109
drop shadows 1109
embossing 1107
extruding 1108
FrameCut 1104
GeoCut 1105
Keep Cut On 1103
PolyCut 1104
RectCut 1104
cuts
adding overlays 1791
changing without affecting grade settings 1779
creating 287
creating new 1774
deleting 1775
inserting between shots 1924
managing 1773
match options 1779
tracking changes 1775

D

Damage 1273, 1275
Damage menu settings 1275
Damage node 1273
Deal 1300, 1301
Deal menu settings 1301
Deal node 1300
definition 227
Deform 1301, 1302
Deform menu settings 1302
Deform node 1301
deformation meshes
adding, in Action 544
deformation models, in Action 544
Degrain 1306, 1307
adjusting response curves 2013
adjusting spatial settings 2012
analysing a frame 2008
applying a filter 2009
applying temporal settings 2012
caching results 2013
copying parameters using Selector 2016
saving and loading presets 2016
using secondaries 2015
using sub-region boxes 2010
workflow 2008
Degrain menu settings 1306, 1307
Deinterlace 1307, 1308
Deinterlace menu settings 1308
Deinterlace node 1307
Delaunay tessellation 575
deleting
change lists 1776
cuts 1775
geometries 2113
grades 1778
grades, from Grade bin 1827
keyframes 2193
shots 1916
shots from Library 1876
source nodes 1004
deleting elements 1941
Deliverables
See also Real-Time Deliverables
about 214
benchmarks 219
creating 221
creating real-time settings 221
deleting 222
displayed information 222
loading 222
managing 221
menu 217
moving to other systems 223
multiple Deliverables, outputting 220
output workflow 219
outputting 221
previewing material 219
processing material 219
renaming 221
saving 222
settings 221
timings 223
transferring to other systems 223
viewing from the library 222
virtual clip 214
Wire 223
working with 219
Denoise 1308, 1309
Denoise menu settings 1309
Denoise node 1308

Depth of Field 1312, 1313
Depth of Field menu settings 1313
Depth of Field node 1312
depth of field, in Action cameras 646
Description element 1571
destination, specifying render files for 2336
developing, Sparks 2171
Diamond Keyer
extracting keys 2472
scaling geometries 2133
setting tolerance range 2129
skewing geometries 2133
viewing secondaries 2120, 2128
Difference Matte 1318
Difference Matte menu settings 1318
Difference Matte node 1318
different resolutions 237
diffuse maps, in Action 615
Direction brush attribute 1009, 1042, 1045
directories, setting up for projects 1734, 1735
disabling
dissolves 1931
reframing settings 2020
Sparks 2170
discrete archive 229
displacement maps, in Action 595
displaying, sources as overlays 1007
dissolves
adding between shots 1927
adding in the Timeline 1957
animating 2188
changing duration 1928
changing orientation in the Timeline 1958
changing speed 1929
creating 1927
creating additional handles for 1928
disabling and re-enabling 1931
editing 1929
editing in the Timeline 1957
in EDLs 174
in the Timeline 1957
overview 1927
removing in the Timeline 1957
retrimming 1928
Distort 1319, 1320
about 1157
adding nodes 1177
blending clips 1183
correspondence points 1185
garbage mask splines 1165
linking source nodes 1179
morphing workflow 1159
schematic, using 1172
setups, saving and loading 1199
source interpolation 1183
splines, animating 1180
splines, drawing 1166
splines, editing 1170
splines, instancing 1171
splines, transforming 1171
tracking and stabilizing 1181
warping workflow 1158
Distort menu settings 1320
Distort node 1319
DLT 229
DLT, archiving clip 234
DLT, archiving project 232
DLT, configuration 231
DLT, restoring clips 235
DLT, restoring project 233
double-speed tape 186
Drag medium 1050
drawing
basic geometries 2093
free-form geometries 2093
split screens 2107
drawing. *See* painting
drop shadows
adding to cutouts 1106
in Action 478
Dual view
enabling 1986
in a stereoscopic project 2212
viewing on the SDI monitor 1989
dual-serial link I/O, configuring 213
Duplicate 1325
Duplicate tool 1325
dust removal
choosing a repair type 1998
choosing manual or automated 1999
dealing with problem dust artefacts 2003
drawing boxes 2004
hot keys 2346
optimizing parameters 1999
overview 1998
performing automated 2001
performing manual 2001
rendering 2007, 2223
saving metadata 2006
using the infrared channel 2005
DynamicParameter element 1601

E

ECParams element 1590
Edge Detect 1325, 1326
Edge Detect menu settings 1326
Edge Detect node 1325
edges, modifying for keys 2134
editing
accessing the edit menu 1913, 2328, 2335
duplicating shots in the Storyboard 1916
EDLs 153
frames by trimming 1917
hot keys 1914, 2347, 2358
in a stereoscopic project 2213
in and out points interactively 1922
in Timeline 1936
keycode in EDLs 175

overview 1912, 2315
shot order in the Storyboard 1915
shots from the Storyboard 1916
shots from the Timeline 1916
start timecode 1964
timecodes in EDLs 175
timewarps in EDLs 177
tracks in EDL 173
transitions in EDLs 174
with the Storyboard 1914
XML-based files 2362
editing import settings 105
edits
insert 268
overwrite 269
replace 270
EDL 152
EDL files 2209
assembling 1892, 1898
assembly workflow 1894
auto-conforming 2273
copying into Lustre 1897
EDL display and functionality 1893
exporting 1903
marking shots for later capture 1905
EDLs
assembly options 155
audio patching 171
display options 167
editing 153
editing comments 164
editing events 163
editing options 167
editing titles 164
editing transitions 165
importing options 153
keycode 175
marking events for capture 165
other options 166
preferences, importing 158
search events 159
tape name, editing 171
timecode 175
timewarps, editing 177
tracks, editing 173
transitions, editing 174
Undo list 168
view 170
wipes 174
Effects and Tools reference 1223
effects, creating with Sparks 2168
elements
copying 1939
deleting 1941
moving 1938
selecting 1937
snapping 1937
swapping 1939
trimming 1952
emission maps, in Action 611
enabling
stereoscopy 2207
event list 150
events list 151
export 52
export, default 60
export, remote Gateway 63
exporting, EDL files 1903
exposure
displayed images 255
with colour transforms 1560
Exposure 1329, 1330
Exposure menu settings 1330
Exposure node 1329
ExposureContrast element 1589
expressions
applying 1496
CascadeChannel keyword 1500
frame keyword 1502
functions 1501, 1505
linking channels 1498
modifying 1497
operators 1503
overview 1496
samples 1528
SelectionOrder keyword 1499
setups 1528
user-defined functions 1526
vectors 1501
extracting
elements from an edit sequence 289
extrapolation, setting 2181

F

FBX
cameras, in Action 646
exporting 3D tracker results 646
importing 534
materials 541
textures 541
FBX cameras, in Action
exporting 669
fcp 72
Field Merge 1331
Field Merge menu settings 1331
Field Merge node 1331
field-based processing, setting the active field 1974
file archive, to restore 229
file browser
in Grade view 1829
file browser, accessing 1856
file formats, supported for playback 1856
fill objects, drawing 1081
filling areas, in Paint 1059
Filter 1331, 1332
Filter menu settings 1332
Filter node 1331

filtering
Animation Channel Editor 1485
filters
in graphics 1089
in Paint 1046
final cut pro 179
Final Cut Pro 72
fit to fill edit 270
flagging shots 1808
automatically 2216
for rendering 2216
manually 2217
Flare 45
Flip 1333, 1334
Flip menu settings 1334
Flip node 1333
flipbook, previewing animation in Paint 1032
flipping, shots 2021
flopping, shots 2021
FontMapping keyword 1133
fonts
in Text module 1152
Format options 330
format preset, default 60
format, setting the output for render files 2225
frame capture 181
frame grab 181
FrameCut command 1104
Fresnel shading, in Action 593
full-resolution images, displaying 1974
full-screen Player
activating 1970
panning and zooming 1972
using hotkeys 1996
functions, in expressions 1501

G

gain
adjusting in Colour Corrector 952
adjusting luminance of key in Keyer 846
gain, modifying 2069
gamma
adjusting for the monitor manually 1853
adjusting in Colour Corrector 952
adjusting, in Colour Warper 989
applying 1541
measuring for monitor calibration 1850
modifying 2069
modifying for input LUTs 2035
Gamma element 1586
GammaParams element 1588
ganging clips 39
gaps
on timeline 286
Gaps 330
Garbage Mask
Garbage Mask menu settings 1335
GMask node 1334
garbage masks
Action 3d geometry 549
animating 906
animating in Tracer 923
animating with tracking 911
defined 891
Distort, importing 1165
drawing 897
drawing order 902
interlaced rendering 914
loading and saving setups 927
motion blur, applying 913
moving 903
opacity 894
order they are drawn 902
ROI (Region of Interest), using with multiple garbage masks 915
rotating 903
scaling 903
schematic, viewing 902
setups, preset 929
setups, raw 929
softness 894
softness, with advanced gradient 917
straight edges 904
tangent handles, moving 904
tangent handles, selecting 903
Tracer 916
turning on and off 894
vertices, activating 909
vertices, adding 909
vertices, adjusting number drawn freehand 901
vertices, deactivating 909
vertices, deleting 911
vertices, moving 904
vertices, removing 909
vertices, selecting 903
garbage masks, loading setups 2155
Gateway clip
modifying 1683
Gaussian blur
applying, in Colour Warper 982
in Action 445
general preferences 1674
Generate Composite
in Batch FX 346
generating, proxies 2227
GeoCut command 1105
geometries
adding and deleting axes 2109
adjusting shapes 2098
animating with the point tracker 2144, 2147
animating with the shape tracker 2144
assigning tracking data 2150
blurring 2104
changing priority order 2111
copying shapes and grades 2090
creating 2091
creating a parent axis 2109
creating with the Autodesk control surface 2477

creating with the Tangent CP100 2503
deleting 2113
deleting an axis 2109
deleting point tracker data from 2153
drawing basic 2093
drawing free-form 2093
drawing split screens 2107
grading inside and outside 2090
inverting 2112
linking across secondaries 2114
modifying 2097
moving axes 2098
scaling 2133
setting colour and opacity 2105
showing and hiding 2112
skewing 2133
softening edges 2101
soloing and ganging point tracker 2151
transforming 2100
geometry
importing in Action 532
painting with 1036
getting started 13
Glow 1338, 1339
Glow menu settings 1339
Glow node 1338
GLU tessellation 575
GPU preview processing
GpuNoAutoSwitch parameter 2046
playing out from source with 2295
real-time 2043
Grade bins
accessing with the Autodesk control surface 2440
applying grades 1824
deleting grades 1827
Marry files 1821
saving grades 1822
types 1821
Grade menu, selecting a scene 1770
Grade view
applying grade to shots 1835
deleting grades 1836
displaying 1829
loading from 1835
loading presets 1844
moving files 1835
Quick folder 1835
viewing media file information 1843
grades
applying from the Grade bin 2440
applying to a new assembled timeline 1901
comparing with the Autodesk control surface 2439
comparing with the Tangent CP100 2490
copying in the colourist timeline 2055
copying unrendered changes to real-time deliverables 2295
copying with the Autodesk control surface 2442
copying with the Tangent CP100 2492
deleting 1778
emulating telecine-style tape 2290
in a stereoscopic project 2214
loading backup copies 1783
loading from Grade view 1835
locking 2230
maintaining when swapping cuts 1779
reverting to previous with the Autodesk control surface 2439
reverting to previous with the Tangent CP100 2490
saving and loading using Marry controls 1784
gradient
advanced, applying to garbage masks 916
animating in Paint 1098
Gradient 1346, 1347
gradient mattes, creating in Paint 1073
Gradient menu settings 1347
Gradient node 1346
grading. *See* colour grading
grain removal
hotkeys 2347
grain removal. See Degrain
graphics, in Paint
adding 1075
anamorphic geometry 1087
animating 1096
animating gradients 1098
Auto Point 1093
Bezier curves 1093
blending 1108
brush tools 1076
buffer, recalling from 1100
buffer, storing to 1100
changing order for objects 1085
deleting objects 1084
drawing objects 1076
drop shadows 1109
editing objects 1084, 1091
embossing 1107
extruding 1108
fill tool 1081
gradients 1089, 1098
hiding objects 1085
hiding vertices 1087
increasing speed 1087
menu 1075
moving objects 1091
object attributes 1087
outlining objects 1087, 1088
resizing objects 1091
resolution, setting for objects 1085
rotating objects 1091
rubber banding 1081
saving 1099
selecting objects 1083
shape animation 1096
shapes 1092
solid and soft edge objects 1087, 1088
tacking 1100
text object 1082
transforming objects 1091
using AutoPaint with 1095

wireframe 1085, 1087
wireframe options 1086
grids and guides
displaying the proper aspect ratio 138
in Paint 1029
grouping
applying changes to shots as a group 1812
creating a shot group 1811
soloing shots in a group 1813, 1815

H

HDCAM 236
HDCAM SR
capture 186
output 198
head frames, rendering 2224
hiding strokes 1005
hiding, Storyboard 1979
highlights
colour correcting, in Colour Corrector 948
matching, in Colour Warper 994
modifying, in Colour Warper 983
removing colour, in Colour Warper 984
histogram
in Colour Corrector 956
luma, viewing in Colour Warper 970
History list, loading for a grade 1783
HLS Keyer
colour sampling 2118
extracting keys 2467
selecting channels for keys 2116
setting softness range 2121, 2123
using 2116
viewing secondaries 2120, 2128
hot keys
accessing editing hot keys globally 1914
assigning notes to shots 2345
capturing and playing out with the numeric keypad 2359
colour grading 2350, 2356
creating animations 2360
curves colour grading with the numeric keypad 2357
editing 2347
editing with the numeric keypad 2358
miscellaneous functions 2361
navigating the Storyboard 2341
navigating with the playback controls 1976
numeric keypad 2356
removing dust 2346
switching view modes 2342
using calculators 2346
hotkeys
colourist timeline 2355
for Player and full-screen Player 1996
removing grain 2347
secondary layers 2353
hue
adjusting, in Colour Corrector 949
cube, in Colour Warper 982
shifting in Keyer 852
shifts, in Colour Warper 995
hues
shifting for secondaries 2088
shifting with curves 2078

I

image window
displaying the proper aspect ratio 138
multiple views 262
reference area 260
image-based lighting
adding, in Action 527
images
applying colours and filters 1057
applying random paint strokes 1065
import 51, 72, 94, 105
import history, to modify 138
import options, to modify 138
import, history 138
importing
audio files 2306
EDL, ALE, and Cutlist files 1897
FBX 534
half-float media 2332
subtitle XML 142
importing tips 51
Impressionist medium 1051
individual files 229
Info element 1568
infrared channel, using for dust removal 2005
ingest 51
init.config 1720
Input Clip menu 181
input LUTs
about 2026
applying existing 2029
comparing colour values 2033
creating conversion LUTs 2030
importing 2027
modifying gamma of film footage 2035
modifying reference black and white levels 2034
modifying reference soft clip values 2036
viewing reference images 2031
InputDescriptor element 1572
insert edits 268
installing, Sparks 2156
Interface menu, customizing Lustres appearance 1789
Interface overview 15
Interlace 1354
Interlace menu settings 1354
Interlace node 1354
interlaced video
garbage masks, rendering 914
interlaced video, setting the active field 1974

interpolation
expressions 1496
interpolation, setting 2179
introduction 227
inverting
geometries 2112
keyframe values 2191
keys 2136
inverting mattes 1072

J

jitter
in Stabilizer, removing 875
removing from animation curve 1482
Jitter brush attribute 1009, 1041
joining, shots together 1925

K

kerning text 1137
keyboard
using in Text 1134
keyboard shortcut
determining for buttons 1664
keyboard shortcuts 1657
catalogues 1665
changing 1666
creating 1668
displaying errors 1664
Keyboard Shortcut Editor 1664
keyboard type 1666
keycode
assembly 1904
assembly workflow 1894
editing in EDLs 175
keyer
Diamond Keyer using 2124
HLS Keyer using 2116
Keyer
accessing from Action 448
garbage masks, using with 891
luminance curve, adjusting 846
Matte Edge controls 848
solid colour background, using 855
Keyer 3D 1354, 1355
Keyer 3D menu settings 1355
Keyer 3D node 1354
Keyer Channel 1359, 1360
Keyer Channel menu settings 1360
Keyer Channel node 1359
Keyer HLS 1360, 1361
Keyer HLS menu settings 1361
Keyer HLS node 1360
Keyer Luma 1362, 1363
Keyer Luma menu setting 1363
Keyer Luma node 1362
Keyer RGB 1363, 1364
Keyer RGB menu settings 1364
Keyer RGB node 1363
Keyer RGBCMYL 1365, 1366
Keyer RGBCMYL menu settings 1366
Keyer RGBCMYL node 1365
Keyer YUV 1368, 1369
Keyer YUV menu settings 1369
Keyer YUV node 1368
keyframes
adding 2177
adding and moving with the Tangent CP100 2493
adding from outside the Animation menu 2204
animating 1471
averaging 1482
baking 1481
changing values 1475
copying 1476, 2196, 2197, 2198
copying and pasting 2194
deleting 2193
editing 2184
inverting values 2191
keyframe indicators 1472
modifying curves 2186
navigating between 2173
pasting 2199
reversing order 2190
scaling to adjust animation duration 2203
selecting 2184
setting extrapolation 2181
setting interpolation 2179
viewing channel ranges between 2174
viewing selected 2174
keying
animating in Keyer 855
colour models, selecting in Keyer 839
edges of key, adjusting in Keyer 828
grey areas, removing from key in Keyer 846
hue, shifting in Keyer 852
in Action 448
with Diamond Keyer 2472
with HLS Keyer 2467
with the Autodesk control surface, Diamond Keyer 2472
with the Autodesk control surface, HLS Keyer 2467
with the Tangent CP100 2500
workflow in Master Keyer 828
keys
blurring 2134
creating to isolate ranges of colours 2116
excluding from secondaries 2136
extracting 2466
inverting 2136
removing stray pixels from 2135
saving and loading presets 2137
selecting channels 2116
setting softness range, Diamond Keyer 2131
setting softness range, HLS Keyer 2123
setting tolerance range, HLS Keyer 2121
shrinking edges 2134
viewing results 2120, 2128

keywords
FontMapping 1133
Video 180
VTR 180

L

layer order, of garbage masks 894
learning the new workflow 13
Lens Distort 1370, 1371
Lens Distort menu settings 1371
Lens Distort node 1370
lens flares
adding presets 512
adding, in Action 510
letterbox formats
outputting clips with letterbox 193
letterbox, scaling 2016
levels, selecting in the image for colour grading 2067
Library
deleting shots from 1876
loading shots from the file browser 1856, 1874
selecting shots for the Storyboard 1915
lift
adjusting luminance of key 846
lift, modifying colour values 2067
lifting elements from an edit sequence 289
lightness, modifying using the Lightness curve 2082
lights
adding, in Action 495
flipping normals, in Action 503
selective lighting, in Action 497
specular highlights, in Action 501
limitations, XML-based files 2362
limiting strokes 1006
linear data
converting to log 1541
Linear mode
defining luminance 2071
grading with the Autodesk control surface 2457
grading with the Tangent CP100 2496
improving contrast 2071
modifying gain 2069
modifying gamma 2069
modifying lift 2067
modifying saturation 2070
monitor calibration 1846
using in Lustre 2040
Linear mode, selecting levels 2067
linking, geometries across layers 2114
live video signals, inputting and outputting 200
loading
change lists 1776
projects 1730
shots to the Library 1874
shots to the storyboard 1876
stereoscopic footage 2209
users 1755
location, OTOC 242
locking
clips duration 1027
colour grade 2230
sources 1007
log data
converting to linear 1541
Log element 1591
Logarithmic mode
adjusting printer lights 2061
grading with the Autodesk control surface 2451
grading with the Tangent CP100 2495
improving contrast 2064
modifying brightness for low, mid, or high levels 2062
modifying brightness for the entire image 2060
modifying saturation 2066
monitor calibration 1846
resetting brightness 2063
setting the pivot point 2065
using in Lustre 2040
Logic Op 1373, 1374
Logic Op menu settings 1374
Logic Op node 1373
logos
loading into Text module 1139
overlay controls 194
overlay requirements 193
overlay, inserting in real-time 197
LogParams element 1592
long-term storage 227
look
with colour transforms 1560
look-at
3D path in Action 638
axis in Action 454
looping
using the Autodesk control surface 2428
using the Tangent CP100 2487
LTC chase option
use explicit timecode 2314
LTO, archiving clip 234
LTO, archiving project 232
LTO, configuration 231
LTO, restoring clips 235
LTO, restoring project 233
luma
adjusting after colour correction 996
adjusting ranges 982
adjusting, in Colour Warper 989
defining ranges, in Colour Corrector 959
viewing with 2D luma histogram 970
luminance
mattes, values for 1074
modifying in Linear mode 2071
modifying using the Luminance curve 2081
Lustre
exiting 1801
starting 1801
lustre, interoperability 139

LUT Editor 1374, 1375, 1539
accessing 1540
comparing colours 1554
LUT Editor menu settings 1375
LUT Editor node 1374
LUT1D element 1574
LUT3D element 1577
LUTs
advanced editing 1551
applying 1541
applying on import (Lustre) 1884, 2026
applying to broadcast monitor or projector 1559
applying to display 1556
applying to display (Lustre) 1853
applying to export (Lustre) 2250
applying to render (Lustre) 2226
basic modification 1544
controlling the look 1560
exporting 1555
file locations 1562
preferences 1677
transcode options (Lustre) 1883
viewing 1975

M

Magnet Curve Editor, in Action 475
manual dust removal
overview 1999
performing 2001
Map Convert 1380, 1381
Map Convert menu settings 1381
Map Convert node 1380
Marry grades
saving and loading using Marry controls 1784
saving in Grade bin 1821
masks
See also garbage masks
Master Keyer 1383, 1385
See also mattes, keying, Keyer, patches
steps for creating and refining a key 828
Master Keyer menu settings 1385
Master Keyer node 1383
match grade
match options 1901
Matchbox 1386, 1387
Matchbox menu settings 1387
Matchbox node 1386
matching
colours between shots 2072
colours, in Colour Corrector 946
colours, in Colour Warper 993
grade settings with a new assembled timeline 1901
material node 541
material node, in Action 587
materials 541
Matrix element 1579
Matte Curves 1390, 1391
Matte Curves menu setting 1391
Matte Curves node 1390
Matte Edge 1391, 1392
Matte Edge menu settings 1392
Matte Edge node 1391
matte overlay
adjusting colour 2106
adjusting opacity 2106
matte overlay settings 1790
mattes
Auto Matte in Paint 1074
changing colour 1071
creating 1072
creating gradients 1073
deleting from animation curve, in Colour Warper 982
deleting, in Colour Warper 982
enabling 1070
garbage masks, removing parts 891
generating for selective colour correction 977
generating in Master Keyer 829
inverting 1072
inverting, in Colour Warper 973
limiting strokes with matte 1014
limiting strokes with source matte 1006
loading 1072
luminance values 1074
resetting, in Paint 1073
saving 1072
transparency 1071
using 1070
viewing 1070
maxInValue element 1584
maxOutValue element 1585
media
See also special effects media
adding, in Action 442
grouping, in Action 482
loading, in Action 441
slipping, in Action 448
media composer 178
Medial Axis tessellation 577
memory
caching with the Tangent CP100 2487
menus
bypassing with the Autodesk control surface 2448
resetting with the Autodesk control surface 2448
resetting with the Tangent CP100 2491
meshes
for morphing 1216
in Warper 1201
metadata, saving for dust removal 2006
midtones
colour correcting, in Colour Corrector 948
defining range, in Colour Warper 985
removing colour, in Colour Warper 984
minInValue element 1584
minOutValue element 1585
mixed resolution 237
mixed timeline 237
Modifying a Mask's Shape 321

Modifying Timeline FX 331
Modular Keyer
accessing from Action 450
monitor calibration 1846
adjusting black level and picture settings 1847
adjusting manually 1853
calibrating 1849
curves 1851
for dual-head mode 1853
gamma 1850
Logarithmic and Linear mode 1846
Print LUTs 1853
reverting to previous settings 1852
selecting project colour space 1849
using a colourimeter 1850
workflow 1847
monitoring during clip I/O 199
monitoring location of current frame 1977
monitoring options 2311
Mono 1396, 1397
Mono menu settings 1397
Mono node 1396
morphing
meshes 1216
one image into another 1216
with Distort 1158
with the Warper 1199, 1466
Motif 1397, 1399
Motif menu settings 1399
Motif node 1397
Motion Analysis 1401, 1402
Motion Analysis menu settings 1402
Motion Analysis node 1401
motion blur
garbage masks, applying 913
garbage masks, applying to stationary 914
in Action 653
Motion Blur 1402, 1403
Motion Blur menu settings 1403
Motion Blur node 1402
Motion Convert 1405
Motion Convert menu settings 1405
Motion Convert node 1405
Motion Convert node in Batch 1405
motion paths
creating with text 1147
mouse
using in Paint 1028
mouse functionality
activating on the Autodesk control surface 2428
moving elements 1938
moving, geometries 2100
Multi view
enabling 1988
viewing on the SDI monitor 1989
multi-camera edits
ganging to view clips 39
Multi-Layer Timeline 1932
muted layers 1959
shot priority 1960
solo mode 1963
Top Vertical Priority 1958
multi-part archive 240
multi-volume 240
multiple views in tools 262
multiversion clips, in Action 407
muted layers. *See* Multi-Layer Timeline
Muting and Deleting Timeline FX 333
MUX 1406, 1407
MUX menu settings 1407
MUX node 1406
MXF, supported codecs on import 1869

N

navigating
between keyframes 2173
the Storyboard 1975, 1979
using the timebar and Storyboard 1975
with playback controls and hot keys 1976
with the Autodesk control surface 2425
with the Tangent CP100 2488
with Timeline navigation controls 1934
node bins
in Batch 399
Node bins
in Action 409
nodes
custom, creating 400
Noise
generating patterns 1449
noise, adding with Sparks 2157, 2165
normal maps, in Action 604
notes
assigning to shots 1807
hot keys 2345

O

Object Group node 535
Offset element 1597
offsets
adjusting, in Colour Corrector 952
animating, in Stabilizer 881
offsetting
cloned sources 1017
sources in Paint node 1020
Opacity brush attribute 1008, 1040
opacity of garbage masks, controlling 894
opacity, adjusting for geometries 2105
OpenEXR
configuring 1743
in Action 407
OpenEXR, importing 2332
Optics 1407, 1408
Optics Creating Glow Effects 1409
Optics menu settings 1408
Optics node 1407
OTOC 241

OTOC, definition 242
OTOC, location 242
output 52
output, view mode 1973
OutputDescriptor element 1572
outputting
assemble mode 198
letterbox overlay 193
live video signals 200
multiple clips 192
single clips 191
YUV material, using a VTR 209
overlay
user interface in Colour Corrector 935
overlay edits 269
overlays, adding to cuts 1791
overwrite edits 269

P

P2 MXF
See MXF
Paint
See also graphics, in Paint
about 1026
animating Paint strokes 1068
Auto Matte command 1074
AutoPaint 1063
brush attributes 1040
brush setups 1113
colour correcting the clips 1120
colour palette 1030
copying graphics 1084
creating a matte 1072
custom brushes 1114
filling an area with colour 1059
filters 1046
full-resolution film images 1033
library 1111
loading a matte 1072
media. *See* special effects media, brush attributes
node 1002
previewing animation 1032
recording brush strokes 1066
reference image overlay 1024
reference images 1044
revealing the back image 1052
Roll command 1036
saving a matte 1072
saving and restoring images 1034
selecting brushes 1035
selecting colours 1030
setups 1111
tracking with AutoPaint 1067
using a matte 1070
Wipe command 1057, 1064
Paint node
brush attributes, setting 1007
canvas, clearing strokes 1022
canvas, wiping 1023
cloning 1017
custom brushes, using 1016
saving setups 1026
sources, clearing strokes 1005
sources, deleting a node 1004
sources, displaying reference image overlays 1007
sources, locking 1007
sources, removing 1004
sources, revealing 1019
strokes, applying 1013
strokes, blending with current image 1020
strokes, clearing 1005
strokes, hiding 1005
strokes, limiting with matte 1014
strokes, limiting with source matte 1006
painting
circles with geometry 1038
circles with object tool 1079
ellipses 1078
full-resolution film images 1033
in Paint node 1013
limiting strokes with matte 1014
limiting strokes with source matte 1006
lines 1036, 1077
on the canvas 1034
rectangles 1037, 1077
squares 1077
triangles 1078
triangles with geometry 1038
with geometry 1036
panning
animation curves 2174
in Player and full-screen Player 1972
with the Autodesk control surface 2429
parallax maps, in Action 600
parameters
bypassing 1805
optimizing for dust removal 1999
resetting 1804
resetting with the Tangent CP100 2491
setting for projects 1756
particles
adding presets 717
applying textures 761
bouncing 783
creating custom particles 759
creating manipulators 765
generating streams 749
setting Channel Editor properties 759
using functions 775
pasting, keyframes 2199
patching
on timeline 282
perferences
user interface 1681
Perspective Grid, in Action 491
Photoshop files
importing, in Action 434

pickers
advanced gradient, using with 919
animating 924
assigning to vertices 921
defined 892, 918
interpolation between keyframes 925
selecting 904
picture settings, adjusting 1847
Pixel Analyser, sampling images 2047
Pixel Spread 1412, 1413
Pixel Spread menu settings 1413
Pixel Spread node 1412
pixels, removing from keys 2135
play modes
toggling with the Autodesk control surface 2428
toggling with the Tangent CP100 2487
playback,changing speed 1926
Player
activating full-screen 1970
aspect ratio, displaying 138
displaying the user interface 2042
setting Colour View 1984
using hotkeys 1996
viewing multiple shots 1984
playheads
assigning reference images 1972
assigning user interface to active playhead 2042
toggling with the Autodesk control surface 2437
playing
shots in the Player 1968
shots rendered with the Slave Renderer 2232
playing out
audio 2315
defining in and out points when 2282
from source with GPU preview processing 2295
hot keys 2359
in Assemble mode 2285
in Insert mode 2283
in Record mode 2287
real-time deliverables 2294
supported video formats 2298
to a VTR 2280
video, overview 2257
workflow 2281
plotting, sampled colour values 2075
plugins. *See* Sparks
point tracker
animating geometries with 2144
animating the shape of geometries 2147
deleting 2153
loading data 2154
soloing and ganging 2151
polygons, drawing 1079
position maps, in Action 621
post-production pipeline
integrating Lustre into 2321
Potential Matches 150
Power element 1597
Pre-processing options 330
preferences
audio 1671
general 1674
LUTs 1677
support 1678
tablet 1676
timeline 1678
Preferences 1671
premiere 179
prepend edit 270
preset 105
presets
for real-time deliverables 2292
loading colour grading parameters 2052
saving and loading for Degrain 2016
saving and loading for grading 1844
saving and loading for keys 2137
saving colour grading parameters 2051
Pressure brush attribute 1008, 1041
mode 1045
primary colour grading
applying secondaries to 2091
in Linear mode with the Autodesk control surface 2457
in Linear mode with the Tangent CP100 2496
in Logarithmic mode with the Autodesk control surface 2451
in Logarithmic mode with the Tangent CP100 2495
overview 2040, 2058
view mode 1973
Print LUTs 1853
print, view mode 1973
printer lights
accessing the calculator from 1819
adjusting brightness 2061
adjusting for secondaries 2089
using in Log mode 2041
Priority Editor, in Action 480
priority, garbage masks
drawing order 902
order 894
processing
in Colour Corrector 962
processing, image pipeline order 1701
ProcessList element 1567
projectors, in Action 627
projects
creating and loading 1730
creating the directory structure 1734, 1735
setting colour space architecture 1786
setting parameters 1756
setting the resolution 1787
setting up for soft-imported clips 2326
ProRes support 72, 2319
ProRes support , native 72, 2319
proxies
displaying 1974
generating 2227
generating at clip capture 181
outputting in Action 423

quality box 183
viewing 2047
pulldown
removing for capture 2265
Pulldown 1416, 1417
Pulldown menu settings 1417
Pulldown node 1416

Q

Quick Composite 1417, 1418
Quick Composite menu settings 1418
Quick Composite node 1417
Quick folder
defining 1835
QuickTime
supported codecs on import 1870

R

racking, to reframe footage 2019
Range element 1582
raster
selecting for stereoscopy 2207
Rate brush attribute 1008, 1040
rays
adding, in Action 521
rdesktop, monitoring results 2237
real-time deliverables
copying changes to or from the grade file 2295
creating presets 2292
overview 2291
playing out 2294
resizing 2294
Real-Time Deliverables
See also Deliverables
about 213
AJA limitations 215
formats 213
requirements 214
use case, film to video 215
use case, PAL to video 216
real-time playback
using GPU preview processing 2043
with frame caching 1996
recording
Assemble mode 2285
Insert mode 2283
Record mode 2287
recovering archives 243
Recursive Clone medium 1052
Recursive Ops 1418, 1419
Recursive Ops menu settings 1419
Recursive Ops node 1418
RED files 90
redoing
actions 1805
in animation 2206
with the Autodesk control surface 2439
with the Tangent CP100 2489
Ref element 1606
reference area
displaying 260
loading clip into 261
reference clip
viewing, in Colour Warper 966
Reference element 1599
reference images
assigning to playheads 1972
toggling playheads with the Autodesk control surface 2437
viewing 1970
viewing results of logarithmic to linear conversion 2031
reference, setting the view mode 1973
ReferenceTable element 1605
reflection maps, in Action 619
reframing
aspect ratio output frame 2018
disabling settings 2020
flipping and flopping shots 2021
racking the image 2019
using reposition controls 2020
refreshing thumbnails 1982
regions, rendering selected 2227
Regrain 1422, 1423
Regrain menu settings 1423
Regrain node 1422
Release element 1570
relink 150, 151
remote Gateway 63
remote project 141
removing, dust and other artefacts 1998
render files, removing 2230
Render passes, in Action 435
rendering
BackBurner Manager, starting 2234
burning in timecode 2223
colour space, setting 2226
dust removal data 2007, 2223
file destination, specifying 2336
flagging shots automatically 2216
flagging single shots manually 2217
head and tail frames 2224
in a stereoscopic project 2215
in the background 2231
locally with Burn for Lustre 2233
monitoring with Backburner Monitor 2235
output format, specifying 2225
overview 2215
region, specifying 2216
remotely with Burn for Lustre 2234
resolution, setting 2218
selected region 2227
setting quality for repositioned shots 2025
shots 2216
Timeline 1964
with Burn for Lustre 2233
with the Command Line Renderer 2237

with viewing LUTs 2224
repair type, selecting for artefact removal 1998
replace edits 270
replace media edit 270
replacing, shots in the Storyboard 1926
repositioning
after adjusting the project aspect ratio 2020
cropped frames 2017
interlaced footage using Sparks 2160
setting rendering quality after 2025
shots with the Autodesk control surface 2449
shots with the Tangent CP100 2494
using animation controls 2024
with the tracker 2022
resetting
in Colour Corrector 942
parameters 1804
secondaries 2113
with the Autodesk control surface 2448
with the Tangent CP100 2491
Resize 1424
Resize menu settings 1424
Resize node 1424
resizing, real-time deliverables 2294
resolution
display 1974
project 1787
rendering 2218
restore clips, archive 235
restore project, archive 233
restore setups 233
restoring
archives 244
restricting strokes 1006
retiming,playback speed for shots 1926
Reveal medium 1052
revealing, sources in Paint node 1019
reverting to previous
grades with the Autodesk control surface 2439
grades with the Tangent CP100 2490
reverting to previous, calibration settings 1852
RGB channels
adjusting, in Colour Warper 989
RGB channels, adjusting individually with curves 2076
ripple replace edit 270
rmd in Final Cut Pro 90
ROI (Region of Interest), using with multiple garbage masks 915
Roll brush attribute 1010, 1043
roll edit
on timeline 273
rolls, text 1141
rotating the canvas 1024
rotating, geometries 2100
rotoscoping 906, 1028

S

Safe Title
in Text 1138
SAIT, archiving clip 234
SAIT, archiving project 232
SAIT, configuration 231
SAIT, restoring clips 235
SAIT, restoring project 233
sampling
adjusting in the Tracer 924
colours, in Colour Corrector 944
sampling, pixels for autograding 2073
SatNode element 1598
saturation
adjusting with curves 2079
adjusting, in Colour Corrector 949
correcting colour suppression, in Colour Warper 997
in Colour Warper 988
matching, in Colour Warper 994
modifying in Linear mode 2070
modifying in Logarithmic mode 2066
Saturation element 1599
saving
grades 1822
setups in Paint node 1026
scaling
geometries 2100
letterbox format 2016
scaling geometries 2133
Scene menu, selecting a scene 1770
scenes
detecting 1966
selecting 1770
schematic
creating branches, in Action 420
garbage mask, viewing 902
in Action 413
selecting nodes 369
schematic view, toggling to 2110
SDI monitor,viewing multiple shots 1989
Searching
on the timeline 283
secondary colour grading
accessing grading menus 2084
adjusting printer lights 2089
applying Sparks 2169
applying to the entire image 2091
changing layer order 2111
creating geometries 2091
creating geometries with the Autodesk control surface 2477
creating geometries with the Tangent CP100 2503
creating keys with HLS Keyer 2467
creating keys with the Autodesk control surface, Diamond Keyer 2472
creating keys with the Tangent CP100 2500
excluding keys 2136
extracting keys with Diamond Keyer 2472

extracting keys with the Autodesk control surface, HLS Keyer 2467
grading outside the secondary 2090
keying ranges of colours 2116
loading garbage mask setups 2155
loading tracker and stabilizer data 2154
overview 2040, 2083
performing colour grading 2088
removing grain 2015
resetting 2113
shifting hue 2088
showing and hiding geometries 2112
view mode 1973
viewing key results 2120, 2128
with the Tangent CP100 2498
secondary layers 2085, 2465
hotkeys 2086, 2353
selecting and activating 2087
segment 240
segmented archive 240
selecting
keyframes 2184
raster for stereoscopy 2207
selecting elements 1937
selection sets, in Channel Editor 1487
selective colour grading. *See* secondary colour grading
selectives
creating, in Colour Warper 977
resetting, in Colour Warper 982
Selector
CDL parameter 1909
copying parameters 1816
Degrain parameters 2016
sequence 72, 94
setups
Action, loading 433
brushes 1113
garbage masks 927
Paint 1111
saving in Paint node 1026
Shade medium 1056
shader node, in Action 588
shadow cast
adding, in Action 507
shadows
See also softness
colour correcting, in Colour Corrector 948
matching, in Colour Warper 994
modifying, in Colour Warper 983
removing colour, in Colour Warper 984
shape animation 1096
Shape channel, used for garbage mask animation 906
shape tracker
animating geometries with 2144
point cloud 2139
region of interest 2139
tracking through translation, rotation, scaling, partial occlusions 2139
shapes, in Graphics (Paint) 1092
share 139
Shared Folders 140
SharedHome element 1604
sharing, changes between grades and real-time deliverables 2295
sharing, clips 140
shot priority. *See* Multi-Layer Timeline
shots
adding dissolves 1927
adding in Timeline 1936
adding to the Storyboard 1915
applying changes to a group 1812
applying grades 1824
assigning notes 1807
comparing multiple in the Player 2046
comparing using a reference 2046
deleting 1916
deleting from Library 1876
dividing into scenes 1966
duplicating in the Storyboard 1916
flagging 1808
grouping 1811
identifying selection colours on the Storyboard 1982
joining back together 1925
moving to the storyboard 1876
playing in the Player 1968
rendering 2216
reordering in the Storyboard 1915
replacing in the Storyboard 1926
repositioning with the Autodesk control surface 2449
repositioning with the Tangent CP100 2494
selecting in the Storyboard 1980
setting to half resolution 2047
slipping in the timeline 1921
soloing 1813, 1815
trimming head or tail frames 1918
viewing information 1917
shrinking
edges of keys 2134
single file 229
Single view, enabling 1984
Size brush attribute 1008, 1040
size, setting the Storyboard view 1979
skewing geometries 2133
Slave Rendering
caching degrained frames 2014
playing shots rendered with 2232
sliders, modifying with the calculator 1819
sliding
on timeline 273
Sliding Timeline FX Keyframes 332
sliding, cut point in a shot 1923
slipping
clips, in Action 448
on timeline 273
slipping, shots in the timeline 1921
Slope element 1596
Smear medium 1053

SMPTE Bars
setting for 2D vectorscope 966
snapping
keyframes 1475
snapping elements 1937
Snapshots
in Batch 384
softclip values, modifying for input LUTs 2036
softness
applying to geometries 2101
in Colour Warper 978
refining garbage masks 916
setting range for keys, Diamond Keyer 2131
setting range for keys, HLS Keyer 2123
softness range, defined in Keyer 838
softness border 916
gradient in Tracer, removing 920
separating from mask border 920
software initialization configuration file
configuring VTRs 180
Solo mode. *See* Multi-Layer Timeline
SOPNode element 1595
source 151
Source Area
ganging clips 39
source nodes in Action
anti-aliasing 703
cameras 705
creating 699
motion blur 703
replacing front or mattes 701
sources in Paint node
about 1003
adding 1003
blending with current image 1020
cloning 1017
displaying overlays 1007
limiting strokes 1006
locking 1007
removing 1004
revealing 1019
wiping 1023
Sparks 1427, 1429
applying to secondaries 2169
creating effects with 2168
descriptions 2156
developing 2171
disabling and re-enabling 2170
installing 2156
overview 2156
using to add noise 2157, 2165
using to blur images 2157, 2161, 2166
using to reposition interlaced footage 2160
spatial Degrain settings 2012
special effects media
Blur 1048
Clone 1049
Drag 1050
Impressionist 1051
in graphics 1089
Recursive Clone 1052
Reveal 1052
Shade 1056
Smear 1053
Stamp 1054
Warp 1055
Wash 1056
specular highlights, on 3D models 543
specular maps, in Action 608
speed
dissolves 1929
playback 1926
spell checking 1141
Splatter
generating patterns 1449
splice. *See* cuts
splice-in edits 268
splicing
by adding a shot to the Timeline 1956
clips on Desktop 303
shots 1924
splines
3D path, in Action 635
splines, creating in Distort 1165
split archive 240
split screens, drawing 2107
Stabilizer
correcting errors 881
gang option 866
importing and exporting data 890
jitter, removing 875
offsets 880
reference point, selecting 862
shift channels 889
shift values 860, 1431
smoothing camera motion 877
tracker and reference boxes 862
tracker box 882
tracker, selecting 863
Stamp medium 1054
Stereo 1437, 1438
Stereo menu settings 1438
Stereo node 1437
stereo tape
capture 186
output 198
stereoscopy
accessing features with Autodesk control surface 2482
assembling an EDL 2209
bypassing menu parameters 2215
capture from tape 186
creating a project 2206
editing 2213
grade files 2214
grading 2214
in Action 485
loading footage into timeline 2209
output to tape 198
rendering 2215

repositioning shots 2210
stereo camera 658
stereo footage in Action 485
stereo objects 486
viewing an eye 2211
Stone filesystem
playing clips located on 2316
storyboard
updating thumbnails 1980
Storyboard
adding a virtual clip 1931
adding shots 1915
collapsing 1983
deleting shots from 1916
duplicating shots in 1916
editing shots 1914
hot keys 2341
inserting cuts between scenes 1924
joining shots together 1925
navigating 1975, 1979
navigating using Timebar and Storyboard 1975
navigating with the Autodesk control surface 2425
navigating with the Tangent CP100 2488
refreshing thumbnails 1982
reordering shots 1915
replacing shots 1926
selecting shots 1980
selection colours 1982
setting view size 1979
showing and hiding 1979
sliding the cut point between shots 1923
trimming frames interactively 1922
viewing shot information 1917
strict matching 150
strokes in Paint node
applying 1013
blending with current image 1020
clearing 1005
clearing canvas 1022
hiding 1005
limiting with source matte 1006
strokes in Paint Node
limiting with matte 1014
Stylize 1440, 1441
Stylize menu settings 1441
Stylize node 1440
subsetups in Colour Warper
comparing 1000
naming 999
replacing 1000
saving and loading 1000
storing and retrieving 999
Substance Designer 579
Substance Noise node in Batch 1449
Substance Splatter node in Batch 1449
Substance textures, from Maya 579
Substance textures, in Action 579
Subtitle XML
about 142
importing 142
sample code 148
syntax 144
text setup files 142
support preferences 1678
supported file formats, in Lustre 1856
supported formats
MXF import codecs 1869
QuickTime import codecs 1870
suppressing
colours, in Colour Corrector 954
surfaces in Action
adding 457
bicubic 465
bilinear 465
changing order 480
swapping elements 1939
symphony 178
sync options 2312
synchronizing eyes in a stereoscopic project 2213
SynColor configuration file 1602
SynColorConfig element 1603
syntax
example 2239
for the Command Line Renderer 2237

T

tab stops, setting in Text 1140
table of contents
deleting 243
using to recover an archive 243
tablet preferences 1676
tail frames, rendering 2224
Tangent CP100
overview 2484
tangent handles
automatic adjustment 905
breaking and reconnecting in garbage masks 904
moving, in garbage masks 904
removing in garbage masks 904
scaling in garbage masks 904
selecting, in garbage masks 903
tangents
adjusting to reshape geometries 2098
breaking for animations 2188
hiding handles 2190
modifying 1483
resetting broken handles 2189
tape archive 231
tape backup 231
tape device 229, 231
tape device, archive clip 234
tape device, archive project 232
tape device, restoring clips 235
tape device, restoring project 233
telecine tape
emulating grading with 2290
temporal Degrain settings 2012

Text 1454
accessing 1124
adding text to a clip 1130
animating 1144
Asian character sets 1133
copying character channels 1138
creating text layers 1130
fonts, Asian 1133
fonts, using 1152
formatting in Text module 1135
grouping layers 1139
in Paint 1082
loading files 1129
loading logos 1139
modifying layer properties 1135
on-screen keyboard 1134
resizing 1138
safe title 1138
saving files 1128
setup files 142
spell checking 1141
tab stops 1140
text columns 1140
text motion paths, creating 1147
text rolls and crawls 1141
text styles, defining 1150
Text menu settings 1454
Text node 1454
text objects, in Paint 1082
texture maps, using in FBX 541
textures
applying in Action 578
particles, applying 761
projecting in Action 627
Substance Designer 579
Substance presets in Action 579
using Maya textures 579
thumbnails
refreshing in the Storyboard 1982
updating in Storyboard 1980
timecode
burning into render files 2223
calculating 1977
editing in EDLs 175
modifying the start value 1964
viewing information 1917
timeline
containers, creating 285
cue marks, adding 285
cuts, creating 287
gaps 286
patching 282
preferences 1678
segment marks, adding 285
Timeline
accessing the Timeline menu 1932
adding a dissolve in 1957
adding black media to 1940
adding shots 1936
adding shots to 1936
changing orientation of a dissolve in 1958
checking against an offline cut 1965
colourist 2054
copying elements in 1939
creating 1936
deleting elements in 1941
deleting shots from 1916
displaying Multi-Layer Timeline 1958
dissolves 1957
editing 1936
editing a dissolve in 1957
gaps, overview 1941
modifying the starting timecode 1964
moving elements in 1938
navigation controls 1934
removing a dissolve in 1957
rendering 1964
searching 283
selecting elements in 1937
sliding an element 1954
slip-and-slide Ripple Mode 1955
slipping an element 1954
slipping-and-sliding an element 1955
snapping elements in 1937
sort mode 1990
splices 1956
swapping elements in 1939
transitions 1956
trimming cursors 1953
trimming with Ripple Mode 1953
view in Lustre 1932
viewing element information 1935
timeline effects preferences 1679
timeline elements
extracting 289
lifting 289
swapping 288
Timeline FX 329
Timeline menu, accessing 1932
timeline preferences 1678
Timeline sort mode 1990
A-mode 1991
accessing Timeline sort mode 1990
C-mode 1993
hot keys 1993, 1995
Timeline transitions 315
timewarps
in EDLs 177
timing
animation curves 1476
timing, adjusting for animations 2201
tolerance
in Colour Warper 978
setting for keys, Diamond Keyer 2129
setting range for keys, HLS Keyer 2121
Tools bin, customizing 396
Top Vertical Priority. *See* Multi-Layer Timeline
Tracer
advanced gradient mode 916
animating 923

animating pickers 924
defined 892
matte, fine-tuning 922
pickers, defined 918
pulling keys 916
troubleshooting 926
when to use 916
track
adding to timeline 281
deleting 281
moving 281
resizing 281
tracking
animating garbage masks 911
animating selected vertices on garbage masks 912
by axis, in Distort 1181
by axis, in Warper 1211
by vertices, in Distort 1182
by vertices, in Warper 1212
geometries 2150
in Paint 1067
repositioned shots 2022
Tracking with Masks 324
tracks
on timeline 280
transcode
colour settings 1882
format settings 1879
output settings 1887
template 1891
workflow 1877
transforming
garbage masks 903
objects in Paint 1091
particles using manipulators 768
text 1144
transitions
in EDLs 174
triangles, painting 1038
trimming
cursors in the Timeline 1953
elements 1952
frames in the timeline 1917
head or tail frames 1918
in and out points 1922
interactively 1922
on timeline 273
shots for dissolves 1928
with Ripple Mode in the Timeline 1953
troubleshoot 152
troubleshooting, problem dust artefacts 2003

U

UI overview 15
UMID 72, 94
undoing
actions 1805
in animation 2206
with the Autodesk control surface 2439
with the Tangent CP100 2489
unlocking clip duration, in Paint 1027
updating Storyboard thumbnails 1980
user interface
differences in Logarithmic and Linear mode 1786
preferences 1681
setting appearance of 1789
user interface preferences 1681
users, creating 1755
Using a Sparks Plug-in 1427
Using Sparks in Batch 1429
UV maps, in Action 624

V

vectors, in expressions 1501
vectorscope, in Colour Warper 966
version
adding to timeline 281
versioning, for film and video 2297
versions
on timeline 280
vertices in garbage masks
activating and deactivating 909
adding and removing 909
adjusting number drawn freehand 901
animating 907
applying tracking data 912
deleting 911
moving 904
selecting 903
setting keyframing options 908
video
See also clip I/O
capture and playout, overview 2257
capturing from timecode 2267
capturing with timecode breaks 2270
formats, supported 2298
outputting multiple versions 2297
video I/O timing
setting initial 180
view modes
hot keys 2342
setting 1973
view options
accessing with the Autodesk control surface 2430
accessing with the Tangent CP100 2486
viewing
animated curves 2174
channel ranges between keyframes 2174
multiple shots in the player 1984
multiple shots on the SDI monitor 1989
selected animated channels 2174
selected keyframes 2174
stereoscopic footage 2211
viewing LUTs
rendering with 2224
selecting 1975

viewports, displaying 262
virtual clips, Deliverables 214
vtr 237
VTR
See also clip I/O
capturing from 2264
controlling for video I/O 2262
emulation 2289
inputting clips 181
outputting clips 187
playing out to 2280
transport controls 204
VTR archive 235
vtr configuration 236
VTR Output module 187
vtr-based archive 236

W

Warp medium 1055
Warper
adding a vertex 1207
animation curves 1218
creating a freehand mesh 1208
keyframes, setting manually 1218
loading clips 1199
locking the back clip 1216
Magnet Curve Editor 1209
meshes, creating 1201
modelling controls 1204
morphing one image into another 1216
patches, adding 1203
proportional mesh 1203
source and destination meshes 1202
source interpolation 1217
subdividing a patch 1207
tangent handles 1204
tracking and stabilizing 1211
tracking by vertices 1212
using the magnet 1209
vertices, moving 1204
warping a still clip 1215
warping and morphing, defined 1199, 1466
warping
colours, in Colour Warper 996
with Distort 1157
with the Warper 1199, 1466
Wash medium 1056
what is 227
What's New, new features 1
white level, modifying for input LUTs 2034
Wipe command
in AutoPaint 1064
in Paint 1057
wipes
in EDLs 174
masks, creating 930
masks, drawing options 930
wiping 1023
wire 139, 141
wireframe
converting, in Action 502
display options 1086
in garbage masks 897
Wiretap
caching degrained frames 2014
importing audio 2308
playing clips located on Stone FS 2316
rendering 2316
rendering options 2336
setting up projects for soft-imported clips 2326
wiring clips 141
workflows
audio 2306
creating animations 2176
Degrain 2008
EDL and keycode assembly 1894
integrating Lustre into post-production 2321
Lustre general 1701
monitor calibration 1847
playout 2281
processing pipeline 1701

X

XDCAM MXF
See MXF
xml 179
XML
CUT file data types 2376
CUT file example 2377
limitations 2362
modifying CUT files 2362
modifying Gateway clips 1683
subtitle 142
viewing and editing files 2362
XML declaration 1566

Y

YUV material, inputting and outputting 209

Z

Z-buffer
setting in Action 482
zooming
animation curves 2174
in Paint 1033
in Player and full-screen Player 1972
with the Autodesk control surface 2429
with the calculator 1819