

Autodesk®
Lustre® 2009

A Discreet® Systems product

User Guide



Autodesk®

© 2008 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 relating to ADOdb Copyright © 2000, 2001, 2002, 2003, 2004 John Lim. All rights reserved. Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met: Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. Neither the name of the John Lim nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission. **DISCLAIMER: THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL JOHN LIM OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.**

Portions relating to GetOpt Copyright © 1987, 1993, 1994. The Regents of the University of California. All rights reserved. Redistribution and use in source and binary forms, with or without modification, are permitted provided that the following conditions are met: 1. Redistributions of source code must retain the above copyright notice, this list of conditions and the following disclaimer. 2. Redistributions in binary form must reproduce the above copyright notice, this list of conditions and the following disclaimer in the documentation and/or other materials provided with the distribution. 3. All advertising materials mentioning features or use of this software must display the following acknowledgement: This product includes software developed by the University of California, Berkeley and its contributors. 4. Neither the name of the University nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission. **THIS SOFTWARE IS PROVIDED BY THE REGENTS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED. IN NO EVENT SHALL THE REGENTS OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND ON ANY THEORY OF LIABILITY, WHETHER IN CONTRACT, STRICT LIABILITY, OR TORT (INCLUDING NEGLIGENCE OR OTHERWISE) ARISING IN ANY WAY OUT OF THE USE OF THIS SOFTWARE, EVEN IF ADVISED OF THE POSSIBILITY OF SUCH DAMAGE.**

Portions relating to GetOpt Copyright ©1999, 2000 Politecnico di Torino. All rights reserved. Redistribution and use in source and binary forms, with or without modification, are permitted provided that: (1) source code distributions retain the above copyright notice and this paragraph in its entirety; (2) distributions including binary code include the above copyright notice and this paragraph in its entirety in the documentation or other materials provided with the distribution, and (3) all advertising materials mentioning features or use of this software display the following acknowledgement: "This product includes software developed by the Politecnico di Torino, and its contributors." Neither the name of the University nor the names of its contributors may be used to endorse or promote products derived from this software without specific prior written permission. **THIS SOFTWARE IS PROVIDED "AS IS" AND WITHOUT ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, WITHOUT LIMITATION, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE.**

Portions relating to LibXML2 2.6.20 Copyright © 2005 Daniel Veillard. Permission is hereby granted, free of charge, to any person obtaining a copy of this software and associated documentation files (the "Software"), to deal in the Software without restriction, including without limitation the rights to use, copy, modify, merge, publish, distribute, sublicense, and/or sell copies of the Software, and to permit persons to whom the Software is furnished to do so, subject to the following conditions: The above copyright notice and this permission notice shall be included in all copies or substantial portions of the Software. **THE SOFTWARE IS PROVIDED "AS IS", WITHOUT WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO THE WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE AND NONINFRINGEMENT. IN NO EVENT SHALL THE AUTHORS OR COPYRIGHT HOLDERS BE LIABLE FOR ANY CLAIM, DAMAGES OR OTHER LIABILITY, WHETHER IN AN ACTION OF CONTRACT, TORT OR OTHERWISE, ARISING FROM, OUT OF OR IN CONNECTION WITH THE SOFTWARE OR THE USE OR OTHER DEALINGS IN THE SOFTWARE.**

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

The following are registered trademarks or trademarks of Autodesk Canada Co. in the USA and/or Canada and other countries: Backburner, Discreet, Fire, Flame, Flint, Frost, Inferno, Multi-Master Editing, River, Smoke, Sparks, Stone, and Wire.

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

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.

Images courtesy of:	Hungarian Academy of Film & Theatre, 3rd year, Moviworld/UK Film & TV Company/Videolab, Technicolor (formerly Toybox), The House
Title:	Autodesk Lustre 2009 User Guide
Document Version:	1
Date:	October 10, 2008

contents

Contents

1	Introduction	1
	Summary	1
	Welcome to Lustre	1
	Lustre Configurations	2
	Related Documentation	2
	Notational Conventions	5
	Contacting Customer Support	6
2	Lustre Workflow	7
	Summary	7
	How to Use this Chapter	7
	Image Processing Pipeline	8
	Setting Up Users and Projects	8
	Monitor Calibration	8
	Applying Print LUTs	9
	Editing	9
	Rendering Proxies	9
	Removing Dust	9
	Repositioning Shots	10
	Applying Input LUTs	10
	Colour Grading	10
	Using Sparks	11
	Animating Colour Effects	11
	Outputting the Result	11

3	Project Management	13
	Summary	13
	About Project Management	13
	Configuring System Settings	14
	Project Configuration	15
	Project Configuration Settings	20
	User Configuration	33
	User Configuration Settings	37
	Loading Users and Projects	45
	Working With Scenes	47
	Working With Cuts	49
	Working With Grades	54
	Setting Up the User Interface	67
	Working with Templates	68
	Recommended Directory Structure for Projects	78
4	Basics	85
	Summary	85
	Overview of Basic Functionality for Lustre	85
	Starting and Exiting Lustre	85
	Understanding the Lustre User Interface	86
	Working with Control Surfaces	88
	Resetting Parameters	89
	Bypassing Menu Parameters	90
	Undoing and Redoing Actions	91
	Confirming and Cancelling Actions	92
	Assigning Notes to Shots	93
	Flagging Shots	94
	Creating Shot Groups	97
	Copying Parameters with the Selector	101
	Assigning Values With the Standard Lustre Calculator	103
	Grade Bins: Saving, Applying, and Deleting Intermediary Grades	105
	Using Audio	113
	Using Text	113
5	Monitor Calibration	117
	Summary	117
	About Monitor Calibration	117
	Adjusting Black Level and Picture Settings	121

Selecting the Project Colour Space.....	123
Calibrating a Monitor.....	123
Calibrating for Dual-Head Mode.....	127
Applying Print LUTs for Viewing.....	127
Applying 3D Mesh LUTs.....	129
6 Browsing for Footage	131
Summary.....	131
About Browsing for Footage.....	131
Accessing the File Browser.....	131
Loading Shots into the Shot Bin.....	134
Viewing Media File Information.....	139
Matching Shots.....	142
Working with EDL, ALE, and Cutlist Files.....	143
Colour Decision List (CDL).....	158
7 Editing	165
Summary.....	165
About Editing.....	165
Accessing the Edit Menu.....	166
Editing with the Storyboard.....	168
Trimming from the Edit Menu.....	171
Changing Playback Speed.....	182
Replacing Shots.....	183
Creating Dissolves.....	184
Adding a Virtual Black Clip using the Storyboard.....	187
Multi-Layer Timeline.....	188
Editing in the Timeline.....	193
Trimming Elements.....	198
Transitions.....	202
Displaying the Multi-Layer Timeline.....	205
Rendering the Timeline.....	212
Modifying the Timeline's Starting Record Timecode.....	212
Performing a Confidence Check.....	213
Using Scene Detection.....	215

8	Playing, Viewing and Sorting Shots	219
	Summary	219
	About Playing, Viewing, and Sorting	219
	Playing Shots	219
	Panning and Zooming the Image.....	222
	Setting Image View Options	222
	Navigating through Shots	226
	Storyboard Viewing Options	230
	Viewing Reference Images.....	236
	Viewing Multiple Shots in the Player.....	238
	Timeline Sort Mode	242
	Caching Frames to Memory for Real-Time Playback.....	249
	Player Hot Keys.....	250
9	Removing Dust	253
	Summary	253
	About Dust Removal	253
	Preparing for Dust Removal	254
	Removing Dust Manually	257
	Removing Dust Automatically.....	258
	Single Frame Dust Removal	260
	Working with Boxes.....	261
	Infrared Channel Dust Detection.....	262
	Options for Saving and Loading Dust Removal Metadata.....	263
	Rendering the Result	264
10	Repositioning Images	267
	Summary	267
	About Repositioning	267
	Adjusting the Frame Aspect Ratio	269
	Reframing an Image	270
	Flipping or Flopping a Shot.....	272
	Tracking Repositioned Shots	273
	Animating Reposition Values.....	276
	Setting Rendering Quality.....	277

11	Input LUTs	279
	Summary	279
	About LUTs	279
	Accessing the Input LUT Menu	280
	Applying an Existing Input LUT	280
	Linear Mode: Creating Conversion LUTs.	281
12	Colour Grading: Basics	291
	Summary	291
	About Colour Grading.	291
	Accessing Colour Grading Menus	294
	GPU Acceleration	295
	Control Surfaces	299
	Viewing Multiple Shots	299
	Reference Images	299
	Viewing Proxies.	300
	Analysing the Image.	300
	Saving and Loading Presets Using the Presets Lists	303
	Colour Grade Animation.	306
	Grade Bin Storage	307
	Colourist Multi-Layer Timeline	307
13	Primary Colour Grading	313
	Summary	313
	About Primary Grading.	313
	Accessing the Grading Menu	314
	Logarithmic Mode: Modifying the Brightness	315
	Logarithmic Mode: Improving Contrast	320
	Linear Mode: Selecting Levels of the Image	323
	Linear Mode: Modifying Lift.	324
	Linear Mode: Modifying Gain	326
	Linear Mode: Modifying Gamma.	326
	Linear Mode: Modifying Saturation	327
	Linear Mode: Modifying Contrast	328
	Linear Mode: Setting Black Clip and White Clip.	329
	Matching Colours	330
	Autograding Shots	331

14	Colour Grading: RGB and Hue Curves	333
	Summary	333
	About RGB and Hue Curves	333
	Accessing the Curves Menu	334
	Plotting Colour Values when Adjusting Curves	334
	Modifying Red, Green, and Blue Channels	336
	Modifying Hue, Saturation, Luminance, and Lightness	337
15	Secondary Colour Grading	345
	Summary	345
	About Secondary Colour Grading	345
	Accessing Secondary Grading Menus	346
	Creating a Secondary by Keying a Range of Colours	347
	Saving and Loading Key Presets	359
	Importing Mattes	360
	About Geometries	361
	Accessing the Geometry Controls	362
	Drawing Basic Geometries	363
	Drawing Free-Form Geometries	364
	Modifying Geometries	368
	Adding a Split Screen	376
	Adding and Deleting Axes	377
	Changing the Priority Order of Geometries	380
	Showing and Hiding Geometries	381
	Inverting Geometries	381
	Deleting Geometries and Resetting Secondaries	382
	Linking Geometries across Secondary Layers	383
	Shape Tracker	385
	Animating Geometries with the Point Tracker	389
	Loading Tracker and Stabilizer Data from Autodesk Applications	400
	Loading Garbage Mask Setups from Autodesk Applications	401
	Colour Grading Secondaries	402
16	Creating Lustre Sparks Effects	407
	Summary	407
	About Lustre Sparks Plugins	407
	Installing Lustre Sparks Plugins	408
	Licensing Lustre Sparks Plugins	408
	Creating an Effect with a Lustre Sparks Plugin	408

Lustre Sparks Plugin Descriptions	411
Disabling and Re-enabling Lustre Sparks Plugins	420
Developing Lustre Sparks Plugins	421
17 Animation	423
Summary	423
About Animation	423
Accessing the Channel Editor	424
Animating with Keyframes	428
Editing Keyframes	438
Copying and Pasting Keyframes	449
The Track Editor	456
Animating in Other Menus	460
Specifying Setup Options	462
Undoing and Redoing Operations	464
18 Rendering	465
Summary	465
About Rendering in Lustre	465
Rendering Shots	466
Saving and Loading Colour Mapping Presets	482
Generating and Viewing Proxies	484
Locking Colour Grading	486
Cleaning Up Render Files	487
Rendering Shots as You Work	488
Rendering with Burn for Lustre	490
19 Stereoscopy	499
Summary	499
About Stereoscopy	499
Creating a Stereoscopic Project	499
Viewing From One Eye to the Other	503
Colour Grading the Left and Right Eye	504
Rendering Stereoscopic Footage	505

20	Video Capture and Video Playout	507
	Summary	507
	About Capture and Playout.	507
	Capture Menu Options	508
	Controlling the VTR.	512
	Capturing Material	514
	Playout Menu Options.	525
	Playing Out to a VTR	529
	Writing Telecine-Style Tape	541
	Playout and GPU Acceleration.	541
	About 3:2 Pulldown	548
	Converting Logarithmic to Linear	549
	Supported Video SDI and GFX SDI Rasters	550
21	Audio	557
	Summary	557
	About Audio	557
	Audio Workflow	558
	Importing an Audio File	558
	Capturing an Audio File	559
	Playing Back Audio with the Timeline	559
	Playback Using the LTC Chase Option	561
	Playout of Audio and Video	562
22	Working with Wiretap	563
	Summary	563
	About Lustre and Wiretap.	563
	Before You Begin	564
	Supported Filesystems and Media Types	566
	Sample Workflows	568
	Preparing the Wiretap Timeline for Lustre	570
	Configuring Wiretap and Lustre	570
	Setting Up the Lustre Wiretap Project.	574
	Browsing for Remote Footage.	576
	Importing the Wiretap Timeline	580
	Working with Soft-Imported Media	580
	Rendering Shots	582
	Troubleshooting	588

23	Hot Keys	591
	Summary	591
	Storyboard Navigation.....	591
	View Modes	592
	Timeline Sort Modes	594
	Notes	594
	Calculators	594
	Dust Removal	595
	Editing	595
	Colour Grading.....	598
	Numeric Keypad.....	602
	Stereoscopy	605
	Animation	605
	Miscellaneous Functions	605
 A	 XML in Lustre	 607
	Summary	607
	Lustre XML-Based Files.....	607
	Limitations.....	607
	CUT File Tags	608
	CUT File Data Types	617
	CUT File Example.....	618
 	 Index	 619

1

Introduction

Summary

Welcome to Lustre	1
Lustre Configurations	2
Related Documentation	2
Notational Conventions	5
Contacting Customer Support	6

Welcome to Lustre

Autodesk® Lustre® is a scene-based digital colour grading solution that provides an intuitive environment for matching colour across shots and for creating colour-based effects on film and high-quality HD or SD video source material.

Lustre enables complete control of the final colour of your media. It ensures consistency between shots by providing a single means of grading footage originating from diverse sources, such as live-action shots, CG sources, stock footage, and photographic plates. It eliminates the need to use filters to adjust for different lighting conditions when shooting at different times on a location.

Complete with media acquisition capabilities that allow you to digitize from a VTR, you can auto-conform an EDL or manually build a timeline with a full range of editing and trimming tools.

You can use the primary colour tools in Lustre to create a particular look or mood for a scene. For example, desaturate a scene to create a monochromatic look, or apply a subtle orange cast to add a sense of warmth. You can also use the secondary colour grading and tracking tools to isolate and emphasize a part of the image. For example, selectively adjust the brightness to direct the attention of the audience to a particular part of the scene.

Further enhance the look of your media with image repositioning controls for adjusting the pan, scan, scale, rotation and aspect ratio. Use Sparks® to create assorted effects. Dust removal tools make it easy to automatically clean up unwanted scratches, hair, and dust.

Film and video footage can be graded in Lustre in either logarithmic or linear colour space. You can also output using a selected LUT or create one. The output from Lustre is a high-quality, graded digital data master that can be used to prepare the digital negative or digital projection masters. You can also master to HD and SD video, DVD, or the Internet.

Lustre Configurations

When you purchase Lustre, you can choose from several configurations, depending on the type of workstation you want and the options you require. The following table describes each Lustre configuration.

Station	Configuration
Master Station	Default — All features are available including the DI Pack, which consists of infrared channel dust removal, input resolutions greater than 2040x1556, output resolutions greater than 1920x1080, and bit depths greater than 10-bit. However, add-ons may be required to enable certain features. In addition, SD and HD I/O, as well as dual link and HSDL video formats, are default for the Master Station. Add-Ons — The following features can be added to the Default configuration: the Slave Renderer, and up to three panels for the Autodesk control surface. The Slave Renderer requires a separate license.
Lustre Station	Default — All features are available except for primary and secondary colour grading. The DI Pack, explained above, is also included, along with full dust removal functionality, and the ability to create geometries and masks. With Primary Colour Correction — Includes all features of the default option as well as primary colour grading. Add-Ons — The following features are available for either the Default configuration or the With Primary Colour Correction configuration, and require an additional license: SD and HD I/O, dual link and HSDL video formats, up to three panels for the Autodesk control surface, and the Slave Renderer.
Lustre HD Station	Default — Most features are available, although support is not available for the DI Pack (explained above). Certain features require add-on licensing in order to be enabled. Add-Ons — The following features can be added to the Default configuration: SD and HD I/O, dual link and HSDL video formats, the Slave Renderer, and up to three panels for the Autodesk control surface. The Slave Renderer requires a separate license.

Related Documentation

The following table describes the documentation associated with your application. For a detailed list of the latest documentation, see your release notes.

User Guides	Provides
<i>Autodesk Lustre 2009 User Guide</i>	Detailed instructions on using the software.
<i>Autodesk Control Surface User Guide</i>	Detailed instructions on using the Autodesk control surface and the Tangent CP100.
<i>Autodesk Lustre 2009 New Features Guide</i>	Information about the new features for this release.
<i>Autodesk Lustre 2009 Hot Keys Card</i>	A list of the most frequently used hot keys.
<i>Autodesk Lustre 2009 Release Notes</i>	A complete list of documentation and information on late-breaking features.
<i>Autodesk Lustre 2009 Fixed and Known Bug List</i>	A complete list of fixed and known bugs for this release.
Installation and Configuration Guides	Provides
<i>Hardware Setup Guide</i> (for your workstation)	Information on how to setup your workstation and video I/O peripherals.
<i>Stone Direct Configuration Guide</i>	Provides detailed connectivity diagrams and configuration procedures for your Autodesk Stone® storage arrays.
<i>Stone and Wire Filesystem and Networking Guide</i>	Procedures for configuring your Autodesk Wiretap™ services.
<i>Autodesk Lustre 2009 Software Installation Guide</i> (for your operating system)	Information about installing and licensing your Autodesk Lustre software.
Other Guides	Provides
<i>Autodesk Lustre Sparks API Reference Guide</i>	Instructions for developing Sparks plugins for Lustre.
<i>Autodesk Backburner 2008.1 Installation Guide</i> <i>Autodesk Backburner 2008.1 User Guide</i>	Information on how to install, set up, and use Autodesk Backburner™.

Consult the Autodesk Web site at www.autodesk.com/lustre-documentation for the latest version of guides, release notes, and fixed and known bugs documents.

Accessing Online Help

Autodesk provides complete documentation in an accessible HTML help system that is displayed in a Web browser. The Help is automatically installed unless specified otherwise during the software installation.

You can install the Help on another system without installing Lustre. For Windows®, select Online Help only when prompted to select components in the Lustre Installer. For Linux®, you can copy the *Documentation/help* directory from the CD onto another workstation.

To start the help system from Lustre:

- Click the Help button, located in the lower-right corner of all menus, or press **SHIFT+F1**.

**To start the help system from the desktop:**

1. If using the Windows version of Lustre, select Start | Programs | Autodesk | lustre | Online Help from the Windows task bar.

The Help appears in a browser window.

2. If using the Linux version of Lustre, open a shell and type:

```
cd /usr/Autodesk/lustre/online_help2009/online_help
```

To copy the Help to another system:

1. Copy the *Documentation/help* directory from the software CD-ROM to the new location on another system.
2. To start the Help after you copy the help directory, open the *help/html/_start_helpsystem.html* file.

Accessing PDF Documentation

The complete documentation set is available in PDF (Portable Document Format) for online viewing and printing. On Windows systems, use Adobe® Acrobat® Reader™ to view and print the PDF files. On Linux workstations, use either Adobe Acrobat Reader or the Xpdf viewer. You can access the PDF files from the Lustre software CD or from www.autodesk.com/lustre-documentation.

NOTE: If you do not have Acrobat Reader, you can download a free copy from the Adobe Web site (www.adobe.com). If you do not have Xpdf viewer, you can download a free copy from the Xpdf Web site (www.foolabs.com/xpdf/).

From the Lustre Application CD

You can view and print the PDF files from the Lustre CD using Adobe Acrobat Reader. All PDF files are located in a directory called *Documentation* at the top level of the CD.

To view the PDF files on the application CD:

1. Place the Lustre CD in the CD-ROM drive of your system.
2. If using the Windows version of Lustre, in Windows Explorer, go to the *Documentation* folder on the CD.
3. If using the Linux version of Lustre, open a Linux shell and navigate to the *Documentation* folder on the CD. Type:

```
/mnt/cdrom/documentation
```

4. To view one of the documentation PDF files, do one of the following:
 - If using the Windows version of Lustre, double-click the desired file.
 - If using the Linux version of Lustre, type:

```
xpdf <filename>
```

The file opens in Acrobat Reader or the Xpdf viewer.

Accessing PDF Documentation from the Autodesk Web Site

You can also find the PDF files on the Autodesk Web site, www.autodesk.com/lustre-documentation, and download any of the available files.

Notational Conventions

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

Convention	Example
Text that you enter in a command line or shell appears in Courier bold. You must press the Enter key after each command.	xpm -qa
Variable names appear in Courier, enclosed in angle brackets.	<filename>
Feedback from the command line or shell appears in Courier.	limit coredumpsize
Directory names, filenames, URLs, and command line utilities appear in italics.	<i>/usr/autodesk</i>

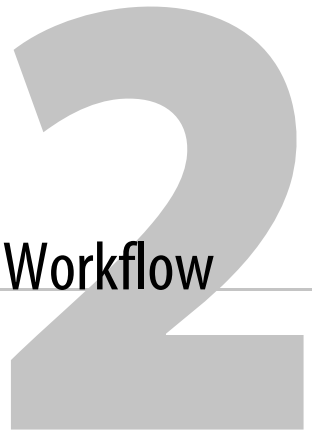
NOTE: Throughout this guide, when referring to Windows-specific file locations, the term *folder* will be used. When referring to Linux-specific file locations, the term *directory* will be used. When referring to both, *directory* will be used.

Contacting Customer Support

A list of contact information for Autodesk Media and Entertainment Customer Support is available at www.autodesk.com/support.

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

Lustre Workflow



Summary

<i>How to Use this Chapter</i>	7
<i>Image Processing Pipeline</i>	8
<i>Setting Up Users and Projects</i>	8
<i>Monitor Calibration</i>	8
<i>Applying Print LUTs</i>	9
<i>Editing</i>	9
<i>Rendering Proxies</i>	9
<i>Removing Dust</i>	9
<i>Repositioning Shots</i>	10
<i>Applying Input LUTs</i>	10
<i>Colour Grading</i>	10
<i>Using Sparks</i>	11
<i>Animating Colour Effects</i>	11
<i>Outputting the Result</i>	11

How to Use this Chapter

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 [Chapter 3, “Project Management,”](#) on page 13.

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 [Chapter 5, “Monitor Calibration,”](#) on page 117.

Applying Print LUTs

For film print simulation, you can apply Print LUTs, which are updated regularly by your film lab. See [“Applying Print LUTs for Viewing”](#) on page 127.

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 Shots into the Shot Bin”](#) on page 134.
- Assembling EDLs in the timeline. See [“Working with EDL, ALE, and Cutlist Files”](#) on page 143.
- Editing shots in a multi-layer Timeline. See [“Editing in the Timeline”](#) on page 193.
- Capturing footage from a VTR. See [“Capturing Material”](#) on page 514.
- Performing basic gestural edits such as trimming, slipping, and sliding shots, as well as creating dissolves. See [Chapter 7, “Editing,”](#) on page 165.

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”](#) on page 484.

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 [Chapter 9, “Removing Dust,”](#) on page 253.

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 [Chapter 10, “Repositioning Images,”](#) on page 267.

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”](#) on page 280.
- 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”](#) on page 281.

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 [Chapter 13, “Primary Colour Grading,”](#) on page 313.
- Modifying RGB and Hue curves to colour grade specific ranges of colour without using keys. See [Chapter 14, “Colour Grading: RGB and Hue Curves,”](#) on page 333.
- Performing secondary colour grading to specific ranges of colour or areas in an image through keys and geometries. See [“Colour Grading Secondaries”](#) on page 402.
- Generating keys for secondary colour grading. See [“Creating a Secondary by Keying a Range of Colours”](#) on page 347.
- Creating and tracking geometries for secondary colour grading. See [“About Geometries”](#) on page 361 and [“Animating Geometries with the Point Tracker”](#) on page 389.

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 [Chapter 16, “Creating Lustre Sparks Effects,”](#) on page 407.

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 [Chapter 17, “Animation,”](#) on page 423.

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”](#) on page 478.
- Rendering a print of the final grade. See [Chapter 18, “Rendering,”](#) on page 465.
- Outputting rendered files to a VTR or any other device. See [“Playing Out to a VTR”](#) on page 529.

Project Management

Summary

About Project Management	13
Configuring System Settings	14
Project Configuration	15
Project Configuration Settings	20
User Configuration	33
User Configuration Settings	37
Loading Users and Projects	45
Working With Scenes	47
Working With Cuts	49
Working With Grades	54
Setting Up the User Interface	67
Working with Templates	68
Recommended Directory Structure for Projects	78

About 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 specific sequence of shots (or a scene) within the project.
- Cut: a group of shots that get rendered

- Grade: the operations performed on a cut.



WARNING: In Lustre, you must create a scene before starting to do any work (importing, capturing or creating data), or you may lose important data.

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”](#) on page 14.
2. Create and set up the project. See [“Project Configuration”](#) on page 15.
3. Create and set up the users for the project. See [“User Configuration”](#) on page 33.
4. Create scenes. See [“Working With Scenes”](#) on page 47.
5. Organize the footage in cuts. See [“Working With Cuts”](#) on page 49.
6. Create grades. See [“Working With Grades”](#) on page 54.

Incinerator-Specific Project Management Configuration

Lustre with Incinerator in-line processing use the same project management concepts. The only difference is that in order to use Incinerator on the Lustre workstation, the Scans Full Home must point to the Gigabit Ethernet (GigE) 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

After Lustre is installed but before launching it for the first time, you must configure system-level settings in the *init.config* file.

You only define these system settings once, unless your system workstation configuration changes. For more information about the *init.config* file, including default settings, refer to the “Software, Project, and User Configuration Files” chapter in the *Lustre Software Installation Guide*.

To configure 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\Lustre2009
Linux	/usr/autodesk/lustre2009

2. Edit the *init.config* file as required to define:
 - BrowseD. See the *Configuring BrowseD* section in the Lustre installation guide for your operating system.
 - Wiretap.
 - Calibration settings
 - Autodesk Control Panel
 - Slave Render
 - Backburner
 - Internal keywords
 - DPX keycode emulsion list
3. Save your changes and exit the text editor.

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 to 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”](#) on page 78.

You can configure the following default settings for your project:

- Project
- Calibration
- Rendering
- Engineering
- Network Rendering

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 the project is loaded, regardless of the logged in user.

While logged in, the user can override some of the project default settings, such as Render Mode in the Local menu. These overrides only last for the duration of the session. Restarting the application resets the settings to the values set at project creation.

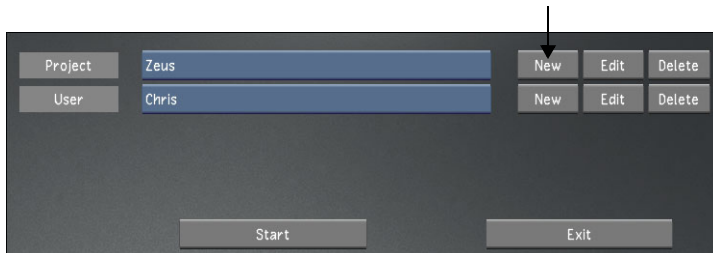
Creating a Project

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”](#) on page 68.

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 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.



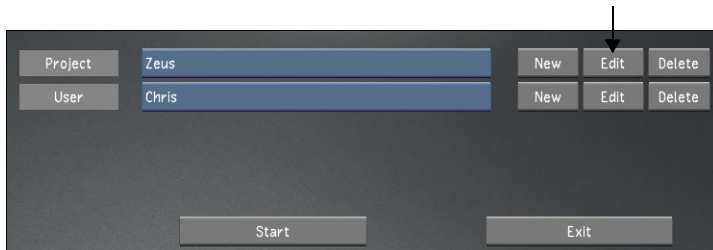
2. Configure the project. Use Reset to reset all settings, from every menu, to their defaults.

Select:	To configure:
Project	The name and location of the project. See "Project menu" on page 20.
Calibration	Default system calibration settings and define the default Print LUTs for a project. See "Calibration menu" on page 22.
Rendering	Default rendering settings for a project. See "Rendering menu" on page 24.
Engineering	Default video and graphics settings for a project. See "Engineering menu" on page 27.
Network Rendering	Default Backburner/Burn, Slave Render, and Wiretap directory paths and settings for a project. See "Network Rendering menu" on page 31.

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 Main menu, click Setup, and then Settings. Select the baseline project, and then click Edit in the Project group.
 - From the Lustre splash screen, 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.

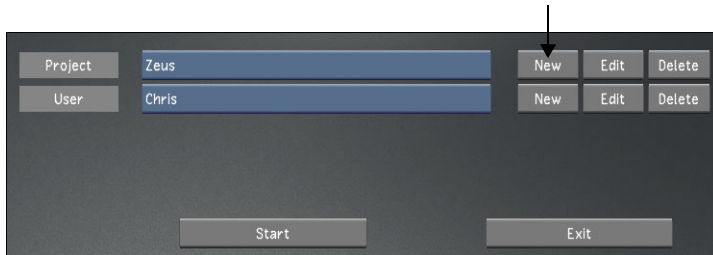
Select:	To configure:
Project	The name and location of the project. See "Project menu" on page 20.
Calibration	Default system calibration settings and define the default Print LUTs for a project. See "Calibration menu" on page 22.
Rendering	Default rendering settings for a project. See "Rendering menu" on page 24.

Select:	To configure:
Engineering	Default video and graphics settings for a project. See "Engineering menu" on page 27.
Network Rendering	Default Backburner/Burn, Slave Render, and Wiretap directory paths and settings for a project. See "Network Rendering menu" on page 31.

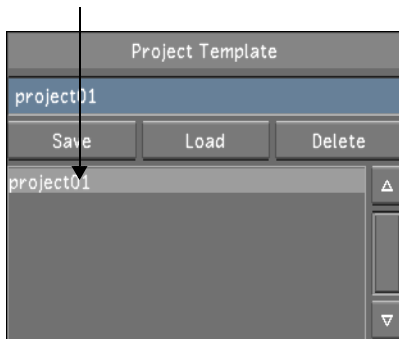
- To save the settings for the project, click Save Project.
- Click Exit Project.

To create a new project using a template:

- 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.



- From the Project Template list, select a template. See ["Creating a Project Template"](#) on page 69.



- 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.

Select:	To configure:
Project	The name and location of the project. See "Project menu" on page 20.
Calibration	Default system calibration settings and define the default Print LUTs for a project. See "Calibration menu" on page 22.
Rendering	Default rendering settings for a project. See "Rendering menu" on page 24.
Engineering	Default video and graphics settings for a project. See "Engineering menu" on page 27.
Network Rendering	Default Backburner/Burn, Slave Render, and Wiretap directory paths and settings for a project. See "Network Rendering menu" on page 31.

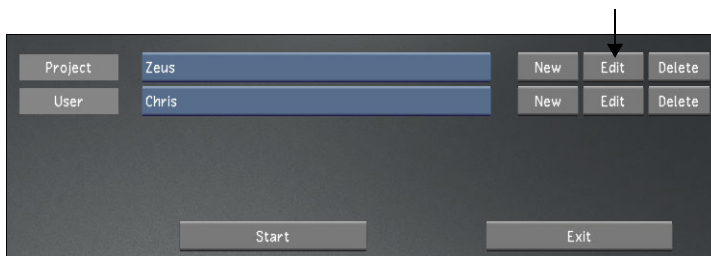
5. To save the settings for the project, click Save Project.
6. Click Exit Project.

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.

Select:	To configure:
Project	The name and location of the project. See "Project menu" on page 20.
Calibration	Default system calibration settings and define the default Print LUTs for a project. See "Calibration menu" on page 22.

Select:	To configure:
Rendering	Default rendering settings for a project. See "Rendering menu" on page 24.
Engineering	Default video and graphics settings for a project. See "Engineering menu" on page 27.
Network Rendering	Default Backburner/Burn, Slave Render, and Wiretap directory paths and settings for a project. See "Network Rendering menu" on page 31.

3. To save the settings for the project, click Save Project.
4. Click Exit Project.

Project Configuration Settings

You have to configure default project settings. Project settings are distributed among 5 menus. Navigate through the Project Configuration by clicking the following buttons:

Project Configuration settings are distributed among five menus.

Project button — Select to configure project name and location, scan and render directories, architecture, scan format, and frame rate settings. See ["Project menu"](#) on page 20.

Calibration button — Select to configure monitor calibration settings and type, and define print LUTs. See ["Calibration menu"](#) on page 22.

Rendering button — Select to configure how render files are organized, set the render output format, configure render options, set the proxy generation size and configure proxy and reposition filters. See ["Rendering menu"](#) on page 24.

Engineering button — Select to configure video device SDI settings, GFX SDI settings, sync mode settings, and video capture and playout settings. See ["Engineering menu"](#) on page 27.

Network Rendering button — Select to configure Backburner/Burn, Slave Render, and Wiretap settings. See ["Network Rendering menu"](#) on page 31.

Project menu

Use the Project menu to configure:

- The project name
- The location directory
- The scan directory
- The render directory
- The architecture

- The scan format
- The frame rate

For the project's directory structure, follow the guidelines described in [“Recommended Directory Structure for Projects”](#) on page 78. When entering a directory path, you can use the path browser. See [“Browsing for Paths”](#) on page 82.

Project Name	Zeus	
Project Home	C:\Footage\New\Projects\Project-Zeus	...
Scans Full Home	C:\Footage\New\Scans	...
Scans Half Home	C:\Footage\New\Scans	...
Renders Full Home	C:\Footage\New\Projects\Project-Zeus	...
Renders Half Home	C:\Footage\New\Projects\Project-Zeus	...

The screenshot shows the Project Configuration Settings dialog box. It is divided into three main sections: Architecture, Output Colour Space, and Project Template. The Architecture section contains buttons for Linear, Scan Type, Progressive, and Frame Rate. The Output Colour Space section contains buttons for Logarithmic, Proxy Width, Percent, Pixel Ratio, and Stereoscopy. The Project Template section contains buttons for Save, Load, and Delete. A slider for Pixel Ratio is visible, with a value of 4.74. A label 'a' points to the Percent/Pixel option box, and a label 'b' points to the Pixel Ratio slider.

a) Percent / Pixel option box b) 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.

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 without selecting a stereoscopic raster.

NOTE: This button becomes unavailable when a stereo raster has been selected.

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.

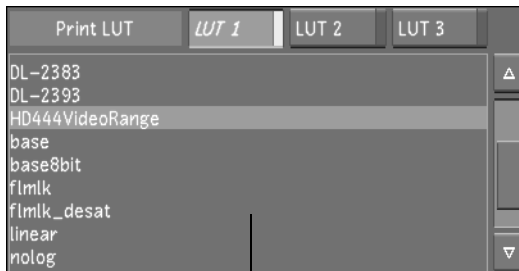
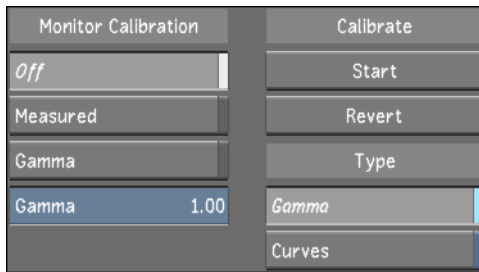
Load button — Click to load a project template after you select a template in the Template name list.

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”](#) on page 68.

Calibration menu

Use the Calibration menu to configure default system calibration settings and to define the default print LUTs for a project.



a



b

a) Print LUT list

b) Input LUT list

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 [Chapter 5, “Monitor Calibration,”](#) on page 117.

Revert button — Click to revert back to the previous automatic calibration.

Gamma button (Calibration Type) — Enable to set the calibration type to ‘Gamma’.

Curves button (Calibration Type) — Enable to set the calibration type to ‘Curves’.

Print LUT buttons — Enable one of LUT 1, 2, or 3 buttons, then select a displayed print LUTs to perform a quick comparison among three LUTs. For example, to designate a LUT as Print LUT 2, enable Print LUT 2 and select one of the displayed print LUTs. This should be done for Print LUT 1, 2 and 3.

Print LUT lists — Select a print LUT from each list after you enable one of the Print LUT buttons.

Custom button — Enable to select a custom input LUT (an input LUT other than the Log-to-Lin LUT). See [Chapter 11, “Input LUTs,”](#) on page 279.

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 [Chapter 11, “Input LUTs,”](#) on page 279.

Input LUT list — Select a custom input LUT from this list to configure Lustre to use this LUT when:

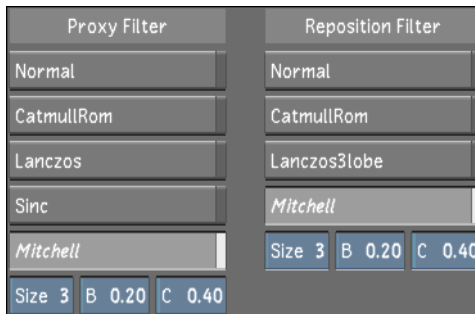
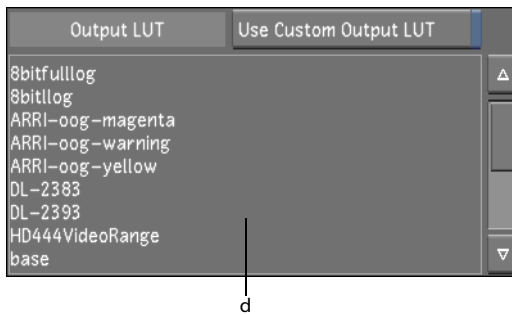
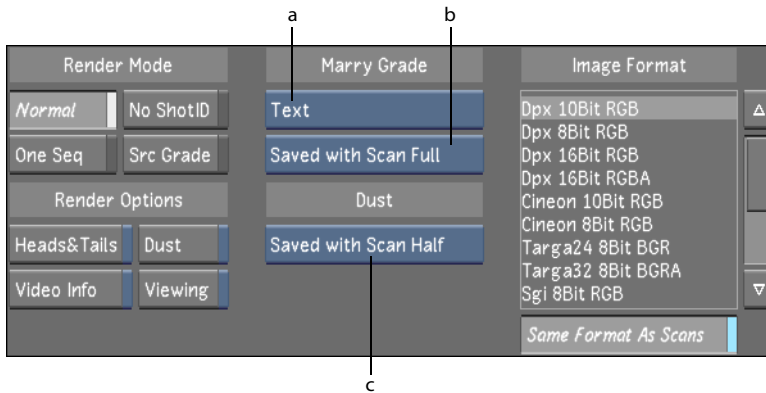
- dragging a shot from the Shot bin or Browser to the timeline, or
- assembling an EDL

NOTE: To select a custom input LUT, enable the Custom button.

Rendering menu

Use the Rendering menu to:

- Organize the render files.
- Set the render output format.
- Configure render options.
- Set the proxy generation size.
- Configure proxy and reposition filters.



- a) Marry Grade Format option box
- b) Marry Grade File Location option box
- c) Dust File Location option box
- d) Output LUT list

Render Mode — Set the default render option.

Select:	To have:
Normal	Each shot of a cut rendered to a separate directory, 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.

Heads & Tails button — Enable to set the project's default to render shots with head and tail frames.

Dust button — Enable to set the project's default to render shots with dust metadata.

Video Info button — Enable to set the project's default to burn in the timecode into the render.

Viewing button — 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.

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 Grades for Single Shots”](#) on page 61.

Select:	To:
Binary	Save marry grade files in a binary format
Text	Save marry grade files in a text format (primary colour correction only)
UID Aware	Save marry grade files in a binary format and make the file 'Unique ID-aware'

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.

Image Format list — Select the default format of the media to be rendered from this list.

Same Format As Scans button — Enable to render media in the same format as the source media.

Output LUT list — Select a default output LUT from this list to use when rendering.

NOTE: Lustre only uses a selected output LUT if the Use Custom Output LUT button is enabled.

Use Custom Output LUT button — Enable to configure Lustre to use the selected output LUT in the Output LUT list when rendering.

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.

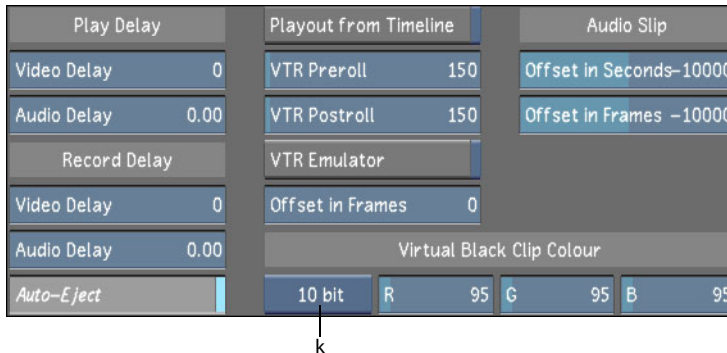
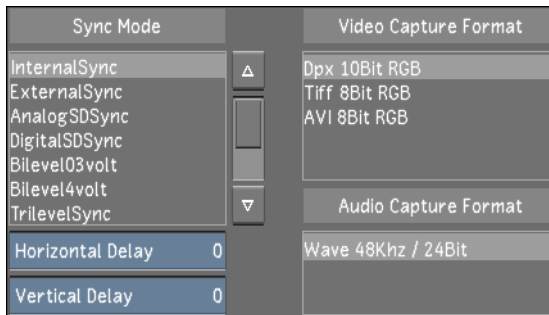
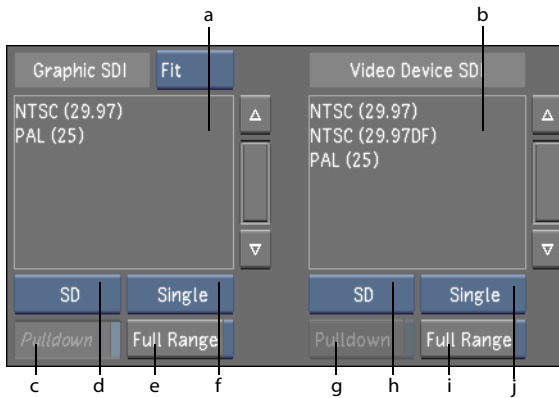
Reposition Filter — Select the reposition filter to use by default when creating proxies.

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.
CatmullRom	The CatmullRom reposition filter.
Lanczos3lobe	The Lanczos3lobe reposition filter.
Mitchell	The Mitchell reposition filter. Use the sliders below to define the filter settings.

Engineering menu

Use the Engineering menu to configure:

- Video device SDI settings
- GFX SDI settings
- Sync mode settings
- Video capture and playout settings



- a) GFX SDI Raster list e) GFX SDI Full Range button i) Video SDI Full Range button
b) Video SDI Raster list f) GFX SDI Link Type option box j) Video SDI Link Type option box
c) GFX SDI Pulldown button g) Video SDI Pulldown button k) Bit Depth option box
d) GFX SDI Format option box h) Video SDI Format option box

Fit button — Enable to enlarge the image as much as possible while preserving the aspect ratio at the SDI out (graphics card).

Centre button — Enable to centre the image without zooming at the SDI out (graphics card).

Stretch button — Enable to fit the image to the dimensions of the monitor by adjusting the aspect ratio at the SDI out (graphics card).

GFX SDI Raster list — Select one of the graphics rasters to set the default format for the graphic card's SDI output.

GFX SDI Link Type option box — Toggle to display, in the Rasters list, rasters using the selected link.

GFX SDI Full Range button — Enable to configure Lustre to use the full range of YUV and RGB when outputting to the Video SDI. When this button is disabled, Lustre uses the Normal mapping matrix for the configured video raster.

GFX SDI Pulldown button — Enable to configure Lustre to add a pulldown frame when outputting through the GFX SDI. See [“About 3:2 Pulldown”](#) on page 548.

GFX SDI Format option box — Toggle to display, in the Rasters list, rasters using the selected resolution format.

Video 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 SDI Link Type option box — Toggle to display, in the Rasters list, rasters using the selected link.

Video SDI Full Range button — Enable to configure Lustre to use the full range of YUV and RGB when outputting to the Video SDI. When this button is disabled, Lustre uses the Normal mapping matrix for the configured video raster.

Video SDI Pulldown button — Enable to configure Lustre to add a pulldown frame when outputting through the video device SDI. See [“About 3:2 Pulldown”](#) on page 548.

Video SDI Format option box — Toggle to display, in the Rasters list, rasters using the selected footage format.

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
AVI 8Bit RGB	AVI files with an RGB colour space at a bit depth of 8 bits

Audio Capture Format list — Currently, the only supported capture format is Wave file (.WAV) with a sampling rate of 48,000 samples per second and a bit depth of 24 bits.

Video Delay slider — Define the default number of video frames to add before the start of the shot when capturing from the VTR.

Audio Delay slider — Define the default number of audio frames to add before the start of the shot when capturing from the VTR.

Video Delay slider — Define the default number of video frames to add before the start of the shot when recording to the VTR.

Audio Delay slider — Define the default number of audio frames to add before the start of the shot when recording to the VTR.

Auto-Eject — Enable to set the project's default to automatically eject the tape during EDL capture.

Playout from Timeline — 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.

Offset in Frames slider — Define the default number of frames you wish to offset the audio.

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 menu

Use the Network Rendering menu to configure default Backburner/Burn, Slave Render, and Wiretap directory paths and settings for a project.

Configuration	<i>Backburner</i>	Slave Render	
Project Home	C:\Footage\New\Projects\Project-Zeus		...
Scans Full Home	C:\Footage\New\Scans		...
Scans Half Home	C:\Footage\New\Scans		...
Renders Full Home	C:\Footage\New\Projects\Project-Zeus		...
Renders Half Home	C:\Footage\New\Projects\Project-Zeus		...

Backburner Manager	Backburner
	Task Size
Slave Render	0
	Priority
Refresh in Seconds 1.00	Skip Error
Renders Full Res	

Wiretap
Local Proxy
Render Library
chorley@wt:/stonefs

a) Backburner Manager host field

b) Task Size slider

c) Slave Render host field

Backburner button — Toggle to display one of the two menus where you can configure the project paths for Backburner and Slave render.

Slave Render button — Enable to set the Slave Render on by default. You must configure the Slave Render paths to use this function. See [“Network Rendering menu”](#) on page 31.

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 of the Backburner Manager node. This address is read from the *init.config* file.

Slave Render Host field — The IP address of the Slave Render 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 Render 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 Render.

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.

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”](#) on page 82.



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.

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.

While logged in, the user can override some of the user default settings. 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 `...\project\<project name>\` folder in Windows and the `.../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 shot
- Resolution
- Crop value
- Assemble frame rate / drop frame mode
- State of the surface control
- Audio file

Context parameters are stored in the `<user>Context.config` file. For information, see the *Autodesk Lustre Software Installation Guide*.

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”](#) on page 33 and [“Project Configuration”](#) on page 15.

Creating a User Profile

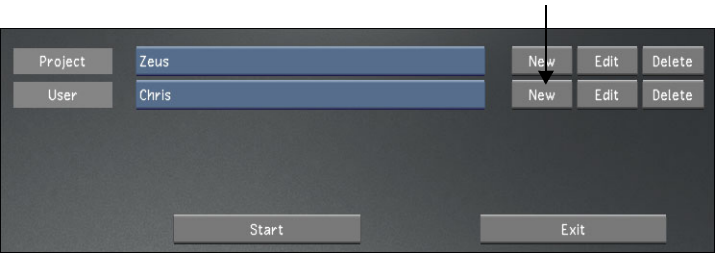
The new user settings menu allows you to customize your settings so they are used whenever you enter the Lustre application. The settings are broken down into three groups: Display & Interface, System & Menu, and Tools. When you create a new user you can set the settings within the three groups to your preference, or you can choose to keep the default settings.

Use templates to accelerate user creation. See [“Working with Templates”](#) on page 68.

To create a new user:

1. Do one of the following:
- From the Main menu, click Setup, then Settings, and then click New in the User group.

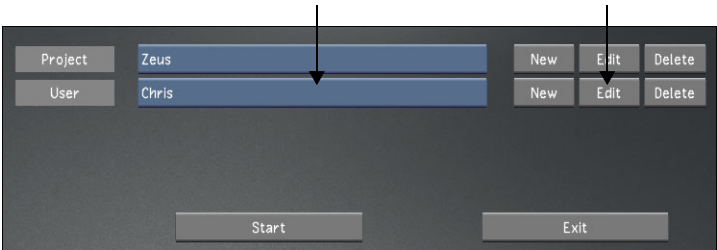
• From the Lustre splash screen, click New in the User group.



2. Configure the new user. Use Reset to reset all settings, from every menu, to their defaults.
- | Select: | To configure: |
|---------------------|---|
| Display & Interface | The default display and interface settings for this user. See “Display & Interface Settings” on page 38. |
| System & Menu | The system, match grade, assemble, and playout settings for this user. See “System & Menu Settings” on page 40. |
| Tools | Keyframe interpolation, animation, and dust settings for this user. See “Tools Settings” on page 43. |
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 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. 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.

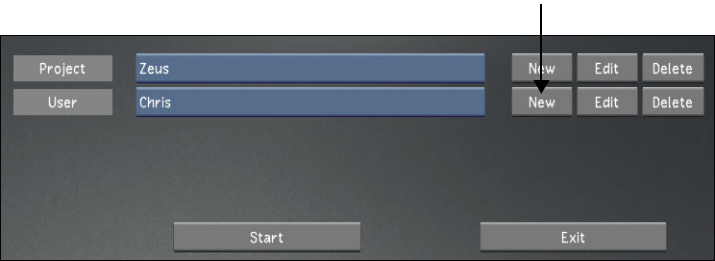
Select:	To configure:
Display & Interface	The default display and interface settings for this user. See "Display & Interface Settings" on page 38.
System & Menu	The system, match grade, assemble, and playout settings for this user. See "System & Menu Settings" on page 40.
Tools	Keyframe interpolation, animation, and dust settings for this user. See "Tools Settings" on page 43.

4. To save the settings for the user, click Save User.
5. Click Exit User.

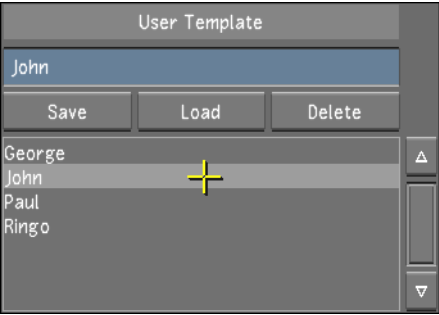
To create a new user using a template:

1.
- Do one of the following:
- From the 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. See [“Creating a User Template”](#) on page 74.



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.



WARNING: Rename the new user, or you will overwrite the template.

Select:	To configure:
Display & Interface	The default display and interface settings for this user. See “Display & Interface Settings” on page 38.
System & Menu	The system, match grade, assemble, and playout settings for this user. See “System & Menu Settings” on page 40.
Tools	Keyframe interpolation, animation, and dust settings for this user. See “Tools Settings” on page 43.

5.
- To save the settings for the user, click Save User.

6. Click Exit User.

Editing an Existing User

As you are working from project to project, one project may require different settings than the other. 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 Main menu, click Setup, and then Settings. Select the user, and then click Edit in the User group.
 - From the Lustre splash screen, 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.

Select:	To configure:
Display & Interface	The default display and interface settings for this user. See “Display & Interface Settings” on page 38.
System & Menu	The system, match grade, assemble, and playout settings for this user. See “System & Menu Settings” on page 40.
Tools	Keyframe interpolation, animation, and dust settings for this user. See “Tools Settings” on page 43.

3. To save the edits, click Save User.
4. Click Exit User.

User Configuration Settings

For any user, you can configure default settings. User settings are distributed among three Settings menus. Navigate through the User Configuration by clicking the following buttons.

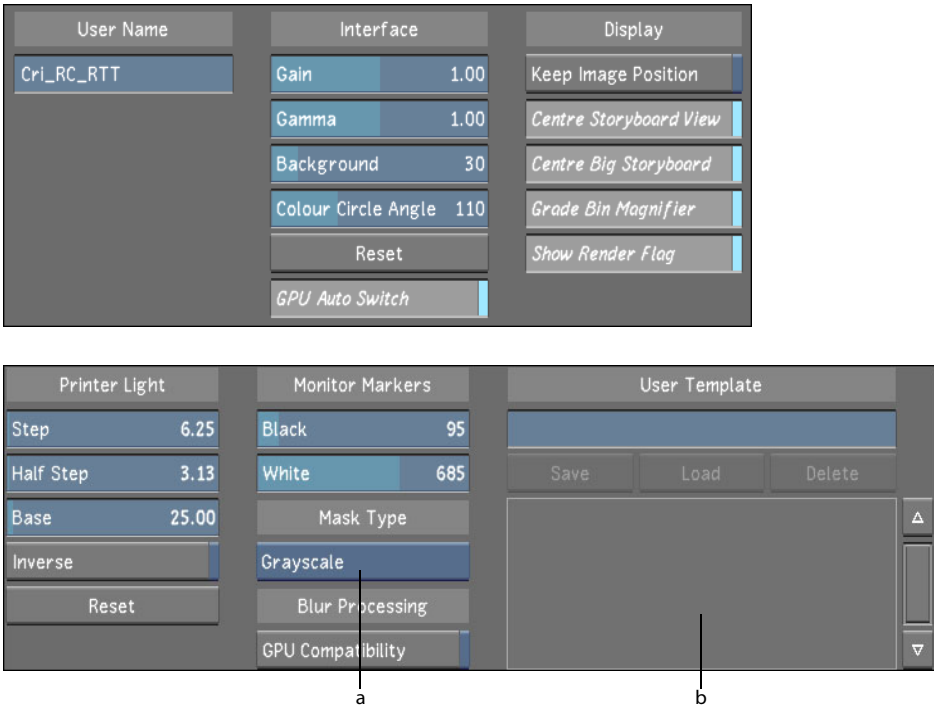
Display & Interface button — Select to configure the interface and display settings associated with this user. See [“Display & Interface Settings”](#) on page 38.

System & Menu button — Select to configure the system, match options, assemble menu, and playout menu setting. See [“System & Menu Settings”](#) on page 40.

Tools button — Select to display the settings for the keyframe, animation, and dust tools. See [“Tools Settings”](#) on page 43.

Display & Interface Settings

Use the Display & Interface Settings page to configure the default display and interface settings for this user.



a) Mask Type option box b) User Template list

User Name field — Enter your user name in this field. This name appears in the list of users after the user profile is saved.

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 increase or decrease 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 button — Enable to allow Lustre to automatically switch to CPU processing for all features that are not GPU compatible.

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.

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.

HalfStep 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 histogram and waveform monitor for white.

Mask Type option box — Toggle to select how you wish to display your masks.

Select:	To display masks:
Grayscale	In grayscale by default.
Saturated	In saturated by default.

GPU Compatibility button — Key blurs and geometry blurs are processed differently depending on whether CPU processing or GPU acceleration is enabled. To ensure that they are always displayed identically, enable the GPU Compatibility button before you start work on keys, key blurs, and geometry blurs.

User Template field — Enter the name for the template you are creating. See [“Working with Templates”](#) on page 68.

Save button — Click this button to save the user settings to the template named in the User Template field.

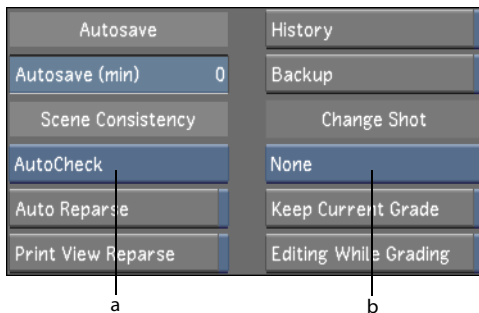
Load button — Loads the user template selected in the User Template list.

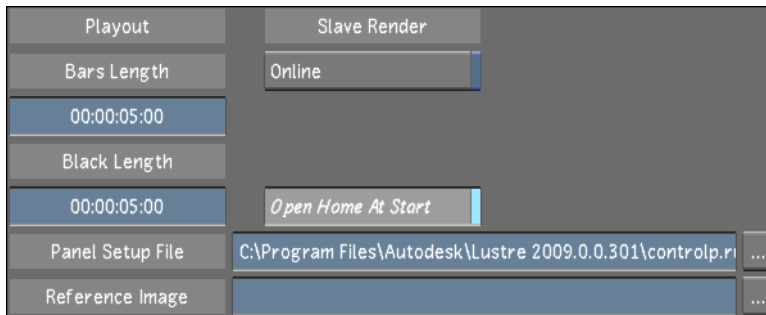
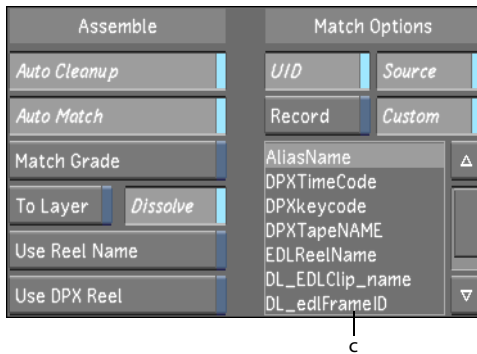
Delete button — Deletes the user template you have selected in the User Template list after you select a template in the Template name list.

User Template list — Displays a list of all the user templates which have been created for Lustre. Select user template from this list before loading or deleting the selected template.

System & Menu Settings

Use the System & Menu Settings page to configure system, match grade, assemble, and playout settings for this user.





a) Scene Consistency option box b) Change Shot option box c) XML Tag Options list

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.

History button — Enable to save a history file for each grade whenever you save the grade.

Backup button — Enable to save a copy of all metadata on the system disk into the Lustre home install directory.

Change Shot option box — Toggle to define the change operation settings.

Select:	To:
Do	Force a Do operation after changing the shot.
DoAndClearUndo	Force a Do operation and flush the undo buffer after changing the shot.
None	Not force a Do operation nor flush the undo buffer after changing the shot.
ClearUndo	Flush the undo buffer after changing the 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 hot keys 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 Shot bin 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. This features is disabled by default.

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 connect to the Slave Render server as it is defined in the *init.config* file. Once this feature is enabled, you can enable or disable Slave Renderer from the Render > Slave Render menu.

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.

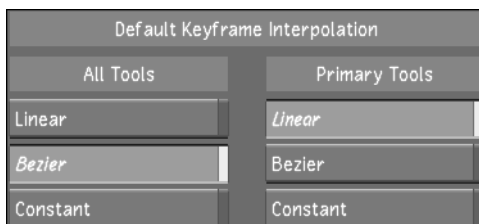
Panel Setup File field — Enter the file path of the control panel rules file or browse and enter a new path for this file. The default path for this file is the User folder of the application. For information about defining paths, see [“Browsing for Paths”](#) on page 82.

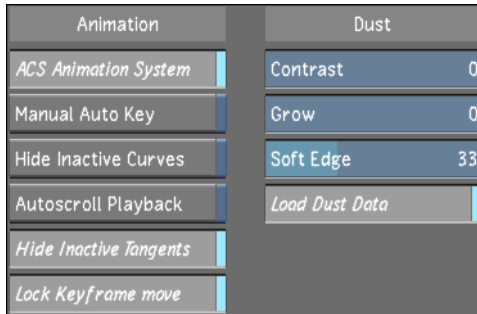
Reference Image field — Enter the path of the reference image or browse and enter a new path for this file. Then, after loading the project and launching the application, you can press **SHIFT+K** to load the image into the single file system for reference and then view the image by pressing **L**. For information about defining paths, see [“Browsing for Paths”](#) on page 82.

Tools Settings

Use the Tools Settings page to configure keyframe interpolation, animation, and dust settings for this user.

NOTE: The settings in the All Tools sub-group do not affect tools in the Colour > Grading menu.





All Tools Linear button — Enable 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.

All Tools Bezier button — Enable 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.

All Tools Constant button — Enable 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.

Primary Tools Linear button — Enable 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.

Primary Tools Bezier button — Enable 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.

Primary Tools Constant button — Enable 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.

ACS Animation System button — Enable to configure Lustre to automatically move the first or last key whenever a keyed parameter is moved. This option is enabled by default.

Manual Auto Key button — Enable to set the Auto Key state to On when moving from shot to shot while grading. Disable this feature so the Auto Key state is disabled. This option is disabled by default.

Hide Inactive Curves button — Enable to hide curves that are not in use. Hiding inactive curves can make it easier for you to manage your keyframes and modify your animations. This option is disabled by default.

Autoscroll Playback button — Enable to configure Lustre's default behaviour to scroll automatically when the positioner plays off the far right edge of the Channel Editor (or the far left edge when playing backward). This option is disabled by default.

Hide Inactive Tangents button — Enable to hide tangent handles for keyframes that are not selected. This option is enabled by default.

Lock Keyframe Move button — Enable to allow only vertical movement when dragging a keyframe. This option is enabled by default.

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. This option is set to 66 by default.

Load Dust Data button — Enable to load the dust removal metadata after you have already loaded the grade. This option is enabled by default.

Loading Users and Projects

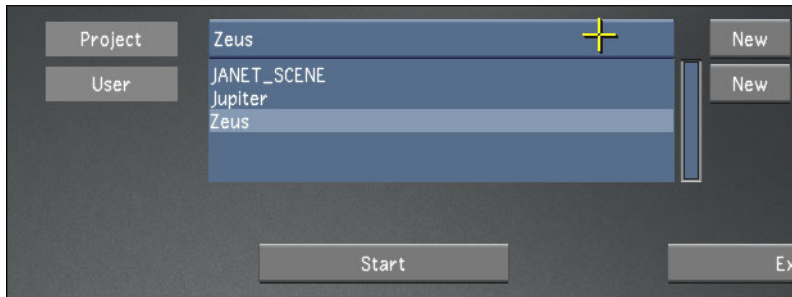
After creating a project and defining users, you can start using it. You do so by loading a project and selecting a user profile. This can be done from the Lustre splash screen, or from an already started session.

When you are already in a Lustre session, you can:

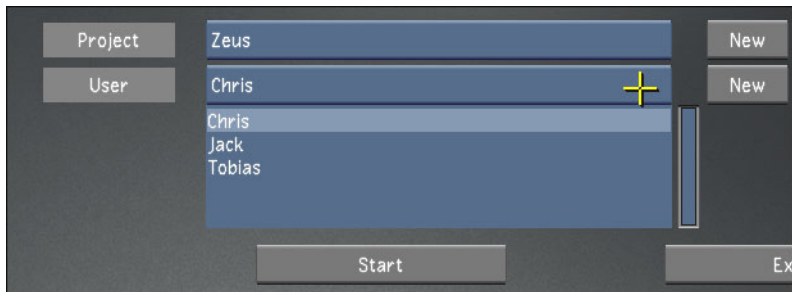
- Keep the current project and load a different user.
- Change the current project and keep the current user.
- Change both the current project and the current user.

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

WARNING: In Lustre, you must create a scene before starting to do any work (importing, capturing, or creating data), or you may lose important data.

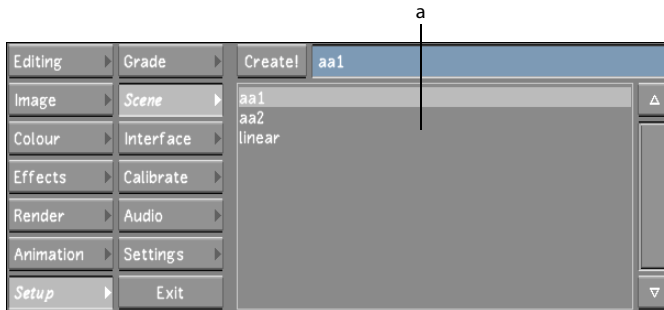
A scene is a specific sequence of shots (or a scene) within the project. You can create as many cuts per scene as needed.

When you create a scene in Lustre, a directory with the same name is created in the Project Home. With the exception of dust removal data and marry grades, the metadata for all changes made to the cuts associated with the scene is stored in a sub-directory of the scene directory. This directory is called *sacc_data* and includes files containing metadata for the grade, cut, as well as render, gang, and bin information. The metadata is contained in 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 grade files associated with the scene. Nothing changes in the application or gets loaded when you select a scene except the contents of the Grade bin, which is associated with the scene. 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.

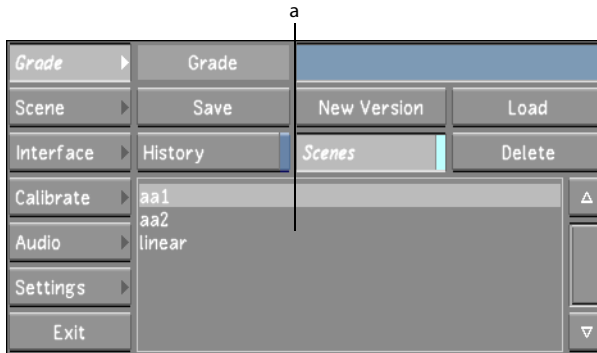
You can select a scene from either the Scene or Grade menu.

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.

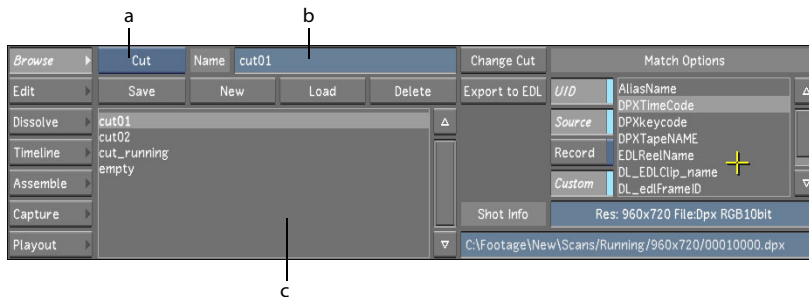
Working With Cuts

WARNING: Do not save a cut until you have created at least one scene.

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 grading, dust removal, or effects.

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”](#) on page 52.

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.



a) Cut/Change List option box b) Cut name field c) Cut Name list

Creating a New Cut

Before you begin a new project, you should create a cut. When you create a cut, the current cut is duplicated and given the name of the new cut.

HINT: An empty cut is automatically created when you create a new scene. You can use this cut as a starting point anytime you need to create a new cut.

To create a new cut:

1. Click Editing, and then click Browse to display the Browse menu.
2. Select the “empty” cut in the Cut Name list, and then click Load.
A cut with no shots is loaded into the timeline.
3. Enter the name for the new cut in the Cut Name field.
4. Click New.

A new cut is created from the current timeline. The name given to the cut appears in the Cut Name list.

Managing Cuts

Use the tools in the Browse menu to manage your cuts. You can load, save, change, and delete cuts from this menu.

Loading a Cut

You can load an existing cut to resume 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.

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. See [“System & Menu Settings”](#) on page 40.

To save a cut:

1. Click Editing, and then click Browse to display the Browse menu.
2. Click Save.

The cut is saved.

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 option box.
3. Enter the new name for the cut in the Cut Name field.
4. Click New.

A new cut is created. The name of the new cut is added to the Cut Name list.

Changing a Cut

You can load a new cut without changing the grade settings. This is called changing a cut and it allows you to apply the grade settings from the current cut to the 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 EDL cut. The grade settings that were in the original cut are transferred to the shots.

The Lustre match function is based on Heuristic Evaluation. This means that even when there is not an exact match, the grade may still be applied if the match is close enough based on Lustre's internal set of rules. For example, if the source timecode has changed on a clip from one cut to the other but the shot IDs still match, the change cut function still works.

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"](#) on page 152.

To change a 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 to in the Cut Name list.
5. 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.

6. Click Setup, and then click Grade to display the Grade menu.
7. Click New Version to save the grade with the newly associated cut.

Deleting a Cut

You can delete a cut when you no longer need it.

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.

The cut is deleted.

Using Change Lists

A change list tracks 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 of shots to the cut
- Removal of shots from the cut
- Addition of layers to the cut
- 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.

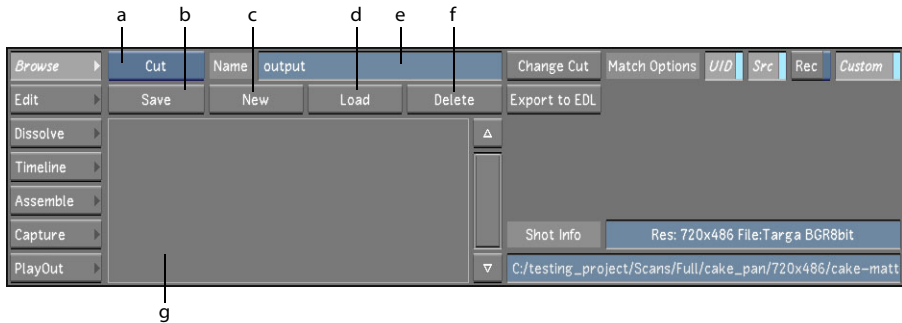
Creating a Change List

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) 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.

Working With Grades



WARNING: Do not save a grade until you have created at least one scene and one cut.

In the grade file, grade data for each shot is linked to the shot. You can edit the cut, for example, 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.

Save the grade to save all modifications made to your footage in Lustre. When you save the grade, all colour grading, animation, Sparks plugins, and reposition modifications made to 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 Grades for Single Shots”](#) on page 61.

Setting Logarithmic or Linear Mode

Use the Colour Space Architecture options to select between the two basic colour grading modes — Linear and Log — 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 log or linear files. For example, film-based projects are usually graded in Log 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 log files, you can convert them to linear after bringing them into Lustre. To convert a shot from log to linear and vice versa, use the Input LUT menu. See [Chapter 11, “Input LUTs.”](#) on page 279.

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”](#) on page 478.

Colour Grading Toolset Differences

The colour grading toolset varies depending on if you are in Linear or Log 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 Log, the controls reflect the workflow of a film-based project. See [Chapter 13, “Primary Colour Grading,”](#) on page 313.
- Printer lights only appear in Log mode. See [“Adjusting Printer Lights for Primary Grading”](#) on page 317.
- In the Input LUT menu, the controls for creating a Log to Lin conversion LUT are only available in Linear mode. See [“Linear Mode: Creating Conversion LUTs”](#) on page 281.

To set the colour space architecture for the project:

- In the Grade menu, enable Log or Linear.

Creating a Grade for a Cut

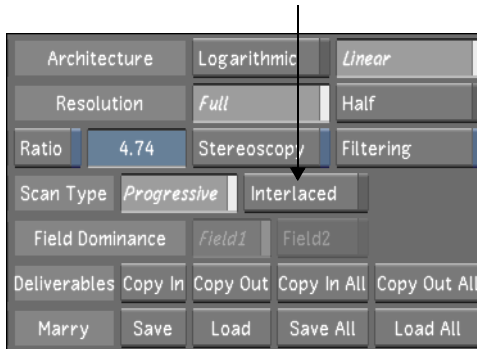
Save the grade for the cut you are working on to save all your work on the cut. You can save up to 100 grades per cut. Grades are named using sequential numbers, but you can optionally add a text name to the number.

NOTE: You can also 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”](#) on page 50.

To create a grade for a cut:

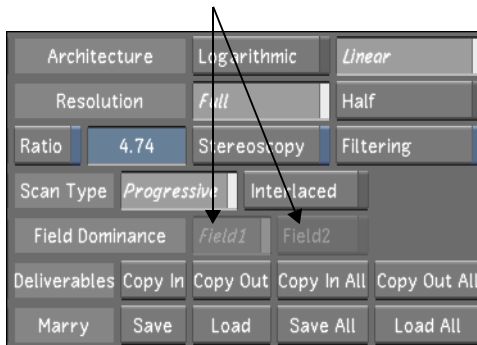
1. Create or load a cut for which you want to save a grade. See [“Working With Cuts”](#) on page 49.
2. Work on the grade as needed.
3. In the Main menu, click Setup, and then click Grade.
The Grade menu appears.

4. Indicate the scan type. For film-based projects, enable Progressive. For video-based projects, enable Interlaced.



NOTE: Interlaced support availability depends on the version of the graphics card installed on your system. See your release notes for details.

5. If working in Interlaced mode, select the dominant field by enabling either Field 1 or Field 2.



The dominant field is rendered first, and is the first field to which animations are applied. By correctly setting the dominant field, you can avoid introducing noticeable jitter that would result if the field recorded earlier in time is displayed after the field that was recorded later in time.

NOTE: Field-based projects display a half-pixel downshift on the image's second field when viewed on the data monitor.

6. (Optional) Type a name for the grade in the Grade field and press **ENTER**.
7. Click Save or press **S**.

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.

To save a new version of a grade:

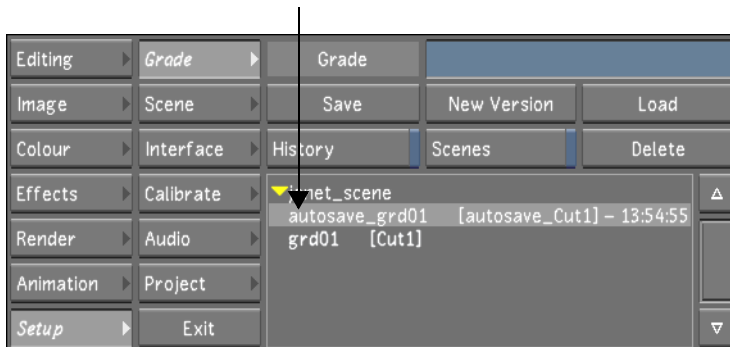
- In the Grade menu, click New Version.

The current settings are saved to a new grade version, which appears in the list. The new grade is named with a number one above the highest existing number.

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”](#) on page 40.

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 *sacc_data* folder.

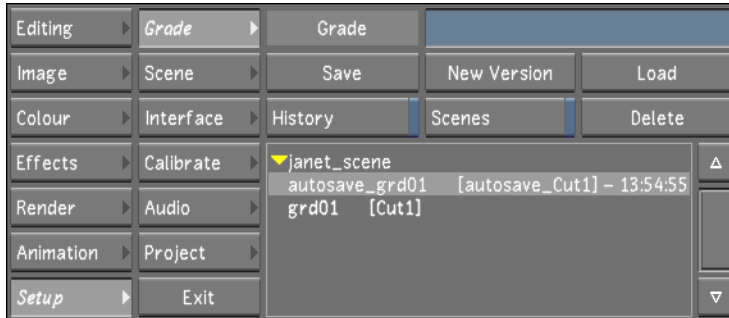
There are certain scenarios where an Autosave is performed before its scheduled time interval. These scenarios include the following:

- 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 [“Changing a Cut”](#) on page 50. 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.



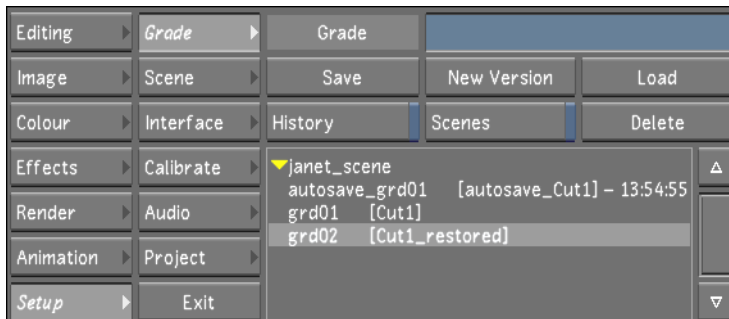
2. Press **CTRL+R** to make sure the most current autosave information is displayed in the Grade list.
3. Select the autosaved grade and cut.
4. Click Load.

NOTE: The autosaved version of the cut file appears in the cut name list (e.g., *autosave_Cut1*).

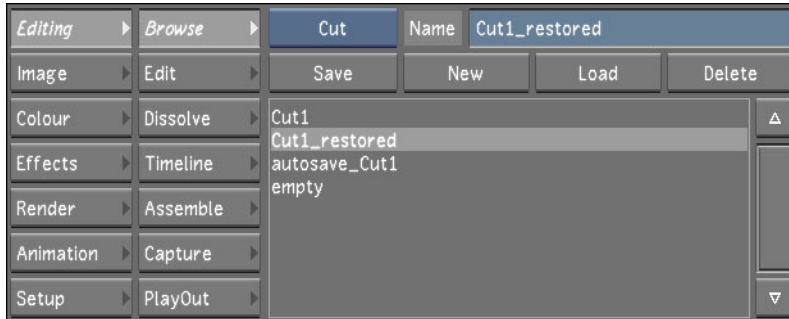
5. Click New Version.

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. This is to differentiate between the original cut and 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 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. If it is not already selected, select the scene containing the grade you want to load. See [“Selecting Scenes”](#) on page 48.
3. If needed, in the Grade list, click the arrow next to the scene name to display the list of grades associated with the scene.
4. Click the grade you want to load.
The grade is highlighted in the list.
5. Click Load.
The grade and associated cut are loaded.

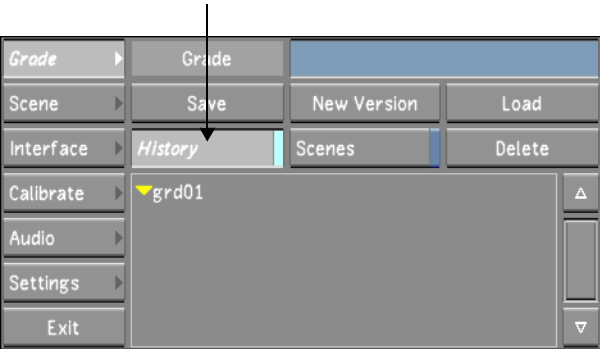
Loading Backup Copies of a Grade

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.

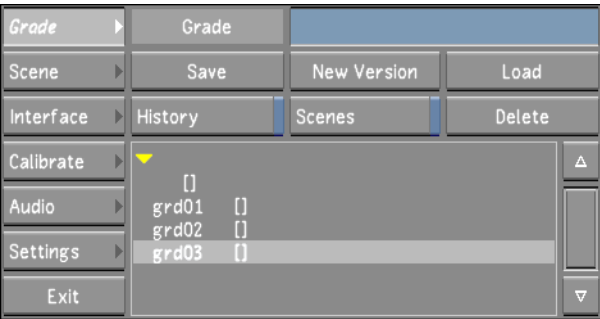
NOTE: To use this feature, you need to first enable it while configuring the user.

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”](#) on page 59.
2. In the Grade menu, click History.



The Grade History list appears.



This is a list of backup copies of the current grade.

3. Click the backup version you want to load.
- The backup grade is loaded.

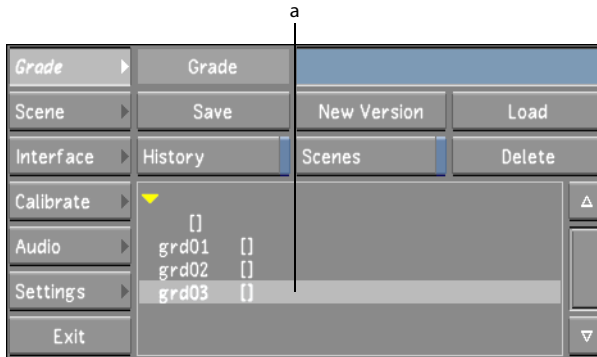
Deleting Grades

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.



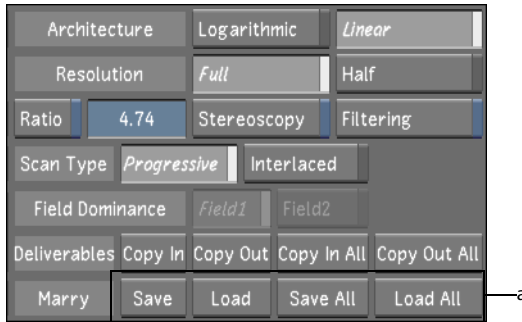
a) Grade list

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. Press left **CTRL**, click Delete, and then confirm by clicking Delete a second time.

Saving and Loading Grades for Single Shots

You can save the grade for a single shot rather than for the entire cut. This is useful when you want to apply a grade created for a shot in one cut to the same shot in another cut. The shots in the two cuts do not have to be in the same order and can have different edits applied to them. One situation where this could 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.

You can save an individual grade for the current shot or for all the shots in the timeline at once. Similarly, you can load the grade back to the current shot or to all shots in the timeline.



a) Marry controls

NOTE: To use 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 [“Grade Bins: Saving, Applying, and Deleting Intermediary Grades”](#) on page 105.

When you save a grade using the Marry controls, the colour grading metadata is saved separately from the regular grade file; it is stored with the Scans Half Home shot footage or Scans Full Home shot footage in a Marry Grade file. When configuring a project, you can specify the format of the Marry Grade file.

- **Binary** — When you select Binary and save a Marry Grade file, Lustre saves all grading information (across all channels) to one *marry.grade* file for each shot. The *marry.grade* file is in a binary format.
- **Text** — When you select Text and save a Marry Grade file, Lustre saves only primary grading information to a text file (*.dsc*). This data is saved as offset in code values for red, green, and blue. The purpose of this option is to be able to transfer primary grading information for individual shots between Lustre and other applications. In order to load this text file, its associated grade file must be loaded already. You can have only one *.dsc* file per grade. This file is named as follows:

grd<grade number>.*dsc*

(for example: *grd11.dsc*)

- **UID Aware** — Every shot has a Unique ID. When you use the Scene Detect function to divide a larger shot into smaller, scene-detected shots, each scene-detected shot is also assigned a Unique ID. When you select UID Aware in the Marry Grade group (see [“Project menu”](#) on

page 20) and save the grade using the Marry Grade function, Lustre saves all grading information:

- for a single shot or scene-detected shot to a UID-based Marry Grade file, or
- for all shots and scene-detected shots in the cut to their respective UID-based Marry Grade files

The format of this file is binary and is named as follows:

marry_grade.id<unique ID of the scene>

(for example: *marry_grade.id484D7232000B0*)

NOTE: If you select Binary format instead of UID Aware, you will not be able to save grading information for individual scene-detected shots through the Marry Grade function. There is only one *marry_grade* file allowed per shot folder. Saving a grade twice on the same shot using Marry controls would result in only one version of the grade being saved. For more information about Marry Grade configuration settings, see [“Rendering menu”](#) on page 24.

NOTE: You can find the UID for each shot in the cut file located in your scene’s *sacc_data* folder.

To save the grade of one or all shots in a cut:

1. In the Main menu, click Setup, and 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”](#) on page 226.
3. Using the Marry controls, do one of the following:
 - Click Save to save the grade for the current shot.
 - Click Save All to save individual grades for each shot in the cut.

The grade is saved to the *marry_grade* file in the location defined during project configuration. See [“Rendering menu”](#) on page 24. If the MarryType PrimText keyword was added to the configuration file, then the grade is saved to a *.dsc* file with the current grade.

To load the grade of one or all shots in a cut:

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 to load the grade of the current shot.
 - Click Load All 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.

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:

- Press **SHIFT+G**.

The *primary.txt* file is generated in the Lustre export directory, e.g. *C:\Program Files\Autodesk\lustre 2009\export* on Windows, and */usr/autodesk/lustre2009/export* on Linux.

The **SHIFT+G** hot key has another function. In addition to saving the primary grading data for the timeline, the hot key 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 2009\export* directory on Windows, and */usr/autodesk/lustre2009/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

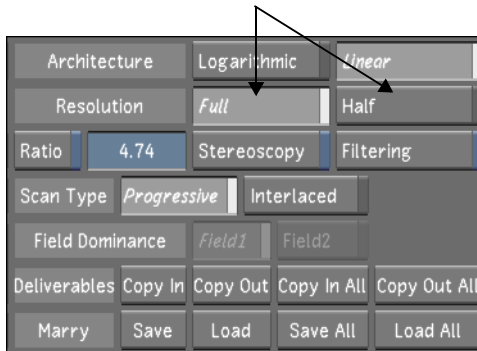
You can work on the full-resolution version of shots or proxies. Using proxies speeds up interaction and playback of shots containing many effects. Proxies are also useful for playing to projectors.

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”](#) on page 484 and [“Recommended Directory Structure for Projects”](#) on page 78.

To set the resolution:

- In the Grade menu, enable Full or Half resolution.

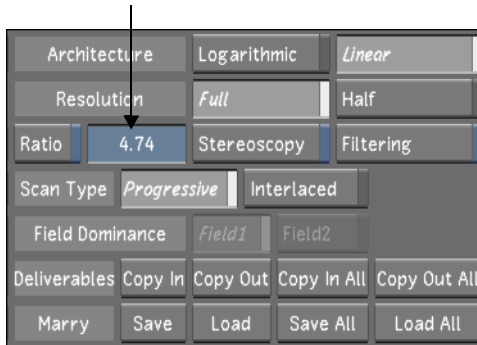


Setting the Aspect Ratio

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, enter the aspect ratio in the Ratio field.



2. Enable Ratio.

The shots are stretched to fit in the specified aspect ratio.

Setting the Filtering

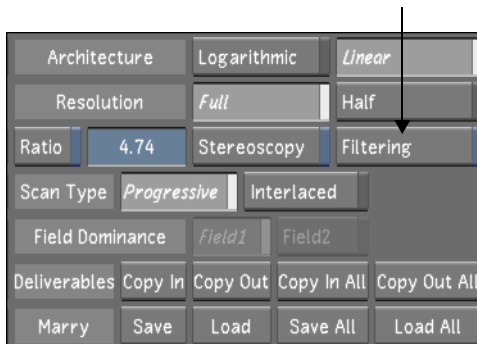
The Filtering option allows you to soften your shot so when you zoom in, the pixels are not as sharp or obvious.

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 keyer.

To enabled the filtering option:

- In the Grade menu, enable Filtering.



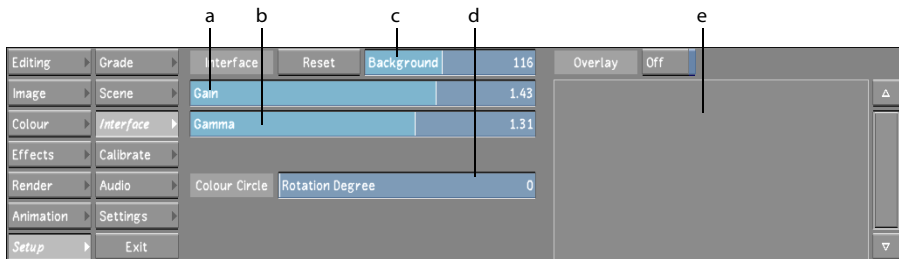
Setting Up the User Interface

Use the controls in the Interface menu to set the appearance of the user interface to your liking. For example, set the user interface to suit the room lighting conditions. From the Interface menu, you can also add an overlay, such as a logo, to your cut.

To access the Interface menu:

- In the Main menu, click Setup, and then click Interface.

The Interface menu appears.



- a) Gain slider
- b) Gamma slider
- c) Background slider
- d) Colour Circle slider
- e) Overlays field

The Interface menu is made up of the following elements.

Background field — Sets the shade of grey of the area surrounding the image 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”](#) on page 67.

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 2009*, where *C* is the drive where Lustre is installed.
 - If using the Linux version of Lustre, type:

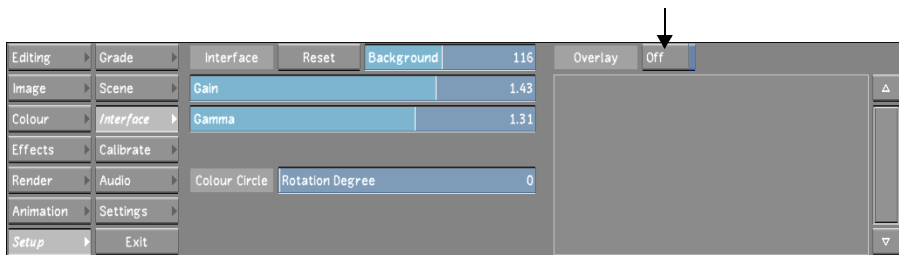

```
cd /usr/autodesk/lustre2009
```
3. Create a directory called *overlay*. To do this in Linux, type:


```
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/lustre2009/overlay
```
5. Restart Lustre.

To apply an overlay to a cut:

1. Load the cut to which you want to apply an overlay. See [“Loading a Cut”](#) on page 50.
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.

Working with Templates

Use templates to define re-usable presets for future projects or users.

Creating a Project Template

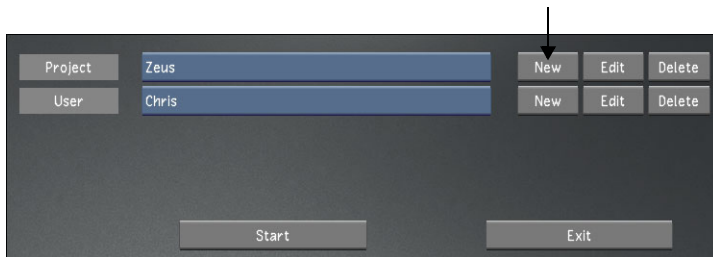
When you are choosing your project settings, you can create a template for your future projects.

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:

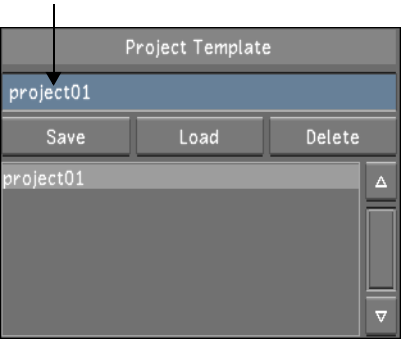
1. Do one of the following:
 - From the Main menu, click Setup, then Settings, and then click New in the Project group.
 - From the Lustre splash screen, click New in the Project group.



2. Configure the settings for the template. Use Reset to reset all settings, from every menu, to their defaults.

Select:	To configure:
Project	The name and location of the project. See "Project menu" on page 20.
Calibration	Default system calibration settings and define the default Print LUTs for a project. See "Calibration menu" on page 22.
Rendering	Default rendering settings for a project. See "Rendering menu" on page 24.
Engineering	Default video and graphics settings for a project. See "Engineering menu" on page 27.
Network Rendering	Default Backburner/Burn, Slave Render, and Wiretap directory paths and settings for a project. See "Network Rendering menu" on page 31.

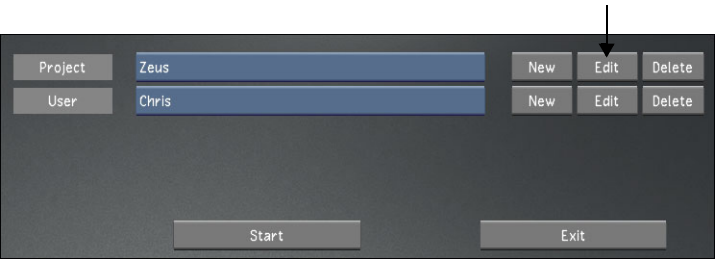
- When you are done editing the settings, click Project, and enter a project name in the Project Template field.



- In the Project Template group, click Save.
- To exit, click Exit Project.

To create a new project template from an existing project:

- 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.



- Configure the project. Use Reset to reset all settings, from every menu, to their defaults.

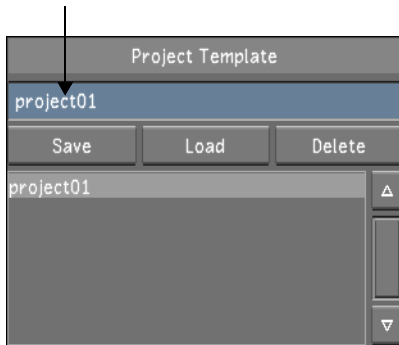


WARNING: Rename the new project, or you will overwrite the original project’s settings with the new settings.

Select:	To configure:
Project	The name and location of the project. See “Project menu” on page 20.
Calibration	Default system calibration settings and define the default Print LUTs for a project. See “Calibration menu” on page 22.

Select:	To configure:
Rendering	Default rendering settings for a project. See "Rendering menu" on page 24.
Engineering	Default video and graphics settings for a project. See "Engineering menu" on page 27.
Network Rendering	Default Backburner/Burn, Slave Render, and Wiretap directory paths and settings for a project. See "Network Rendering menu" on page 31.

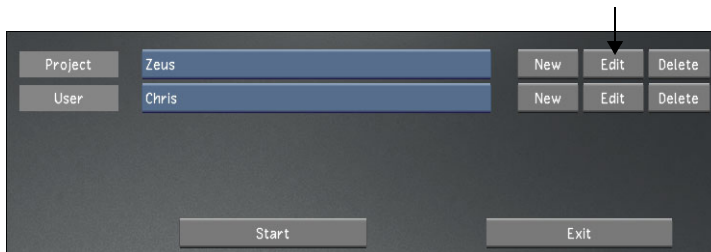
3. Click Project and enter a name in the Project Template field.



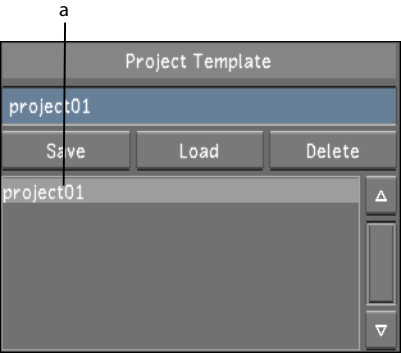
4. To save the template, click Save in the Project Template group.
5. Click Exit Project.

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.



- From the Project Template list, select a template.



a) Project Template list

- Click Load.
- 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.

- Configure the template. Use Reset to reset all settings, from every menu, to their defaults.

Select:	To configure:
Project	The name and location of the project. See "Project menu" on page 20.
Calibration	Default system calibration settings and define the default Print LUTs for a project. See "Calibration menu" on page 22.
Rendering	Default rendering settings for a project. See "Rendering menu" on page 24.
Engineering	Default video and graphics settings for a project. See "Engineering menu" on page 27.
Network Rendering	Default Backburner/Burn, Slave Render, and Wiretap directory paths and settings for a project. See "Network Rendering menu" on page 31.

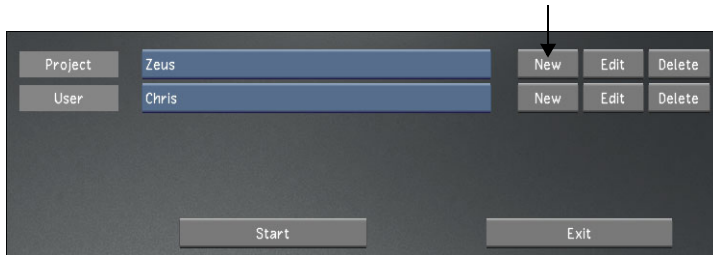
- To save the template, click Save in the Project Template group.
- Click Exit Project.

Editing a Project Template

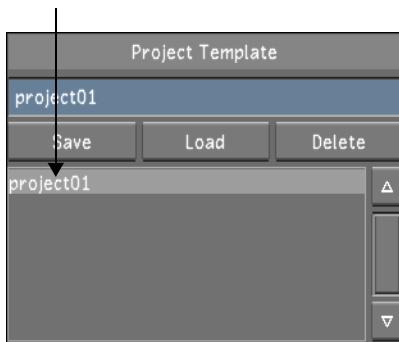
As you create project after project, there may come a time when you need to change a few of the settings and you do not want to create an entire new template. You can accomplish this by using the edit feature.

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.

Select:	To configure:
Project	The name and location of the project. See "Project menu" on page 20.
Calibration	Default system calibration settings and define the default Print LUTs for a project. See "Calibration menu" on page 22.
Rendering	Default rendering settings for a project. See "Rendering menu" on page 24.
Engineering	Default video and graphics settings for a project. See "Engineering menu" on page 27.
Network Rendering	Default Backburner/Burn, Slave Render, and Wiretap directory paths and settings for a project. See "Network Rendering menu" on page 31.

5. To save the template, click Save in the Project Template group.

6. Click Exit Project.

Creating a User Template

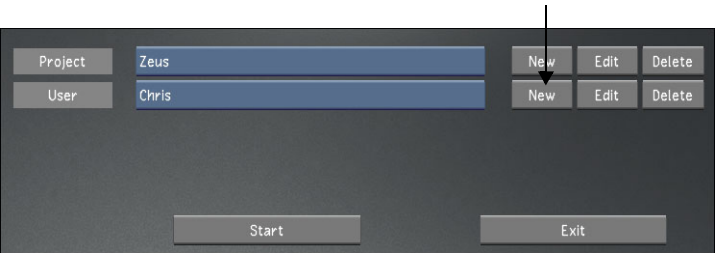
When creating a user, you can save the setup as a template for other users.

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:

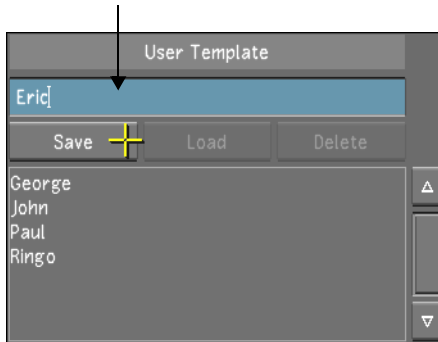
1. Do one of the following:
 - From the Main menu, click Setup, then Settings, and then click New in the User group.
 - From the Lustre splash screen, click New in the User group.



2. Configure the new user. Use Reset to reset all settings, from every menu, to their defaults.

Select:	To configure:
Display & Interface	The default display and interface settings for this user. See "Display & Interface Settings" on page 38.
System & Menu	The system, match grade, assemble, and playout settings for this user. See "System & Menu Settings" on page 40.
Tools	Keyframe interpolation, animation, and dust settings for this user. See "Tools Settings" on page 43.

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:

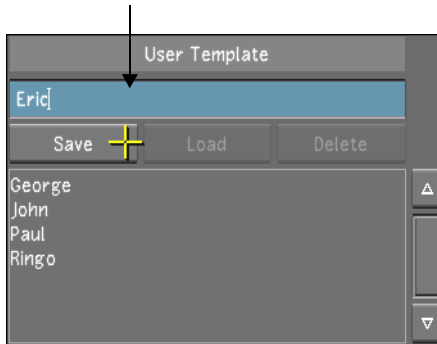
1. Do one of the following:
 - From the Main menu, click Setup, and then Settings. Select the baseline user, and then click Edit in the User group.
 - From the Lustre splash screen, select the baseline user, and then click Edit in the User group.



2. Configure the new template. Use Reset to reset all settings, from every menu, to their defaults.

Select:	To configure:
Display & Interface	The default display and interface settings for this user. See "Display & Interface Settings" on page 38.
System & Menu	The system, match grade, assemble, and playout settings for this user. See "System & Menu Settings" on page 40.
Tools	Keyframe interpolation, animation, and dust settings for this user. See "Tools Settings" on page 43.

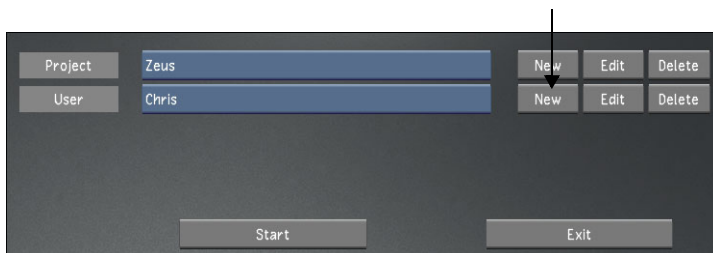
3. Click Display & Interface and enter a new name in the User Template field.



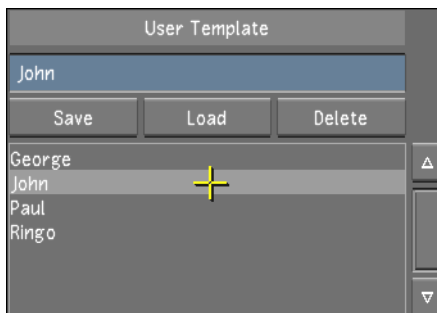
4. To save the template, click Save in the User Template group.
5. Click Exit User.

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.

Select:	To configure:
Project	The name and location of the project. See "Project menu" on page 20.
Calibration	Default system calibration settings and define the default Print LUTs for a project. See "Calibration menu" on page 22.
Rendering	Default rendering settings for a project. See "Rendering menu" on page 24.
Engineering	Default video and graphics settings for a project. See "Engineering menu" on page 27.
Network Rendering	Default Backburner/Burn, Slave Render, and Wiretap directory paths and settings for a project. See "Network Rendering menu" on page 31.

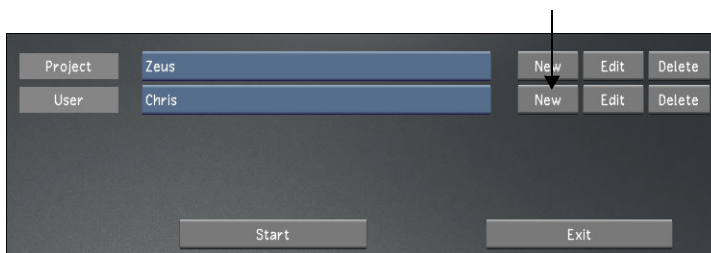
6. From the User Template list, select the new template.
7. To save the template, click Save in the User Template group.
8. Click Exit User.

Editing a User Template

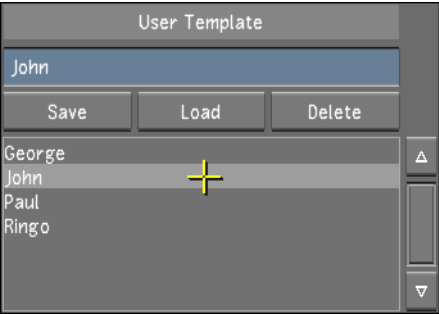
The template editing feature is available for you to make minor changes to existing templates as they are needed.

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.

Select:	To configure:
Project	The name and location of the project. See "Project menu" on page 20.
Calibration	Default system calibration settings and define the default Print LUTs for a project. See "Calibration menu" on page 22.
Rendering	Default rendering settings for a project. See "Rendering menu" on page 24.
Engineering	Default video and graphics settings for a project. See "Engineering menu" on page 27.
Network Rendering	Default Backburner/Burn, Slave Render, and Wiretap directory paths and settings for a project. See "Network Rendering menu" on page 31.

5. To save the template, click Save in the User Template group.
6. Click Exit User.

Recommended Directory Structure for Projects

Lustre stores all project metadata in sub-directories of the main project directory, Project Home. The metadata is saved per scene, each scene having its sub-directory. The sub-directory is created automatically and is named `sacc_data`.

Original footage and render files, both full and half resolution, can be stored per scene in the Project Home directory, or they can be stored in separate directories, as decided at project configuration. 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.

The following tables provide examples of each of these possible scenarios.

NOTE: If configuring Lustre on an Autodesk Incinerator™ high-speed network, see [“Incinerator-Specific Project Management Configuration”](#) on page 14.

Scanned Footage and Renders Stored in Project Home

In this example, the Project Home is defined in Windows as the *H:* drive, whereas in the Linux version it is defined as the */mnt/md0/* mount point.

When using this structure, you can first create scenes in Lustre, which then generates directories for these scenes in the Project Home. You create the directories for your scans manually. The render directories related to the scenes are created automatically by Lustre. Alternatively, you can manually create the scene’s directory under the Project Home in Lustre. Later you can define scenes in Lustre with the exact names you gave to the directories. Lustre will recognize them as existing scene directories and create the required metadata directories and files inside them. See [“Working With Scenes”](#) on page 47.

Windows Folder:	Linux Directory:	Description:
<i>H:\<project name></i>	<i>/mnt/md0/<project_name></i>	Project Home directory. You do not necessarily have to create this off the root.
<i>H:\<project name>\<scene name></i>	<i>/mnt/md0/<project_name>/<scene_name></i>	Scene directory.
<i>H:\<project name>\<scene name>\sacc_data</i>	<i>/mnt/md0/<project_name>/<scene_name>/sacc_data</i>	Directory where most metadata associated with the scene is stored.
<i>H:\<project name>\<scene name>\<scans folder></i>	<i>/mnt/md0/<project_name>/<scene_name>/<scans_directory></i>	Directory in which you create sub-directories to store original footage.
<i>H:\<project name>\<scene name>\grd</i>	<i>/mnt/md0/<project_name>/<scene_name>/grd</i>	Directory in which Lustre creates sub-directories for render files.

The following table shows the recommended sub-directory structure for scans. In this table, *H:\... * in the Windows column, and */mnt/md0/.../* in the Linux column, respectively, represent *H:\<project name>\<scene name>* and */mnt/md0/<project_name>/<scene_name>*, as shown in the previous table.

Windows Folder:	Linux Directory:	Description:
<i>H:\...\<scans folder>\<shot name></i>	<i>/mnt/md0/.../<scans_directory>/<shot_name></i>	Directory in which sub-directories for different resolutions of the scans are located.
<i>H:\...\<scans folder>\<shot name>\<resolution></i>	<i>/mnt/md0/.../<scans_directory>/<shot_name>/<resolution></i>	Directory in which the actual scans are stored. Half- and full-resolution footage should both be at this level, in different directories. The directory name must consist of the resolution expressed in numbers with a lowercase x in between. Example: 2048x1556. If you generate proxies in Lustre, the proxies directory is created automatically.

The following table shows the sub-directory structure that Lustre creates for renders. The structure differs depending on the Render Place option specified when rendering. See [“Specifying the Destination for Local Render Files”](#) on page 471. In this table, *H:\...* in the Windows column and */mnt/md0/.../* in the Linux column, respectively, represent *H:\<project name>\<scene name>* and */mnt/md0/<project_name>/<scene_name>*, as shown in the first table.

Windows Folder:	Linux Directory:	Description:
<i>H:\...\grd\<shot name_grdxx></i>	<i>/mnt/md0/.../grd/<shot_name_grdxx></i>	Directory in which sub-directories for different resolutions of the renders are located when using the Normal or No Wedge Render Place option. In the example, xx is the grade number.
<i>H:\...\grd\<shot name_grdxx>\<resolution></i>	<i>/mnt/md0/.../grd/<shot_name_grdxx>/<resolution></i>	Directory in which the actual renders are stored when using the Normal or No Wedge Render Place option. The directory name corresponds to the resolution of the render files, for example, 1280x1024.
<i>H:\...\grd\<grdxx></i>	<i>/mnt/md0/.../grd/<grdxx></i>	Directory in which the actual renders are stored when using the One Sequence Render Place option. In the example, xx is the grade number.

Scanned Footage and Renders Stored Separately from Project Home

In this example, the Project Home is defined in Windows as the *H:* drive, whereas in the Linux version it is defined as the */mnt/md0/* mount point.

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.

Windows Folder:	Linux Directory:	Description:
<i>H:\<project name></i>	<i>/mnt/md0/<project_name></i>	Project Home directory.
<i>H:\<project name>\<scene name></i>	<i>/mnt/md0/<project_name>/<scene_name></i>	Scene directory.
<i>H:\<project name>\<scene name>\sacc_data</i>	<i>/mnt/md0/<project_name>/<scene_name>/sacc_data</i>	Directory where most metadata associated with the scene is stored.
<i>G:\films\myfilm\<scans folder></i>	<i>/mnt/fileserver/films/myfilm/<scans_directory></i>	Directory in which you create sub-directories to store original footage.
<i>G:\films\myfilm\grd</i>	<i>/mnt/fileserver/films/myfilm/grd</i>	Directory in which Lustre creates sub-directories for render files.

The following table shows the recommended sub-directories structure for scans. For the sake of simplicity, in this table, *G:\...* represents *G:\films\myfilm*, and */mnt/fileserver/.../* represents */mnt/fileserver/films/myfilm* as shown in the previous table.

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*.

Windows Folder:	Linux Directory:	Description:
<i>G:\...\<scans folder>\<shot name></i>	<i>/mnt/fileserver/.../ <scans_directory>/ <shot_name></i>	Directory in which sub-directories for different resolutions of the scans are located.
<i>H:\...\<scans folder>\<shot name>\<resolution></i>	<i>/mnt/fileserver/.../ <scans_directory>/ <shot_name>/<resolution></i>	Directory in which the actual scans are stored. Half- and full-resolution scans should both be at this level, in different directories. The directory name must consist of the resolution expressed in numbers with a lowercase x in between. Example: 2048x1556. If you generate proxies in Lustre, the proxies directory is created automatically.

The following table shows the sub-directory structure that Lustre creates for renders. The structure differs depending on the Render Place option specified when rendering. See [“Specifying the Destination for Local Render Files”](#) on page 471. For the sake of simplicity, in this table, *G:\...* represents *G:\films\myfilm*, and */mnt/fileserver/.../* represents */mnt/fileserver/films/myfilm* as shown in the first table in this section.

Windows Folder:	Linux Directory:	Description:
<i>G:\...\grd\ <shot name_grdxx></i>	<i>/mnt/fileserver/.../grd/ <shot_name_grdxx></i>	Directory in which sub-directories for different resolutions of the renders are located when using the Normal or No Wedge Render Place option. In the example, xx is the grade number.
<i>G:\...\grd\ <shot name_grdxx>\<resolution></i>	<i>/mnt/fileserver/.../grd/ <shot_name_grdxx>/ <resolution></i>	Directory in which the actual renders are stored when using the Normal or No Wedge Render Place option. The directory name corresponds to the resolution of the render files, for example, 1280x1024.
<i>H:\...\grd\<grdxx></i>	<i>/mnt/fileserver/.../grd/ <grdxx></i>	Directory in which the actual renders are stored when using the One Sequence Render Place option. In the example, xx is the grade number.

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 '...'.

Project Name	New_Project	
Project Home	C:\	...
Scans Full Home	C:\	...



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 Name	Zeus	
Project Home	<HOME>\Projects\[PROJECT_NAME]	...
Scans Full Home	<HOME>\Scans	...
Scans Half Home	<HOME>\Scans	...
Renders Full Home	<HOME>\Projects\[PROJECT_NAME]	...
Renders Half Home	<HOME>\Projects\[PROJECT_NAME]	...

Summary

<i>Overview of Basic Functionality for Lustre</i>	85
<i>Starting and Exiting Lustre</i>	85
<i>Understanding the Lustre User Interface</i>	86
<i>Working with Control Surfaces</i>	88
<i>Resetting Parameters</i>	89
<i>Bypassing Menu Parameters</i>	90
<i>Undoing and Redoing Actions</i>	91
<i>Confirming and Cancelling Actions</i>	92
<i>Assigning Notes to Shots</i>	93
<i>Flagging Shots</i>	94
<i>Creating Shot Groups</i>	97
<i>Copying Parameters with the Selector</i>	101
<i>Assigning Values With the Standard Lustre Calculator</i>	103
<i>Grade Bins: Saving, Applying, and Deleting Intermediary Grades</i>	105
<i>Using Audio</i>	113
<i>Using Text</i>	113

Overview of Basic Functionality for Lustre

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. The chapter also explains how to use audio and text, features that exist outside the colour grading pipeline.

Starting and Exiting Lustre

To start Lustre, use any of the standard methods for opening an application in Windows or Linux.

To start Lustre:

- 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, open a shell and navigate to the Lustre directory. Type:

```
cd /usr/Autodesk/lustre2009/
```

Next, start Lustre. Type:

```
./lustre to start the Master Station, or ./assistant to start the Lustre Station.
```

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.

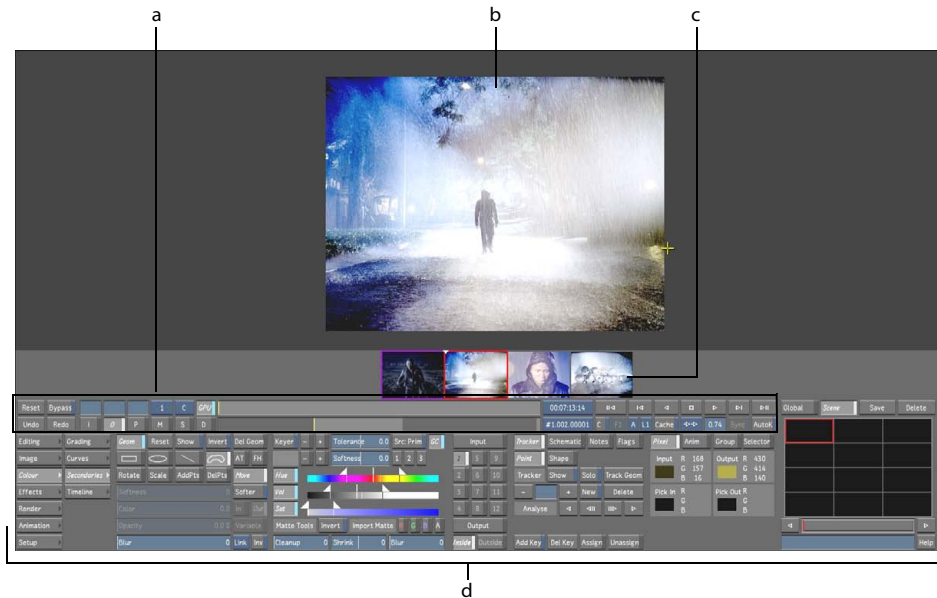
NOTE: If the Windows version of Lustre cannot find a license, the Flex LM® wizard appears, requesting the location of the license file. If the Linux version of Lustre cannot find the license for the application, it will not launch. If plugin licenses are missing, these will be displayed in the Linux shell.

To exit Lustre:

1. In the Main menu, click Setup.
2. Click Exit and confirm.

Understanding the Lustre User Interface

The Lustre user interface is divided into four main sections.



a) Player display controls

b) Player

c) Storyboard

d) Menu

Image courtesy of Moviworld / UK File & TV Company / Videolab

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 Shot bin and file browser in order to select footage to add to the Storyboard. See [Chapter 6, “Browsing for Footage.”](#) on page 131.
- View and modify EDL details. See [“Working with EDL, ALE, and Cutlist Files”](#) on page 143.
- Display Colour menus. See [“Displaying Colour Menus in the Player”](#) on page 225.
- Monitor capture progress. See [“Capturing Material”](#) on page 514.

The Storyboard

Use the Storyboard to select shots, arrange shots, navigate through shots, and isolate certain types of shots. See [“Navigating through Shots”](#) on page 226 and [“Storyboard Viewing Options”](#) on page 230.

Player Display Controls

Below the Storyboard, there are controls for:

- Resetting the current menu. See [“Resetting Parameters”](#) on page 89.
- Bypassing the current menu. See [“Bypassing Menu Parameters”](#) on page 90.
- Undoing and redoing actions. See [“Undoing and Redoing Actions”](#) on page 91.
- Setting play and view options for shots. See [Chapter 8, “Playing, Viewing and Sorting Shots,”](#) on page 219.
- Adjusting printer light settings. These display only when a Colour menu is active. See [“Adjusting Printer Lights for Primary Grading”](#) on page 317.
- Setting keyframes automatically. See [“Animating Reposition Values”](#) on page 276 and [“Animating with Keyframes”](#) on page 428.

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”](#) on page 93.
- The Flags system. See [“Flagging Shots”](#) on page 94.
- Grouping functionality. See [“Creating Shot Groups”](#) on page 97.
- The Selector. See [“Copying Parameters with the Selector”](#) on page 101.
- The standard Lustre calculator. See [“Assigning Values With the Standard Lustre Calculator”](#) on page 103.
- The Grade bin. See [“Grade Bins: Saving, Applying, and Deleting Intermediary Grades”](#) on page 105.

Working with Control Surfaces

Lustre supports the Tangent CP100 control surface and the Autodesk control surface. Both types of control surfaces improve interactivity when colour grading film and video footage, and both allow you to accomplish many of the tasks you do in the Lustre user interface.

The Tangent CP100 control surface is available for the Master Station only. You can use it for a wide range of tasks, such as customizing the view, navigating the Storyboard, and performing primary, curves, and secondary grading.

The Autodesk control surface divides its functionality among three separate surfaces. The control surface designed for performing primary colour grading is available for any Lustre configuration. The Master Station supports all three control surfaces.

See the *Autodesk Control Surface User Guide*.

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.

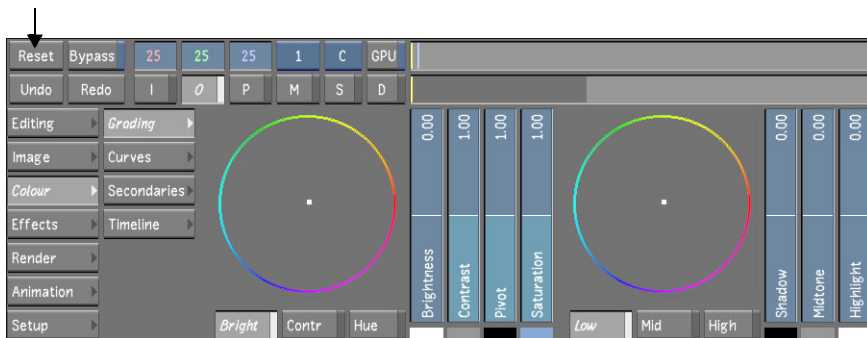
You can also reset parameters with the Autodesk control surface or with the Tangent CP100 control surface. See the *Autodesk Control Surface User Guide*.

To reset a parameter:

- 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 (or press **R**), 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.

HINT: 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.

To reset multiple shots to their original state:

1. Right-click to select the shots you want to reset.
2. Double-click an empty grade in the Grade bin.

The selected shots are reset to their original state.

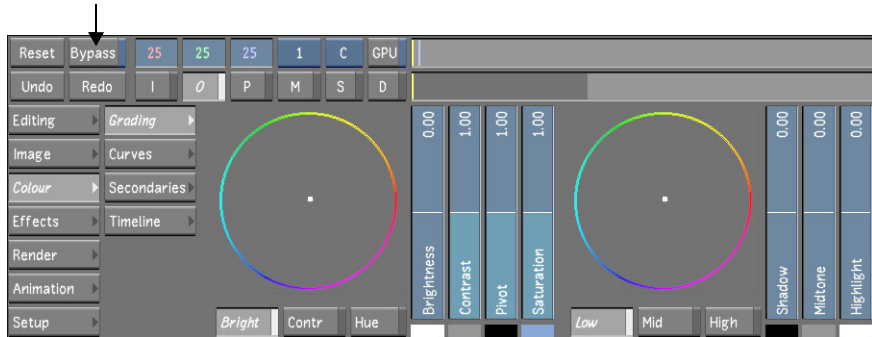
Bypassing Menu Parameters

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**.
 - Press **F9** on the control panel.

Changes made to the current menu are temporarily turned off.

4. Enable Bypass again to turn the menu back on.

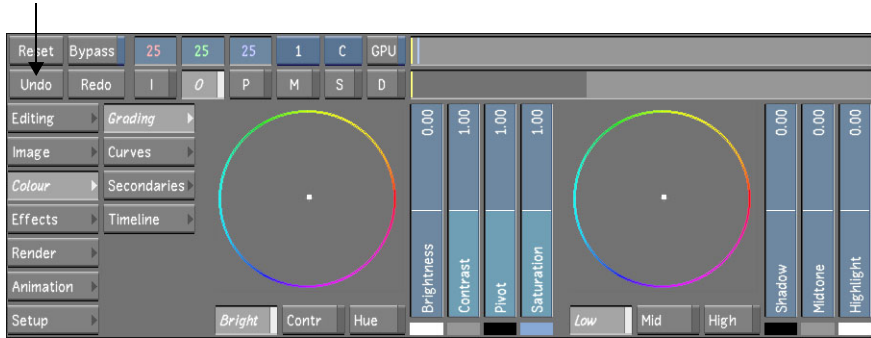
Undoing and Redoing Actions

Each shot in Lustre has its own undo and redo history.

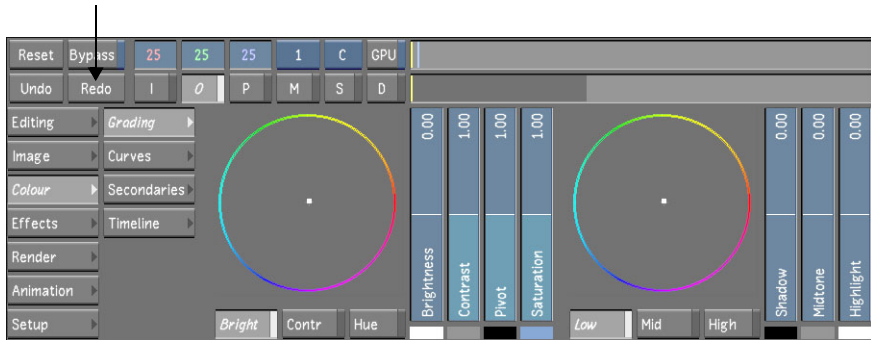
You can undo most operations in Lustre. You can use Redo immediately after undoing an operation.

To undo an operation:

- After performing an operation that you want to undo, click Undo (or press **U**).

**To redo an operation:**

- After undoing an operation, click Redo.



You can also undo and redo operations with the Autodesk control surface or with the Tangent CP100 control surface. See the *Autodesk Control Surface User Guide*.

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 command button turns red after you click it. If you click the button again, the action commences.

If you do not want the action to begin (for example, you clicked the button in error), click anywhere in the grey area of the menu. This cancels the action.

Assigning Notes to Shots

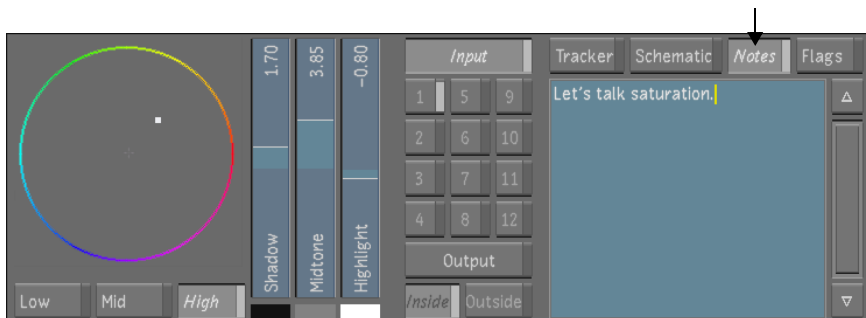
The Notes window is available from almost every menu and allows you to enter unlimited wrapping text pertaining to the current shot. Notes are retained as you jump from one shot to another, and saved when you save the grade.

Notes are saved to an .XML file in the scene's *sacc_data* 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 *sacc_data* directory if any changes were made. See [Appendix A, “XML in Lustre,”](#) on page 607.

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.
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 for a Cut”](#) on page 55.

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 four types:

Render — Flags the shots to be rendered.

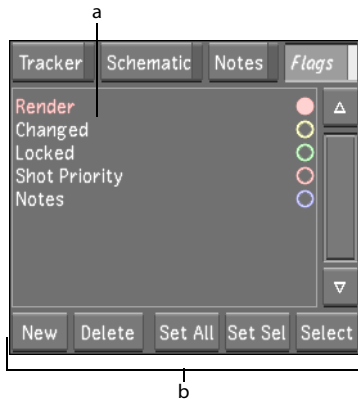
Changed — Indicates whether or not the current shot has been modified since the last save.

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. See [“Shot Priority”](#) on page 207.

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”](#) on page 93.



a) Flags list

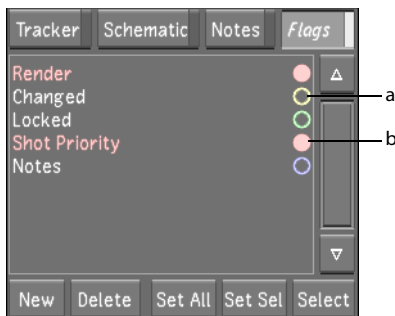
b) Flag controls

To access the Flags window:

- Enable the Flags button.
This displays the flags for the current shot.

To enable or disable a flag:

- 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

HINT: 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 permitted 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 *sacc_data* 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:

- 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”](#) on page 193.
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”](#) on page 97.
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.

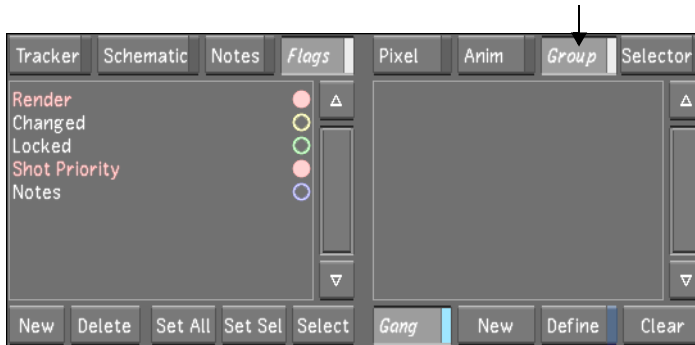
Grouping Shots

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.



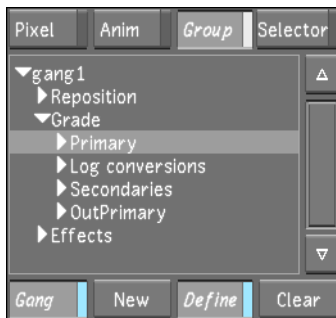
NOTE: You can access the Group button from any menu except the Animation menu.

2. Click New.

A new group called gang1 is created.

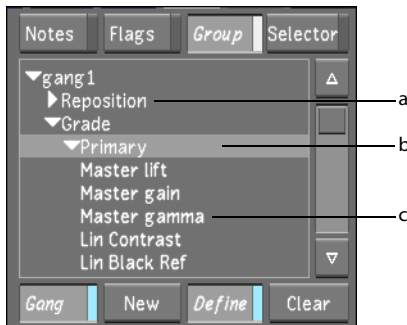
3. Enable Define.

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

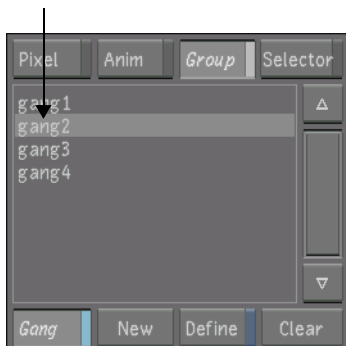
6. When you have finished setting up the group, disable Define.
7. You can name the group by middle-clicking the group and entering a new name.
NOTE: Spaces are not permitted in naming conventions. It is recommended that underscores be used instead of spaces when working on either Linux or Windows.
8. You can delete a group by selecting it and pressing **CTRL+BACKSPACE**.
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.

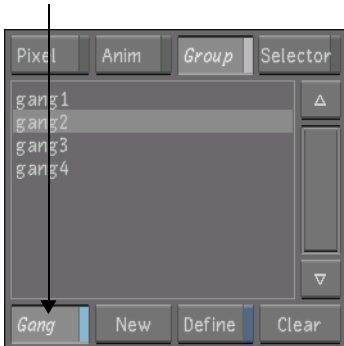
To apply changes to all shots in a group:

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.

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 all too dark. You can select all the shots, adjust the brightness for the current shot, 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”](#) on page 308.

You can define which channels are copied between shots. For example, if you blurred a shot and adjusted its hue, you can choose to copy only the hue change to the other selected shots. You can copy the following channel parameters between shots using the Selector:

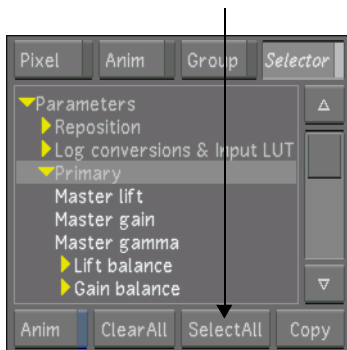
- Colour grading parameters
- Input LUT selections
- Repositioning values
- Tracker data
- Effects
- Render flag settings

To copy channel parameters from the current shot to selected shots:

1. Click Selector.



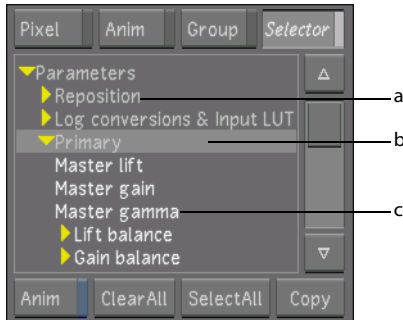
2. Right-click the shots that you want to select. To select all the shots in the Storyboard, click Select All.



The selected shots are highlighted in aqua.

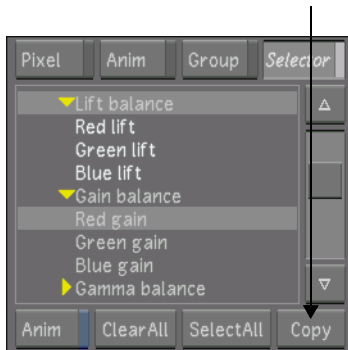
3. Click the shot to which you want to make modifications.
This shot is the current shot and it is highlighted in red.
4. Specify what parameters to copy by selecting their channels in the Selector channel list:
 - To select all the channels in a directory, select the directory.
 - To select specific channels in a directory, expand the directory and select any of its channels.

Selected channels turn white.



a) Collapsed directory b) Selected expanded directory c) Selected channel

5. Adjust any of the parameters for the current shot or apply a grade to the current shot from the Grade bin. See [“Grade Bins: Saving, Applying, and Deleting Intermediary Grades”](#) on page 105.
6. Click Copy or press **C**.



The parameters for the selected channels are copied to the selected shots.

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.

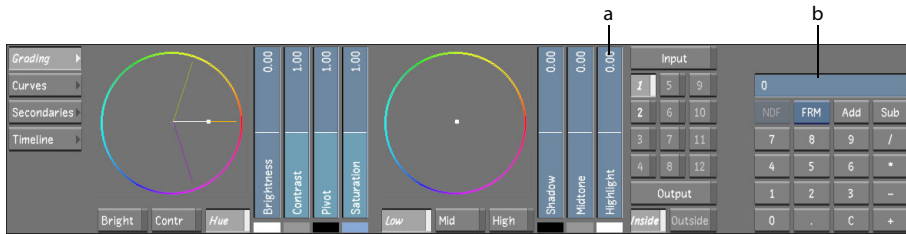
Sliders — Adjust the value of the currently selected slider.

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”](#) on page 229.



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 hot keys 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.

Press:	To:
ENTER or click the calculator's active numeric field	Calculate the value, apply it, and close the calculator.
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.

Grade Bins: Saving, Applying, and Deleting Intermediary Grades

Store shot settings to a Grade bin when you need intermediary grades or scratch pad memories. Intermediary grades are useful when you want to experiment with different looks. Intermediary grades are saved on a shot-by-shot basis and contain all shot settings.

You can also use Grade bin thumbnails as reference images. See [“Viewing Reference Images”](#) on page 236.

Saving Intermediary Grades

You can save intermediary grades to the Grade bin. When you save intermediary grades, all the settings are saved with a thumbnail for identification. Save your shot settings for use within a scene or throughout the entire project—by saving them to the Scene Grade bin or the Global Grade bin, respectively.

To save intermediary grades to a Grade bin:

- 1. Using the various options on the menus, modify a shot to your satisfaction.
- 2. Select the Grade bin you want to use.

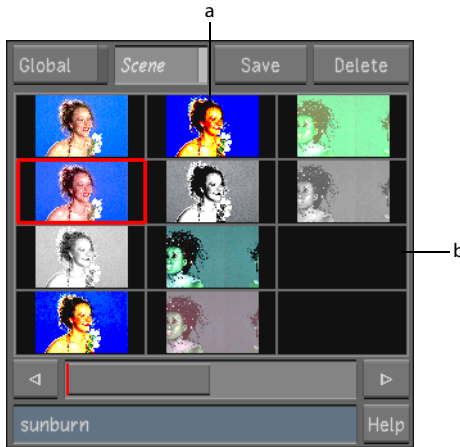


Image courtesy of The House

Click:	To save:
Scene	Shots for use within the current scene.
Global	Shots for use throughout the entire project.

- 3. 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.

4. Do one of the following:
 - To create an intermediary grade, select an unused storage container.
 - To save a new version of an intermediary grade, select the last saved version.



- a) Used storage container b) Unused storage container

Image courtesy of The House

The red line in the slider indicates the current position.

NOTE: When you add a grade to a position 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.

5. Enter a name in the Name field.

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.

6. Click Save.

The settings from the current shot in the timeline are saved to the intermediary grade and a thumbnail of the current frame appears in the Grade bin.

HINT: You can use the thumbnail as a reference image while you work on another shot. See [“Viewing Reference Images”](#) on page 236.

To save intermediary grading information from several shots to the Grade bin:

1. From the Storyboard or the Multi-Layer Timeline, select 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”](#) on page 193.

NOTE: If you do not select any shots, the entire timeline’s grading information is saved.

2. In the Grade bin, select the storage container where you wish to save your grade.
3. Make sure the positioner is over a shot. If you are in the Storyboard view, see [“Navigating Using the Timebar and Storyboard”](#) on page 227. If you are in the Multi-Layer Timeline view, see [“Timeline Navigation Controls”](#) on page 191.
4. Hold down the **SHIFT** key while pressing the Save button.



WARNING: If you are saving more than one grade to the Grade bin, each grade is saved into one storage container starting from the selected container and proceeding column by column from left to right. Therefore, be aware of the storage containers whose 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 this grade.

5. If prompted, confirm or cancel the action, as required. See [“Confirming and Cancelling Actions”](#) on page 92.

The selected grades (or all grades if no shots are selected) are saved to the Grade bin.

Applying Intermediary Grades to Shots

Apply intermediary grades to your shots when you want to experiment with different looks. You can apply intermediary grades to one or multiple shots in the Storyboard. 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.

HINT: 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”](#) on page 231 and [“Creating Shot Groups”](#) on page 97.

2. Select the Grade bin you want to use.

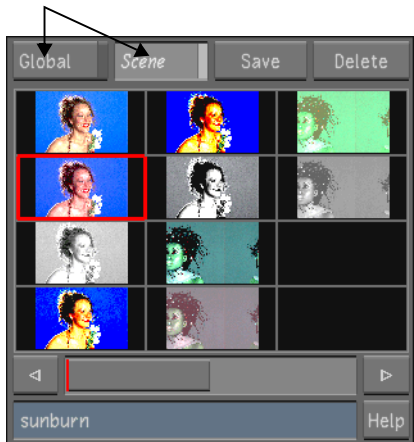


Image courtesy of The House

Click:	To access:
Scene	Intermediary grades saved for the current scene only.
Global	Intermediary grades saved for the entire project.

3. Scroll through the Grade bin to the thumbnail representing the intermediary grade you want to load. To scroll through a Grade bin, middle-click and drag right or left, or use the horizontal slider located immediately below the Grade bin.

HINT: To display an enlarged image of the thumbnail, place your mouse cursor over a thumbnail without clicking.

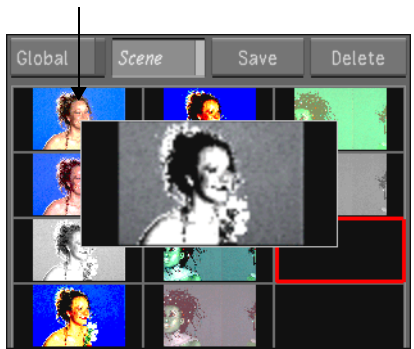


Image courtesy of The House

4. To view the name of a thumbnail, click the thumbnail.

The thumbnail is outlined in red and its name appears in the Name field.

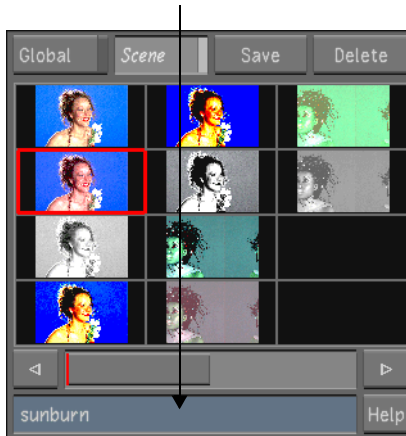


Image courtesy of The House

5. Do one of the following:

- To load all the settings in the intermediary grade to the current shot only, double-click a thumbnail.

HINT: Double-click an unused storage container in the Grade bin to reset selected shots to their default settings.

- To load all the settings in the intermediary grade to selected shots, press **SHIFT** and double-click a thumbnail.
- To load the intermediary grade settings to the current shot only from the current menu only, press **CTRL** and double-click a thumbnail. For example, if the Grading menu is displayed, the saved settings from the Grading menu are applied.
- To load the intermediary grade settings to selected shots from the current menu only, press **CTRL+SHIFT** and double-click a thumbnail. For example, if the Grading menu is displayed, the saved settings from the Grading menu are applied.
- To load the intermediary grade settings that are selected in the Selector to the current shot only, press **ALT** and double-click a thumbnail. See [“Copying Parameters with the Selector”](#) on page 101.
- To load the intermediary grade settings that are chosen in the Selector to a selection of shots, press **ALT+SHIFT** and double-click a thumbnail. See [“Copying Parameters with the Selector”](#) on page 101.

The intermediary grade is applied. In the Player, the image is updated to reflect the current settings.

HINT: To revert to the previous settings, click Undo.

Deleting Intermediary Grades

You can delete intermediary grades one at a time, or delete all grades from the Grade bin simultaneously. When you delete an intermediary grade, the thumbnail is also deleted.

To delete intermediary grades from a Grade bin one at a time:

1. Select the Grade bin from which you want to delete grades.

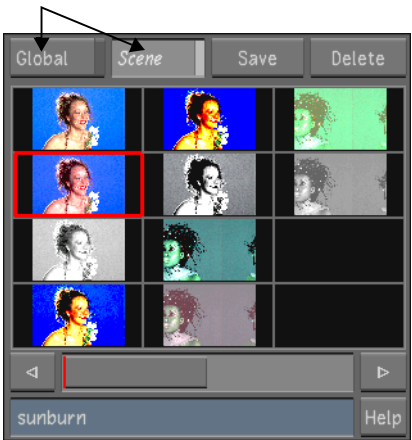


Image courtesy of The House

Click:	To delete:
Scene	Intermediary grades saved for the current scene only.
Global	Intermediary grades saved for the entire project.

2. Scroll through the Grade bin to the thumbnail you want to delete. To scroll through a Grade bin, middle-click and drag right or left.
3. Click the thumbnail you want to delete.
The name of the selected thumbnail appears in the Name field.
4. Click Delete.
The intermediary grade and associated thumbnail are deleted.

To delete all intermediary grades from the Grade bin simultaneously:

1. Select the Grade bin from which you want to delete grades.

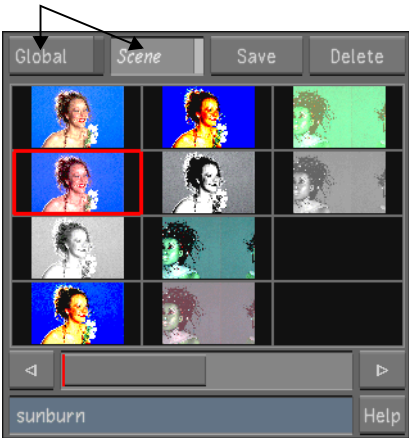


Image courtesy of The House

Click:	To delete:
Scene	Intermediary grades saved for the current scene only.
Global	Intermediary grades saved for the entire project.

2. Hold down **SHIFT** while clicking Delete.
- The Delete All button appears as a red confirmation button.

3. Do one of the following:
 - Click Delete All to clear the Grade bin.
 - Click outside the button to cancel the operation.

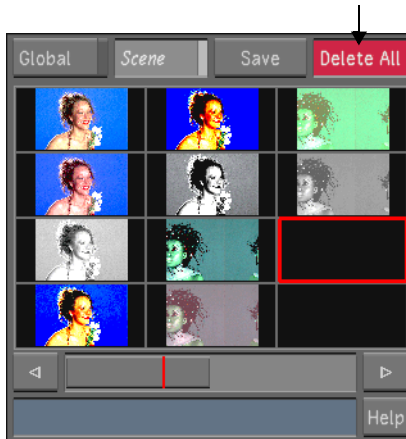


Image courtesy of The House

Using Audio

You can hear audio during timeline playback if you import or capture a WAV 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 WAV file can be associated with the timeline.

The supported PCM (pulse-mode modulation) WAV encoding type is 48.00kHz/24 bit stereo.

Using Text

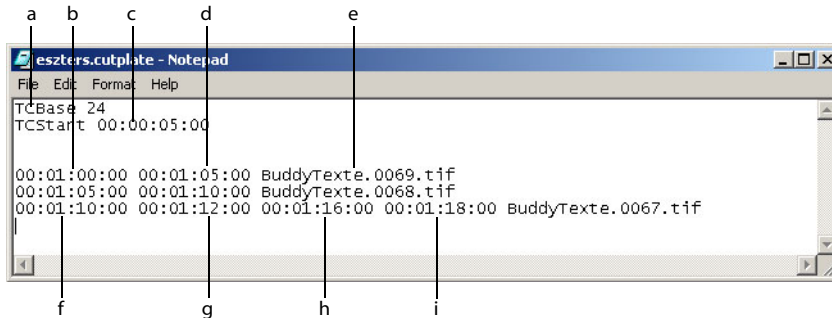
When you want to add credits to a cut loaded to the timeline, you can automatically key text files containing an alpha channel over shots in the timeline. To do so, use a text plate EDL that describes how the text files are arranged in the timeline.

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 *sacc_data* directory—where the loaded cut file is stored. Store the text files in the same scene's *sacc_data\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)	d) Record out timecode	g) Fade-in end timecode
b) Record in timecode	e) Text file name	h) Fade-out start timecode
c) TCStart (timecode offset)	f) Record in timecode and start of fade-in	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.

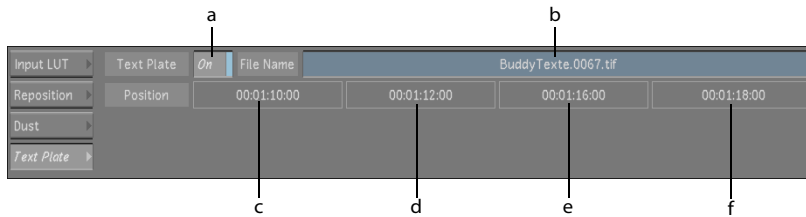
Timecode — List the record timecode for each referenced text file. Set the record in timecode and record out timecode, as well as durations for fade-ins and fade-outs.

Use the Text Plate menu to turn text plates on or off.

To turn a text plate on:

1. Load the cut file with the text plate to the timeline. See [“Loading a Cut”](#) on page 50.
2. In the Image menu, click Text Plate.

The Text Plate menu appears.



- a) On/Off button c) Record in timecode and start of fade-in e) Fade-out start timecode
- b) Text file name d) Fade-in end timecode f) Record out timecode and end of fade-out

3. In the timeline, go to the timecode where the text plate appears.

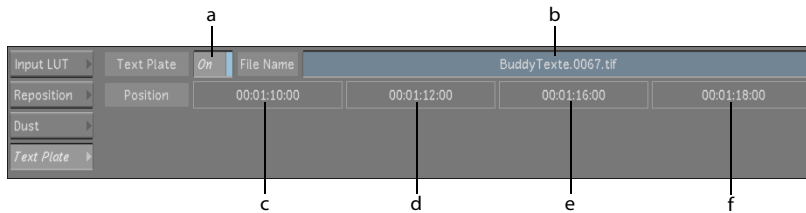
4. Set the On/Off button to On.

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.



- a) On/Off button c) Record in timecode and start of fade-in e) Fade-out start timecode
- b) Text file name d) Fade-in end timecode f) Record out timecode and end of fade-out

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.

5

Monitor Calibration

Summary

About Monitor Calibration	117
Adjusting Black Level and Picture Settings	121
Selecting the Project Colour Space	123
Calibrating a Monitor	123
Calibrating for Dual-Head Mode	127
Applying Print LUTs for Viewing	127
Applying 3D Mesh LUTs	129

About Monitor Calibration

All monitors display colours differently. The colours that you see on your monitor change slightly over time because the intensity of light generated by Cathode Ray Tubes (CRTs) can drift. 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 monitor calibration settings will default back to the settings configured in User and Project Management.

When you calibrate your monitor in Lustre, the image that you view is an exact 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 accurately displayed—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 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.

Alternatively, you can apply a 3D Mesh LUT to set up the proper display environment. With a 3D Mesh 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.

Lookup Tables

A LUT is an ASCII (*.lut*) file that describes specific pixel values used for image data conversion both on input and output. A Print LUT is obtained from your film lab and is updated on a regular basis. If running the Windows version of Lustre, save print LUTs in the *C:\Program Files\Autodesk\lustre 2009\lut* folder. If running the Linux version of Lustre, save print LUTs in the */usr/Autodesk/lustre2009/lut* directory.

A monitor calibration LUT is applied to the monitor only and affects the display of images, not the image data itself. It is generated from one measure of gamma (white, grey, and black) or a series of progressive measures for each colour channel. The number of steps in the series is defined in the *init.config* file. See the *Autodesk Lustre Software Installation Guide*.

3D Mesh LUTs

Use 3D Mesh LUTs for advanced film simulation in Lustre. After you perform basic monitor calibration and take calibration measurements using the appropriate calibration device, you can work with your film lab to generate a 3D (*.lut*) Mesh LUT file to amalgamate all the stages of calibration into the LUT.

The following is an example of a 3D Mesh LUT.


```

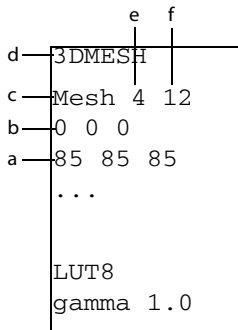
3DMESH
# 3 columns rgb
# 4913 rows
# input [0..1023]
# output [0..4095]

# 0 64 128 192 256 320 384 448 512 576 640 704 768 832 896 960 1023
Mesh 4 12

    0      0      0
    85     85     85
   170    170    170
   255    255    255
   341    341    341
   426    426    426
   511    511    511
   597    597    597
   682    682    682
   767    767    767
   853    853    853
   940    940    940
  1023   1023   1023

```

The following format is used for 3D Mesh LUTs.



- a) Mesh definition with R, G, B triplet values (second output value)
- b) Mesh definition with R, G, B triplet values (first output value)
- c) Mesh keyword (case sensitive)
- d) 3DMESH keyword (case sensitive)
- e) Input bit depth
- f) Output bit depth

In this example, the 3D Mesh LUT has an input bit depth of 4 bits and an output bit depth of 12 bits. You use the input value to calculate the RGB triplet to be $17 \times 17 \times 17$ (where $17 = 2^4 + 1$ where 4 is the input bit depth). The first triplet is the output value at (0,0,0);(0,0,1);...;(0,0,16) r,g,b coordinates; the second triplet is the output value at (0,1,0);(0,1,1);...;(0,1,16) r,g,b coordinates; and so on. You use the output bit depth to set the output bit depth range (12 bits or 0-4095).

3D LUTs are highly accurate in measuring the characteristics of a specific monitor—they cannot be used on other monitors. You should update 3D LUTs as often as necessary to

maintain an accurate display environment. No further monitor calibration is necessary. See [“Applying 3D Mesh LUTs”](#) on page 129.

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”](#) on page 281.

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” on page 121.
2. Select the colour space you intend to use for the project—logarithmic or linear.	“Selecting the Project Colour Space” on page 123.
3. If you are using a colourimeter, automatically calibrate the monitor.	“Calibrating the Monitor Automatically” on page 124.
4. If you do not have a colourimeter, manually calibrate the monitor.	“Calibrating the Monitor Manually” on page 127.
5. If your video card is set to Dual-Head mode, set LUTs for each monitor.	“Calibrating for Dual-Head Mode” on page 127.
6. Select and compare up to three Print LUTs obtained from your lab to determine which best calibrates the display with the printer.	“Applying Print LUTs for Viewing” on page 127.

If you are applying a 3D LUT, see [“Applying 3D Mesh LUTs”](#) on page 129.

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:

www.poynton.com/notes/brightness_and_contrast/index.html

To adjust the black level setting:

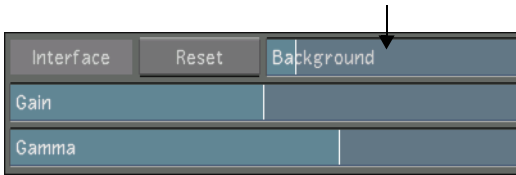
1. Start Lustre.
2. Load a black frame (RGB 0, 0, 0) to the Storyboard. See [“Moving Shots to the Storyboard”](#) on page 139.

The black frame appears in the Player. Use this image to set your black level.

HINT: You can also create a black frame (RGB 0, 0, 0) in the Curves menu by dragging the RGB curves down.

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 subtle contrast between the background colour and the black frame.

5. On the monitor, set Contrast to the minimum value.
6. On the monitor, set Brightness to the minimum value.
7. 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.

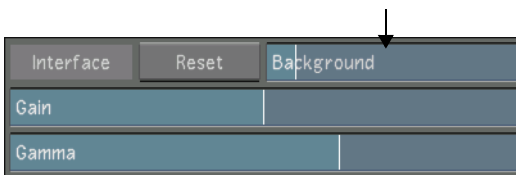
To adjust the picture setting:

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 [“Moving Shots to the Storyboard”](#) on page 139.

The white frame appears in the Player. Use this image to set your picture settings.

HINT: 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.

- 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.

HINT: 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.

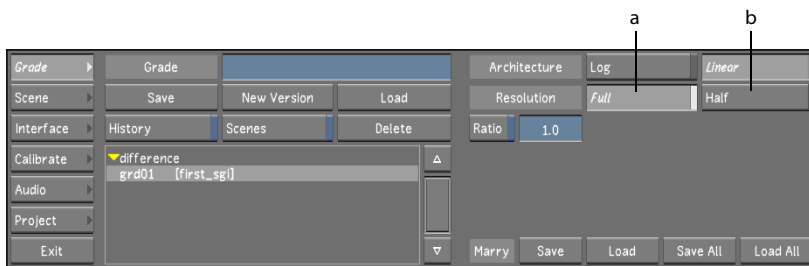
Selecting the Project Colour Space

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:

- In the Main menu, click Setup, and then click Grade.

The Grade menu appears.



a) Logarithmic colour space button

b) Linear colour space button

- Select a colour space:
 - Click Log to select the logarithmic colour space.
 - Click Linear to select the linear colour space.

Calibrating a Monitor

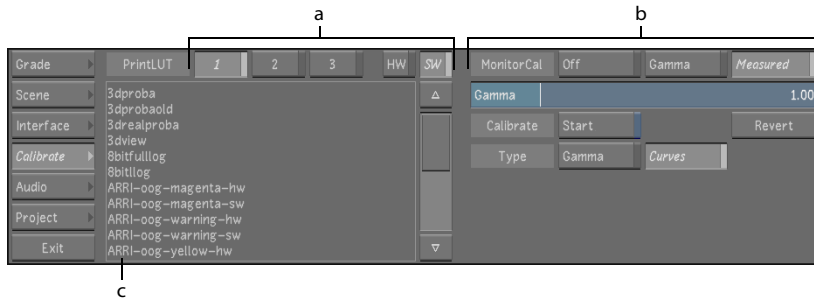
In Lustre, you can calibrate your monitor automatically or manually. Automatic monitor calibration is much more accurate than manual monitor calibration methods because manual methods hinge on perception, and your perception of the monitor can vary depending on the surrounding environment.

Access the Calibration tools in the Calibrate menu.

To access the Calibrate menu:

- 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:

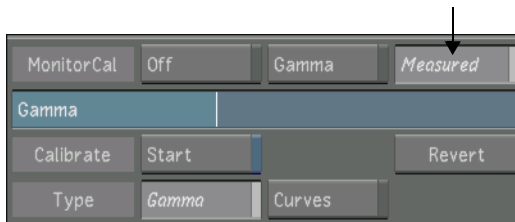
1. In the Calibrate menu, click Off to disable existing monitor calibration settings.



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.

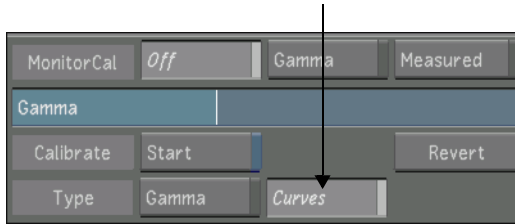
NOTE: You can set the colour steps in the *init.config* file. See the *Autodesk Lustre Software Installation Guide*.

To automatically calibrate the monitor based on a curves measure:

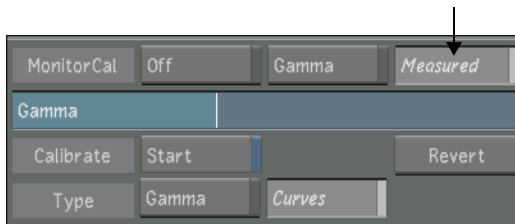
1. In the Calibrate menu, click Off to disable existing monitor calibration settings.



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.

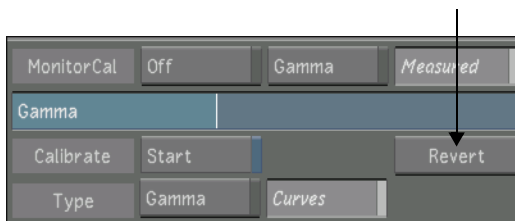


Reverting to a Previous Calibration Settings

If you are not satisfied with the results from the automatic calibration, you can revert to the previous automatic calibration.

To revert to the previous calibration settings:

- In the Calibration menu, click Revert.



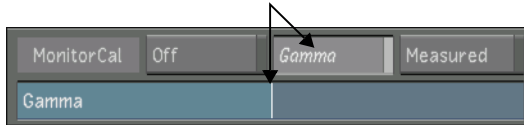
The LUT that was previously saved to the *monitor.calib* file is used to calibrate the monitor. If you click Revert again, you will apply the settings from the latest LUT.

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:

- In the Calibrate menu, enable Gamma, and then drag the Gamma slider.



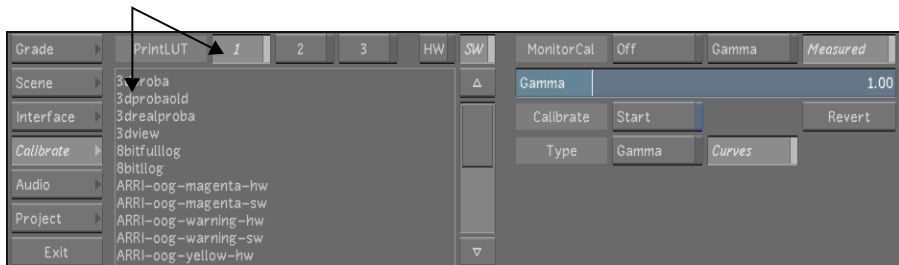
Calibrating for Dual-Head Mode

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. In the Calibrate menu, click 1 for the main monitor, enable HW, and select a hardware 3D Mesh for the main LUT.

NOTE: A hardware LUT is processed directly on the video card and therefore ensures better performance in Lustre. Only 3D Meshes are available for hardware LUTs.



2. Click 2 and select any LUT for the second monitor.

NOTE: This setup will not affect playback speed unless a 3D Mesh is also chosen for the second monitor. In this case, the playback rate will be approximately 18 fps.

Applying Print LUTs for Viewing

Use Print LUTs, obtained from your film lab, to apply preset gamma correction. Print LUTs are specially designed by your lab to ensure an accurate display of your film stock images by

simulating a film print look. Print LUTs may be updated several times a day to ensure proper calibration with the lab.

NOTE: In Lustre, LUTs used to calibrate the monitor are applied from the 16-bit result buffer and have a higher quality than those produced on the graphics card.

Selecting Print LUTs

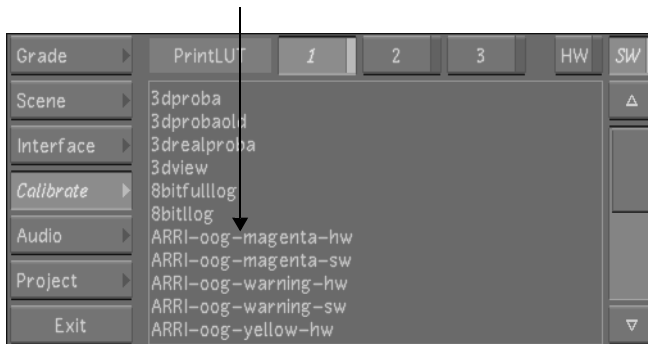
You can select up to three Print LUTs for quick comparison in a project. When your lab sends you the latest LUT, simply select it from the Calibrate menu. You can have any number of Print LUTs on the system.

To select Print LUTs for comparison:

1. In the Calibrate menu, click 1.



2. Select a Print LUT from the Print LUT list.



The Print LUT is assigned to L1.



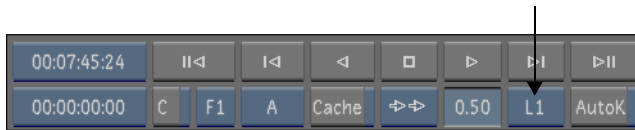
3. Repeat steps 1 and 2 to assign a Print LUT to L2 and L3.

Switching between Print LUTs

You can switch between Print LUTs to change the display of your images anytime during a session.

To switch between selected Print LUTs:

- Do one of the following:
 - In the Calibrate menu, click 1, 2, or 3.
 - Click the LUT button to toggle through the selected Print LUTs.



The selected Print LUT is applied to the display.

Applying 3D Mesh LUTs

When you use 3D Mesh LUTs, you apply either the hardware 3D Mesh LUT or software 3D Mesh LUT. The hardware 3D LUT is an interactive viewing LUT that works through the NVIDIA® graphics card. The hardware 3D LUT displays how the colour graded images will look on film stock as you work, without impeding processing speed.

The software 3D LUT is a viewing LUT that can also be applied to the video output (such as a preview monitor). Use the software 3D LUT when you want to view your work on a video monitor—the software 3D LUT will simulate one destination medium on another medium on output. For example, if you want to create a particular film look when you output your colour graded shots to video, you can simulate the film stock on video by selecting the software 3D LUT. When you use the software 3D LUT rather than the hardware 3D LUT while you colour grade, playback speed will slow down as the colour grade becomes more complex.

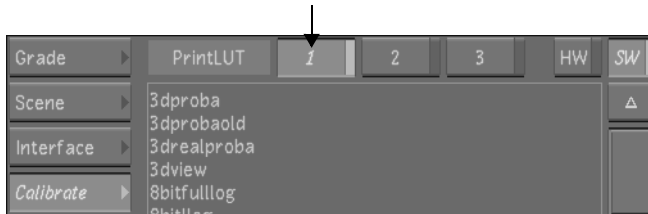
HINT: To output the colour grade with the 3D LUT to video, render the images with Render Viewing On. The 3D LUT will be included in the render without slowing down playback speed.

To apply a 3D Mesh LUT:

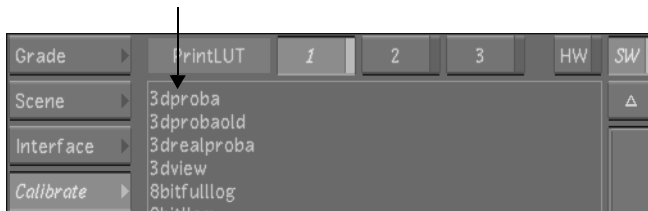
1. Save the 3D LUTs in the `C:\Program Files\Autodesk\lustre 2009\lut` folder if running the Windows version of Lustre, or in the `/usr/Autodesk/lustre2009/lut` directory if running the Linux version of Lustre.
2. In the Calibrate menu, click Off to disable existing monitor calibration settings.



3. Click 1.



4. Enable HW for hardware or SW for software.
5. In the Print LUT list, select a 3D LUT.



The selected 3D LUT is assigned to L1.



6. If necessary, repeat steps 3 and 4 to assign the other 3D LUT to L2.

HINT: For real-time playback with a 3D LUT applied, enable GPU acceleration by clicking the GPU button or pressing **Y**. See [“GPU Acceleration”](#) on page 295.

Browsing for Footage

Summary

About Browsing for Footage	131
Accessing the File Browser	131
Loading Shots into the Shot Bin	134
Viewing Media File Information	139
Matching Shots	142
Working with EDL, ALE, and Cutlist Files	143
Colour Decision List (CDL)	158

About Browsing for Footage

You use the file browser to locate your captured footage and load it into the Shot bin or directly onto a Timeline. This is one of the first steps in the colour grading process. Once your frames are loaded, you can import a CMX3600 format Edit Decision List (EDL), or an ALE or Cutlist file for Keycode assembly, and then assemble your footage into a cut.

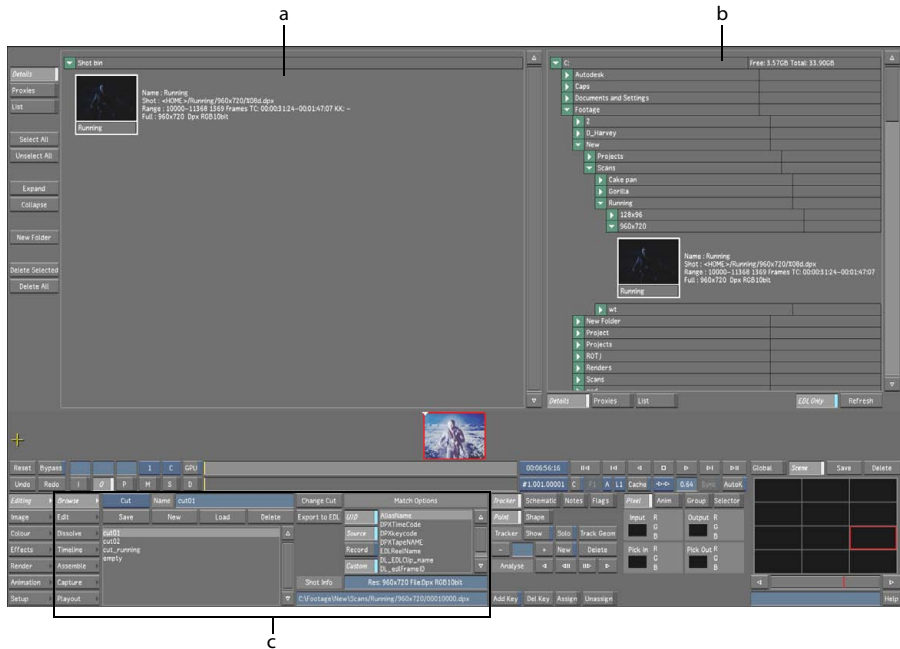
Accessing the File Browser

Use the file browser to locate shots and load them into the Shot bin.

To access the file browser:

- Click Editing, and then click Browse.

The Browse menu and the file browser appear.



a) Shot bin

b) File browser

c) Browse menu

Image courtesy of Moviworld / UK File & TV Company / Videolab

When you are in the Browse menu, the work area is divided into three areas.

Shot bin — Use to organize the shots you are using in your cuts.

Click:	To:
Proxies	Display shots in the Shot bin as thumbnails. Traditional methods of multi-selecting (SHIFT-click , CTRL-click) are enabled.
Details	Display shots in Thumbnail and Text view. Traditional methods of multi-selecting (SHIFT-click , CTRL-click) are disabled.
List	Display shots in Text view.
Select All	Select all shots in the Shot bin so that they can be added to the Storyboard simultaneously.
Unselect All	Deselect the selected clips in the Shot bin.
Expand	Display the contents of the selected folder.
Collapse	Hide the contents of the selected folder.

Click:	To:
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, <i>Shot bin</i> , cannot be renamed). Shots already in the Shot bin cannot be moved to the new folder.
Delete Selected	Delete selected shots.
Delete All	Delete all shots from the Shot bin.

File browser — Use to locate footage and bring it into the Shot bin. When copying captured footage into Lustre, always place the files into the Scans-Full home directory. In the file browser, you can see if the files are correctly located in the Scans-Full home directory. If they are, the path to the media file starts with the word <HOME>.

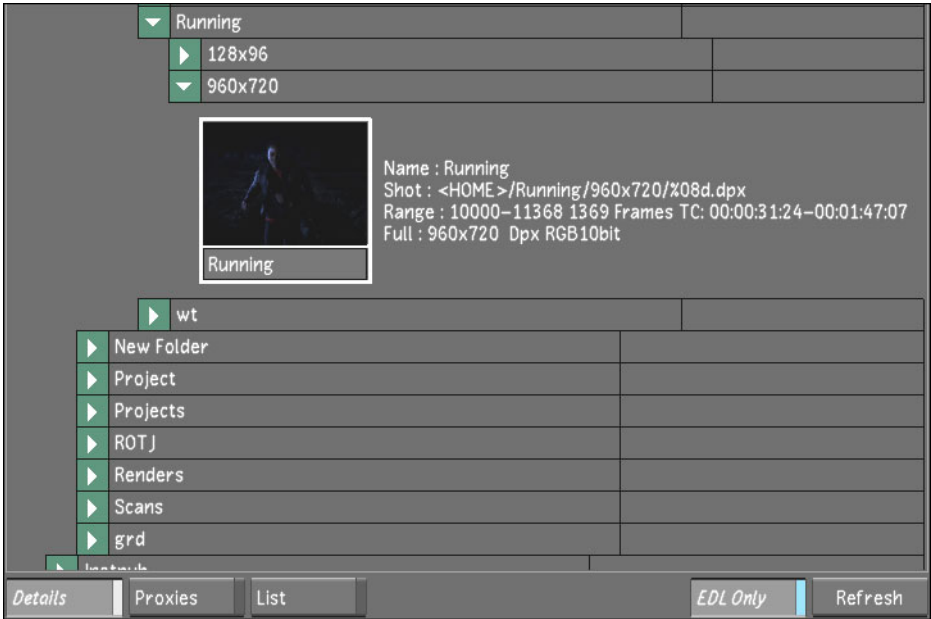


Image courtesy of Moviworld / UK File & TV Company / Videolab

Click:	To:
Details	Display details about all clips.
Proxies	Display only the clip thumbnails (without details).
List	Display shot information in a tabular Text view.
Refresh	Rescan the file systems and update the file browser with up-to-date information.

The EDL Only button is specific to Wiretap footage.

Browse menu — Use to create and manage the cuts in the project.

Supported File Formats

The following uncompressed file formats are all valid for use in Lustre:

- DPX (8-bit, 10-bit, 16-bit)
- Cineon (8-bit, 10-bit)
- TGA (8-bit, 10-bit)
- TIFF (8-bit, 10-bit, 16-bit)
- SGI (8-bit, 10-bit)
- BMP (8-bit, 8-bit w/ alpha)

NOTE: You cannot work with compressed files in Lustre.

Loading Shots into the Shot Bin

Once you locate your shots in the file browser, you move them into the Shot bin and then into the Storyboard so that you can use them in your cuts. Shots in the Shot bin are pointers to the actual media files on the filesystem. You can view information about the shots in both the file browser and the Shot bin.

To move shots from the file browser to the Shot bin:

1. Locate your shots. Expand the drive by clicking the arrow to the left of the drive letter, and then navigate to the appropriate folder.

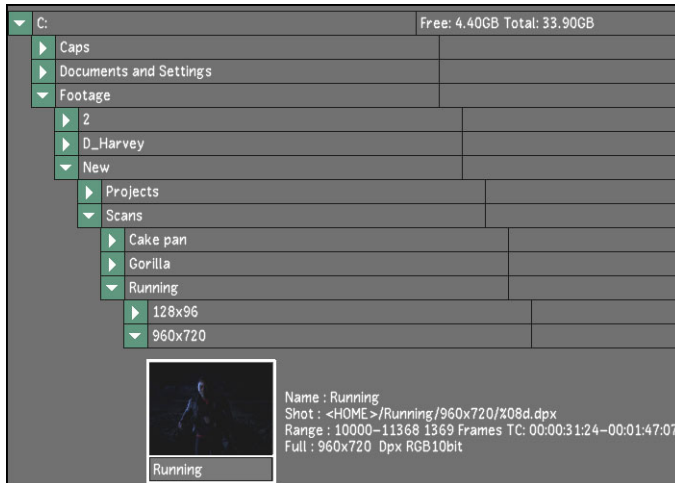


Image courtesy of Moviworld / UK File & TV Company / Videolab

NOTE: You can open all of a folder's sub-directories by **SHIFT**-clicking the arrow to the left of the folder.

The available shots appear in the browser with information about their contents.

- In the file browser, select Details to view detailed shot information, Proxies to view only the proxy thumbnails, or List to view a tabular text view of shot information.

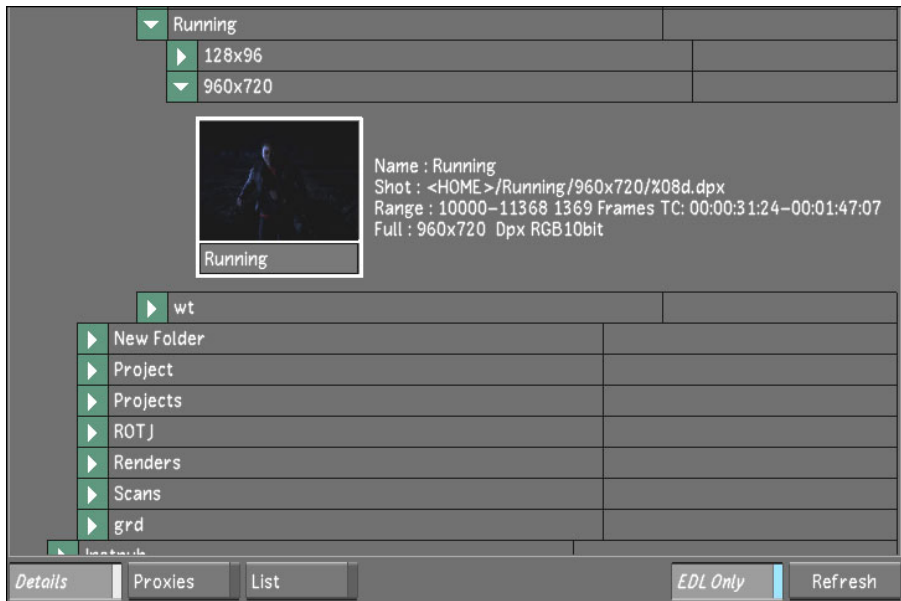


Image courtesy of Moviworld / UK File & TV Company / Videolab

- Click the shots you want to move to the Shot bin.
Each time you click a shot, the cursor picks it up. If you click more than one shot, the cursor picks them all up.

HINT: To select all the shots from a folder, select the folder.

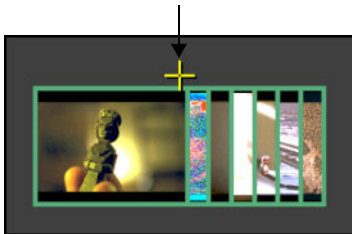
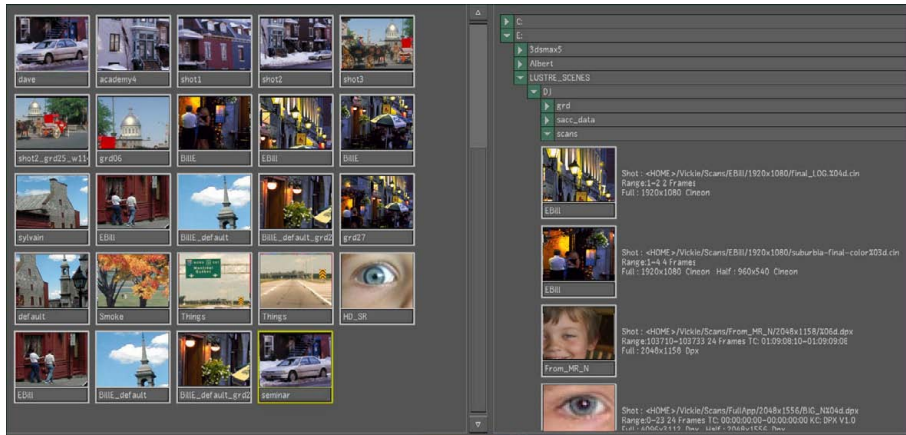


Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

- Drag the selected shots to the Shot bin, and then click to release the shots.

HINT: You can drag shots directly from the file browser to the Storyboard. When you do this, the shots are also placed in the Shot bin.

Newly added shots are highlighted. Shots already in the Shot bin are not duplicated.



5. If you pick up the wrong shots, click the grey area under the Export to EDL button to cancel your selection of shots.

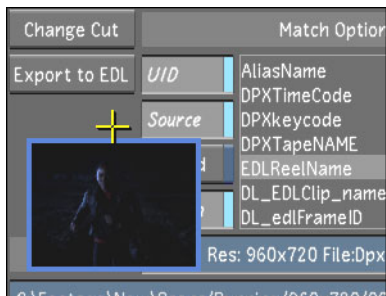


Image courtesy of Moveworld / UK File & TV Company / Videolab

6. Use the following Shot bin buttons to switch between Shot bin view modes.

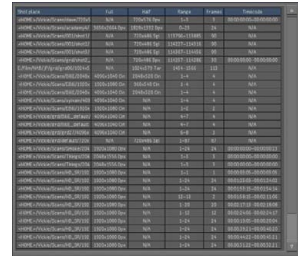
Click:	To:
Proxies	Display shots as thumbnails.
Details	Display shots as thumbnails with accompanying detail text.
List	Display shots in text format only.



Proxies



Details



List

NOTE: You can also press **B** to switch between view modes in the Shot bin as well as in the file browser.

Deleting Shots From the Shot Bin

You can delete all or selected shots from the Shot bin.

NOTE: Deleting shots from the Shot bin does not delete them from the Storyboard.

To delete selected shots from the Shot bin:

1. Select the shots you want to delete by doing one of the following:

- Click **Select All** in the Shot bin.
All the shots are selected.
- Press **CTRL** and click the name of the shot.
Selects only the shots you choose.

2. Click **Delete Selected**, and then confirm the action.

All the selected shots are deleted.

To delete all shots from the Shot bin:

- Click **Delete All**, and then confirm the action.
All the shots are deleted.

Moving Shots to the Storyboard

You can select shots in the Shot bin and move them to the Storyboard to create a cut. You can either move one shot at a time, or you can select multiple shots and drag them to the Storyboard all at once.

To move shots from the Shot bin to the Storyboard:

1. Click the shots in the Shot bin.
The cursor picks up the shot. If you click more than one shot, the cursor picks them all up.
2. Move the cursor to the Storyboard.
A yellow insertion marker appears in the Storyboard.

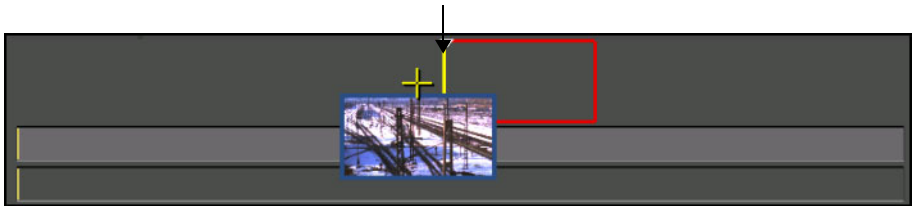


Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

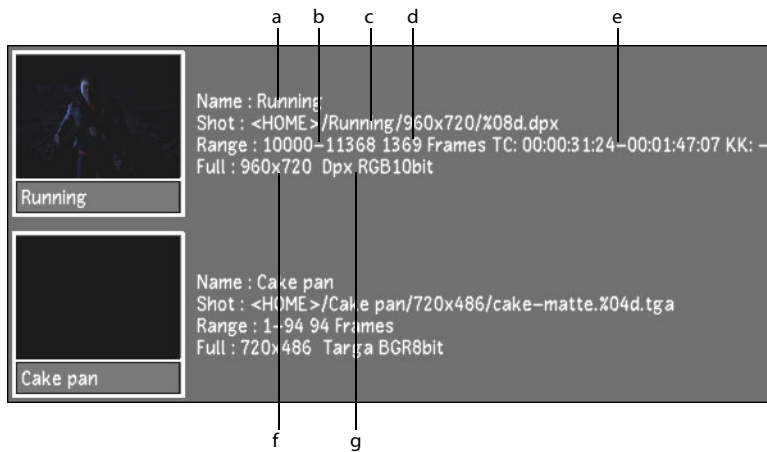
3. Click to drop the shot at the insertion point.

Viewing Media File Information

You can view information pertaining to your shots. When you view shots in the file browser or the Shot bin, information about the media files that make up the shot is displayed.

In Details view, the following information is displayed.

6 Browsing for Footage



- a) Shot name
- b) Frame range
- c) Shot location
- d) Number of frames
- e) Timecode range
- f) Full resolution
- g) File format

Image courtesy of Moviworld / UK File & TV Company / Videolab

In List view, the following information is displayed.

a	b	c	d	e	f
Shot place	Full	Half	Range	Frames	Timecode
<HOME>/Vickie/Scans/dave/720x5	N/A	720x576 Dpx	1-3	3	00:00:00:00-00:00:00:00
<HOME>/Vickie/Scans/academy4/	3656x2664 Dpx	1828x1332 Dpx	0-23	24	N/A
<HOME>/Vickie/Scans/001/shot1/	N/A	720x486 Sgi	113796-113885	90	N/A
<HOME>/Vickie/Scans/001/shot2/	N/A	720x486 Sgi	114227-114316	90	N/A
<HOME>/Vickie/Scans/001/shot3/	N/A	720x486 Sgi	114367-114456	90	N/A
<HOME>/Vickie/Scans/grd/shot2_	N/A	720x486 Dpx	114257-114286	30	00:00:00:00-00:00:00:00
E:/Film/NAB/JP/grd/grd06/1024x5	N/A	1024x579 Tar	1454-1566	113	N/A
<HOME>/Vickie/Scans/BILLIE/2048x	4096x1040 Cin	2048x520 Cin	1-4	4	N/A
<HOME>/Vickie/Scans/EBILL/1920x	1920x1080 Cin	960x540 Cin	1-4	4	N/A
<HOME>/Vickie/Scans/BILLIE/2048x	4096x1040 Cin	2048x520 Cin	1-4	4	N/A
<HOME>/Vickie/Scans/sylvain/409	4096x1040 Cin	N/A	1-4	4	N/A
<HOME>/Vickie/Scans/EBILL/1920x	1920x1080 Cin	N/A	1-2	2	N/A
<HOME>/Vickie/grd/BILLIE_default/	4096x1040 Cin	N/A	4-7	4	N/A
<HOME>/Vickie/grd/BILLIE_default/	4096x1040 Cin	N/A	4-7	4	N/A
<HOME>/Vickie/grd/grd27/4096x	4096x1040 Cin	N/A	6-8	3	N/A
<HOME>/Vickie/grd/default/720x	N/A	720x486 Sgi	1-87	87	N/A
<HOME>/Vickie/Scans/Smoke/204	1920x1080 Dpx	N/A	1-24	24	00:00:00:00-00:00:00:23
<HOME>/Vickie/Scans/Things/204	2048x1556 Dpx	N/A	1-3	3	00:00:00:00-00:00:00:00
<HOME>/Vickie/Scans/Things/204	2048x1556 Dpx	N/A	1-3	3	00:00:00:00-00:00:00:00
<HOME>/Vickie/Scans/HD_SR/192	1920x1080 Dpx	N/A	1-1	1	00:00:03:05-00:00:03:05
<HOME>/Vickie/Scans/HD_SR/192	1920x1080 Dpx	N/A	1-24	24	00:01:23:03-00:01:24:02
<HOME>/Vickie/Scans/HD_SR/192	1920x1080 Dpx	N/A	1-24	24	00:01:53:15-00:01:54:14
<HOME>/Vickie/Scans/HD_SR/192	1920x1080 Dpx	N/A	12-13	2	00:01:59:15-00:02:11:06
<HOME>/Vickie/Scans/HD_SR/192	1920x1080 Dpx	N/A	1-20	20	00:02:17:13-00:02:18:08
<HOME>/Vickie/Scans/HD_SR/192	1920x1080 Dpx	N/A	1-12	12	00:02:24:06-00:02:24:17
<HOME>/Vickie/Scans/HD_SR/192	1920x1080 Dpx	N/A	1-24	24	00:00:19:05-00:00:20:04
<HOME>/Vickie/Scans/HD_SR/192	1920x1080 Dpx	N/A	1-24	24	00:00:39:21-00:00:40:20
<HOME>/Vickie/Scans/HD_SR/192	1920x1080 Dpx	N/A	1-24	24	00:00:44:22-00:00:45:21
<HOME>/Vickie/Scans/HD_SR/192	1920x1080 Dpx	N/A	1-24	24	00:00:51:22-00:00:52:21

a) Shot place column

c) Half column

e) Frames column

b) Full column

d) Range column

f) Timecode column

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 shot.

Full — Indicates the resolution and file format of the full-resolution images that make up the shot.

Half — Indicates the resolution and file format of the half-resolution images that make up the shot.

Range — Displays the range of frames in the shot sequence that make up the shot.

Frames — Displays the number of frames that make up the shot.

Timecode — Displays source in and out timecode values that make up the shot.

In the Browse menu, you can also view information about a selected shot in the Storyboard.

NOTE: Text information is not displayed in Thumbnail view.

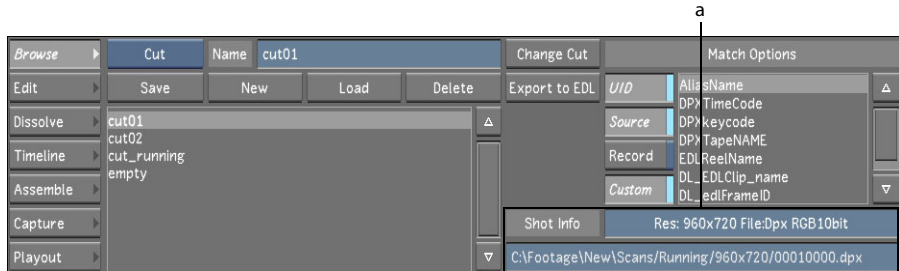
To view information for shots in the Shot bin or file browser:

1. Click Editing, and then click Browse to display the file browser.
2. Switch the focus to either the Shot bin or file browser by clicking it.
3. Press **B** to toggle through the view modes. If the focus is on the Shot bin, you can also use the Proxies, Details, or List button to switch view modes.

To view information for shots in the Storyboard:

1. Click Editing, and then click Browse to display the file browser.
2. Select a shot in the Storyboard.

The following information appears in the Shot Info fields in the Browse menu for the selected shot: resolution, file format, bit depth, and path to the media files.



a) Shot Info fields

Matching Shots

You can use Matchback to select a shot in the timeline and find it in the Shot bin, or click a shot in the Shot bin and see where it is used in the timeline.

To match a shot from the Timeline or Storyboard to the Shot bin:

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 Shot bin.

To match a shot from the Shot bin to the timeline:

- **ALT**-click the shot in the Shot bin.

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 Shot bin, successive instances of the same shot are highlighted in the Storyboard and positioned in the Timeline.

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 name. If the Match Name option 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.
- 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.

NOTE: Currently, when loading an EDL, ALE, or Cutlist file, Lustre does not read comments.

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.

Event	Reel Name	Transition	Source In	Source Out	Record In	Record Out	KeyCode Start	KeyCode End	Media
1	REEL01	Cut	00:00:00:01	00:00:00:02	00:00:00:00	00:00:00:01			Linked
2	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:00:01	00:00:05:08			Linked
3	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:05:08	00:00:10:15			Linked
4	REEL01	Cut	00:00:00:01	00:00:01:11	00:00:10:15	00:00:12:00			Linked
5	REEL01	Cut	00:00:00:01	00:00:02:19	00:00:12:00	00:00:14:18			Linked
6	REEL01	Cut	00:00:00:01	00:00:01:06	00:00:14:18	00:00:15:23			Linked
7	REEL01	Cut	00:00:00:01	00:00:01:06	00:00:15:23	00:00:17:03			Linked
8	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:17:03	00:00:22:10			Linked
9	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:22:10	00:00:27:17			Linked
10	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:27:17	00:00:32:24			Linked
11	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:32:24	00:00:38:06			Linked
12	REEL01	Cut	00:00:05:07	00:00:05:08	00:00:38:06	00:00:38:07			Linked
13	REEL01	Cut	00:00:03:20	00:00:03:21	00:00:38:07	00:00:38:08			Linked
14	REEL01	Cut	00:00:00:01	00:00:00:02	00:00:38:08	00:00:38:09			Linked
15	REEL01	Cut	00:00:00:01	00:01:40:01	00:00:38:09	00:02:18:09			Linked
16	REEL01	Cut	00:00:00:02	00:00:00:03	00:02:18:09	00:02:18:10			Linked
17	REEL01	Cut	00:00:00:01	00:00:00:05	00:02:18:10	00:02:18:14			Linked
18	REEL01	Cut	09:36:00:00	09:36:00:04	00:02:18:14	00:02:18:18			Need Match

a) EDL event buttons

b) EDL events

HINT: 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.

Click:	To:
Mark For Capture	Flag the currently selected event for capture. See “Metadata Maintained for Missing Shots” on page 156.
Mark For Match	Flag the currently selected event to be matched to a shot in the Shot bin.
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” on page 131.
2. Load shots into the Shot bin.	“Loading Shots into the Shot Bin” on page 134.
3. Add CDL data to an EDL file to associate CDL data to individual events (shots).	“Adding CDL Data to an EDL File” on page 146.
4. Copy the EDL into Lustre.	“Copying an EDL, ALE, or Cutlist File into Lustre” on page 148.
5. Assemble the EDL to automatically build your footage into a timeline.	“Assembling an EDL” on page 149.
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.	Chapter 7, “Editing,” on page 165, and “Exporting Shots to an EDL” on page 154.
7. Save the assembled EDL as a cut.	“Creating a New Cut” on page 49.

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: 0054
*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).

Copying an EDL, ALE, or Cutlist File into Lustre

Before you can assemble an EDL, ALE, or Cutlist file, you have to copy the file into the current scene's *sacc_data* directory.

NOTE: The file must have a *.edl, *.ale, or *.ctl file extension or it will not appear in the EDL list.

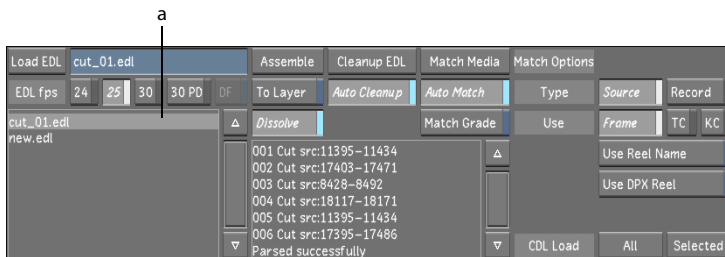
To copy an EDL, ALE, or Cutlist file into Lustre:

1. If using the Windows version of Lustre, use Windows Explorer to copy the EDL into the current scene's *sacc_data* folder.
2. If using the Linux version of Lustre, do the following:
 - Open a shell and navigate to the directory where the EDL was copied or downloaded. For instance, **/usr/tmp**.
 - Copy the EDL to the *sacc_data* directory of the current scene. Type:

```
cp <filename> /usr/autodesk/lustre2009/projects/<project name>/sacc_data
```

3. Click Editing, and then click Assemble.

The Assemble menu appears.



a) EDL list

4. Press **CTRL+R** to refresh the EDL list.

The contents of the *sacc_data* directory are scanned for EDLs, ALE, or Cutlist files. The available files are listed in the EDL list.

5. You are now ready to load and assemble your EDL, ALE, or Cutlist file.

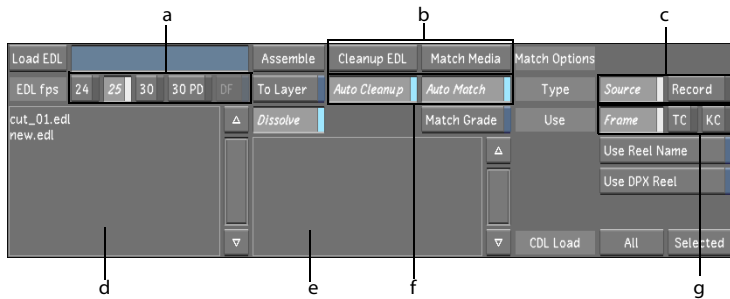
Assembling an EDL

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 Shot bin.
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. Select the EDL you want to load from the EDL list.
4. 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”](#) on page 443.
5. 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.

Enable:	To:
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.

6. 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`

HINT: 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”](#) on page 40.

7. 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.

8. 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”](#) on page 154.

9. 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”](#) on page 154.

10. To perform an EDL cleanup, do one of the following:

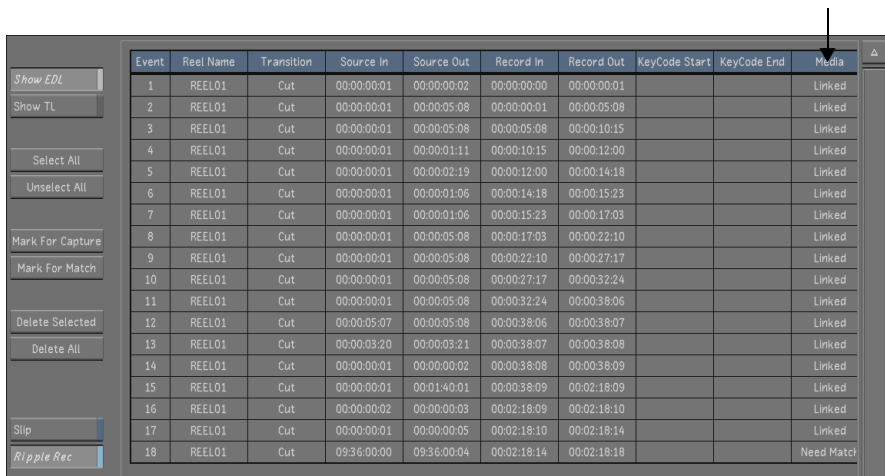
- Click EDL Cleanup to perform cleanup prior to assembling.
- Enable Auto Cleanup to automatically perform cleanup during assembly.

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.

11. To match EDL event timecodes to shot timecodes in the Shot bin, do one of the following:

- Click Match Media to match EDL event timecodes to available shot timecodes in the Shot bin prior to assembling.
- Enable Auto Match to automatically match EDL event timecodes to available shot timecodes in the Shot bin during assembly.

The matching process links EDL event timecodes to media timecodes in the Shot bin. 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.



Event	Reel Name	Transition	Source In	Source Out	Record In	Record Out	KeyCode Start	KeyCode End	Media
1	REEL01	Cut	00:00:00:01	00:00:00:02	00:00:00:00	00:00:00:01			Linked
2	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:00:01	00:00:05:08			Linked
3	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:05:08	00:00:10:15			Linked
4	REEL01	Cut	00:00:00:01	00:00:01:11	00:00:10:15	00:00:12:00			Linked
5	REEL01	Cut	00:00:00:01	00:00:02:19	00:00:12:00	00:00:14:18			Linked
6	REEL01	Cut	00:00:00:01	00:00:01:06	00:00:14:18	00:00:15:23			Linked
7	REEL01	Cut	00:00:00:01	00:00:01:06	00:00:15:23	00:00:17:03			Linked
8	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:17:03	00:00:22:10			Linked
9	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:22:10	00:00:27:17			Linked
10	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:27:17	00:00:32:24			Linked
11	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:32:24	00:00:38:06			Linked
12	REEL01	Cut	00:00:05:07	00:00:05:08	00:00:38:06	00:00:38:07			Linked
13	REEL01	Cut	00:00:03:20	00:00:03:21	00:00:38:07	00:00:38:08			Linked
14	REEL01	Cut	00:00:00:01	00:00:00:02	00:00:38:08	00:00:38:09			Linked
15	REEL01	Cut	00:00:00:01	00:01:40:01	00:00:38:09	00:02:18:09			Linked
16	REEL01	Cut	00:00:00:02	00:00:00:03	00:02:18:09	00:02:18:10			Linked
17	REEL01	Cut	00:00:00:01	00:00:00:05	00:02:18:10	00:02:18:14			Linked
18	REEL01	Cut	09:36:00:00	09:36:00:04	00:02:18:14	00:02:18:18			Need Match

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”](#) on page 156.

12. 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.

13. Click Assemble to conform the EDL and build the cut.

NOTE: When a timecode gap occurs between EDL events, black frames are inserted between the events to fill the gap.

14. (Optional) To manually match shots in the Shot bin to EDL events, use the Replace Shot tool to add the correct footage into the blank shots. See [“Replacing Shots”](#) on page 183.

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”](#) on page 152.

NOTE: You can also assemble an EDL with CDL data. See [“Assembling an EDL with CDL Data”](#) on page 162.

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.

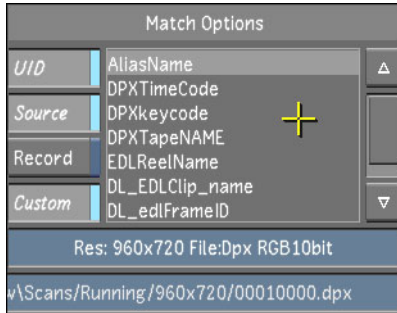
Match Grade

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. The Match Grade feature uses the same Heuristic Evaluation method as the Change Cut function, meaning that an exact match is not required in order for grade settings to be transferred.

When you enable Match Grade during 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 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.

Src — Performs matching based on source data, such as a shot's source timecode.

Rec — Performs matching based on record data, such as an EDL's record timecode.

Custom — Performs matching based on custom XML metadata.

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.

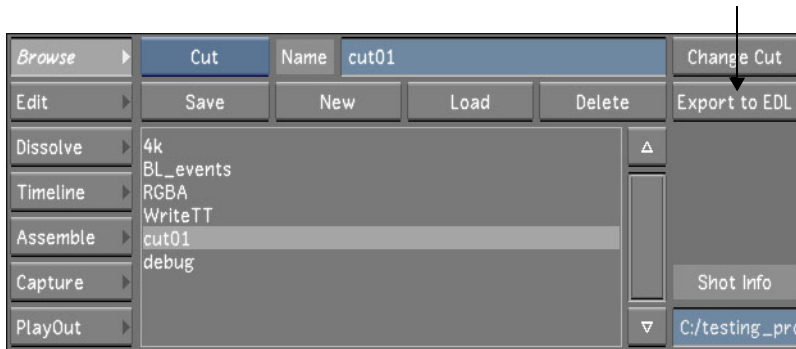
Exporting Shots to an EDL

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 *sacc_data* 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 [“Loading a Cut”](#) on page 50.
3. Click Export to EDL.



A CMX3600 EDL is created from the shots in the current timeline and placed in the scene's *sacc_data* directory.

Keycode Assembly

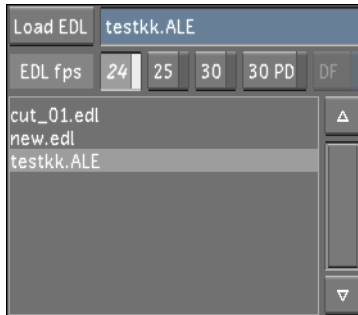
You can load ALE (*.ale) and Cutlist files (*.ctl) as EDLs in the Assemble menu. Once loaded, you can use their keycode information for media assembly.

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 *filmtype.config* file, located in *C:\Program Files\Autodesk\lustre 2009* on the Windows version of Lustre, or in */usr/autodesk/lustre2009* 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 *filmtype.config* file:

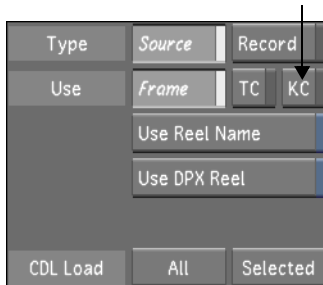
FN 123456

To load and assemble ALE and CTL files:

1. Place the ALE or Cutlist file in the scene's *sacc_data* directory.
2. Load the shots that you want to assemble into the Shot bin.
3. Click Editing, and then click Assemble.
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 Shot bin, do one of the following:
 - Click Match Media to match event timecode/keycode to available shot timecode/keycode in the Shot bin prior to assembling.
 - Enable Auto Match to automatically match event timecode/keycode to available shot timecode/keycode in the Shot bin 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”](#) on page 156.

10. To manually match shots in the Shot bin to ALE or CTL events, use the Replace Shot tool to add the correct footage into the blank shots. See [“Replacing Shots”](#) on page 183.
11. Click Assemble to conform the ALE or CTL file and build the cut.

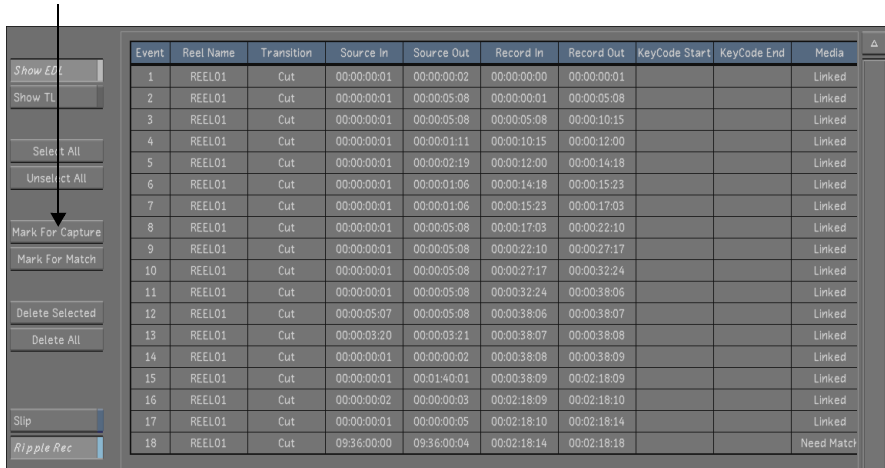
Metadata Maintained for Missing Shots

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:

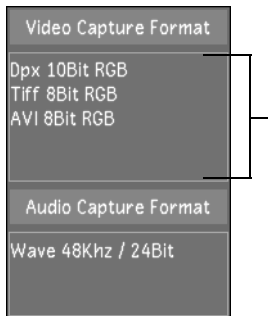
1. In the Editing menu, click Assemble to display the Assemble menu.
2. In the Assembly window, **CTRL**-click each event to be marked for capture.

3. Click Mark For Capture.



Event	Reel Name	Transition	Source In	Source Out	Record In	Record Out	KeyCode Start	KeyCode End	Media
1	REEL01	Cut	00:00:00:01	00:00:00:02	00:00:00:00	00:00:00:01			Linked
2	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:00:01	00:00:05:08			Linked
3	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:05:08	00:00:10:15			Linked
4	REEL01	Cut	00:00:00:01	00:00:01:11	00:00:10:15	00:00:12:00			Linked
5	REEL01	Cut	00:00:00:01	00:00:02:19	00:00:12:00	00:00:14:18			Linked
6	REEL01	Cut	00:00:00:01	00:00:01:06	00:00:14:18	00:00:15:23			Linked
7	REEL01	Cut	00:00:00:01	00:00:01:06	00:00:15:23	00:00:17:03			Linked
8	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:17:03	00:00:22:10			Linked
9	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:22:10	00:00:27:17			Linked
10	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:27:17	00:00:32:24			Linked
11	REEL01	Cut	00:00:00:01	00:00:05:08	00:00:32:24	00:00:38:06			Linked
12	REEL01	Cut	00:00:05:07	00:00:05:08	00:00:38:06	00:00:38:07			Linked
13	REEL01	Cut	00:00:03:20	00:00:03:21	00:00:38:07	00:00:38:08			Linked
14	REEL01	Cut	00:00:00:01	00:00:00:02	00:00:38:08	00:00:38:09			Linked
15	REEL01	Cut	00:00:00:01	00:01:40:01	00:00:38:09	00:02:18:09			Linked
16	REEL01	Cut	00:00:00:02	00:00:00:03	00:02:18:09	00:02:18:10			Linked
17	REEL01	Cut	00:00:00:01	00:00:00:05	00:02:18:10	00:02:18:14			Linked
18	REEL01	Cut	09:36:00:00	09:36:00:04	00:02:18:14	00:02:18:18			Need Match

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”](#) on page 508.



Video Capture Format

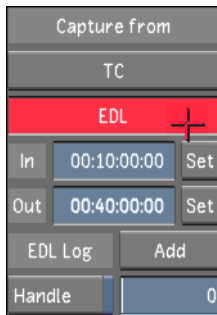
- Dpx 10Bit RGB
- Tiff 8Bit RGB
- AVI 8Bit RGB

Audio Capture Format

- Wave 48Khz / 24Bit

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

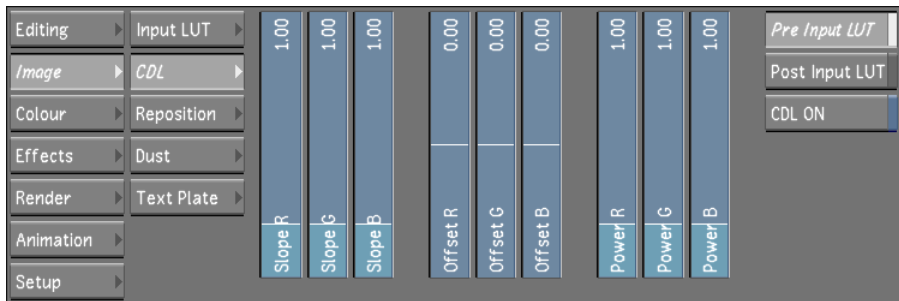
The CDL menu allows you to:

- Change the three basic transfer functions (slope, offset, and power).
- 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.

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 value).
- 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 value).
- 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 value).
- 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.

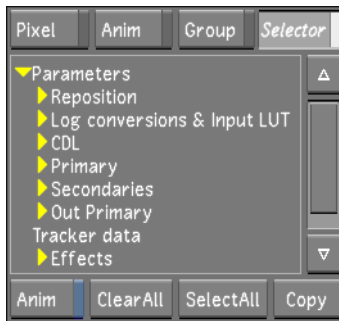
CDL On/Off

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

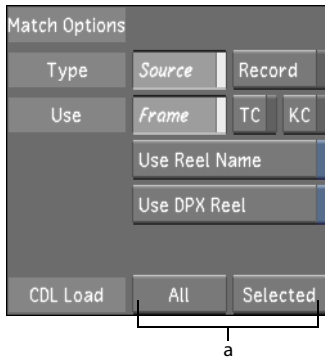
The EDL panel contains a new interface to support loading CDL data.

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 part of the currently loaded EDL, you need to use the CDL Load options to apply the CDL data to the currently opened 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.

NOTE: It is possible to use a different EDL to conform and a different EDL to import CDL data.

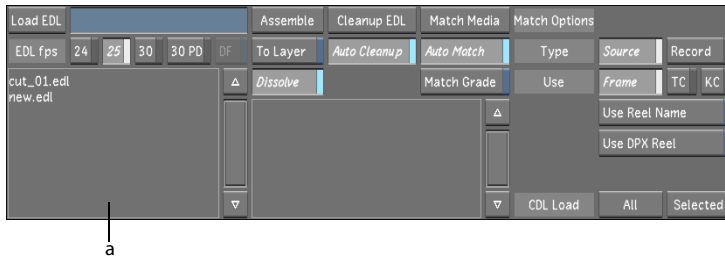
Assembling an EDL with CDL Data

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 Shot bin.
2. Click Editing, and then click Assemble.

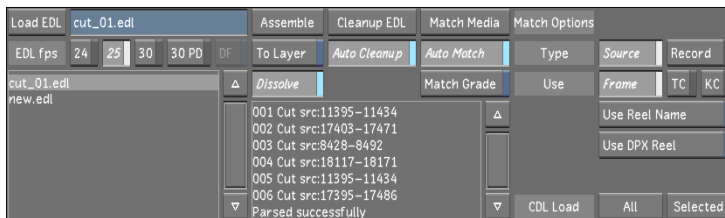
The Assemble menu appears.



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”](#) on page 149.

5. Click Match Media to match the EDL event timecodes to the available shot timecodes in the Shot bin 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”](#) on page 162.
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”](#) on page 146.

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 ACS CDL specifications for more detail.

See [“Adding CDL Data to an EDL File”](#) on page 146.

CDL XML Files Location

To be able to use XML files, place them in an import sub-folder located in the project's `sacc_data` folder:

<my project>/<scene 1>/<sacc_data>/<import>

Summary

<i>About Editing</i>	165
<i>Accessing the Edit Menu</i>	166
<i>Editing with the Storyboard</i>	168
<i>Trimming from the Edit Menu</i>	171
<i>Changing Playback Speed</i>	182
<i>Replacing Shots</i>	183
<i>Creating Dissolves</i>	184
<i>Adding a Virtual Black Clip using the Storyboard</i>	187
<i>Multi-Layer Timeline</i>	188
<i>Editing in the Timeline</i>	193
<i>Trimming Elements</i>	198
<i>Transitions</i>	202
<i>Displaying the Multi-Layer Timeline</i>	205
<i>Rendering the Timeline</i>	212
<i>Modifying the Timeline's Starting Record Timecode</i>	212
<i>Performing a Confidence Check</i>	213
<i>Using Scene Detection</i>	215

About 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”](#) on page 205). 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”](#) on page 168.
- the Multi-Layer Timeline, see [“Multi-Layer Timeline”](#) on page 188.
- Solo mode, see [“Solo Mode”](#) on page 211.
- Mute mode, see [“Muted Layers”](#) on page 206.

Accessing the Edit Menu

To modify shots in the timeline, you use the Edit menu.

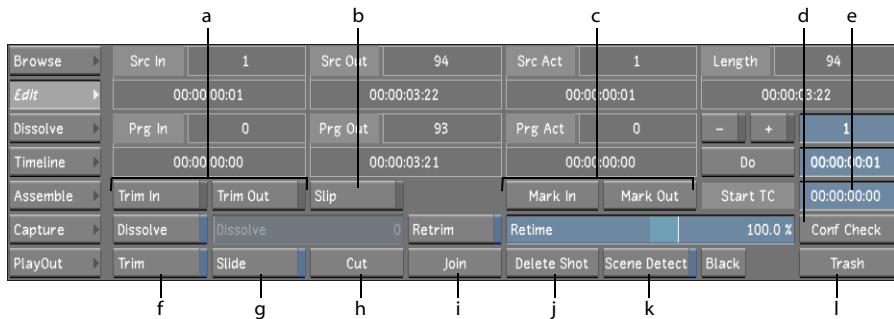
NOTE: If a layer is soloed, the Edit tools (Trash, Delete, Trim In/Trim Out, and Mark In/Out) default to ‘Ripple End’ behaviour, regardless of the Ripple Mode setting. If no layer is soloed, however, the

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”](#) on page 199.

To access the Edit menu:

1. In the Main menu, click Editing.
2. Click Edit.

The Edit menu appears.



a) Trim In/Out buttons	d) Confidence Check button	g) Slide button	j) Delete Shot button
b) Slip button	e) Start TC field	h) Cut button	k) Scene Detect button
c) Mark In/Out buttons	f) Trim button	i) Join 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”](#) on page 173.

Slip — Moves the source frames within a shot without adjusting the shot’s position in the timeline. See [“Slipping a Shot”](#) on page 176.

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”](#) on page 178.

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”](#) on page 212.

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”](#) on page 177.

Slide — Slides a cut point forward or backward. See [“Sliding the Cut Point between Shots”](#) on page 179.

Cut — Cuts one shot into two at the Current Frame positioner. See [“Inserting Cuts”](#) on page 180.

Join — Joins two shots coming from the same source clip with consecutive source timecode. See [“Joining Shots”](#) on page 181.

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”](#) on page 215.

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”](#) on page 213.

Trash — Deletes shots from the Shot bin or Storyboard when you drag them to the Trash button.

Global Access to Editing Hot Keys

When you enable the Editing While Grading button in the User settings, you can access the following editing hot keys 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”](#) on page 40.

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 to the Storyboard by dragging them from the Shot bin.

HINT: 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”](#) on page 188.

To add shots to the Storyboard:

1. Select shots in the Shot bin.

The order in which you click shots in the Shot bin determines their order in the Storyboard.

HINT: 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.

2. Drag the shots to the Storyboard.

A yellow insertion marker appears on the Storyboard.

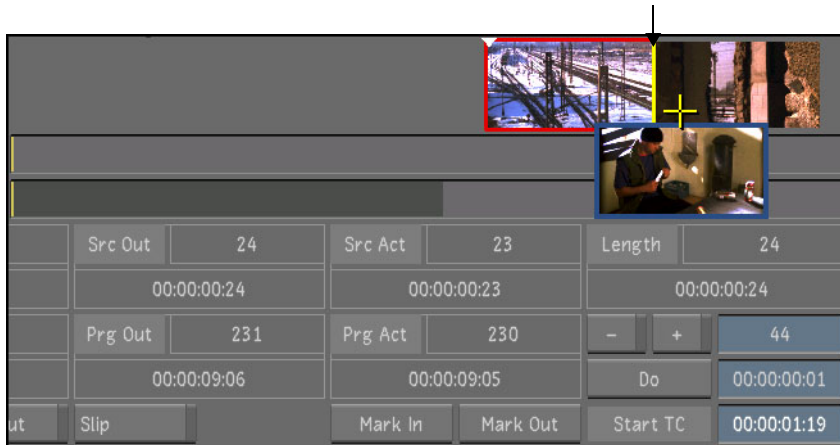


Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

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.

NOTE: You can also drag shots directly from the file browser and drop them into the Storyboard. In this case, the shots are also placed into the Shot bin (if they are not already there).

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.

HINT: 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:

- After adding a shot to the Storyboard, drag the shot from the Shot bin a second time to create a duplicate.

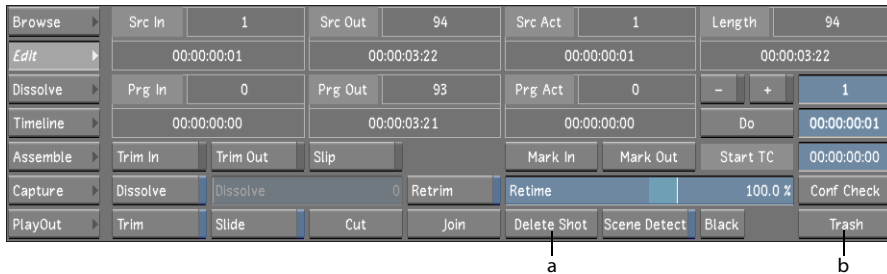
HINT: If you select the wrong shots, click the grey area to the left of the Trash button. This releases the selected shots from the cursor.

HINT: 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”](#) on page 188. Otherwise, the duplicated shot is inserted at the end of the Storyboard.

Deleting Shots

You can delete shots from the Shot bin, Storyboard, or Multi-Layer Timeline (see [“Deleting Elements”](#) on page 197). Deleting shots from the Shot bin 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”](#) on page 199) 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”](#) on page 199 and [“Slip & Slide Ripple Mode”](#) on page 201) 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.

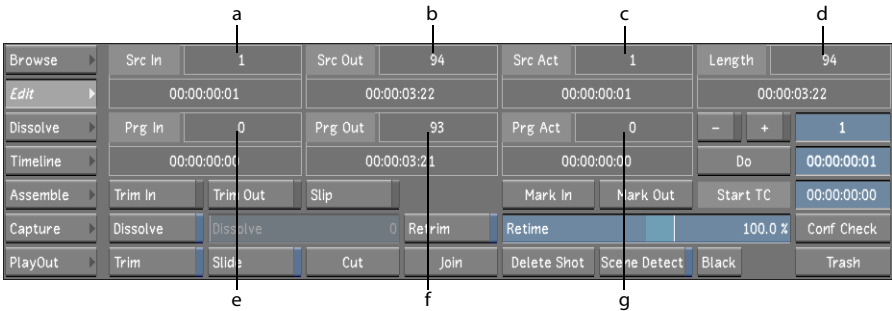
Viewing Shot Information

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”](#) on page 171.

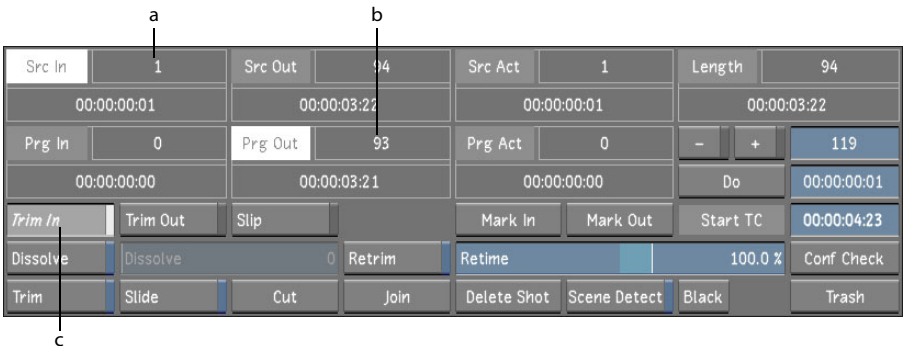
To trim frames at the in point for one or more selected shots:

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.



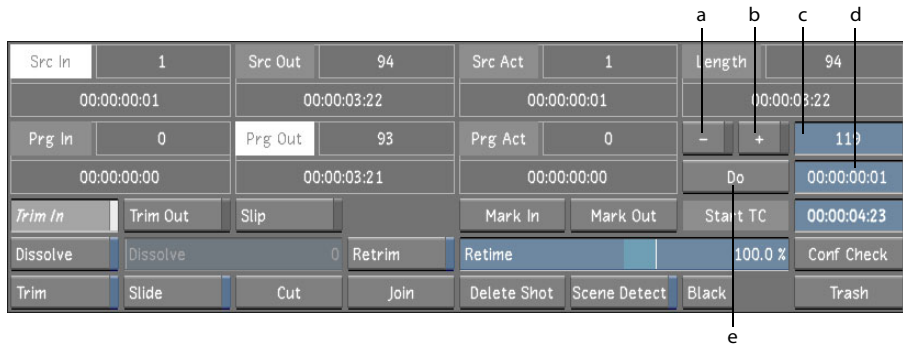
- a) Src In field b) Prg Out field c) Trim In button

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 c) Frame field e) Do button
 b) + (plus) button d) Do field

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.

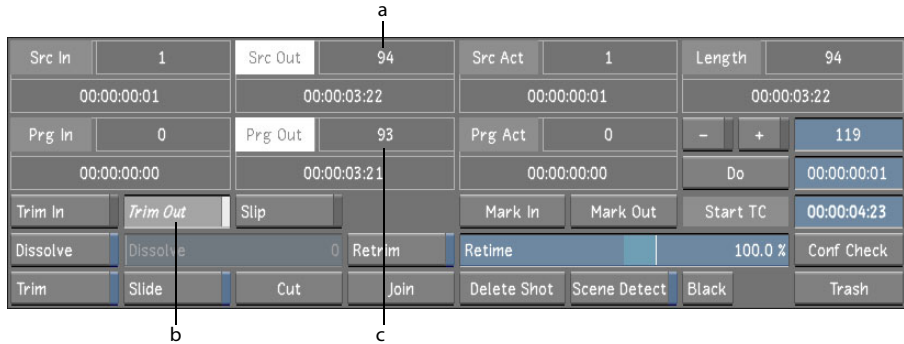
To trim frames at the out point for one or more selected shots:

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.



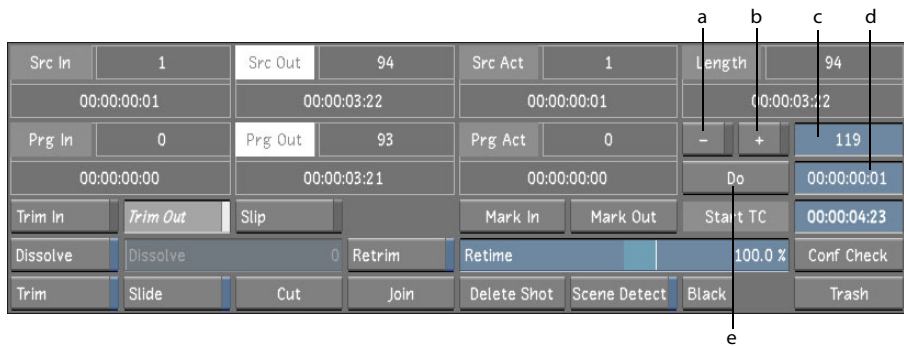
- a) Src Out field b) Trim Out button c) Prg Out field

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 c) Frame field e) Do button
b) + (plus) button d) Do field

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.

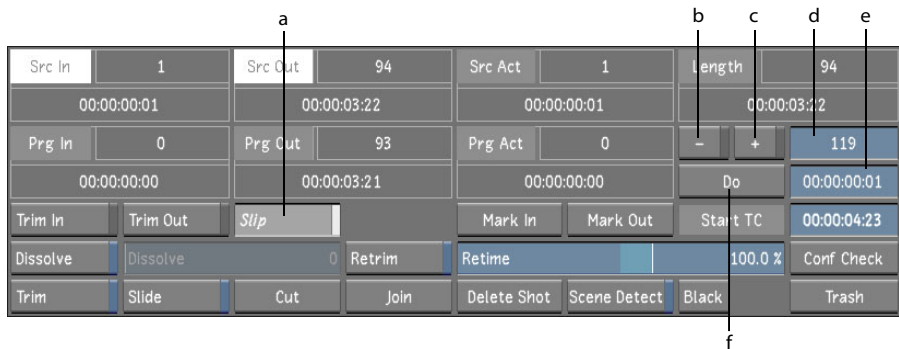
To slip one or more shots:

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

c) + (plus) button

e) Do field

b) - (minus) button

d) Frame 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.

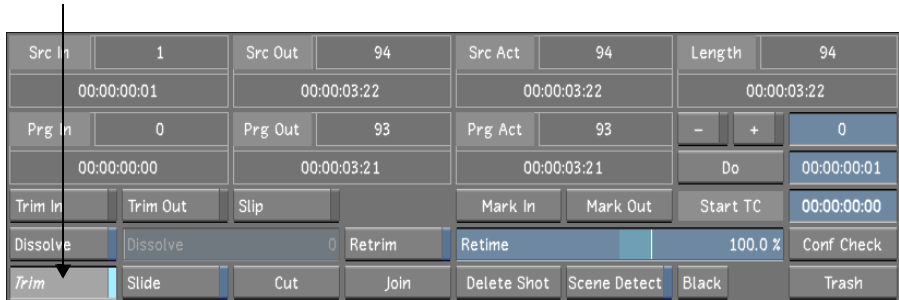
Trimming Interactively

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.



HINT: 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”](#) on page 40.

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

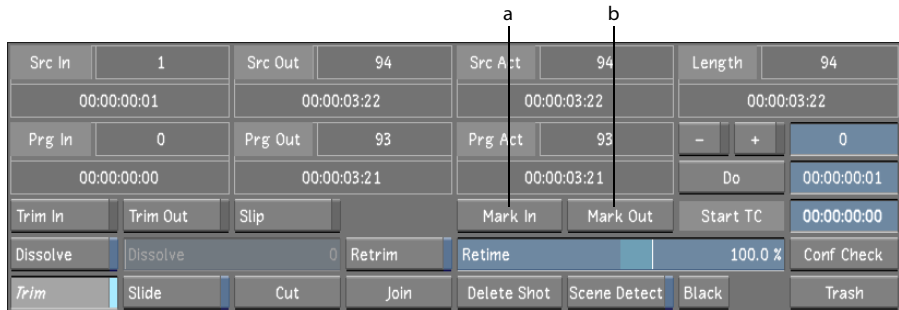
You can change the in and out points for a shot. This is like trimming, except you can only shorten a shot by changing the in or out points.

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

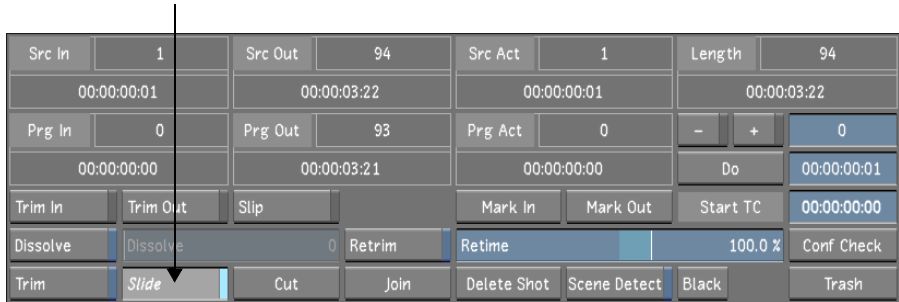
You can interactively move the cut point between two shots forward or backward, if handles are available. Using Slide does not adjust the overall length of your timeline. By removing frames from one side of the cut, you automatically add frames to the other side.

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.



HINT: 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”](#) on page 40.

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.

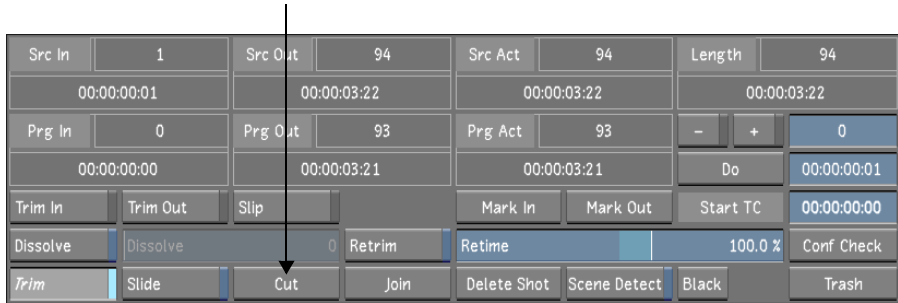
Inserting Cuts

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.

HINT: 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”](#) on page 40.

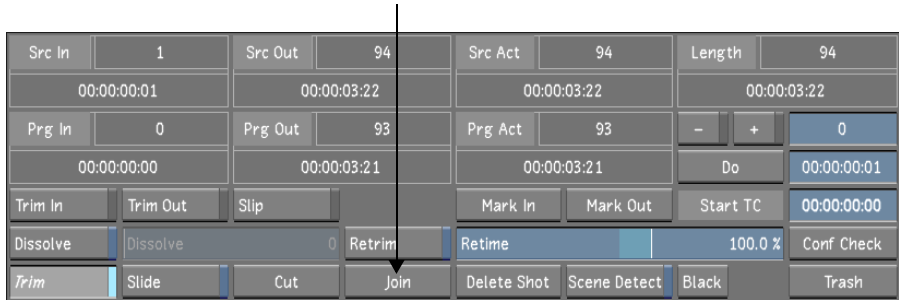
Joining Shots

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.

HINT: 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”](#) on page 40.

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”](#) on page 171.

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 Shot bin (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 Shot bin view, but the swap can only be done in the Storyboard area.

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 Shot bin (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 Shot bin (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 split 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

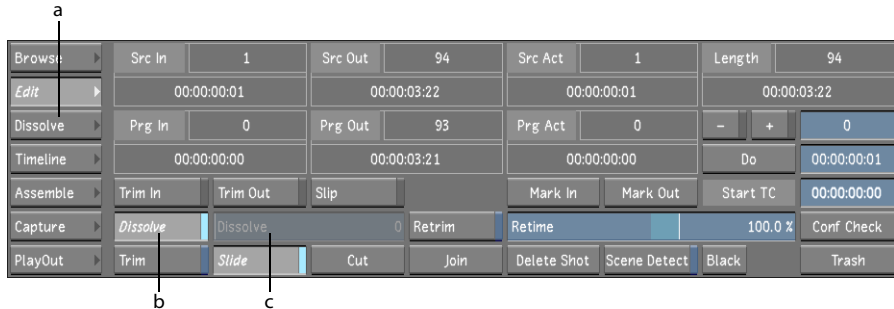
You can add a dissolve between two shots provided that there are a sufficient number of underlying handles.

NOTE: All dissolves in Lustre are centre aligned.

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. Select the shot to the left of the cut to which you want to add a dissolve.

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”](#) on page 186.

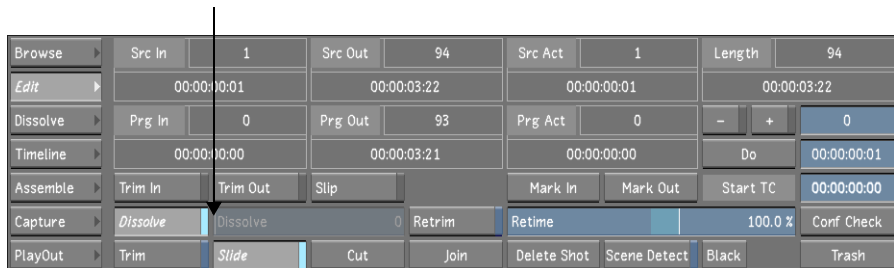
3. Enable Dissolve, and then drag the Dissolve slider to set the dissolve duration.
4. To edit the shape of the Dissolve curve, see [“Editing Dissolves”](#) on page 186.

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.

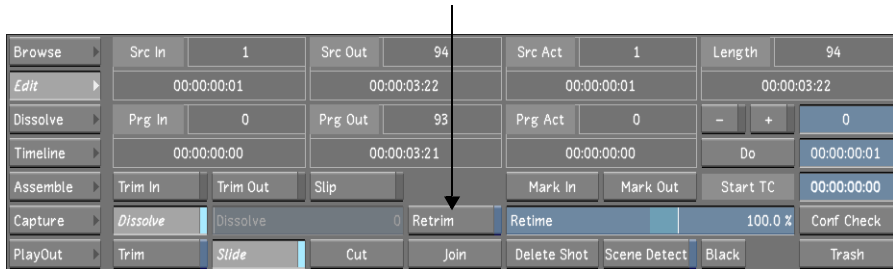


Creating Additional Handles for Dissolves

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.

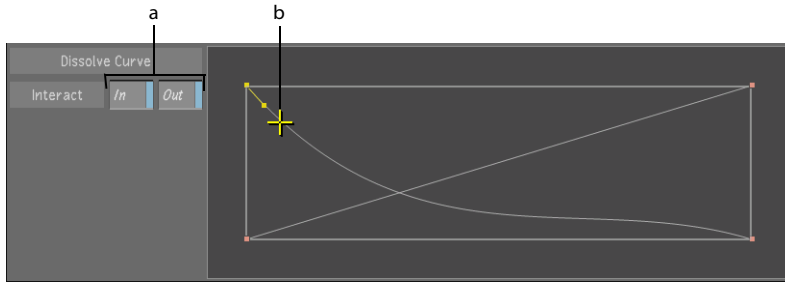
Editing Dissolves

Use the Dissolve Curves window in the Dissolve menu to edit the rate at which the dissolve changes. There are 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.

To modify the dissolve curve:

1. In the Edit menu, click Dissolve to display the Dissolve menu.
2. 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.



a) Interact In/Out buttons

b) Adjusting the Bezier handle

3. Click one of the keyframes and drag the Bezier handle to adjust the rate at which the dissolve takes place.

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 to toggle a dissolve on and off.

Adding a Virtual Black Clip using the Storyboard

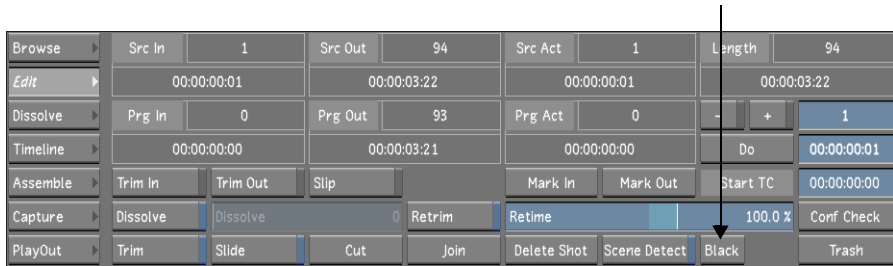
Virtual black clips can be added anywhere on the timeline. The default duration of each clip is one second, according to the current project frame rate. See [“Adding Black Media to the Multi-Layer Timeline”](#) on page 196.

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”](#) on page 188. 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 menu”](#) on page 27.

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.

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”](#) on page 308.

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”](#) on page 205.

Timeline Menu

There are various areas to work with when you access the Timeline menu:

- Storyboard. See [“Storyboard Viewing Options”](#) on page 230.
- File browser. See [“Accessing the File Browser”](#) on page 131.
- Shot bin. See [“Accessing the File Browser”](#) on page 131.

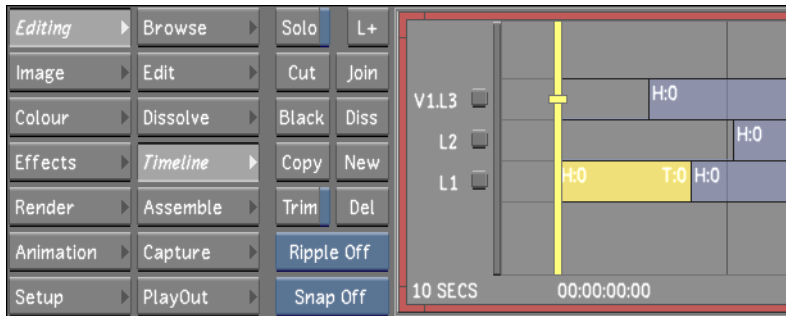
- Player and full-screen Player. See [“The Player”](#) on page 87.

NOTE: From the Timeline menu, you can press **ENTER** to toggle through these different views.

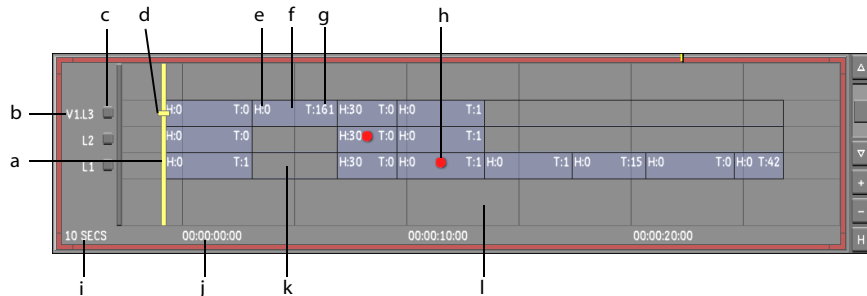
To access the Timeline menu and Multi-Layer Timeline:

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) Positioner	e) Element Head	i) Timeline scale
b) Layer name	f) Element	j) Starting timecode
c) Layer selector	g) Element Tail	k) Gap
d) Focus point	h) Shot Priority indicator	l) Canvas

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”](#) on page 228.

Layer name — The name of the layer, where V stands for Video and L for Layer. Adding a layer generates V1L2, V1L3, etc.

Layer selector — Selects the entire layer in preparation for an operation.

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”](#) on page 228.

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”](#) on page 207.

Timeline scale — The scale of the timeline. Zooming in or out of the timeline allows you to view more or less detail.

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.

For the hot keys to these functions, see [Chapter 23, “Hot Keys,”](#) on page 591.

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.

L+ — Adds a new layer above the current layer(s).

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.

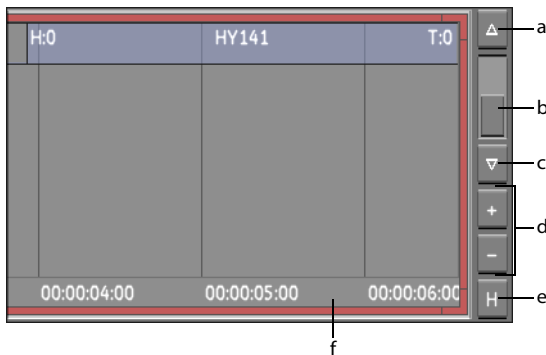
Del — Deletes the current selection.

Timeline Navigation Controls

The following is a list of hot keys and mouse gestures commonly used to navigate within the Timeline menu. See [Chapter 23, “Hot Keys,”](#) on page 591.

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 c) Scroll timeline canvas down e) Home / selected Focus view
 b) Vertical scroll bar d) Vertical zoom control 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:

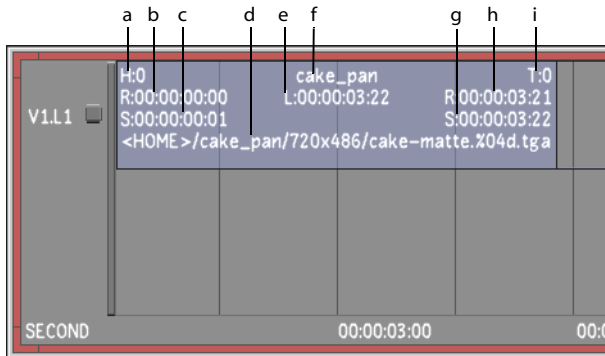
- Click +.
- More information about the element is revealed.

NOTE: Each time you press +, more information is revealed.

To minimize the layer:

- Click -.
- Other layers become visible.

The following information is displayed about an element.



a) Head	d) Shot location	g) End source timecode
b) Start record timecode	e) Length of element	h) End record timecode
c) Start source timecode	f) Name of element	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 Shot bin or Browser to the Multi-Layer Timeline or to the Storyboard. See [“Adding Shots to the Storyboard”](#) on page 169.

NOTE: Snap and Ripple mode affects how shots are added to the timeline.

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.

HINT: 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:

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.

NOTE: The status of Ripple and Snap modes are taken into account when dropping elements.

Snap mode

Snap mode affects how elements are placed when gesturally moved or copied to the timeline. Snap mode consists of the following three options.

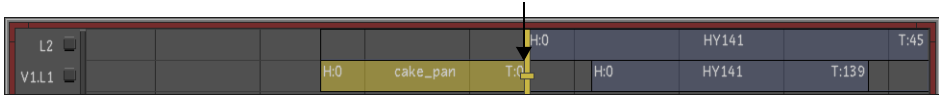
Snap Off — Drops the element at the location that you choose.

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.



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.

To copy elements within a layer:

1. In the Multi-Layer Timeline, select the element(s) you want to copy.
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.
4. To maintain a vertical lock on the copied element, hold down the **SHIFT** key.

NOTE: If you wish to copy the element to a different horizontal position, do not use the vertical lock.

5. Drag the element to the desired layer and horizontal time position.
6. 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 either by:
 - holding down the **SHIFT** key and selecting the far-right element you want to copy, or
 - holding down the **CTRL** key and individually selecting the remaining elements you want to copy

NOTE: Use the **SHIFT** key if you want to select all the elements in between the far-left and far-right elements. Use the **CTRL** key if you want to select only the individually selected elements.

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.
5. To maintain a vertical lock on the copied elements, hold down the **SHIFT** key.
NOTE: If you wish to copy the elements to a different horizontal position, do not use the vertical lock.
6. Drag the elements to the desired layer and horizontal time position on the Timeline.
7. Release the mouse button (and the **SHIFT** key, if required) to complete the copy.

NOTE: How copied elements are inserted depends upon the Snap mode. See [“Snap mode”](#) on page 194.

Adding Black Media to the Multi-Layer 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”](#) on page 187.

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.

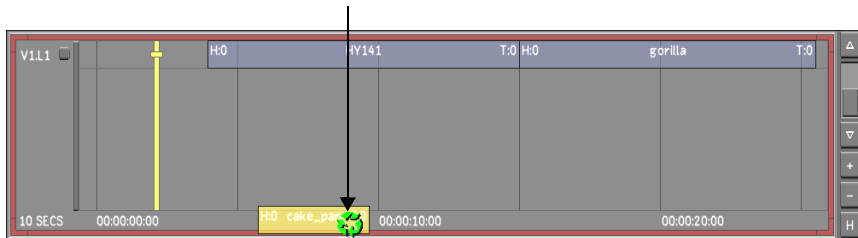
Deleting Elements

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:

- Do one of the following:
 - Select the element(s) (see [“Selecting Elements”](#) on page 193), click Del, and then confirm the deletion.
 - Select the element(s) (see [“Selecting Elements”](#) on page 193), and press **SHIFT+D**.
 - Click Del and confirm the deletion, or press **SHIFT+D** to delete the element with focus (no selection required).
 - Select the element(s) (see [“Selecting Elements”](#) on page 193), and drag it 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”](#) on page 171. 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”](#) on page 199). Their length can also be increased or decreased by modifying the clips around them. Gaps can be selected, copied, or deleted.

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 [Chapter 12, “Colour Grading: Basics,”](#) on page 291.

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”](#) on page 197). 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”](#) on page 201.

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”](#) on page 199.

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”](#) on page 199.
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

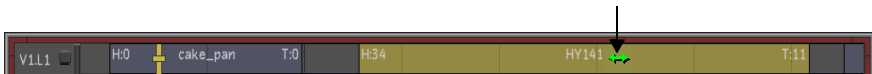
Slipping simultaneously trims the head and tail of the element without changing its position in the sequence. The duration of the shot is not altered. Other elements in the edit sequence are not

affected and the total duration of the sequence does not change. You can slip a single element or a selection.

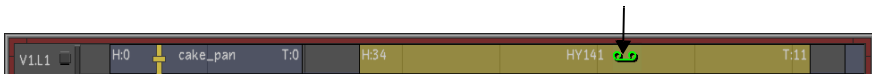
NOTE: Slipping is a non-rippling action.

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.

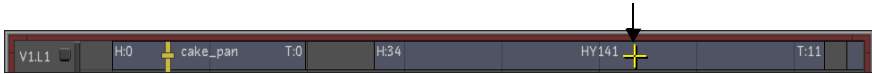
NOTE: Sliding can only occur if there is a gap before and/or after the element.

You can slide an element or a transition. See [“Transitions”](#) on page 202.

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

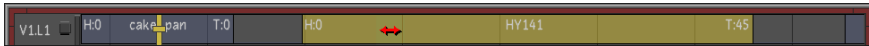
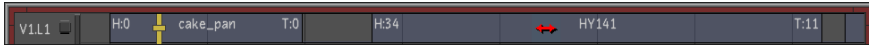
There are three types of Ripple mode when you slip and slide.

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 (or press **SHIFT+C**).

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 (or press **SHIFT+J**).

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.

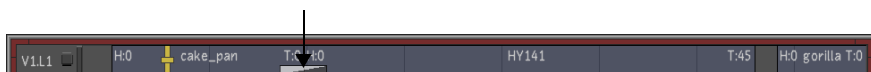
Adding a Dissolve

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”](#) on page 185.

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”](#) on page 204.

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”](#) on page 231.

Top Vertical Priority

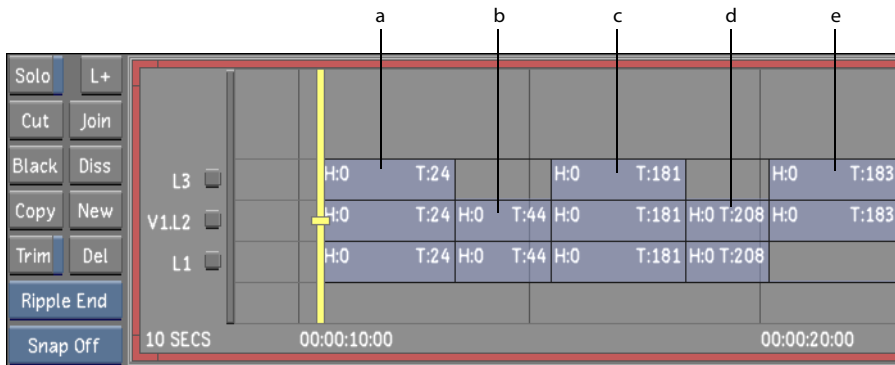
With exceptions, at any given point in a timeline with multiple layers, the top shot in a vertical stack of layers is the shot that is displayed, rendered, or played out to tape. This is the Top Vertical Priority rule and it is the default behaviour of the Multi-Layer Timeline when Solo mode is disabled (See [“Solo Mode”](#) on page 211).

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”](#) on page 211). There are a couple of features which alter the way the Top Vertical Priority rule works; layer muting (see [“Muted Layers”](#) on page 206), and shot priority (see [“Shot Priority”](#) on page 207).

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)

c) Shot 3 (L3)

e) Shot 5 (L3)

b) Shot 2 (L2)

d) Shot 4 (L2)

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” on page 220. For example, in the following screen capture, the shot being displayed is shot 3 on Layer 2, and the current frame is frame 183.

#2.003.00183

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:

- 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:

- 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.

V1.L3	☐	H:16	T:167			
L2	☐	H:16	T:167	H:32	T:166	H:31
L1	☐	H:16	T:167	H:32	● T:166	H:31

In the following example, the dissolve works as expected because the dissolve is on the same layer as the priority shot.

L3	☐	H:16	T:167			
V1.L2	☐	H:16	T:167	H:32	T:156	H:31
L1	☐	H:16	T:167	H:32	● T:156	H:31

- 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.

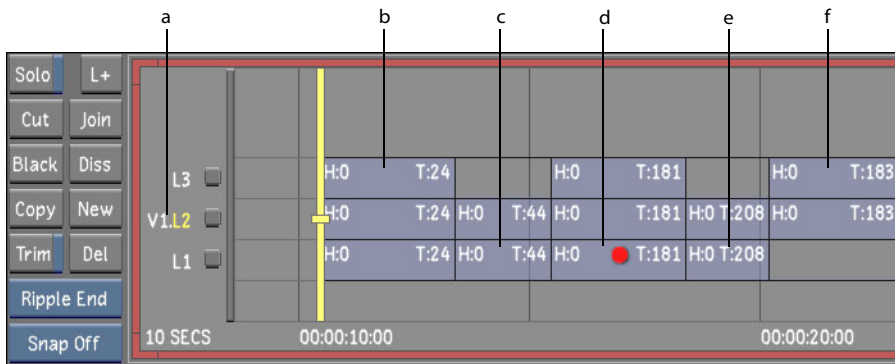
In the following example, the first Layer 3 shot is displayed completely and then the Layer 2 shot is displayed starting somewhere in the middle of the shot.

V1.L3	☐	H:0	☒	T:44	H:0	T:54
L2	☐			H:31	☒	T:13
L1	☐	H:0	☒	T:44		



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:

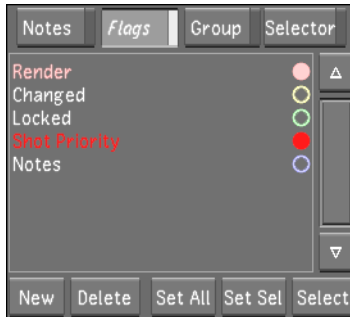
- In the Multi-Layer Timeline, **SHIFT+right-click** the shot (in the desired layer) to which you would like to assign priority.
- A red dot appears on the shot.

To assign shot priority to the shot in focus:

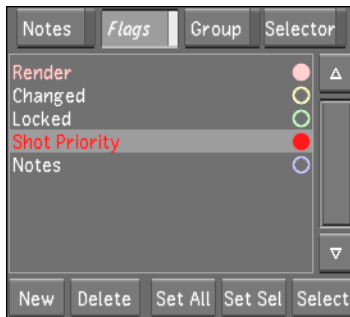
1. In the Multi-Layer Timeline, make sure the positioner and its focus point are focused on the shot to which you apply priority.
2. Press **SHIFT+\.**
- A red dot appears on the shot in focus.

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”](#) on page 193.
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:

- 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”](#) on page 206 and [“Shot Priority”](#) on page 207). 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.

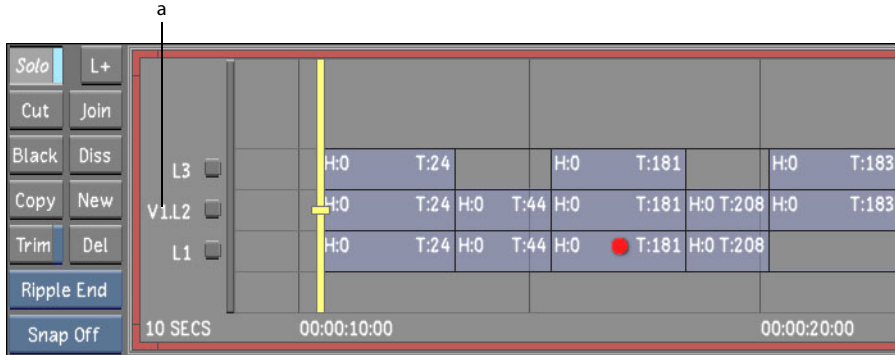
NOTE: If you are using the solo function as a method to create editorial or grade versioning, please keep in mind that the status is not part of the grade file data structure. Therefore, this state is not saved upon exiting Lustre.

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"](#) on page 198.



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:

- 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 [Chapter 18, "Rendering,"](#) on page 465.

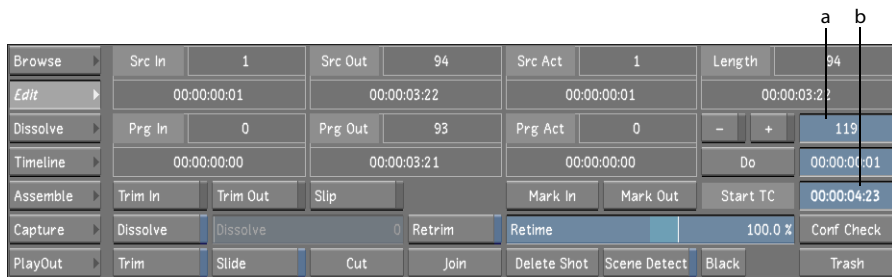
Modifying the Timeline's Starting Record Timecode

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.

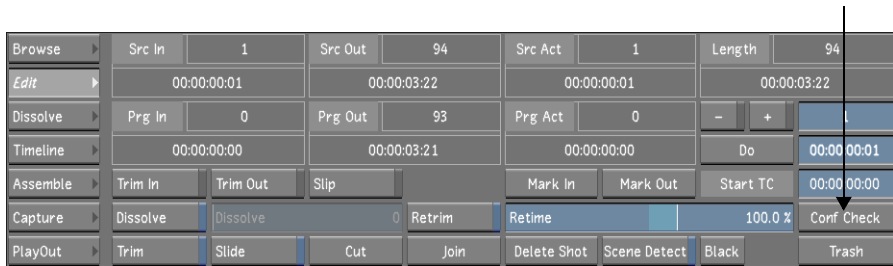
Performing a Confidence Check

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. 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 [“Loading a Cut”](#) on page 50.
 - Creating a newly assembled timeline from an EDL. See [“Working with EDL, ALE, and Cutlist Files”](#) on page 143.
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”](#) on page 514.
4. Click Editing, and then click Edit to display the Edit menu.
5. Drag the digital reference cut from the Shot bin 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”](#) on page 171.

To trim from the Timeline menu, see [“Trimming Elements”](#) on page 198.

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

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.

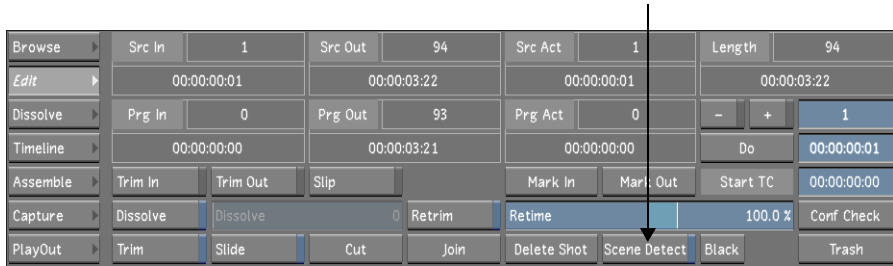
For example, you may have a finished edited HD master tape that needs to be colour graded before it is ready for broadcasting. You can capture the entire tape as one shot, and then use scene detection to cut it into separate shots. You can then colour grade each shot one-by-one.

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”](#) on page 514.
2. Create a new empty cut. See [“Working With Cuts”](#) on page 49.
3. Drag the digitized shot from the Shot bin to the Storyboard.
4. Click Editing, and then click Edit to display the Edit menu.

5. Enable Scene Detect.

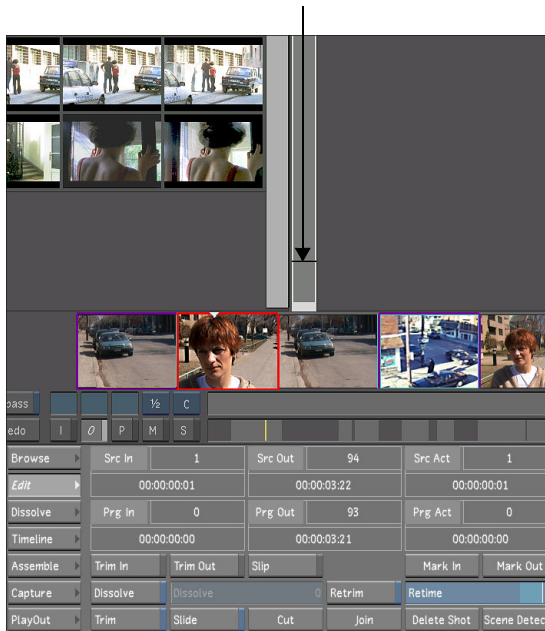


HINT: If the Editing While Grading button is enabled in the User settings, you can start a scene detection from any menu by pressing **SHIFT+S**. For more information about the Editing While Grading button, see ["System & Menu Settings"](#) on page 40.

A threshold indicator bar appears to the right of the Shot bin. 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.



Images courtesy of Hungarian Academy of Film & Theatre, 3rd year

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”](#) on page 180.

8

Playing, Viewing and Sorting Shots

Summary

About Playing, Viewing, and Sorting	219
Playing Shots	219
Panning and Zooming the Image	222
Setting Image View Options	222
Navigating through Shots	226
Storyboard Viewing Options	230
Viewing Reference Images	236
Viewing Multiple Shots in the Player	238
Timeline Sort Mode	242
Caching Frames to Memory for Real-Time Playback	249
Player Hot Keys	250

About Playing, Viewing, and Sorting

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 hot keys. To play shots, you must first load them into the Storyboard. See [“Moving Shots to the Storyboard”](#) on page 139.

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.

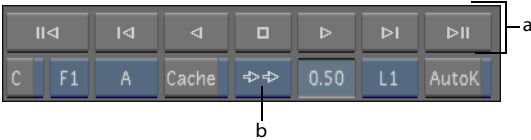
When you play shots in the Player, you can do the following:

- 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”](#) on page 113.

To play shots in a cut:

1. Load your shots into the timeline. See [“Loading a Cut”](#) on page 50.
2. Click the Play Mode button or press the . key (the decimal point on the numeric keypad) 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.

NOTE: For control surface mappings, see the *Autodesk Control Surface User Guide*.

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 hot keys 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.
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”](#) on page 226.

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 hot keys and mouse gestures to play shots, access both playheads, display split view and multi-view, and pan and zoom the image. You can also display the Storyboard, vectorscope, histogram, and waveform monitor. See [“Viewing Colour Distribution”](#) on page 302.

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 hot keys 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 Hot Keys”](#) on page 250 for a list of the hot keys and mouse gestures commonly used in the full-screen Player.
3. To exit the full-screen Player, press **ENTER** again.

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:

- 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:

- 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.



HINT: 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.
- Display colour menus in the Player alongside the image.

For control surface mappings, see the *Autodesk Control Surface User Guide*.

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:

- 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 Chapter 18, “Rendering,” on page 465. 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 current selection for a secondary correction. See “Creating a Secondary by Keying a Range of Colours” on page 347.
S	L	The stored reference image. See “Viewing Reference Images” on page 236.

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”](#) on page 484.

To set the resolution:

- In any menu, click the Full/Half Resolution button or press **F9**.



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 Colour View Mode

You can display all colour channels of an image, or individual channels in grayscale. The grayscale versions of colour channels represent in values of gray the amount of that colour channel found in each part of the image. The darker the gray, the greater the amount of colour present.

To set the colour view mode:

- Click the Colour Mode button one or more times.



The button cycles through the different channels and the image is updated.

Button Label	Channel Displayed
C	Colour (all channels)
R	Red
G	Green
B	Blue
L	Luminance

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:

- Toggle the F1/F2 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 [“Selecting Print LUTs”](#) on page 128.

To cycle between LUTs:

- Click the LUT button one or more times.



The button label changes to L1, L2 or L3, indicating the currently loaded LUT.

Displaying Colour Menus in the Player

You can display the Colour 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”](#) on page 236.

To display Colour menus in the Player:

- Press **F7**.

The Colour menus appear alongside the image, and are disabled in the main user interface.

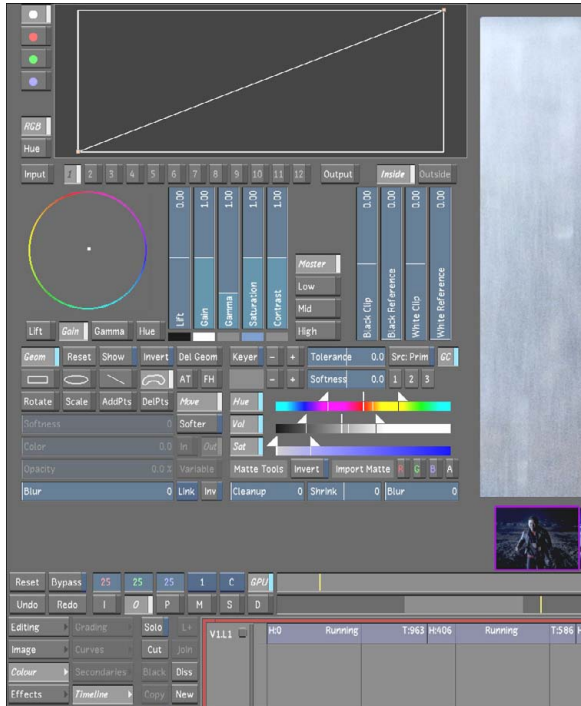


Image courtesy of Moviworld / UK File & TV Company / Videolab

Navigating through Shots

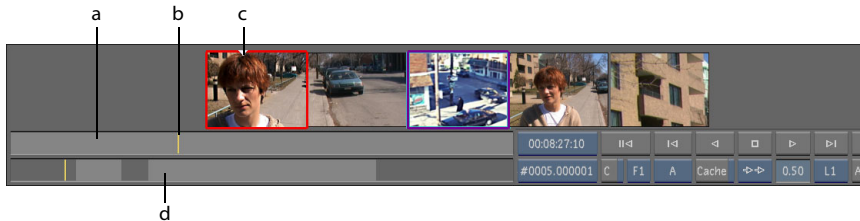
Various methods are available for navigating through the shots in your cut. You can navigate using:

- Storyboard and the shot and scene timebars
- Playback controls and hot keys
- Timecode field
- Timecode calculator

You can also navigate through shots with the control surface. See the *Autodesk Control Surface User Guide*.

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) Shot timebar
- b) Shot positioner
- c) Storyboard position arrow
- 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:

- 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:

- Drag the positioner of either timebar to the left or right.

To pan the Storyboard:

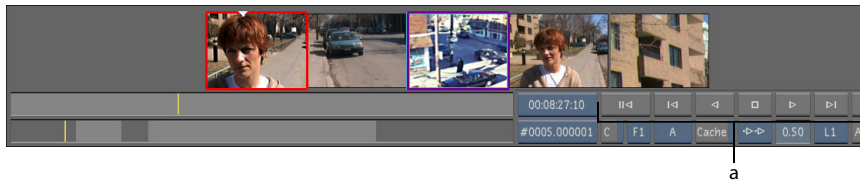
- Place the cursor over any thumbnail, press the middle mouse button, and drag to the left or right.

To go to the first frame in a shot:

- Click the shot thumbnail in the Storyboard.

Navigating with the Playback Controls and Hot Keys

You can use playback controls to navigate through shots. There are hot key equivalents for several of these controls.

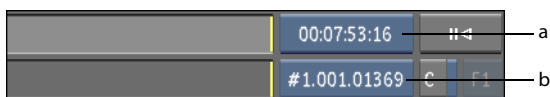


a) Playback controls

Playback Control	Hot Key	Description
	N/A	Goes to the first frame in the cut. To step one frame back, right-click the button or press ALT on the right-hand side of the keyboard.
	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.
	RIGHT ARROW	The playback control goes to the last frame in the shot. The hot key 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 or press CTRL on the right-hand side of the keyboard.

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

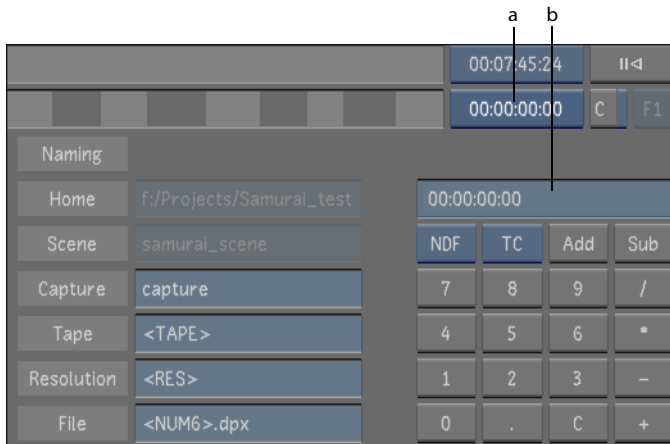
b) Timeline Information field

You can click the timeline information field one or more times to display other information, as described in the following table.

Information:	Indicates:
283	Frame number of the current frame.
0 Frame/Sec	Frame rate (when playing shots).
00:00:11:18	Timecode of the current frame.
000017+10	Frame number in film feet / frames.
#1.004.00001	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 (or press ENTER)	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.

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:

- 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 Shot bin 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 Shot bin always appears to the left of the browser.

To toggle between regular and large Storyboard view:

- 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.

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.

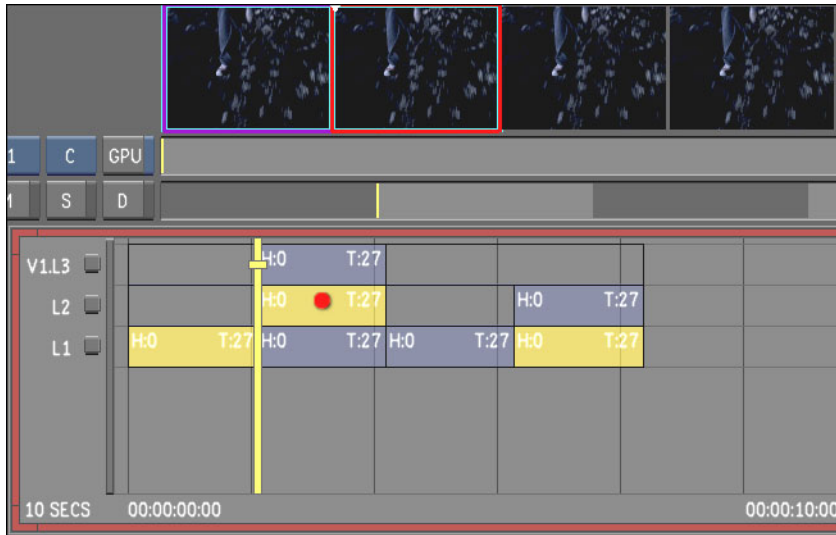


Image courtesy of Moviworld / UK File & TV Company / Videolab

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”](#) on page 205.

To select a single shot:

- 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:

- 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:

- 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 Shot bin 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 Shot bin.
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 or press **SPACEBAR**.

Displaying Only Selected Shots in the Storyboard

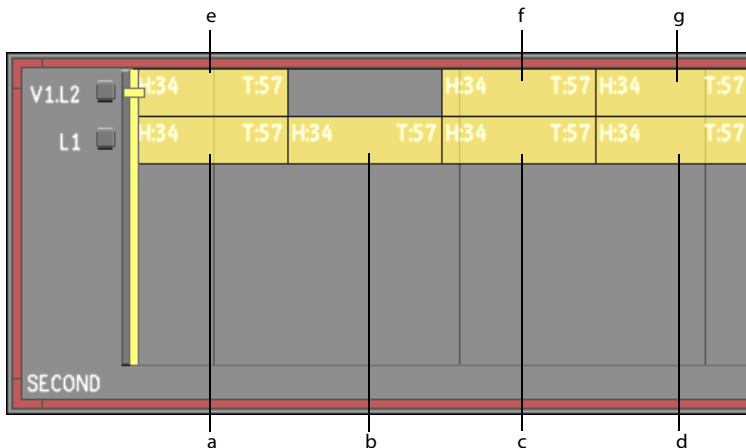
In the Multi-Layer Timeline you can select shots on multiple layers and collapse the Storyboard so that only selected shots are visible. This is useful to isolate certain types of shots for grading, such as outdoor scenes. In addition, in the Multi-Layer Timeline, you can select all the shots on all 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”](#) on page 242.

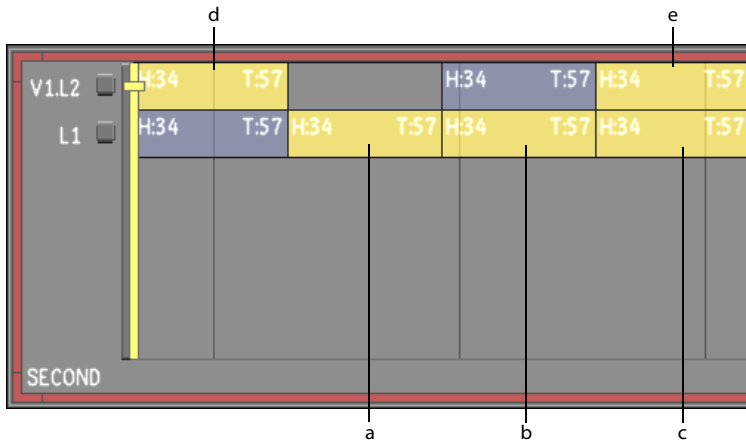
When you enable the collapse function, Lustre disables the Edit, Dissolve, Editing > Timeline, and Colour > Timeline menus. If you are in one of these menus when you enable the collapse function, Lustre takes you to the Colour > Grading menu. This behaviour guards against accidentally modifying the editorial 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 collapse is enabled is from a to g.

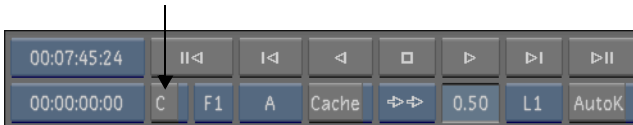


In the following example, several shots are selected and some are not. The order in which shots are displayed in the Storyboard when collapse is enabled is from a to e.



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”](#) on page 193.
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”](#) on page 193.
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.

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.

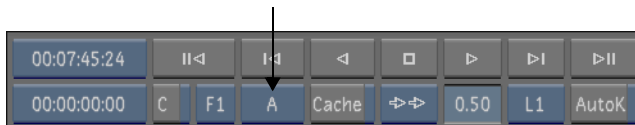
It is recommended that you work on your shot in Playhead A and load the reference images into Playhead B. For example, in Playhead B, you can view a reference image from the shot adjacent to the one you are currently working on in Playhead A. Visually compare the two images to match the saturation, whites 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.

You can also navigate through shots with the control surface. See the *Autodesk Control Surface User Guide*.

HINT: After you have assigned shots to the playheads, you can view them together in split view. See [“Viewing Multiple Shots in the Player”](#) on page 238.

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.

- Press **L** or click the S button to display the reference image in Playhead B.



The reference image appears in the Player.

HINT: 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).

- 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:

- 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.

- 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.

- Press **F2** or select B from the Playhead button.

The shot is assigned to Playhead B.

- Press **L** or click the S button to display the reference image in Playhead B.

The reference image appears in the Player.

HINT: At any time, press **L** to toggle between the original shot (O button) and the reference image (S button).

- 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**.

Viewing Multiple Shots in the Player

Lustre has several viewing modes that allow you to view and work with multiple shots in the Player, full-screen Player, and the SDI monitor.

You can view two frames simultaneously from the same shot or from a reference shot using split view. You can view up to 16 shots at the same time with multi-view.

Using Split View

Split view is useful for matching two shots or for copying colour correction parameters from one shot to another. With split view, you see half of each image; however, you can pan the images. The split can be horizontal or vertical.

The two frames displayed in split view are the current frames on Playheads A and B.

To view a reference image loaded from a cut:

1. Press **F5**.
The last used view is displayed—either split view or multi-view. Split view displays two frames split in half while multi-view displays the entire frame of one or more shots.
2. If multi-view is displayed, press **F4** to activate split view.

The current frames on Playheads A and B are displayed either horizontally or vertically, depending on the last displayed orientation.



Image courtesy of Moviworld / UK File & TV Company / Videolab

3. Press **F4** to toggle horizontal and vertical Split view.

The images assigned to Playheads A and B appear.



Image courtesy of Moviworld / UK File & TV Company / Videolab



Image courtesy of Moviworld / UK File & TV Company / Videolab

HINT: Press **F3** to view the reference image (Playhead B) and the shot you are working on (Playhead A) in separate viewers.

4. If needed, pan either frame by clicking the right mouse button and dragging in the image. Pan both frames simultaneously by dragging in either image with the middle mouse button.

Viewing Multiple Shots

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 toggle between viewing 2, 4, 9, or 16 shots.

To view multiple shots simultaneously in the Player:

1. Select up to 16 shots. See [“Selecting Shots in the Storyboard”](#) on page 231.

NOTE: If you do not select any shots, you will see the first shots in the sequence in multi-view.

2. Press **F5**.

The last used view is displayed—either split view or multi-view. Split view displays two frames split in half while multi-view displays the entire frame of one or more shots.

3. If split view is displayed, press **F3** to activate multi-view.

The last used multi-view mode is displayed. Either 2, 4, 9, or 16 shots are displayed.

4. If needed, press **F3** one or more times to change the number of shots displayed.

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.

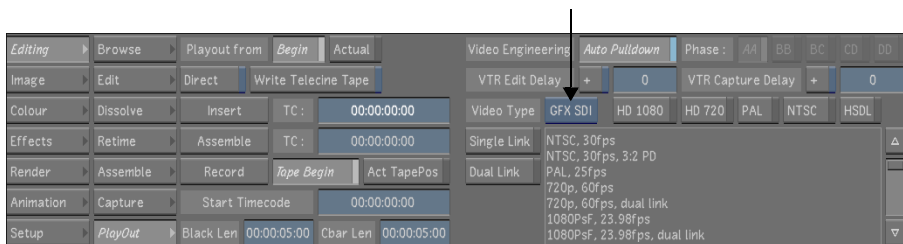
5. If needed, pan the displayed shots by clicking the middle mouse button and dragging in any image.
6. If needed, switch playheads and navigate to another frame on either playhead.
7. If needed, make a new selection of shots while in multi-view, and then press **F6** to update multi-view with the new selection.
8. To turn off multi-view, do one of the following:
 - Press **F5** to return to a single image.
 - Press **F4** to return to split view.

Viewing Multiple Shots on the SDI Monitor

You can use split view and multi-view in the SDI monitor.

To view two frames with split view, or multiple shots on the SDI monitor:

1. Click Editing, and then click PlayOut.
- The PlayOut menu appears.
2. Make sure GFX SDI is enabled.



3. From the Video Type list, select the video resolution and scan mode that corresponds with your footage.
4. Press **F7** to activate SDI output mode.
5. To view two frames with split view on the SDI monitor, perform the procedure for viewing two frames with split view in the Player.
6. To view multiple shots on the SDI monitor, perform the procedure for viewing multiple shots in the Player.

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.

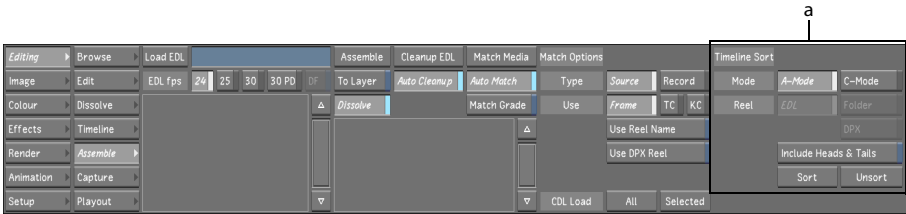
Accessing the Timeline Sort Feature

To sort the shots within your timeline, you can access the Timeline Sort section in the Assemble menu.

To access the Timeline Sort feature:

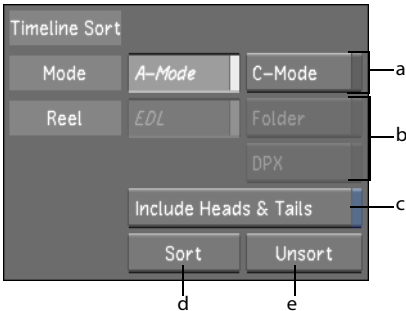
1. In the Main menu, click Editing.
2. Click Assemble.

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”](#) on page 244.

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”](#) on page 246.

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.

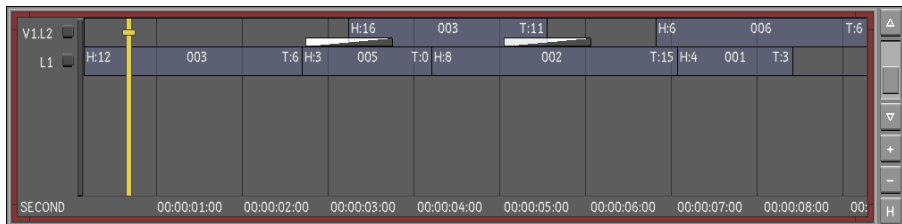
Unsort button — Click Unsort to return the timeline back to its original sequence. The editing tools are enabled.

Sorting with A-Mode

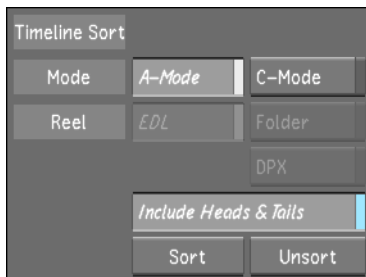
When you use A-mode to sort your timeline, your shots are organized according to their time location within the sequence (i.e., the record timecode). The shots are displayed one after another on a single layer and any dissolves and retimes are not displayed.

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”](#) on page 49.
- Render the sorted shots. See [“Rendering Shots”](#) on page 466.
- Play out the sorted shots to a VTR. See [“Playing Out to a VTR”](#) on page 529 and [“Writing Telecine-Style Tape”](#) on page 541.
- Return the sorted shots to their original order.

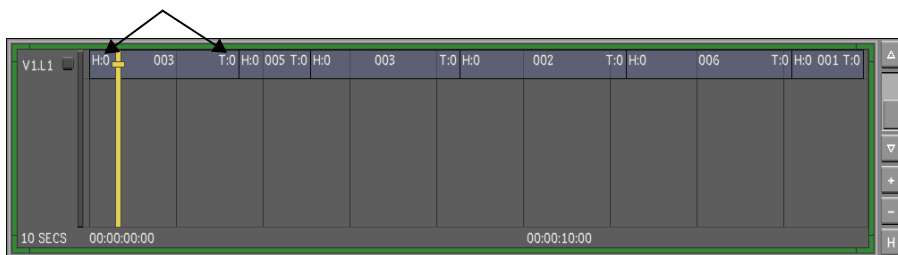
To return the sorted shots to their original order:

➤ Do one of the following:

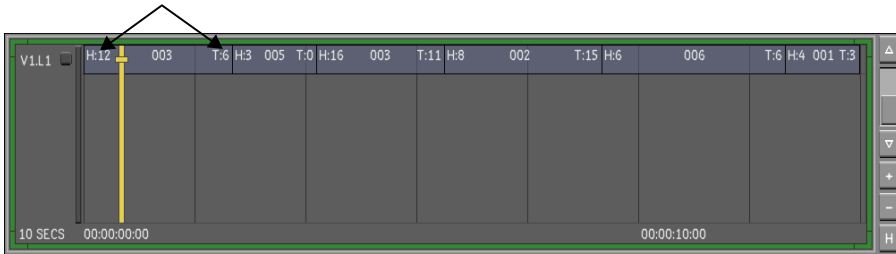
- Click Unsort.
- Press **ALT+F8**.

A-Mode Sort Using Hot Keys

You can also sort the shots, without accessing the Assemble menu, by using hot keys. 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.

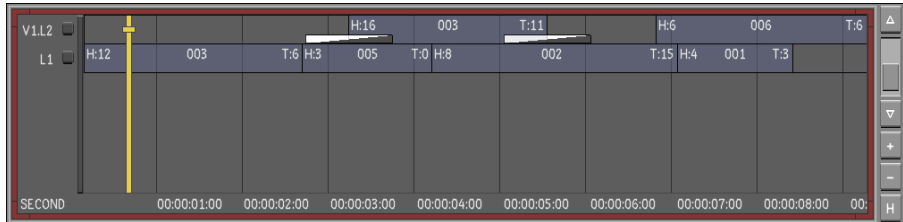
You want to sort the shots by EDL if you are working with a cut that has been assembled from an EDL, or a cut that is based on a Wiretap EDL. When you sort the timeline using the EDL option, the shots are arranged by the EDL reel name in alphanumerical order (numerical before alphabetical), and then 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”](#) on page 78). 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.

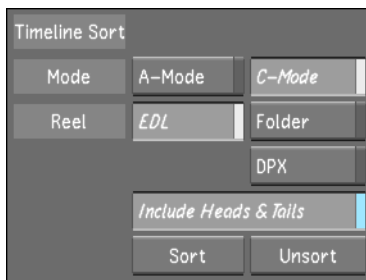
Lastly, you can sort the timeline of DPX files by their reel name. This data is read from the XML tag DPXReelName within the cut file. This XML tag is based on the data in the DPX header of the imported film scans. For more information on XML tags, see [Appendix A, “XML in Lustre,”](#) on page 607.

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.



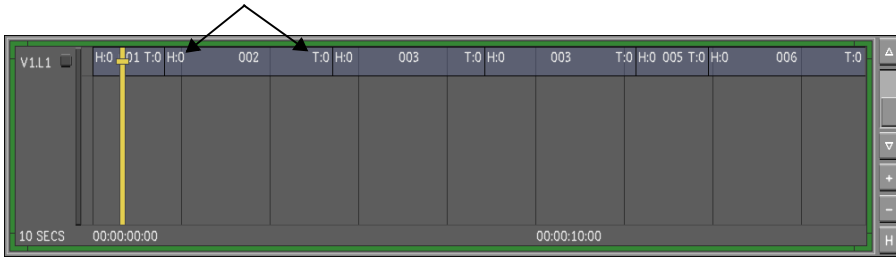
NOTE: If any of your shots have a retime value with heads and tails, the Retrim button (in the Edit menu) must be enabled so the sort function can calculate the handles properly.

4. Click Sort.

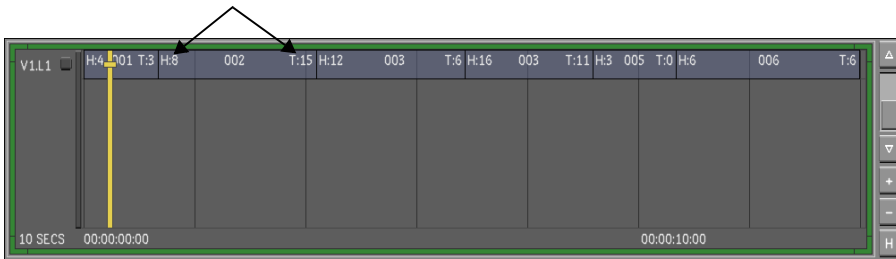
The shots are now sorted according to your criteria. A green border appears around the timeline canvas to indicate that you are working in a sorted timeline.

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”](#) on page 49.
- Render the sorted shots. See [“Rendering Shots”](#) on page 466.
- Play out the sorted shots to a VTR. See [“Playing Out to a VTR”](#) on page 529 and [“Writing Telecine-Style Tape”](#) on page 541.
- Return the sorted shots to their original order.

To return the sorted shots to the original EDL:

- Do one of the following:
 - Click Unsort.
 - Press **ALT+F8**.

C-Mode Sort Using Hot Keys

You can also sort the shots without accessing the Assemble menu. You can sort the shots by using the hot keys.

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.
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. See the *Autodesk Control Surface User Guide*.

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 [Chapter 18, “Rendering,”](#) on page 465 and [“Setting the View Mode”](#) on page 222.

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”](#) on page 219.
3. Enable Cache or press **M**.



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 Hot Keys

Following is a list of hot keys and mouse gestures commonly used in the Player and the full-screen Player.

Press:	To:
ENTER	Toggle full-screen Player mode.
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 split view.
F3	Activate multi-view.
F4	Activate split view.
F6	Update multi-view with the new selection.

Press:	To:
F7	Display the Colour menus in the Player.
F9	Toggle between full- and half-resolution in the Player.
TAB	Display and hide the Storyboard.
Q	Toggle between regular and large Storyboard view.
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.

9 Removing Dust

Summary

About Dust Removal	253
Preparing for Dust Removal	254
Removing Dust Manually	257
Removing Dust Automatically	258
Single Frame Dust Removal	260
Working with Boxes	261
Infrared Channel Dust Detection	262
Options for Saving and Loading Dust Removal Metadata	263
Rendering the Result	264

About Dust Removal

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

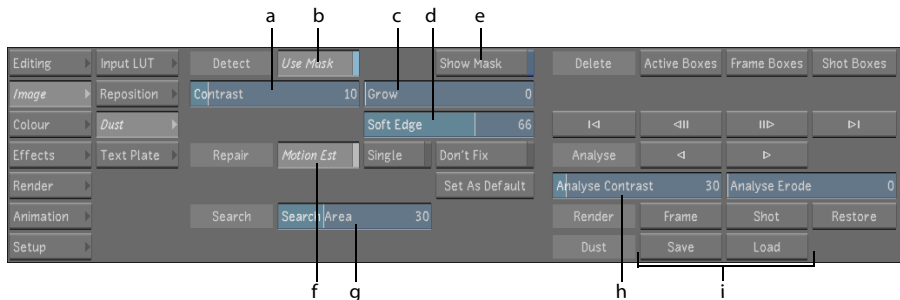
For most shots, when you set dust removal parameters for one dust artefact, the settings will work well for other artefacts in the shot. Prior to removing dust either manually or automatically, set parameters in the Dust menu to the best settings for the amount of motion and the colour composition of 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	d) Soft Edge field	g) Search Area field
b) Use Mask button	e) Show Mask button	h) Analyse Contrast slider
c) Grow field	f) Motion Estimation button	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** hot keys 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”](#) on page 261.

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”](#) on page 254.
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”](#) on page 261.
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”](#) on page 260.

Removing Dust Automatically

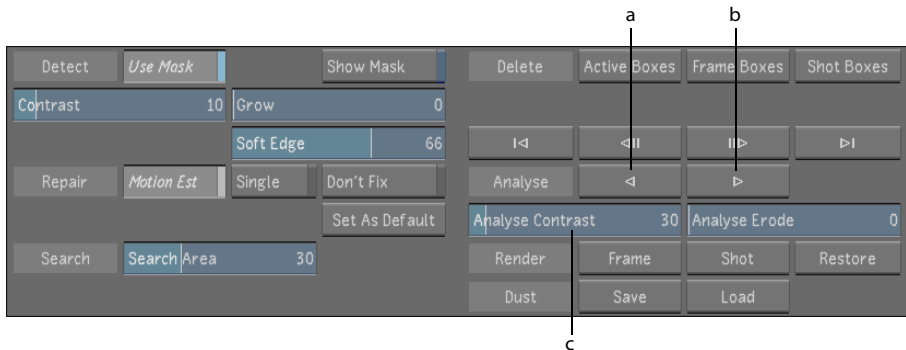
You can have Lustre automatically remove dust from each frame in the sequence by analysing the frames based on specified parameter values. Once the analysis is complete, you will need to verify that each box created is around a legitimate dust artefact, and you may need to tweak the settings for some 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”](#) on page 255.
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:

- 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.

HINT: 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:

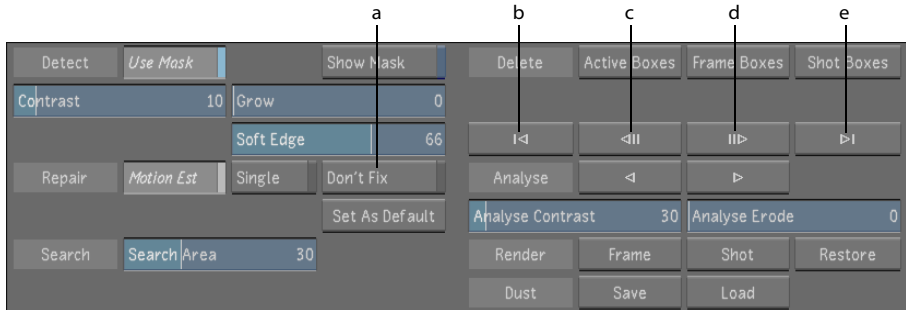
1. Visually scan the first analysed frame.

HINT: 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.

HINT: Click the Previous and Next Box buttons to move from box to box.



- a) Don't Fix button c) Previous Box button e) Last Analysed Frame button
b) First Analysed Frame button d) Next Box 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 or pressing **BACKSPACE**.
 - 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”](#) on page 260.

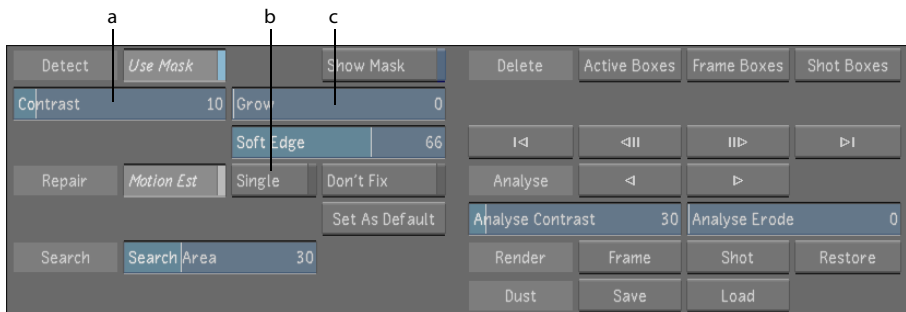
Single Frame Dust Removal

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.

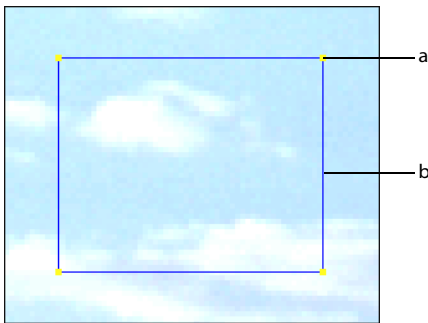
6. (Optional) If the artefact is still visible, set the repair type to Don't Fix.

The box border changes to red, indicating the applied Don't Fix repair type.

HINT: 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:

- 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:

- Right-click inside the box.

To move a box:

- Click inside the box and drag it.

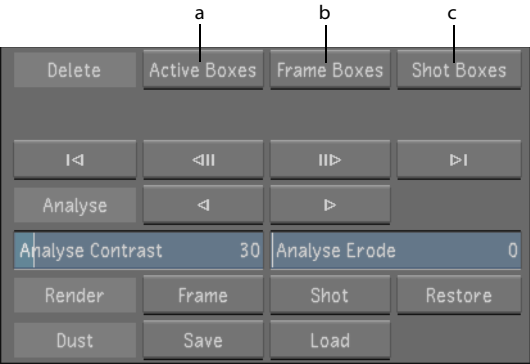
To re-size a box:

- Click on a vertex and drag.

To delete one or more boxes:

- 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.

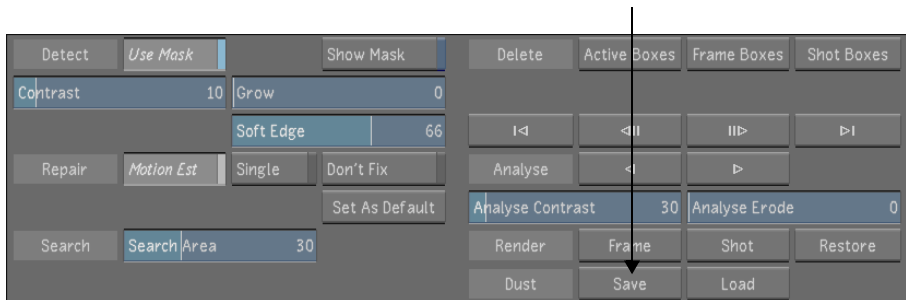
Options for Saving and Loading Dust Removal Metadata

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”](#) on page 59.

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:

- In the Dust menu, click Save.



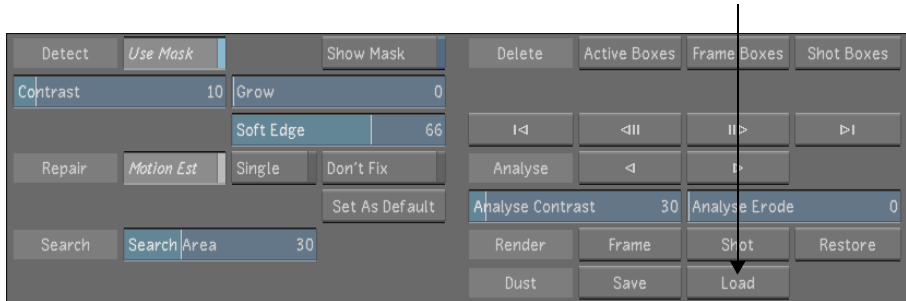
To ensure that dust removal metadata is not automatically loaded with the grade:

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:

- In the Dust menu, click Load.



Rendering the Result

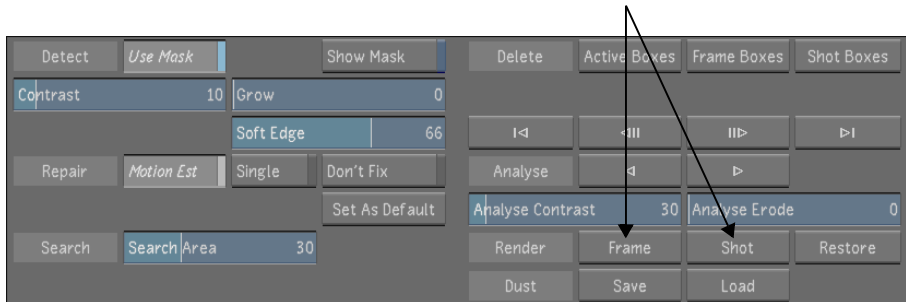
When you are satisfied with the dust removal results, you can render your material. Two rendering options are available depending on whether you render using the Render menu or the Dust menu:

- 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”](#) on page 466 and [“Rendering Dust Metadata”](#) on page 475.
- 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.

10

Repositioning Images

Summary

About Repositioning	267
Adjusting the Frame Aspect Ratio	269
Reframing an Image	270
Flipping or Flopping a Shot	272
Tracking Repositioned Shots	273
Animating Reposition Values	276
Setting Rendering Quality	277

About Repositioning

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.

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 wide-screen 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 wide-screen 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 wide-screen 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 wide-screen 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.

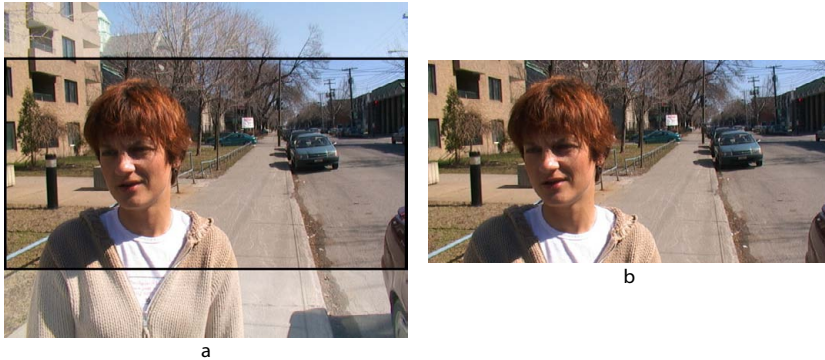


a) Original wide-screen frame

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.

For control surface mappings, see the *Autodesk Control Surface User Guide*.

HINT: For real-time playback of repositioned shots, enable GPU processing by clicking the GPU button or pressing **Y**. See [“GPU Acceleration”](#) on page 295.

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 (aspect ratio 1.33), but will deliver in a wide-screen-format aspect ratio of 1.85. Or, you may have shot in a wide-screen 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”](#) on page 65.

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.

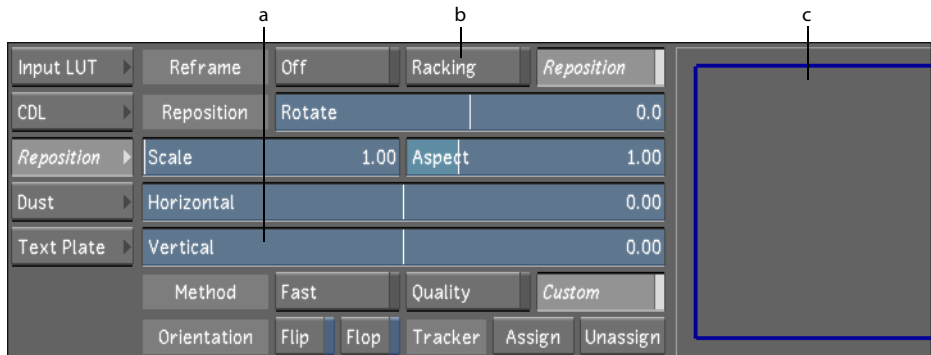
Racking an Image

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. Make sure you have set the aspect ratio to match the intended output format. See [“Setting the Aspect Ratio”](#) on page 65.
2. Select the shot to adjust.
3. Click Image, and then click Reposition.

The Reposition menu appears.



a) Vertical slider

b) Racking button

c) Preview window

4. Enable Racking.
5. Use the Vertical slider to adjust the vertical position of the image.

You can see the viewable area of the image change with respect to the entire frame in the Player.

6. You can animate the vertical position using the Animation controls. See [“Animating Reposition Values”](#) on page 276.

Repositioning an Image

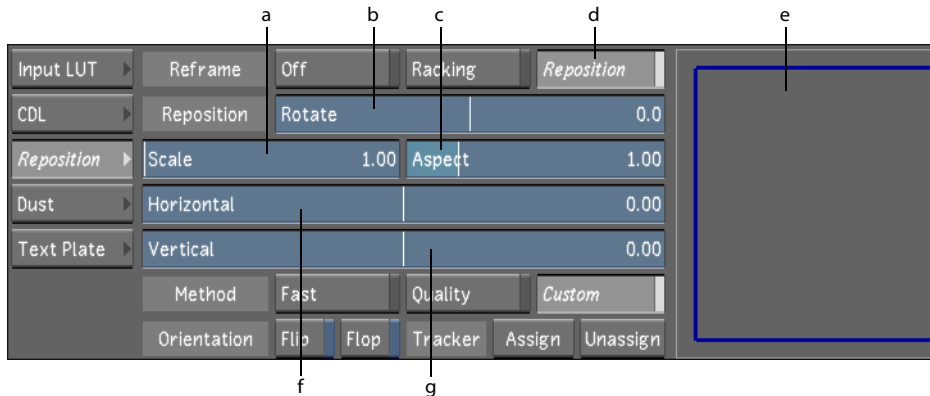
Repositioning involves scaling, rotating, and horizontally and vertically moving images to ensure that the correct elements are visible in the frame. You need to do this after adjusting the project aspect ratio. If you do not, key elements of the shot composition may be cropped or even completely off screen.

You can zoom into the image and offset the position in the horizontal and vertical directions.

To reposition an image:

1. Make sure you have set the aspect ratio to match the intended output format. See [“Setting the Aspect Ratio”](#) on page 65.
2. Select the shot to adjust.
3. Click Image, and then click Reposition.

The Reposition menu appears.



a) Scale slider	d) Reposition button	g) Vertical slider
b) Rotate slider	e) Preview window	
c) Aspect slider	f) Horizontal slider	

4. Enable Reposition.
5. Use the Scale, Horizontal, Vertical, Rotate, and Aspect sliders to adjust the image scale and positioning as needed.

As you modify the image, the Preview window visually shows the relationship of the cropped frame to the original image.

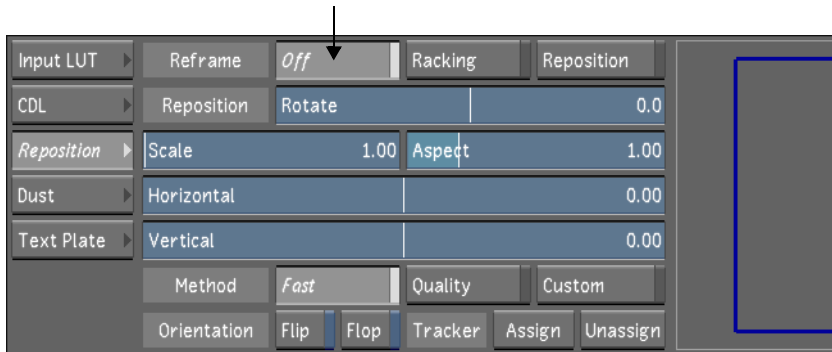
- You can animate Reposition parameters using the Animation controls. See [“Animating Reposition Values”](#) on page 276.

Turning Off Reframing

You can remove Reframing settings from your shots at any time.

To turn off reframing:

- In the Reposition menu, enable Off.



Flipping or Flopping a Shot

Use the Flip or Flop tool to change your shot's composition. Flip turns the picture upside down. Flop mirrors the picture.



Original Image



Flipped image

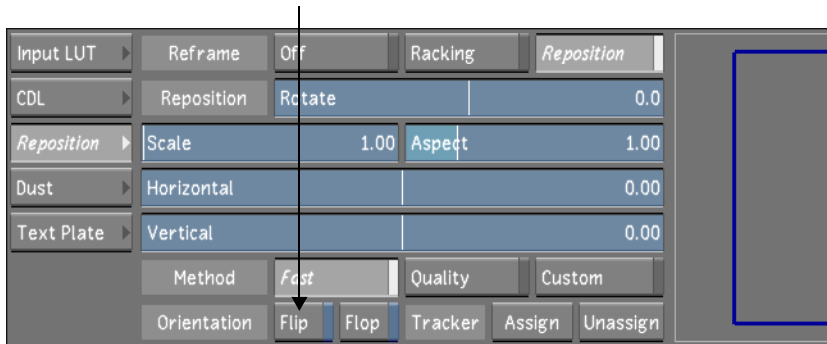


Flopped image

To flip a shot:

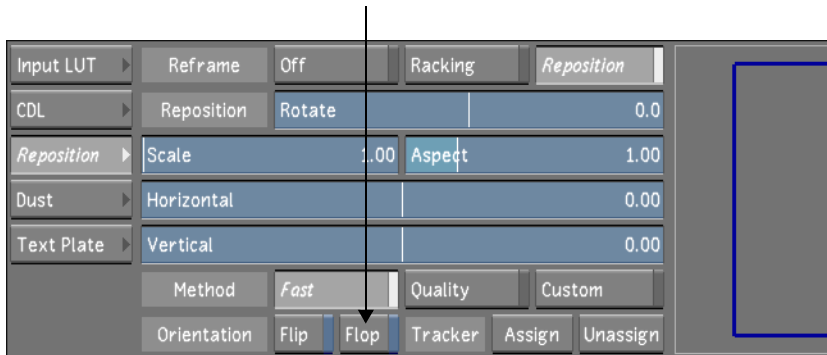
- Select the shot.
- Click Image, and then click Reposition.

3. Enable Flip.



To flop a shot:

1. Select the shot.
2. Click Image, and then click Reposition.
3. Enable Flop.



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 wide-screen 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”](#) on page 400.

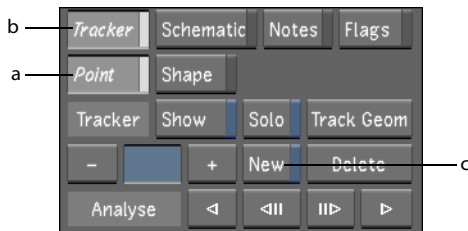
This section is intended to provide an overview of the stabilizing process. See [“Animating Geometries with the Point Tracker”](#) on page 389.

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”](#) on page 65.
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.



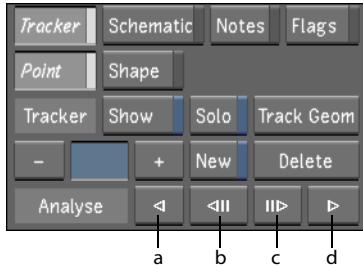
a) Point button

b) Tracker 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 Geometries with the Point Tracker”](#) on page 389.



- a) Analyse backwards
- b) Analyse back one frame
- c) Analyse ahead one frame
- d) Analyse forward

10. Enable either Racking or Reposition.

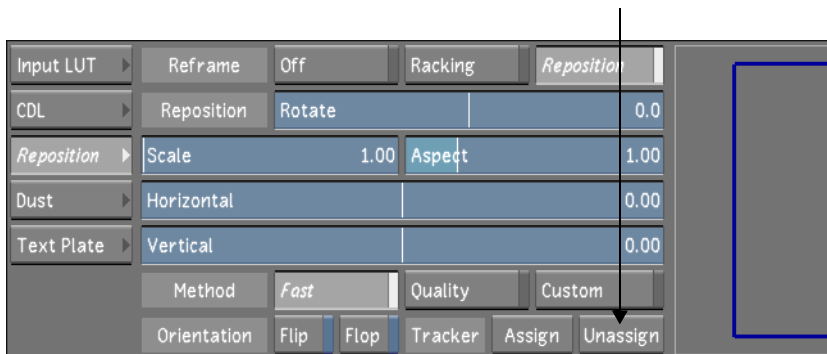
11. If you enabled Reposition, scale the image as needed.

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. Select the shot that contains tracking data.
2. Click Image, and then click Reposition to display the Reposition menu.
3. Click Unassign.

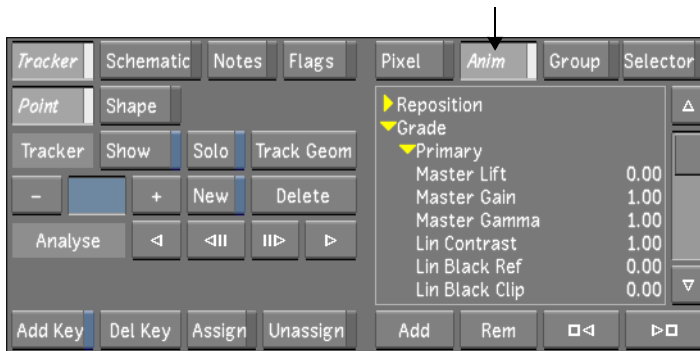


Animating Reposition Values

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:

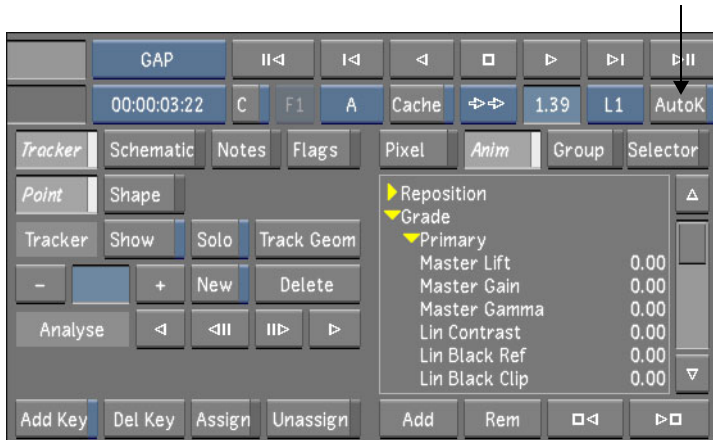
1. Select the shot to modify.
2. Click Image, and then click Reposition to display the Reposition menu.
3. Enable Racking or Reposition.
4. Click Anim to display the Animation controls.



5. In the channel hierarchy, select the Reposition directory.
6. 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.
7. Move to the last frame of the animation.
8. 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.
9. Scrub to view the animation.
10. To modify the animation curve, use the tools in the Animation menu. See [“Accessing the Channel Editor”](#) on page 424.

To animate position and scaling with Autokey:

1. Select the shot to modify.
2. Click Image, and then click Reposition to display the Reposition menu.
3. Enable Racking or Reposition.
4. Enable AutoK (Autokey).



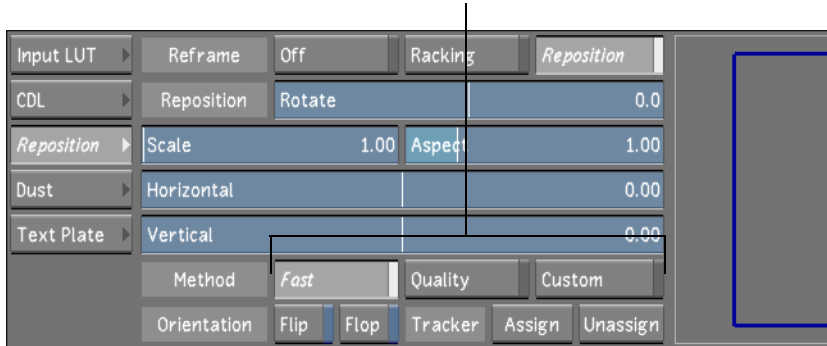
5. Move to the first frame of the shot and then adjust the scaling or the positioning as needed. Keyframes are automatically added at the current frame to the channel parameter you change.
6. Move to the last frame of the animation.
7. Set the scaling and the position. Keyframes are added to the adjusted 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”](#) on page 424.

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. Click Image, and then click Reposition to display the Reposition menu.
2. 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.

11

Input LUTs

Summary

About LUTs	279
Accessing the Input LUT Menu	280
Applying an Existing Input LUT	280
Linear Mode: Creating Conversion LUTs	281

About LUTs

Film projects use digitized film images originating from a datacine conversion process. These are high-resolution scans of the film frames that preserve much of the film's resolution, colour depth, and grain.

Most film scans contain logarithmic data while video footage contains linear data. Logarithmic data allows for more subtlety in the lower luma ranges of an image, whereas linear data maps the luma range in equal intervals from black to white. In Lustre, you can work in either logarithmic or linear colour space.

When you need to convert and gamma correct shots with non-standard code value ranges or characteristic curves into the proper range for the selected colour space, you can use an image conversion Lookup Table (LUT) on input.

A LUT is an ASCII (.lut) file that describes specific pixel values used for image data conversion both on input and output. For example, you can use an input LUT to convert a 10-bit logarithmic film image to a 12-bit linear image when working in the linear colour space. You can then apply an inverse LUT on output to restore the 10-bit logarithmic data, ensuring that your result film prints match the original shots. See [“Setting Colour Space Options”](#) on page 478.

There are as many workflows when working with film and video footage as there are projects. In some cases, you may want to apply an input LUT to film to try to give it the look of different film stock. You may even apply an input LUT to video to try to give video clips a film look.

You can also apply monitor and Print LUTs to display images properly without affecting image data. See [Chapter 5, “Monitor Calibration,”](#) on page 117.

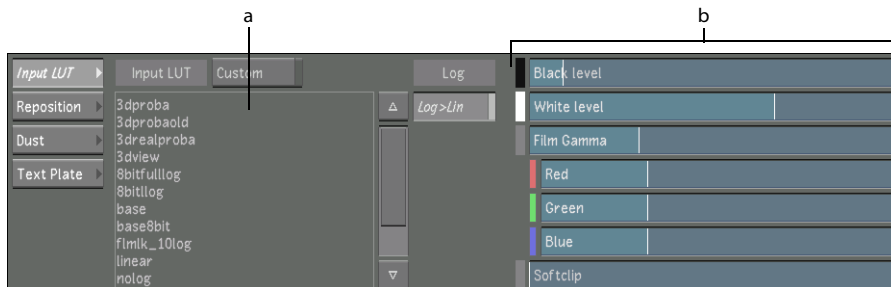
Accessing the Input LUT Menu

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.



a) Input LUT list

b) LUT Editor (Linear mode)

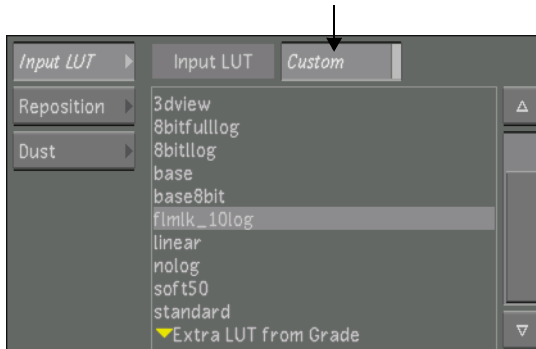
Applying an Existing Input LUT

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. 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.

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 an input LUT from the Input LUT list and then enable Custom.

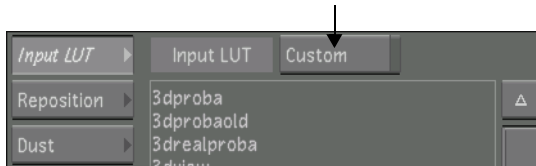


The input LUT is applied to the current shot.

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.

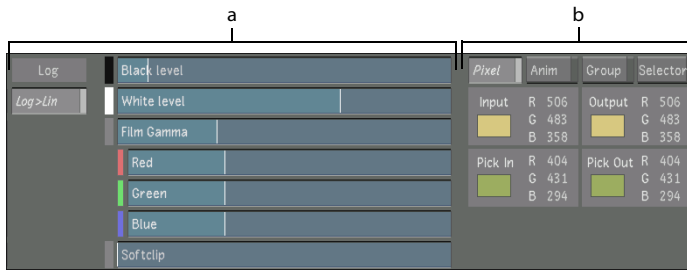


The input LUT is no longer applied to the shot.

3. To remove the input LUT completely, click it in the Input LUT list to deselect it.

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

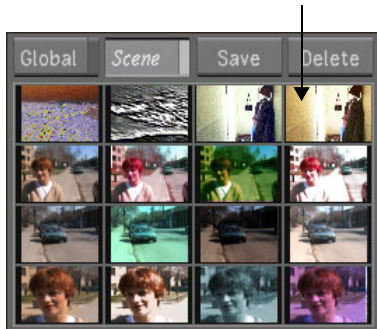
While creating Log to Lin conversion LUTs, it is important to view reference images. For example, view other shots in the cut to ensure continuity.

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 [“Grade Bins: Saving, Applying, and Deleting Intermediary Grades”](#) on page 105.

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:

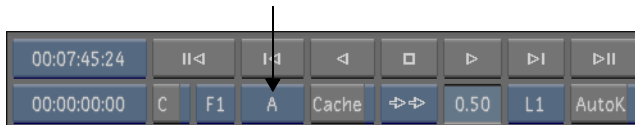
1. Load the reference image to the frame buffer by doing one of the following:
 - Go to the frame you want to use in the Storyboard and then press **K** to load it to the frame buffer.
 - Right-click a Grade bin thumbnail.



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).

HINT: 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 **F5** to enable split view.



6. Press **F4** to toggle horizontal and vertical split view.

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.

HINT: Press **F3** 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.

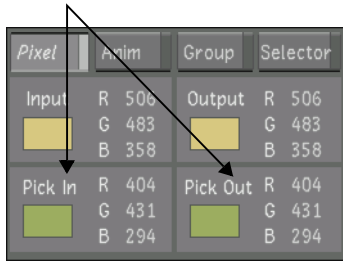
Comparing Colour Values

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:

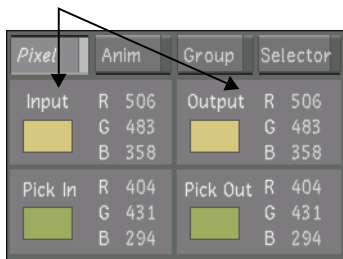
1. Click a pixel in the image.

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.

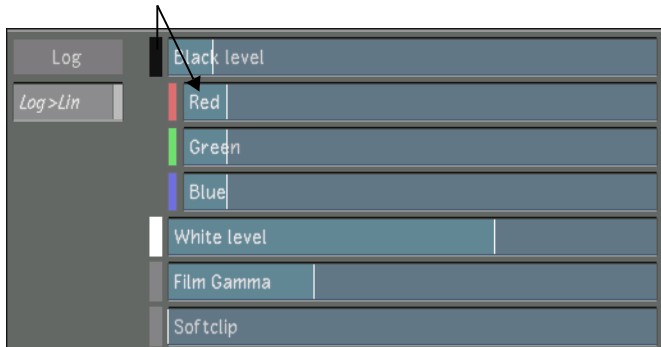


2. To modify the red, green, and blue channels in the reference white or black levels proportionally, drag the White Level or Black Level slider.



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.



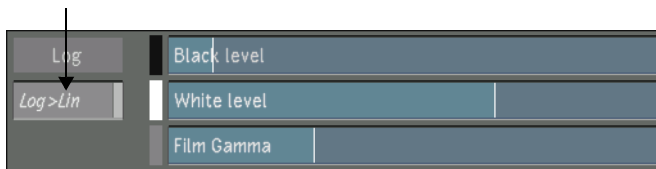
While dragging, the reference white or black level of the selected channel is displayed in the slider.

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.

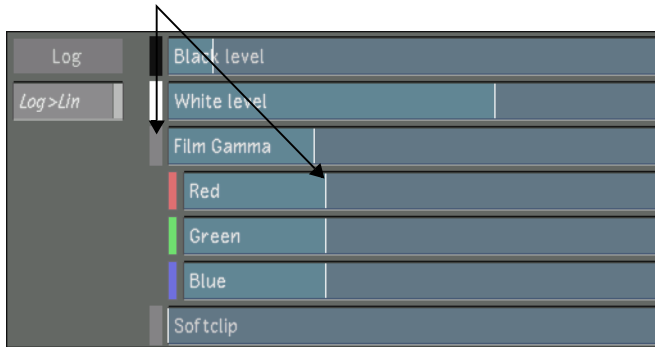


2. To modify the red, green, and blue channels for the film gamma levels proportionally, drag the Film Gamma slider.



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.



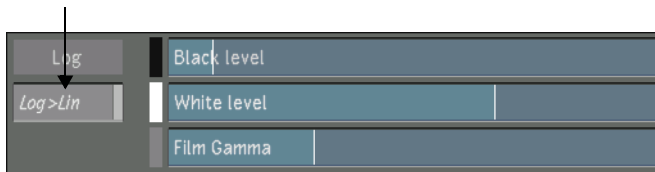
While dragging, the film gamma level for the selected channel is displayed in the slider.

Adjusting Softclip Values

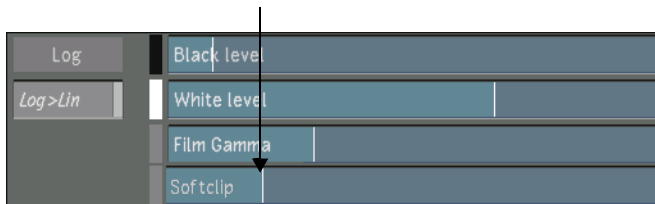
You can soften the shoulder of the gamma correction curve by adjusting the softclip levels. When you soften the shoulder of the conversion curve, you soften the transition of colours toward the reference white value. This results in softer highlights in your shot.

To soften the shoulder of the gamma correction curve:

1. In the Input LUT menu, enable the Log>Lin button.

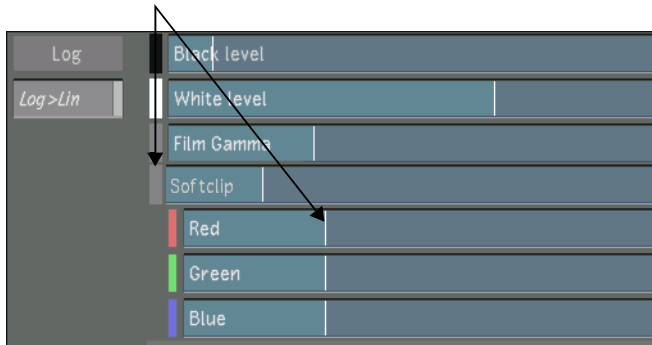


2. To modify the red, green, and blue channels for the softclip levels proportionally, drag the Softclip slider.



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.

12

Colour Grading: Basics

Summary

About Colour Grading	291
Accessing Colour Grading Menus	294
GPU Acceleration	295
Control Surfaces	299
Viewing Multiple Shots	299
Reference Images	299
Viewing Proxies	300
Analysing the Image	300
Saving and Loading Presets Using the Presets Lists	303
Colour Grade Animation	306
Grade Bin Storage	307
Colourist Multi-Layer Timeline	307

About Colour Grading

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, 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. Colour effects are applied to the image in the menu order. For example, modifications you make in the Grading menu are applied to the image before modifications you make in the Curves menu.

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" on page 54.
2. With primary colour grading, balance colours in the shot and develop a continuity grade to create a consistent colour look.	Chapter 13, "Primary Colour Grading," on page 313.
3. Further develop the colour look using the RGB and Hue curves.	Chapter 14, "Colour Grading: RGB and Hue Curves," on page 333.
4. Perform secondary colour grading to colour grade objects and colour ranges in the shot.	Chapter 15, "Secondary Colour Grading," on page 345.
5. Use Sparks plugins to apply image processing effects such as grain reduction.	Chapter 16, "Creating Lustre Sparks Effects," on page 407.
6. Adjust the colour look using the RGB curves.	Chapter 14, "Colour Grading: RGB and Hue Curves," on page 333.
7. Perform any necessary primary colour grading to adjust the colour look and complete the colour grade.	Chapter 13, "Primary Colour Grading," on page 313.

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 hot keys. 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. 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 12 secondaries. 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 colour grading 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”](#) on page 38.

Accessing Colour Grading Menus

Use the Colour menu to access the Grading, Curves, and Secondaries menus.

To access the Colour menu:

- 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 12 secondaries that can then be colour-graded using the Grading and Curves 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.

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.

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?
FX5600	Yes
FX5500	Yes (except Noise plugin 3.1)

GPU acceleration is available for the following features:

- 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:

- [Chapter 13, “Primary Colour Grading,”](#) on page 313
- [Chapter 14, “Colour Grading: RGB and Hue Curves,”](#) on page 333
- [Chapter 16, “Creating Lustre Sparks Effects,”](#) on page 407

GPU acceleration is also available for certain secondary grading features with up to 12 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, Color, and Opacity slider values for geometries, with Softer and Variable optionally enabled
- Key-in shapes based on hue, value (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 [Chapter 15, “Secondary Colour Grading,”](#) on page 345.

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.

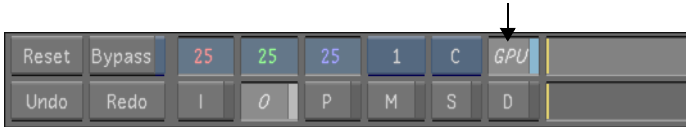
The following parameters cannot be accelerated by the GPU. These parameters are processed by the CPU instead:

- 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:

- Complete 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:

- Complete one of the following:
 - Click GPU
 - Press **SHIFT+Y**

GPU acceleration is disabled.

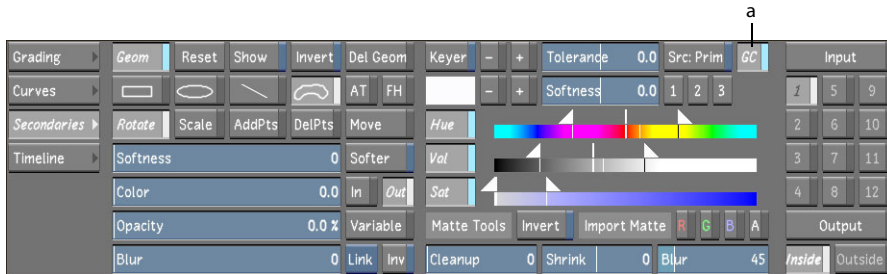
GPU Auto Switch

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”](#) on page 38. 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.

GPU Compatibility

Key blurs and geometry blurs are processed differently depending on whether CPU processing or GPU acceleration is enabled. To ensure that they are always displayed identically, enable the

GPU Compatibility (GC) button before you start work on keys, key blurs, and geometry blurs. The GPU compatibility feature is automatically enabled when GPU acceleration is enabled.



a) GC button

You can also enable the GPU compatibility feature when you set your user settings. See, [“Display & Interface Settings”](#) on page 38.

NOTE: Enabling the GPU compatibility feature does not impact final renders, nor does it jeopardize preview performance. It ensures that GPU acceleration is performed in such a way as to closely match how keys, key blurs, and geometry blurs are displayed in CPU grading mode.

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.

For control surface mappings, see the *Autodesk Control Surface User Guide*.

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 [“Viewing Multiple Shots in the Player”](#) on page 238.

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”](#) on page 236.

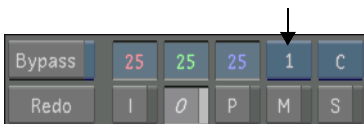
Viewing Proxies

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”](#) on page 484.

To view proxies:

- 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

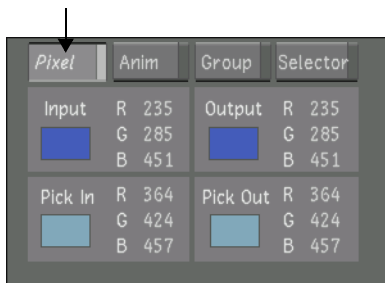
When you generate mattes, modify shots, or match colours between shots, you can sample the current image in the Player. When you sample the image, the input and output colour values are displayed in the Pixel Analyser.

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”](#) on page 347.

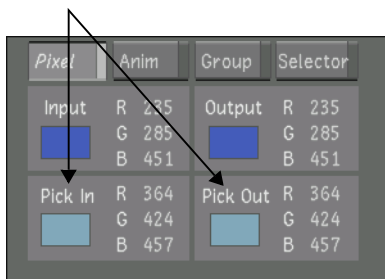
To sample colour values:

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.



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.

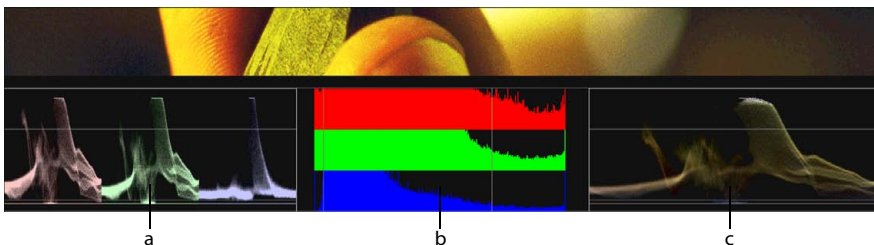
The Analyse tools show the changes you make to the colour content in a shot. Each tool is dynamically updated to reflect your changes as you modify the image.

NOTE: The histogram, vectorscope, and waveform monitor do not dynamically update if GPU processing is enabled. They retain the colour distribution of the image prior to GPU being enabled.

To view colour distributions:

- 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.



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 split view or multi-view. Split view displays two frames split in half while multi-view displays the entire frame of one or more shots.

2. If multi-view is displayed, press **F4** to activate split 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.

HINT: You can also save up to three Key presets to the Secondaries menu. See [“Saving and Loading Key Presets”](#) on page 359.

Saving Presets

Save settings from the current Colour menu to a preset. You can save settings to the following Presets lists:

- Primary presets
- Curves presets
- Key presets
- Geometry presets

NOTE: Primary presets are accessible from the Grading menu.

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.

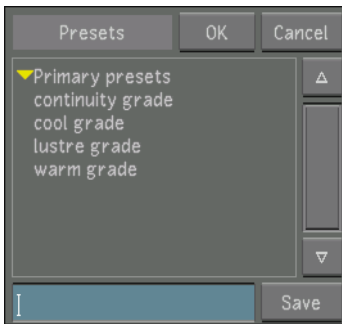
To save a preset:

1. Do one of the following:

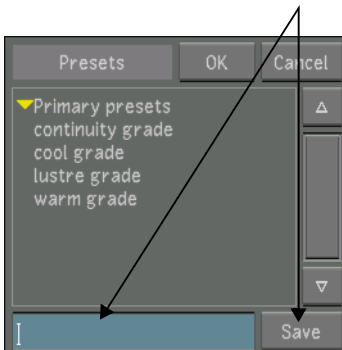
- Right-click or middle-click the Colour menu where you want to set parameters for a preset.
- In the Secondaries menu, right-click the Keyer button to set parameters for a Key preset.

HINT: You can also save a Key preset directly to the Secondaries menu. See [“Saving and Loading Key Presets”](#) on page 359.

The affiliated Presets list appears. For example, if you right-clicked the Grading menu, the Primary 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.

HINT: To save the settings to an existing preset, select the preset in the Presets list.

Loading Presets

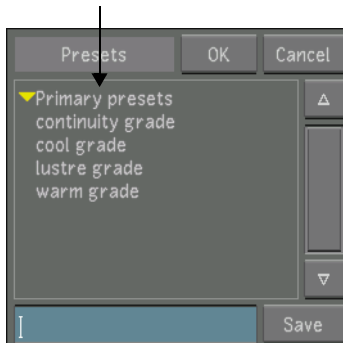
Increase efficiency by loading presets to the current shot when you want to reuse Colour menu settings. Preview the effect a preset will have on a shot before you apply the settings.

To load a preset from a Presets list:

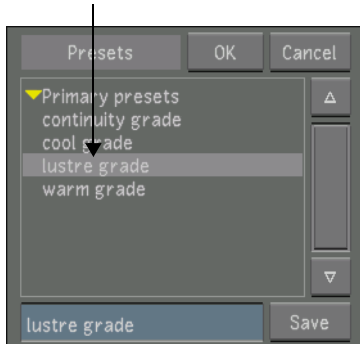
1. Do one of the following:
 - Right-click the Colour menu that contains the preset you want to load.
 - In the Secondaries menu, right-click the Keyer button to set parameters for a Key preset.

HINT: You can also load a Key preset directly from the Secondaries menu. See [“Saving and Loading Key Presets”](#) on page 359.

The affiliated Presets list appears. For example, if you right-clicked the Grading menu, the Primary presets list appears.



2. Select a preset in the list.



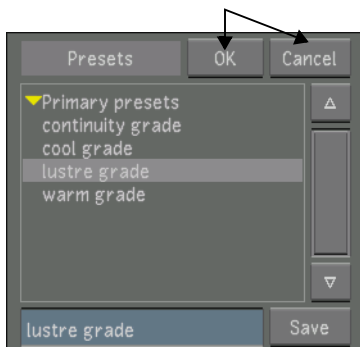
The image in the Player is updated with the settings from the preset so that you can preview the effect.

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 [Chapter 17, “Animation,”](#) on page 423.

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 [“Grade Bins: Saving, Applying, and Deleting Intermediary Grades”](#) on page 105.

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”](#) on page 242.

Accessing the Colourist Timeline Menu

To edit shots in the Colourist Timeline view, you need the Timeline menu.

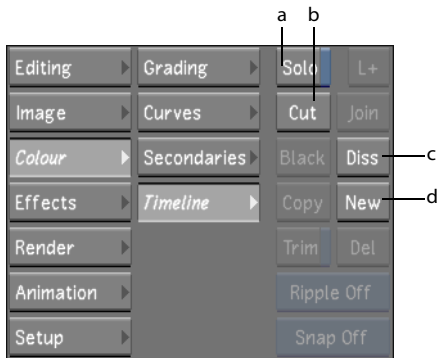
To access the Colourist Timeline menu:

1. In the Main menu, click Colour.
2. Click Timeline.

The Colourist Timeline menu appears.



The Timeline is made up of the following buttons.



- a) Solo button c) Diss button
b) Cut button d) New button

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.

HINT: You can colour grade different layers of the cut by using the dual heads split view. If you want to see the Colourist Multi-Layer Timeline and the grading toolset at the same time, navigate to the Colour > Timeline menu and press **F7**.

For more information about the Multi-Layer Timeline, see [“Multi-Layer Timeline”](#) on page 188.

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).

To copy all grading information from one shot to another:

1. Using the various grading tools, grade a shot to your satisfaction.
2. In the Colourist Multi-Layer Timeline, hold down the **CTRL** key and left-click+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.

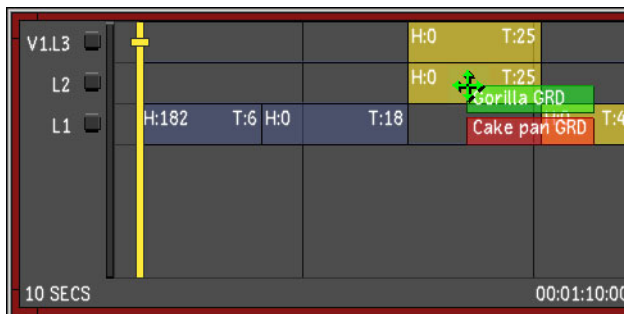


3. Release the **CTRL** key and mouse button when the cursor is over the desired destination shot. The entire shot's grading information is copied to the destination shot.

To copy all grading information from one shot to multiple shots:

1. 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”](#) on page 193.
3. Hold down **CTRL** and **SHIFT** and left-click+drag from the graded shot to any of the selected shots.

A green indicator displays the graded shot's name. When you move your mouse over a shot, a red indicator displays the destination shot name.



4. Release the **CTRL** and **SHIFT** keys and mouse button 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.

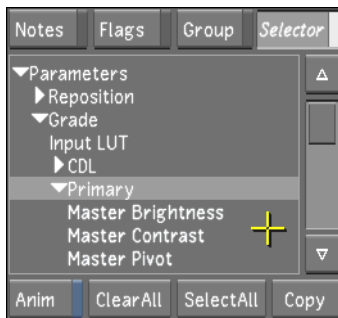
To copy a defined selection of grading information from one shot to another:

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”](#) on page 193.

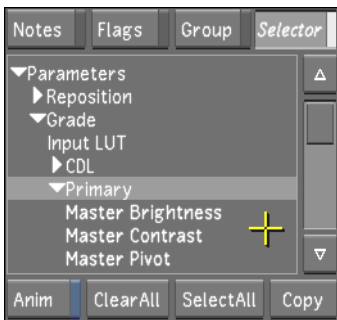
3. In the Timeline menu, enable the Selector button.



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.

13

Primary Colour Grading

Summary

<i>About Primary Grading</i>	313
<i>Accessing the Grading Menu</i>	314
<i>Logarithmic Mode: Modifying the Brightness</i>	315
<i>Logarithmic Mode: Improving Contrast</i>	320
<i>Linear Mode: Selecting Levels of the Image</i>	323
<i>Linear Mode: Modifying Lift</i>	324
<i>Linear Mode: Modifying Gain</i>	326
<i>Linear Mode: Modifying Gamma</i>	326
<i>Linear Mode: Modifying Saturation</i>	327
<i>Linear Mode: Modifying Contrast</i>	328
<i>Linear Mode: Setting Black Clip and White Clip</i>	329
<i>Matching Colours</i>	330
<i>Autograding Shots</i>	331

About Primary 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

gamma, gain, lift, and saturation values, as well as clamp minimum and maximum luminance values.

HINT: For faster playback on shots with input primary grading applied, enable GPU processing by clicking the GPU button or pressing **Y**. See [“GPU Acceleration”](#) on page 295.



WARNING: The Histogram, Waveform, and Vectorscope functions do not update when GPU processing is enabled.

Accessing the Grading Menu

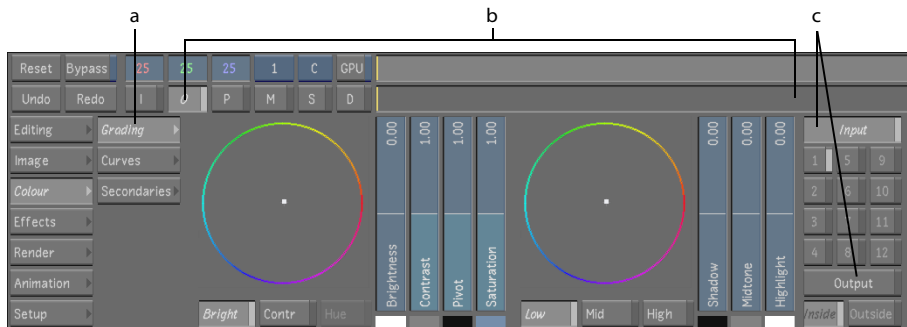
Primary colour grading is done from the Grading menu.

For control surface mappings, see the *Autodesk Control Surface User Guide*.

To access the Grading menu:

1. In the Main menu, click Colour, and then Grading.

The Grading menu appears.



a) Grading button

b) Grading menu (Log mode)

c) Input and Output buttons

2. Enable one of the following buttons.

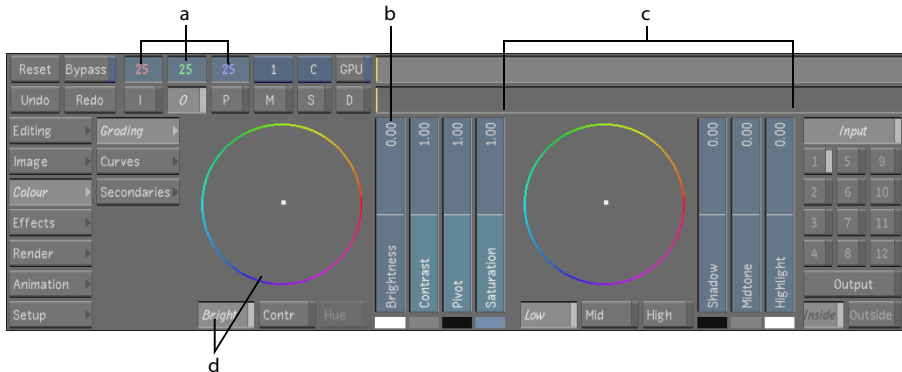
Input — Enables input primary grading. Use it to perform primary colour grading at the beginning of the colour grading process.

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”](#) on page 336.

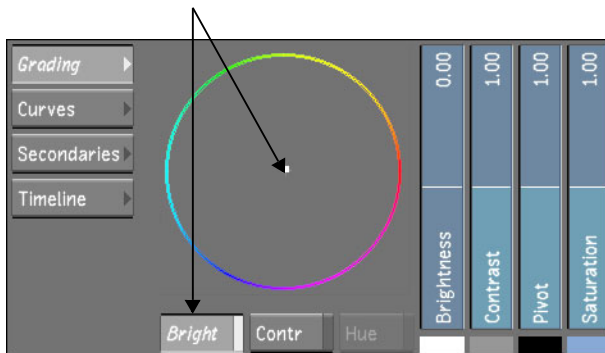
Modifying Brightness in the Entire Image

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.

HINT: 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.

HINT: 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”](#) on page 38.

NOTE: Hot keys for printer light adjustments only work in Log mode.

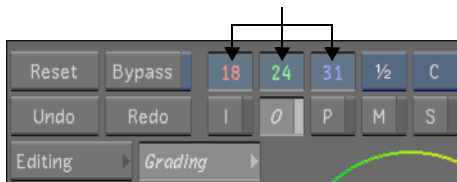
To modify brightness by printer lights:

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 hot keys 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.
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.

Press:	To:
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”](#) on page 318.

HINT: 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:

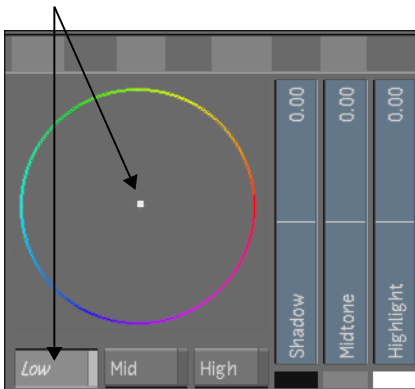
1. Indicate whether this is the initial or final primary grade:
 - To work on the initial primary colour grade, click Input.
 - To make final adjustments to the primary colour grade, click Output.

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 hot key.

To reset Brightness settings:

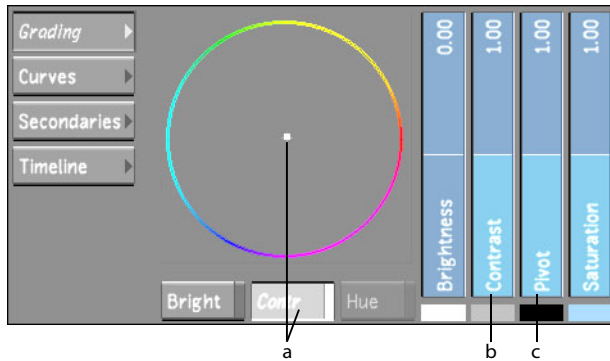
- Press - (minus sign) on the keypad.

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”](#) on page 89.

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”](#) on page 321 and [“Modifying Saturation”](#) on page 322.



- 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

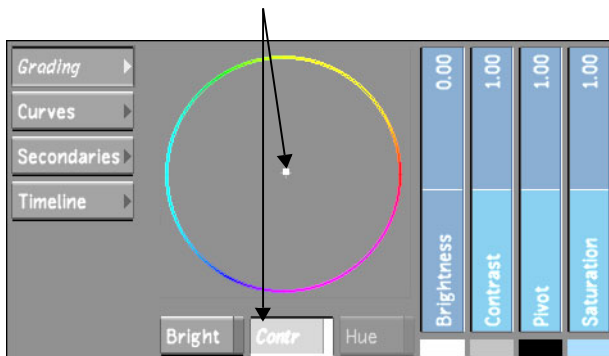
Improve image contrast by modifying the film gamma with the Contrast slider. Use the Contrast Balance wheel to increase the contrast of any channel while decreasing the contrast of other channels. These changes in contrast are relative to the pivot point.

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”](#) on page 321.

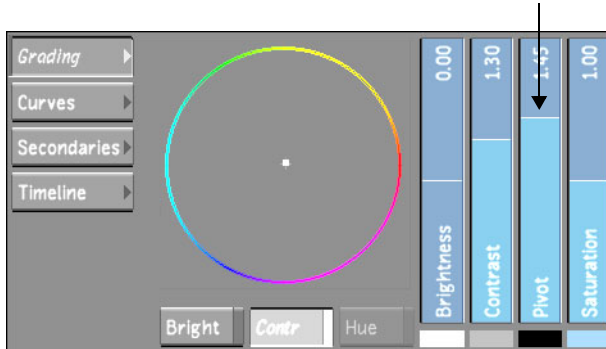
4. Make adjustments to the image using the Contrast slider and Contrast Balance wheel until you are satisfied with the results.

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**

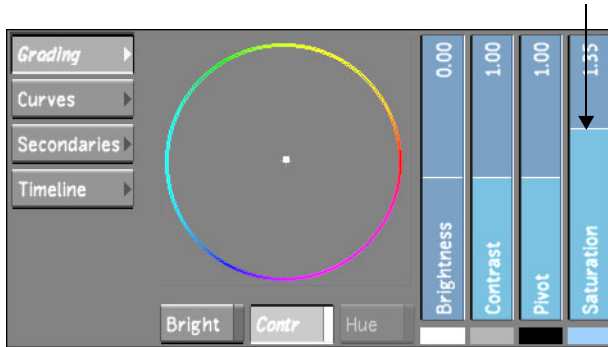
Control the image's colour intensity by modifying saturation levels.

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

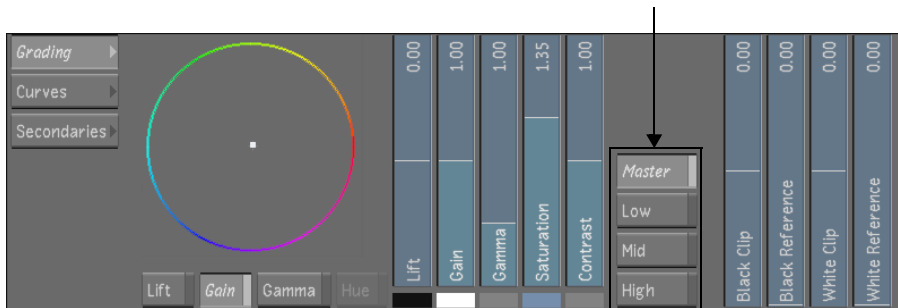
You can modify lift, gain, gamma, and saturation in the low, mid, and high levels of the image, or across the entire image. Select the range you want to modify while performing primary colour grading. You can also define values to clamp minimum and maximum luminance values.

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

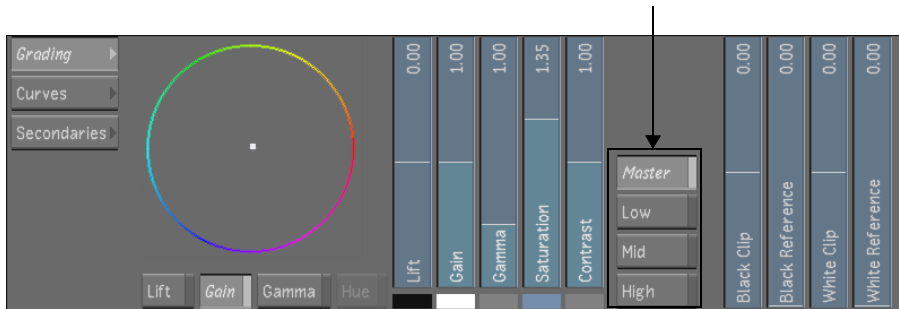
You can lift colour values in a selected range or in the entire image, and then offset ranges of colour. Changes in lift mostly affect black level. Although lift affects the entire image or image level, visually, it mostly affects the shadows.

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:

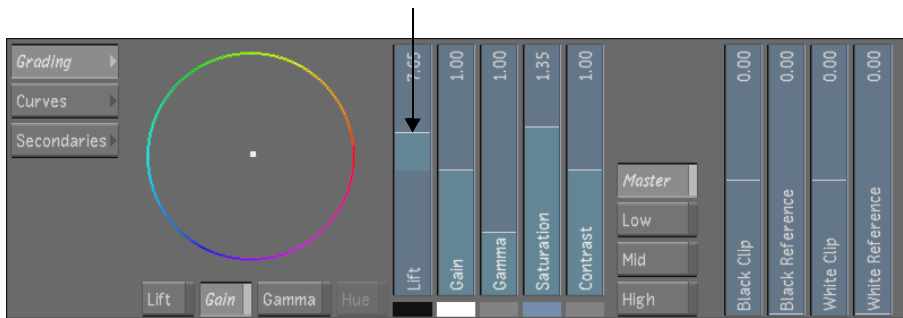
1. Indicate whether this is the initial or final primary grade:
 - To work on the initial primary colour grade, click Input.
 - To make final adjustments to the primary colour grade, click Output.

2. Select the image level you want to modify.

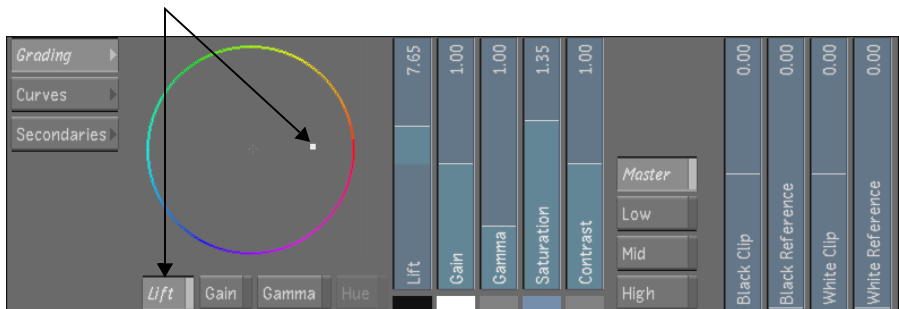


3. Modify the black levels:

- 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.

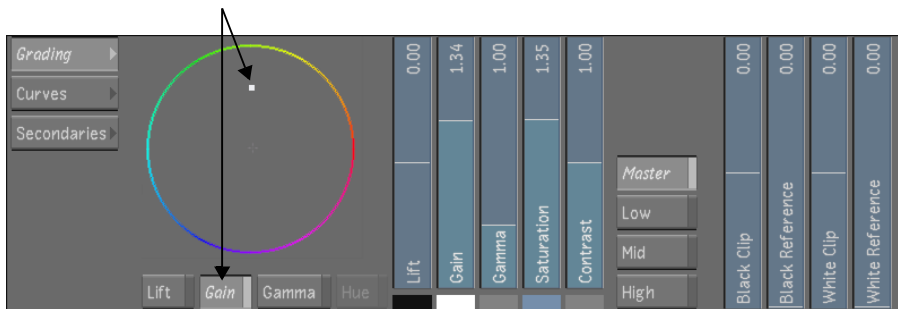
Linear Mode: Modifying Gain

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.

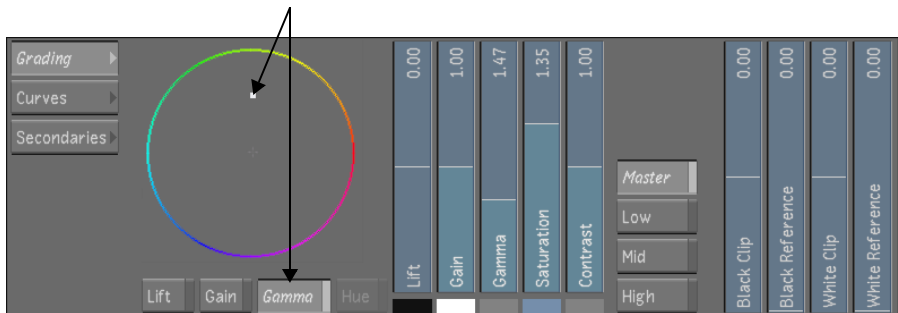
Linear Mode: Modifying Gamma

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.

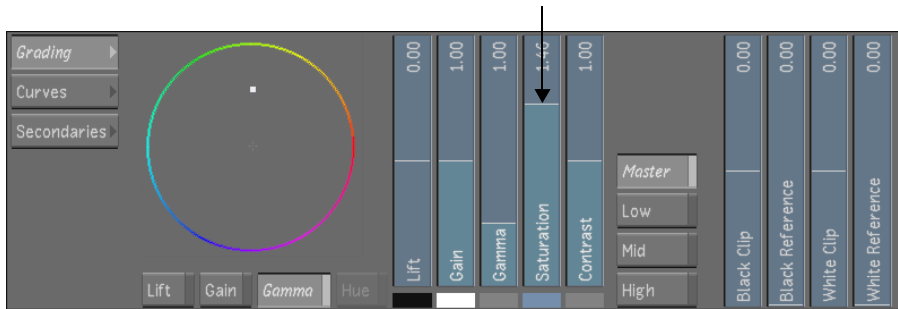
NOTE: Select Low, Mid, or High to further differentiate between the areas of the image that are affected.

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.



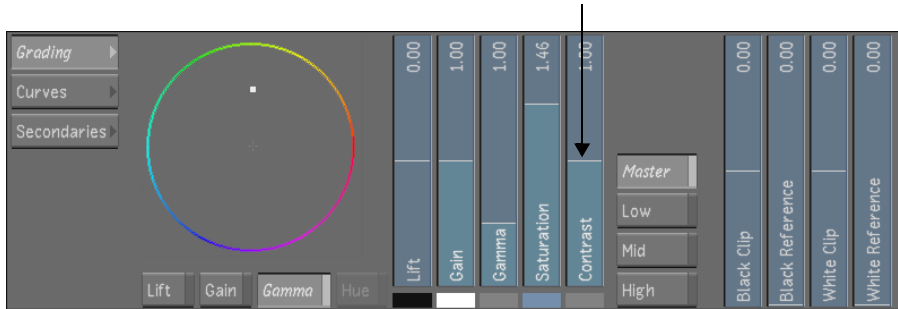
Linear Mode: Modifying Contrast

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:

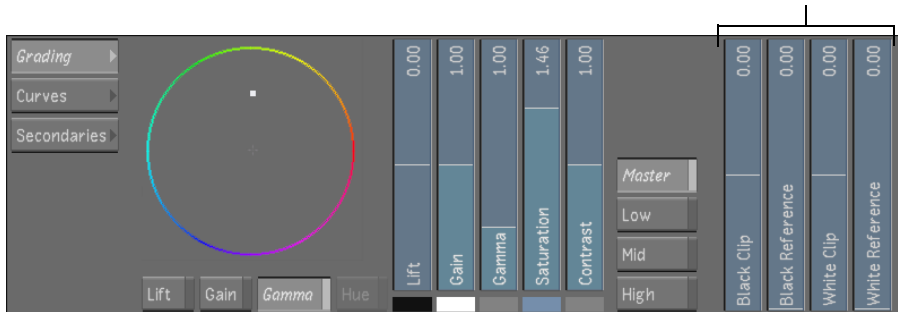
1. Indicate whether this is the initial or final primary grade:
 - To work on the initial primary colour grade, click Input.
 - To make final adjustments to the primary colour grade, click Output.
2. Modify the clip and reference values using the following sliders.

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.

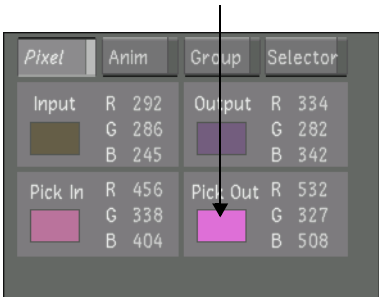
To match colours:

1. In the Grading menu, click Input.
2. Show the clips you want to match in the Player. Press **F5** to toggle split view on and then press **F3** to set the number of viewers. See [“Viewing Multiple Shots in the Player”](#) on page 238.
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

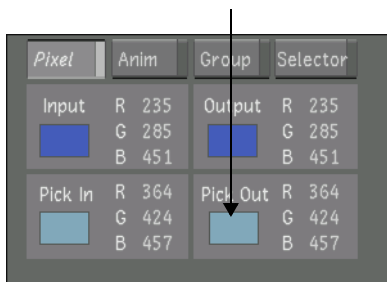
more blue than any other colour, yellow (the colour that is opposite on the colour wheel) is added to the image. To properly autograde a shot, sample an area of the image that should be neutral or have no hue cast. Sample blacks, whites, or greys.

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

Summary

About RGB and Hue Curves	333
Accessing the Curves Menu	334
Plotting Colour Values when Adjusting Curves	334
Modifying Red, Green, and Blue Channels	336
Modifying Hue, Saturation, Luminance, and Lightness	337

About 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 [Chapter 15, “Secondary Colour Grading,”](#) on page 345. The Luminance curve can only be modified on the Curves menu.

HINT: For faster playback on shots with input RGB and Hue curve modifications, enable GPU processing by clicking the GPU button or pressing **Y**. See [“GPU Acceleration”](#) on page 295.

Accessing the Curves Menu

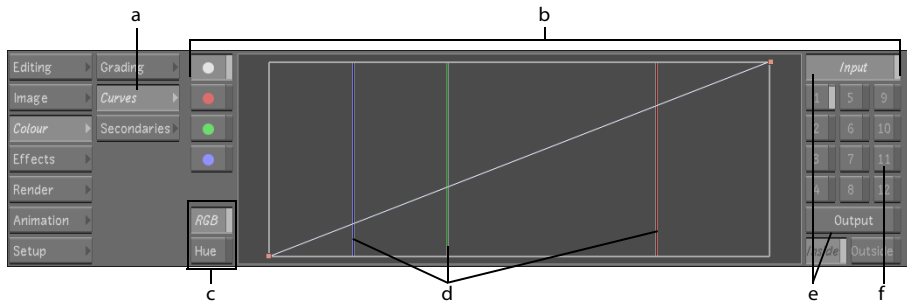
Use the Curves menu to modify RGB and Hue curves.

For control surface mappings, see the *Autodesk Control Surface User Guide*.

To access the Curves menu:

1. In the Main menu, click Colour and then Curves.

The Curves menu appears.



a) Curves button	c) RGB and Hue curve buttons	e) Input and Output buttons
b) Curves menu	d) Sampled colour values	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.

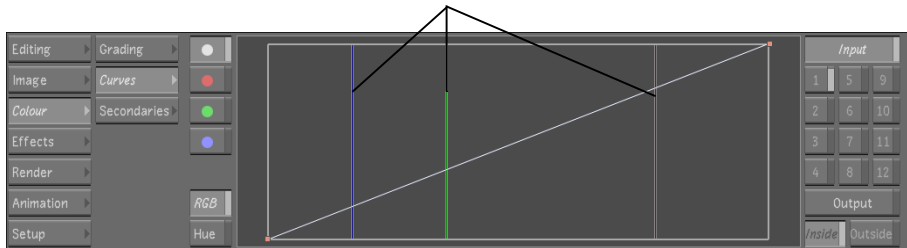
Plotting Colour Values when Adjusting Curves

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.

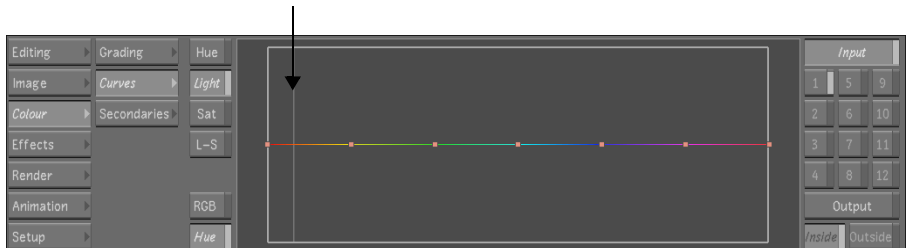
To plot colour values when adjusting curves:

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.



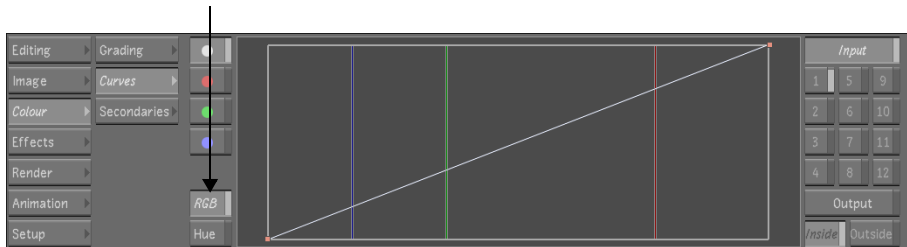
3. To zoom in on a curve using rectangle zoom, drag a selection box around the desired area while pressing **ALT**, and then release the mouse.
The area of the curves contained by the selection box now occupies the entire Curve canvas.
4. To zoom in on a curve using proportional zoom, press **ALT** while 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:

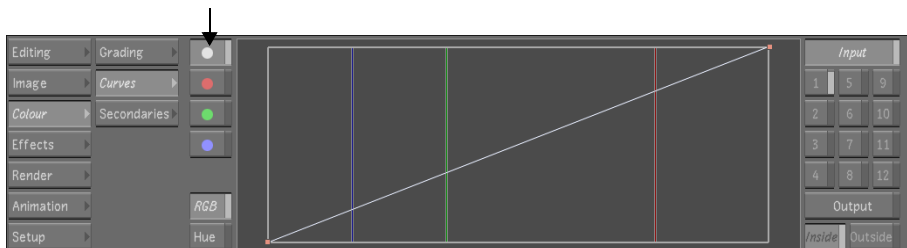
1. Indicate the colour grading stage at which the curve modification is being applied:
 - To work on the initial primary grade, click Input.
 - To make final adjustments to the primary grade, click Output.
 - To make curve modifications for secondary colour grading, enable a secondary layer.
2. Click 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.

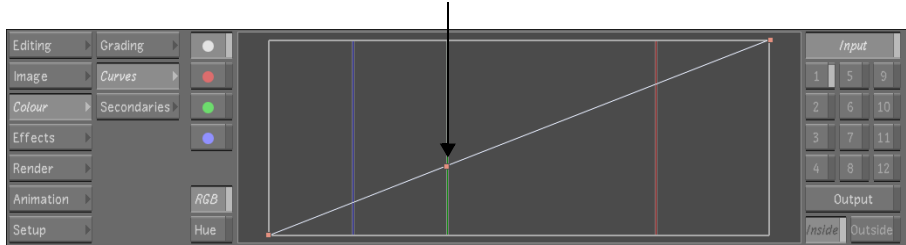
NOTE: To display the RGB curve, click the white button.



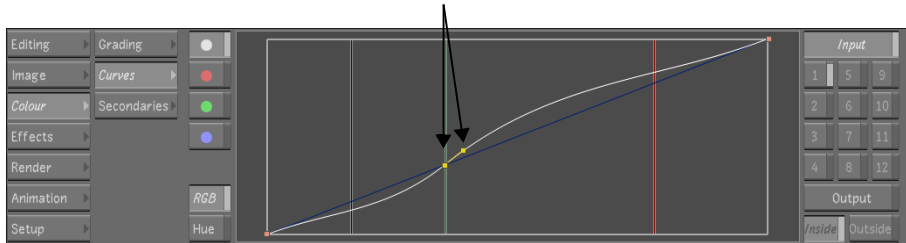
4. Sample the colour you want to use as a reference for your adjustments. See [“Plotting Colour Values when Adjusting Curves”](#) on page 334.

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:

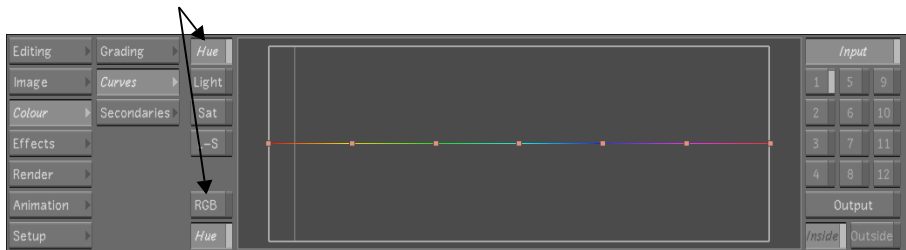
- Perform hue shifts.
- Suppress or saturate colour ranges.
- Suppress or saturate luminance ranges.
- Lighten or darken colour ranges.
- Add vertices to the curves to refine your adjustments.

Shifting Hue

Shift the hue of the entire image or a range of colour in the image.

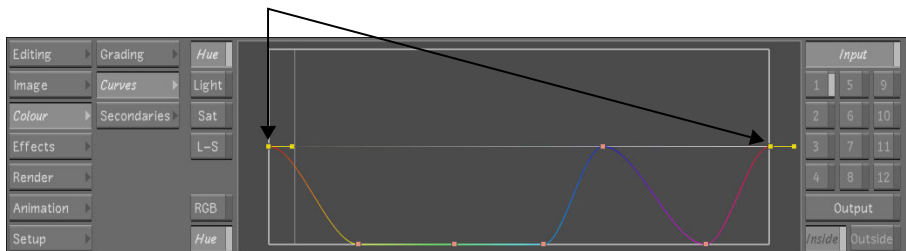
To perform a hue shift:

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”](#) on page 334.
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.

HINT: You can also select a vertex to use tangent handles to adjust the curve.

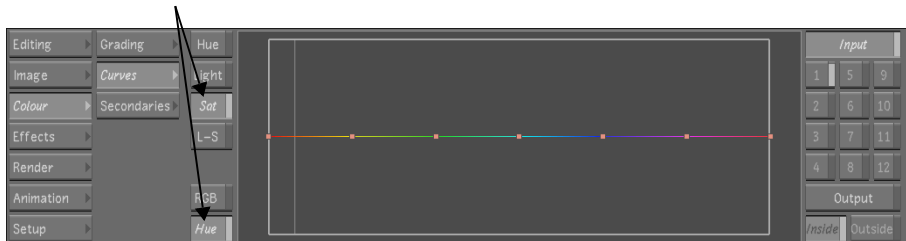
5. Modify the curve until you are satisfied with the results:
 - To select a vertex, click it. To select several vertices, draw a selection box around them.
 - To move several selected vertices, select them and then drag one. To restrict vertex movement to the Y axis, press **SHIFT** while moving the 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.

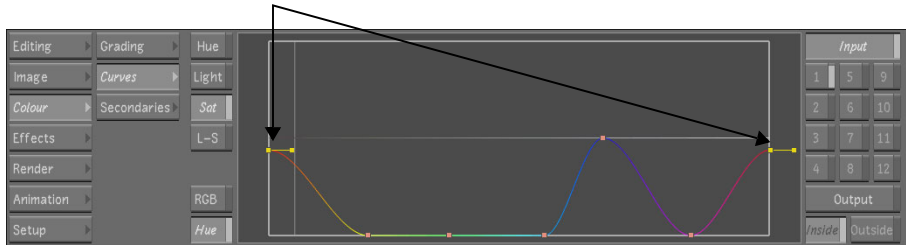


The Saturation curve is plotted to the hue spectrum.

3. Sample the colours you want to use as a reference for your adjustments. See [“Plotting Colour Values when Adjusting Curves”](#) on page 334.
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.

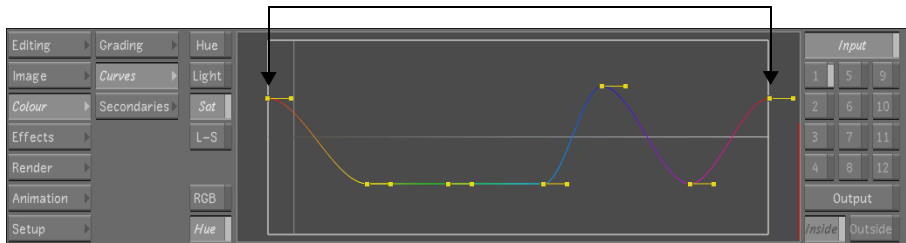
5. Saturate or suppress the colour:

- 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.



HINT: 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**.



HINT: 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.

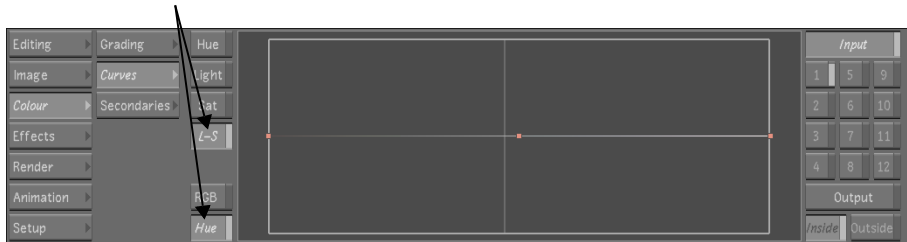
7. To add a vertex to the curve, place the mouse cursor over the area on the curve where you want to add the vertex and then press **A**. To lock the curve in place while adding a vertex, press **SHIFT+A** on the curve.
8. To delete vertices, select them and then press **D**.
9. Edit the curve until 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.

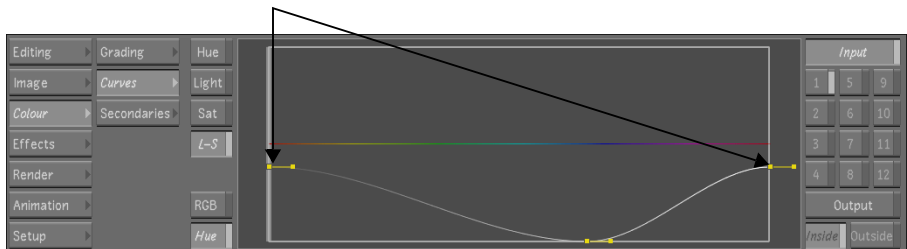
To suppress or saturate luminance areas in the image:

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.



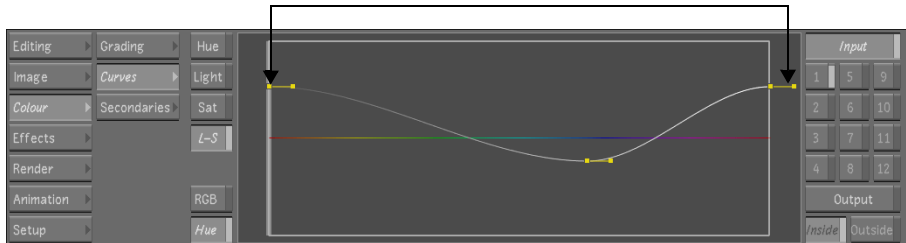
The Luminance curve is plotted to the hue spectrum.

3. Sample the luminance areas you want to use as a reference for your adjustments. See [“Plotting Colour Values when Adjusting Curves”](#) on page 334.
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.



HINT: You can also select a vertex to use tangent handles to adjust the curve.

- 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**.



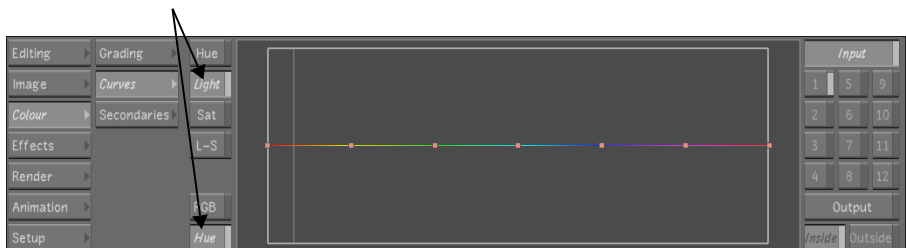
- To add a vertex to the curve, place the cursor over the area on the curve where you want to add the vertex and then press **A**. To lock the curve in place while adding a vertex, press **SHIFT+A** on the curve.
- To delete vertices, select them and then press **D**.
- Edit the curve until you are satisfied with the results.

Modifying Image Lightness

Increase or decrease the lightness in specific colour ranges or across the image.

To modify image lightness:

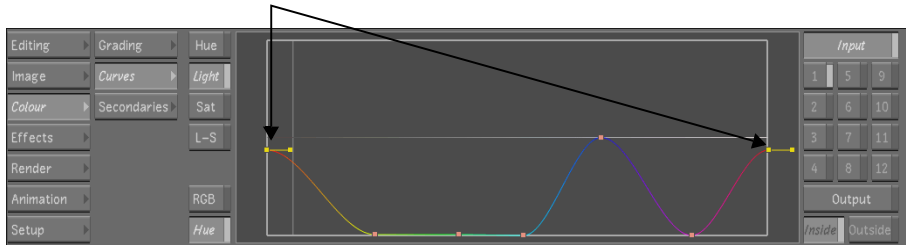
- Indicate the colour grading stage at which the curve modification is being applied:
 - To work on the initial primary grade, click Input.
 - To make curve modifications for secondary colour grading, enable a secondary layer.
- Click Hue and then click Light to display the Lightness curve.



The Lightness curve is plotted to the hue spectrum.

- 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.



HINT: 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.

15

Secondary Colour Grading

Summary

<i>About Secondary Colour Grading</i>	345
<i>Accessing Secondary Grading Menus</i>	346
<i>Creating a Secondary by Keying a Range of Colours</i>	347
<i>Saving and Loading Key Presets</i>	359
<i>Importing Mattes</i>	360
<i>About Geometries</i>	361
<i>Accessing the Geometry Controls</i>	362
<i>Drawing Basic Geometries</i>	363
<i>Drawing Free-Form Geometries</i>	364
<i>Modifying Geometries</i>	368
<i>Adding a Split Screen</i>	376
<i>Adding and Deleting Axes</i>	377
<i>Changing the Priority Order of Geometries</i>	380
<i>Showing and Hiding Geometries</i>	381
<i>Inverting Geometries</i>	381
<i>Deleting Geometries and Resetting Secondaries</i>	382
<i>Linking Geometries across Secondary Layers</i>	383
<i>Shape Tracker</i>	385
<i>Animating Geometries with the Point Tracker</i>	389
<i>Loading Tracker and Stabilizer Data from Autodesk Applications</i>	400
<i>Loading Garbage Mask Setups from Autodesk Applications</i>	401
<i>Colour Grading Secondaries</i>	402

About Secondary Colour Grading

In Lustre, you can 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, hue, and saturation.

NOTE: If you create effects for your image using Sparks, they are processed after secondary colour grading. See [Chapter 16, “Creating Lustre Sparks Effects,”](#) on page 407.

You can create and colour grade up to 12 unique secondary layers for each shot. Secondaries are processed in numerical order. However, you can toggle secondaries on and off and view the results in the Player. You can also save key presets, secondary presets, and geometry presets. You can also import garbage mask setups, tracker data, and stabilizer data from other Autodesk products.

HINT: GPU acceleration makes faster playback available for certain secondary grading features. See [“GPU Acceleration”](#) on page 295.



WARNING: The Histogram, Waveform, and Vectorscope functions do not update when GPU acceleration is enabled.

Accessing Secondary Grading Menus

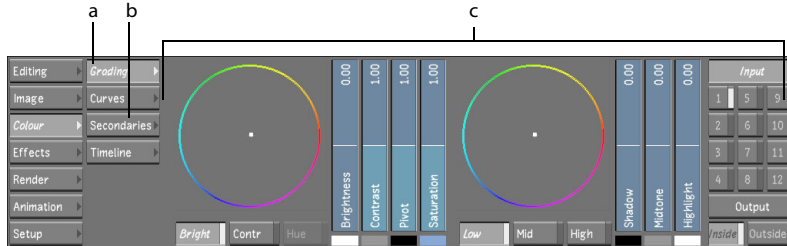
You perform secondary colour grading with the Secondaries and Grading menus.

For control surface mappings, see the *Autodesk Control Surface User Guide* .

To access menus for secondary colour grading:

1. In the Main menu, click Colour.

The Colour menu appears.



a) Grading button

b) Secondaries button

c) Grading menu

2. Click one of the following buttons.

Secondaries — Displays the Secondaries menu, where you can create keys to isolate ranges of colour, and create and track geometries to isolate objects or areas in the shot. Use logical operations to link geometries across secondaries.

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.

NOTE: You can also modify RGB and Hue curves for secondaries using the Curves menu. For information, see [Chapter 14, "Colour Grading: RGB and Hue Curves,"](#) on page 333.

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 generate a key, you can view it in the Player. Keys are based on the hue, value (luminance), and saturation of a selected colour from the original image. You define the softness range and the tolerance range by sampling the original image. When you set the tolerance range, you define which colours are fully keyed for the secondary. When you set the softness range, you define which colours are partially keyed. By softening the edge of the key, you can refine the results to integrate the secondary colour grade 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, use the SRC PRIM button.

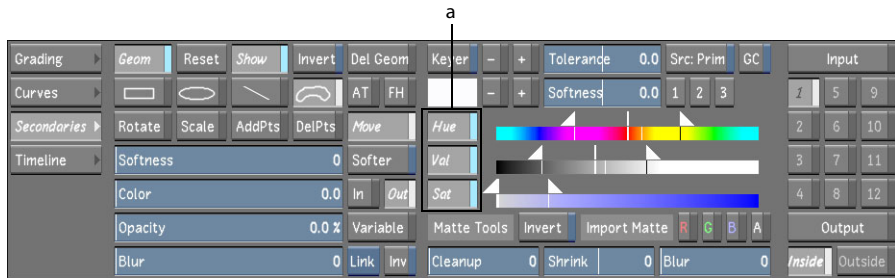
Selecting Channels for a Key: Hue, Luminance, and Saturation

You can base a key on hue, luminance, and saturation channels. You can use one or more of these channels. To determine which channel produces the best key, save separate keys based on each channel to the key presets. You can then toggle between them to compare the keys. See [“Saving and Loading Presets Using the Presets Lists”](#) on page 303 and [“Saving and Loading Key Presets”](#) on page 359.

To set keying parameters:

1. In the Colour menu, click Secondaries.

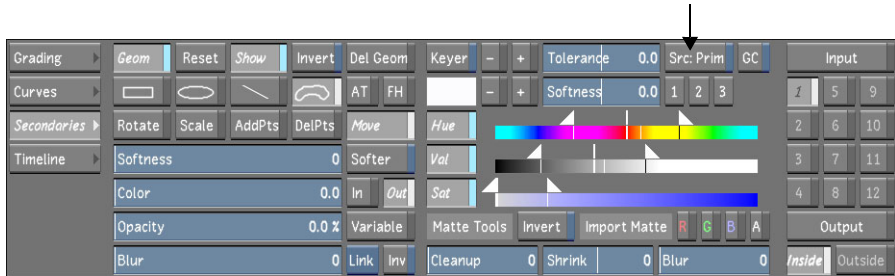
The Secondaries menu appears.



a) Hue, luminance, and saturation buttons

2. To select the basis of the key, do one of the following:
 - To generate a key based on the hue of the selected colour range, enable Hue. To exclude hue from a key, disable Hue.
 - To generate a key based on the luminance of the selected colour range, enable Val. To exclude luminance from a key, disable Val.
 - To generate a key based on the saturation of the selected colour range, enable Sat. To exclude saturation from a key, disable Sat.

- To key based on processing that is happening prior to the secondary subsystem, enable Src: Prim (e.g.: Input LUT, CDL, and Input primary results).



Setting Initial Softness and Tolerance

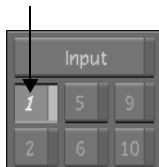
To generate a key, you set the initial softness and tolerance values by sampling a pixel or an average colour in the image.

To set the initial softness and tolerance:

- Set the view to Output. You can click the O button or press **F12**.



- In the Secondaries menu, click a secondary button to enable a layer.



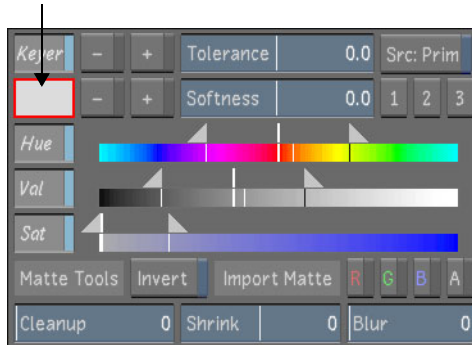
You can also use one of the following hot keys.

Press:	To enable:
1 to 9	Secondaries one to nine.
0	Secondary ten.
- (minus sign)	Secondary eleven.
= (equal sign)	Secondary twelve.

HINT: To toggle a secondary on or off, press \.

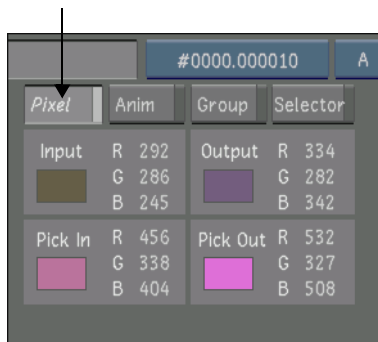
- Click the colour pot.

The Keyer button is enabled and the colour pot is outlined in red, indicating that it is in use.



HINT: You can finish a colour picking operation by clicking the colour pot a second time.

4. Click Pixel to display the Pixel Analyser.



5. Sample the image:

- To sample a single pixel, drag through 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, click the image.
- To sample an average taken from a range of colours in the image, **ALT**-drag the image in the Player.

The mouse cursor becomes 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 hue, luminance, and saturation channels. The colour pot displays the sampled colour.

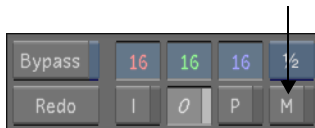


Viewing Secondaries

Once you set the initial tolerance and softness of a key, you can view the resulting secondary in the Player. You can view the key in Secondary view or Matte view. Secondary view displays the range of colour 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 view a secondary:

1. To display the image in Secondary view, click M or press **F11**.



In Secondary view, you see the colours that are included in, and excluded from, the current secondary. The colours in the image that are excluded from a secondary are desaturated—

they appear in greyscale. Colours included in the secondary are visible through the greyscale image.



Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

2. To toggle between Secondary view and Matte view, press **F11**.

In Matte view, the white parts of the matte are fully keyed and can be selectively colour graded. The black parts are excluded from the secondary. Greys indicate the softness parameters.

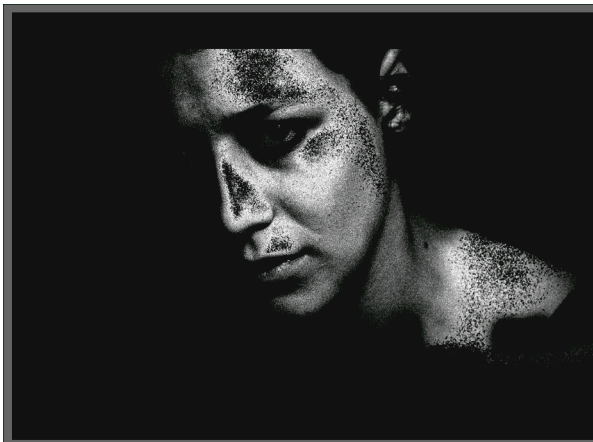
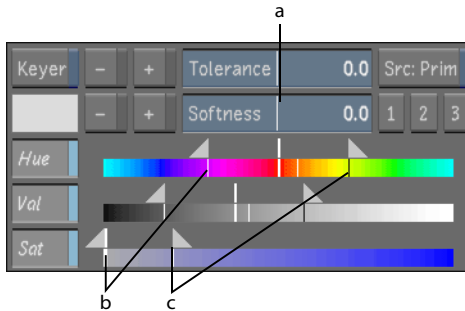


Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

Setting the Softness Range

The softness at the edges of a key increases as you increase softness. Use softness to create a more natural 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 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) Minimum Softness indicators c) Maximum Softness 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 clearly see the modifications you make to softness. You can toggle between Matte view and Secondary view while you develop the key.
2. In the Secondaries menu, enable the + or - button beside the Master Softness slider to increase or decrease softness.
3. Drag in 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:

1. View the secondary in the Player. Press **F11** to toggle between Matte view and Secondary view.
In Matte view, you can clearly see the modifications you make to softness. You can toggle between Matte view and Secondary view while you develop the 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, Value (luminance), or Saturation channel:

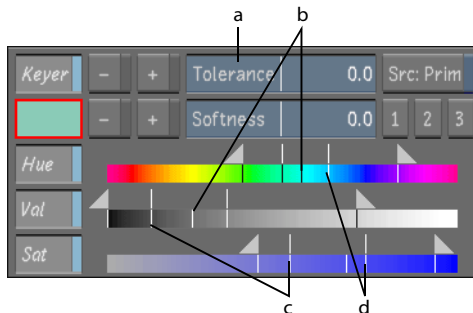
1. View the secondary in the Player. Press **F11** to toggle between Matte view and Secondary view.

In Matte view, you can clearly see the modifications you make to softness. You can toggle between Matte view and Secondary view while you develop the key.

2. For the Hue, Value, or Saturation channel, set the minimum value for the softness range using the Minimum Softness Range indicator. Drag the indicator to the left to increase softness. Drag the indicator to the right to decrease softness.
3. For the Hue, Value, or Saturation channel, set the maximum value for the softness range using the Maximum Softness Range indicator. Drag the indicator to the left to decrease softness. Drag the indicator to the right to increase softness.

Setting the Tolerance Range

Once you define the softness range, you can set the tolerance range to fully key colours and remove unwanted greys from the secondary. The maximum and minimum tolerance values define the range of colours included in the secondary. To set the tolerance, you can use the colour picker, the Master Tolerance slider, or the Tolerance Range indicators. With the Tolerance Range indicators, you set the tolerance range for hue, luminance, and saturation individually.



a) Master Tolerance slider	c) Minimum Tolerance indicators
b) Centre Tolerance indicators	d) 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 + or - button beside the Master Tolerance slider to increase or decrease tolerance.
3. Drag in an area of 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:

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, Value (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, Value, or Saturation channel, set the centre tolerance value using the Centre Tolerance indicator. Make sure you 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.

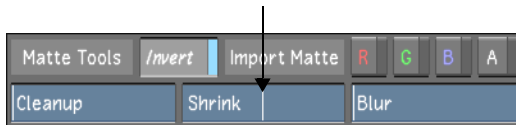
3. For the Hue, Value, or Saturation channel, set the minimum value for the tolerance range using the Minimum Tolerance Range indicator. Drag the indicator to the left to increase tolerance. Drag the indicator to the right to decrease tolerance.
4. For the Hue, Value, or Saturation channel, set the maximum 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.

Modifying the Edges of the Key

You can shrink, erode, or blur a key to modify the edges. 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.

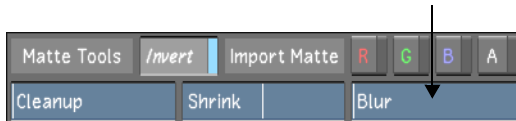
To shrink or erode the edge of the key:

1. View the secondary in the Player. Press **F11** to toggle between Matte view and Secondary view.
2. Modify 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.

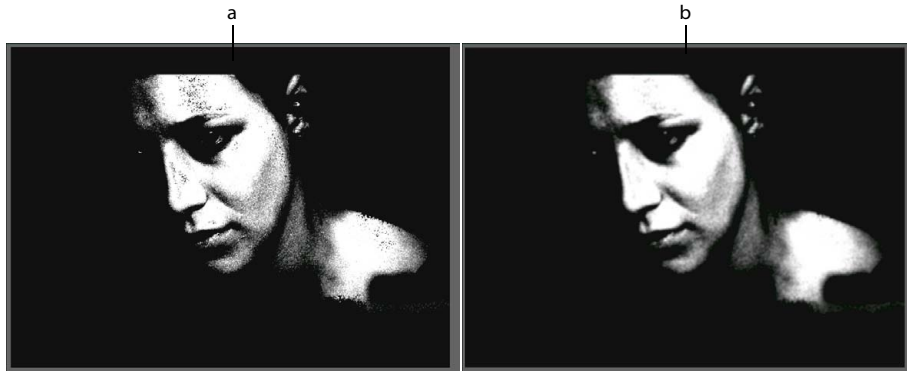


To blur the edge of the key:

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.



a) Before

b) After

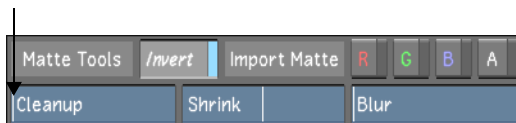
Images courtesy of Hungarian Academy of Film & Theatre, 3rd year

Removing Stray Pixels from a Key

When you generate 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.

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.

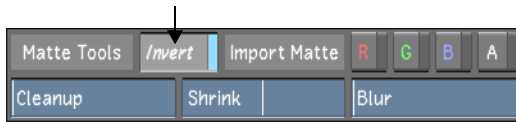


Inverting Keys

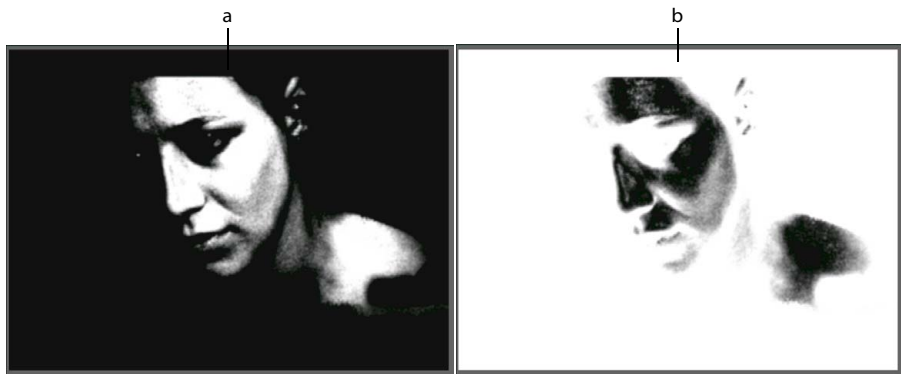
When it is easier to key colours in the image you do not want to include in a secondary than those that you do, you can invert the key.

To invert a key:

- In the Secondaries menu, enable Invert.



The area of the key that was opaque becomes transparent and the area that was transparent becomes opaque.



a) Before

b) After

Images courtesy of Hungarian Academy of Film & Theatre, 3rd year

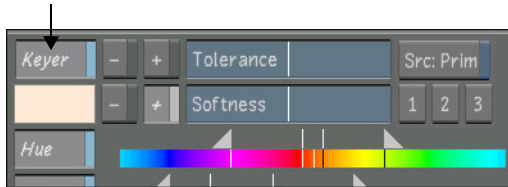
Excluding Keys from Secondaries

You can exclude keys from secondaries. When you exclude a key from a secondary that includes a geometry, only the geometry is colour graded.

To exclude a key from a secondary:

1. In the Secondaries menu, click the Secondary button that contains the key you want to exclude.

If a key is included with the secondary, the Keyer button is enabled.



HINT: You can click M in the View Mode buttons to view the range of colour defined by the key.

2. Disable Keyer.



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.

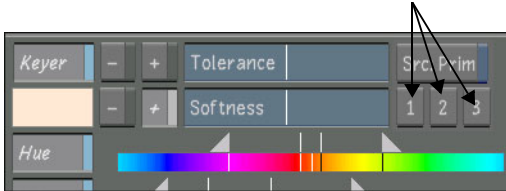
Saving and Loading Key Presets

You can save up to three user-defined keys, or key presets. You can then quickly load them to the current shot for comparison.

You can also save key presets to the Key Presets list. See [“Saving and Loading Presets Using the Presets Lists”](#) on page 303.

To save a key preset:

1. In the Secondaries menu, create a key using the Keyer.
2. Right-click the 1, 2, or 3 button.

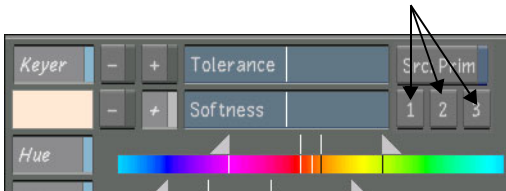


The key is saved to the button as a key preset. If a key was previously saved to the button, it is overwritten by the current key.

To load a key preset:

- Click the 1, 2, or 3 button to apply the saved key to the current shot.

NOTE: To display the key preset list, right-click the Keyer button.



The key preset is applied to all selected shots.

HINT: If the key does not appear to be applied, make sure the secondary is enabled.

Importing Mattes

Use the Import Matte button to import mattes stored on the four channels that are available in a TGA or TIFF file. You can then use the matte in place of generating a key for the current secondary.

HINT: Using the four channels in TGA and TIFF files, you can apply up to four predefined secondaries to the shot.

Matte sources must be saved in a directory named *matte*. The *matte* directory must be stored in the directory that contains the original footage you are working on. For example, if the shot footage is stored in *scans/tigris01/914x666/15700.dpx*, the matte must be stored in *scans/tigris01/914x666/matte/15700.tif*.

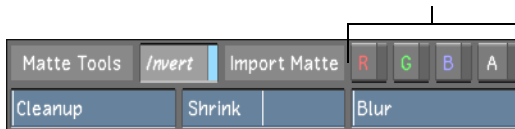
NOTE: The matte file cannot be compressed and must be present at the first image of the file sequence.

To import a matte:

1. In the Storyboard, select a shot with a TIFF or TGA matte.
2. In the Secondaries menu, right-click a Secondary button to enable it.



3. Enable the R, G, B, or A (alpha) channel.



The matte is applied to the shot and the opaque areas of the matte can be combined with a geometry for secondary colour grading.

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 Geometries with the Point Tracker”](#) on page 389.

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, click a Secondary button to enable it.

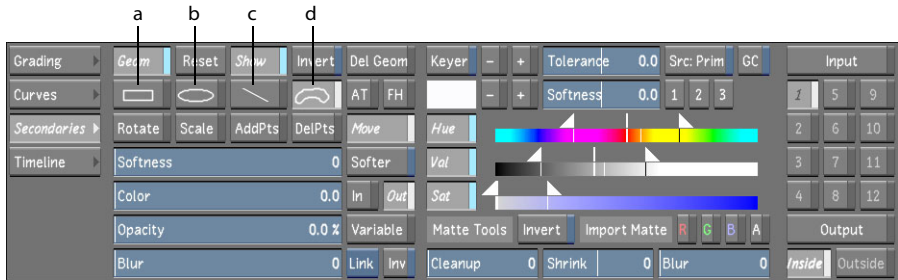


2. Enable Show.



If a geometry exists, it is displayed in the Player.

3. Enable a geometry.



- a) Rectangle geometry c) Split screen geometry
 b) Circle geometry d) Free-form (bezier) geometry

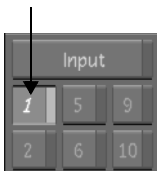
NOTE: The Keyer picker is automatically deactivated when a geometry is being created.

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, click a Secondary button to enable it.

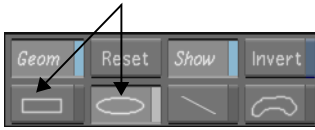


2. Enable Show.



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.

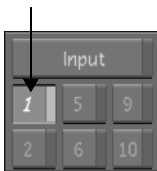
5. To resize the geometry, drag vertices or a vertex point. To scale it, drag the Z-axis.
6. To convert a basic shape to a free-form (Bezier) shape, right-click the shape in the Player.

Drawing Free-Form Geometries

For each secondary, 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, click a Secondary button to enable 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.

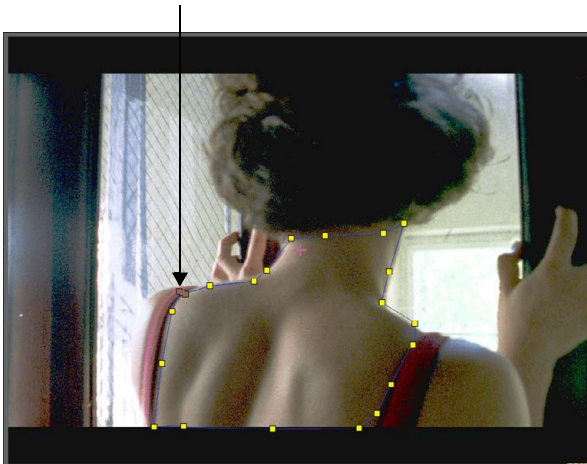
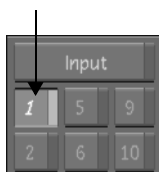


Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

To draw a free-form geometry using AutoTangent mode:

1. In the Secondaries menu, click a Secondary button to enable 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.

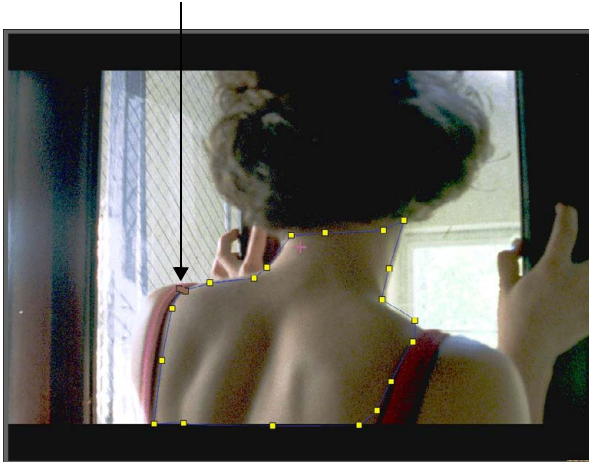


Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

To draw a free-form geometry using Freehand mode:

1. In the Secondaries menu, click a Secondary button to enable 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, 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.

HINT: 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:

- 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.

HINT: To deselect multiple vertices, click away from the selected ones.

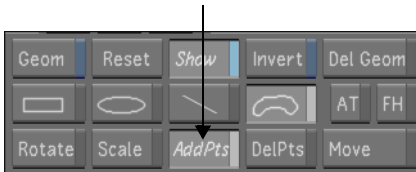
- To modify the shape or curve of a side, move the adjacent tangents. Drag a tangent handle to change its position.

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:

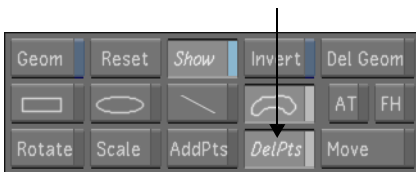
1. If you are working with a basic shape, right-click it in the Player to convert it to a free-form geometry.
2. In the Secondaries menu, enable Add Pts or press **A**.



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 or press **D**.



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:

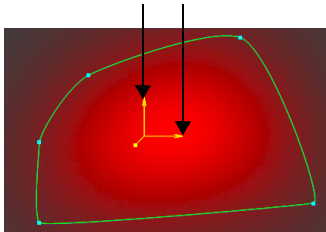
- Move geometries.
- Rotate geometries.
- Scale geometries.

To move a geometry:

1. In the Secondaries menu, enable Move or press **T**.



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.



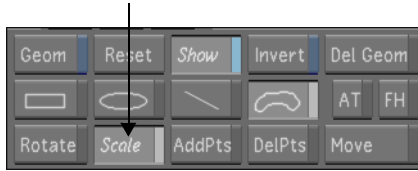
To rotate a geometry:

- 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 or press **Z**. 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 or press **X**.



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.

HINT: 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

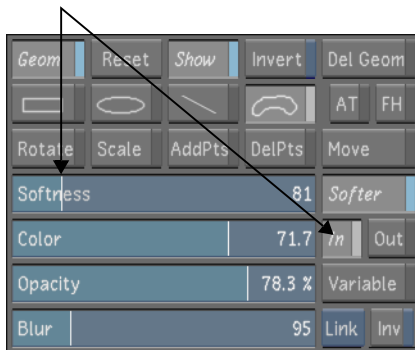
You can apply a constant range of softness to the edge of a geometry on a secondary. You can apply constant softness to the inside or outside of the mask border. You do not need to convert a basic shape to a free-form geometry in order to apply constant softness.

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.



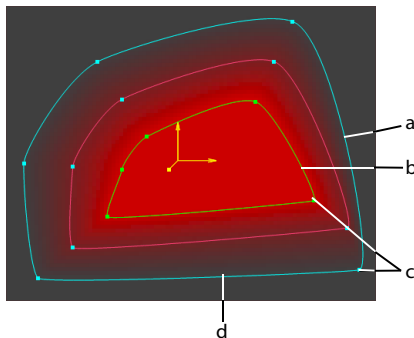
2. In the Secondaries menu, do one of the following:
 - To apply softness to the inside of the mask border, enable In, and then drag the Softness slider. Drag right to increase softness. Drag left to decrease softness.
 - To apply softness to the outside of the mask border, enable Out, and then drag 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	c) Softness vertices
b) Inner softness border	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.

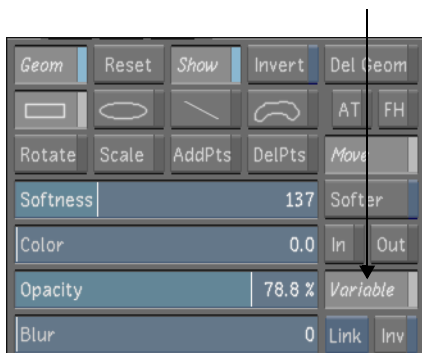
Variable softness can only be applied to free-hand geometries, or to basic geometries that have been converted to a free-form geometry.

To apply variable softness to the edges of the geometry:

- Do one of the following
 - In the Player, select a geometry.
 - From the Axes list, select a geometry or its axis.

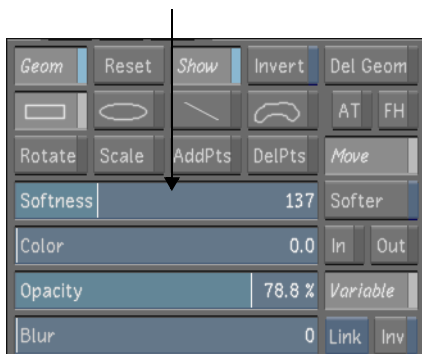


- In the Secondaries menu, enable Variable.



Softness is applied inside and outside the mask border.

- 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”](#) on page 369.

Blurring Geometries

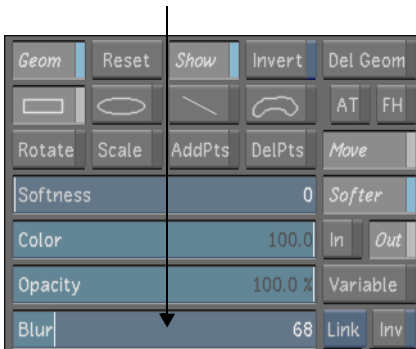
You can apply a Gaussian blur to soften all geometries on the current secondary layer.

To apply a Gaussian blur:

1. In the Secondaries menu, click a Secondary button to enable 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.

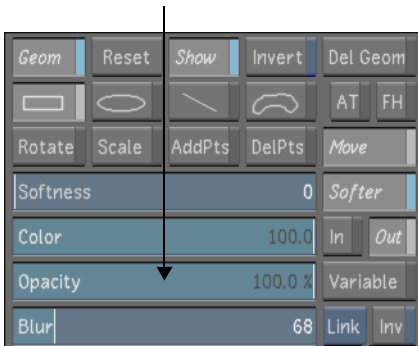
If you are applying only one geometry or several non-overlapping geometries to a secondary, modify either the colour or opacity to adjust the matte—in this case, colour and opacity will have the same effect on the image. If you are applying several overlapping geometries to a secondary, modify both colour 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, use the Colour controls:
 - Click B to set a colour value of 0% (black).
 - Click W to set a colour value of 100% (white).
 - Drag the Colour slider to set a colour value between 0% and 100%. A value of 50% means the area is 50% grey.

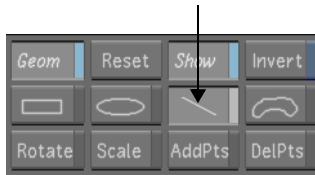
Adding a Split Screen

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, enable the Split Screen button.



2. 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.

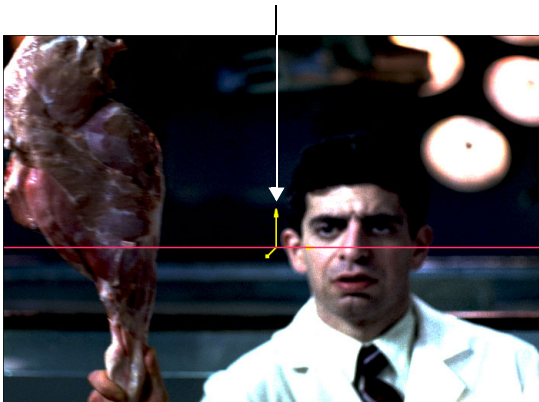
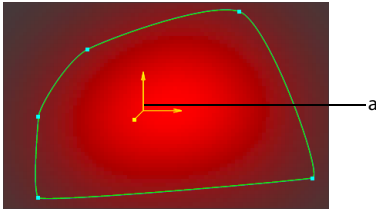


Image courtesy of Technicolor (formerly Toybox)

3. To grade the other half of the split screen, enable Outside (see [“Colour Grading Inside and Outside Secondaries”](#) on page 404).
4. To move the split screen up or down, click and drag the split screen's axis.
5. 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.



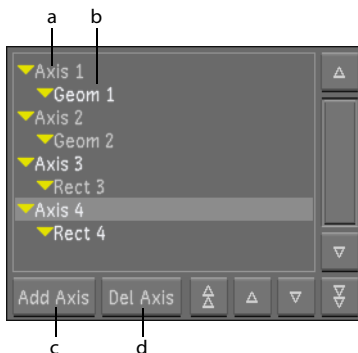
a) Geometry axis

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	c) Add Axis button
b) Geometry	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.

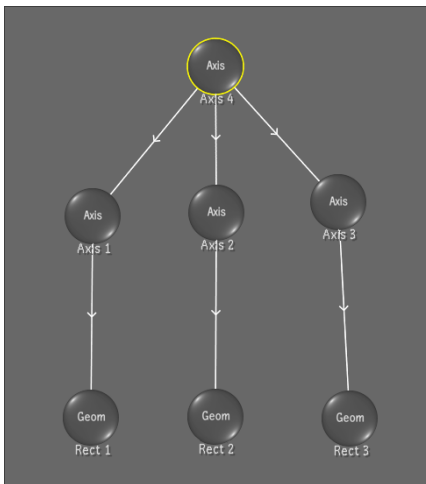
To delete an axis:

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.

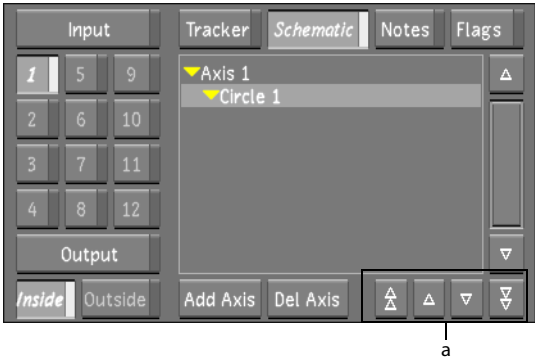
3. Switch back to default view by pressing ~ again.
The changed hierarchy is displayed in the schematic.

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.
	To the bottom layer.

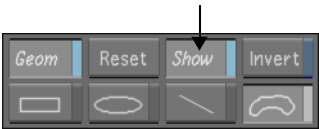
Click:	To move the geometry:
	Down one layer.
	Up one layer.

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.

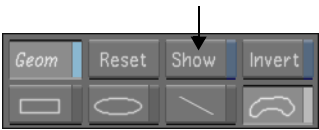
To show a geometry:

- In the Secondaries menu, enable Show.



To hide a geometry:

- Disable Show.

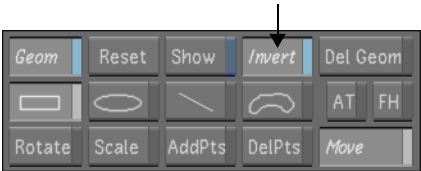


Inverting Geometries

You can invert geometries on the current secondary layer.

To invert geometries:

- 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, or press **SHIFT+D**.



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.

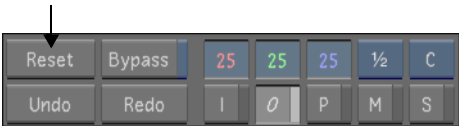
NOTE: You can also delete geometries and their axes from the Axes list. To delete a geometry or axis, select it in the Axis list (or **SHIFT**-click to select several), and then click Del. Axis.

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 box, located in the Secondaries menu. Use the Link box when you have already applied a key, a geometry, or both to the current secondary layer.

NOTE: You can use the Link box without applying any key or geometry to the current secondary layer.

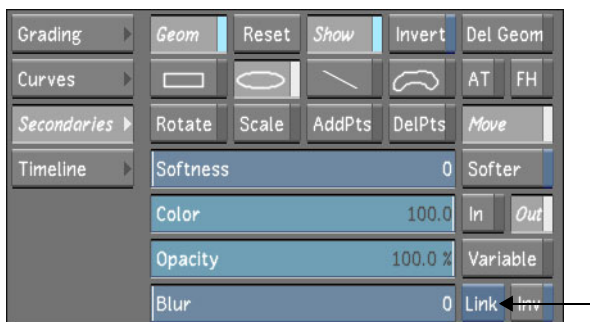
When you use the Link box, a logical operation is applied to determine how the geometries from the previous layer are combined with the key and geometries on the current secondary layer, and what areas of the image are colour graded. Depending on what you want to accomplish with the secondary, you can 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.

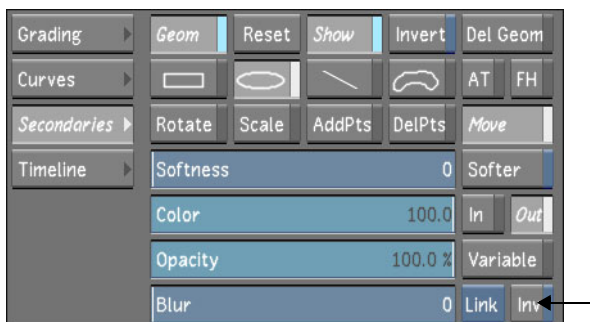
2. Select a logical operation from the Link box. The logical operation you select determines what is colour graded by the current secondary.



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 Invert.



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.

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.

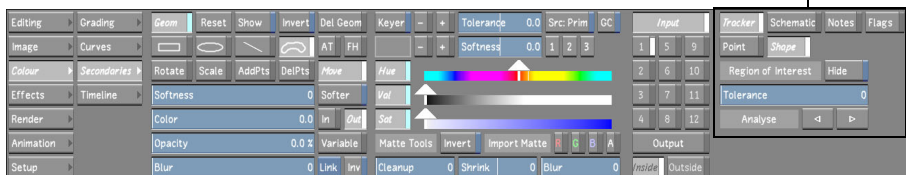
The Shape Tracker Menu

To track 2D shapes within a shot, you need to enable the Shape Tracker tool and create an ROI.

To access the Shape Tracker menu and create an ROI:

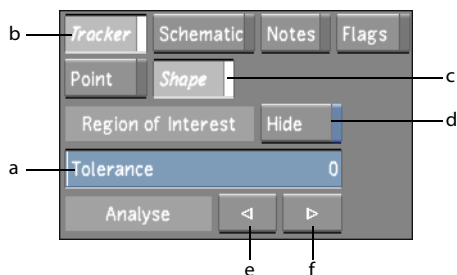
1. In the Main menu click Colour.
2. Click Secondaries.

The Secondaries menu with the Shape Tracker menu appears.



Parts of the Shape Tracker

The Shape Tracker is made up of the following components.



a) Tolerance slider	d) Hide button
b) Tracker button	e) Analyse backward button
c) Shape 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 hot key, 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.

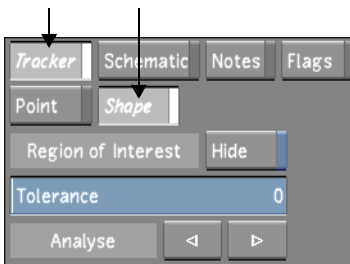
The following procedure is used when the object has a large enough ROI to track.

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, enable the Secondary button.
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”](#) on page 363 and [“Drawing Free-Form Geometries”](#) on page 364.

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.

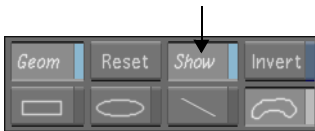
6. Make sure the geometry you want to track is selected.
The axis of the geometry should be yellow.
7. Set the tolerance slider between 0-10.
8. Click the analyse backward, or analyse forward button, to analyse the shot segment.
When the analysis is complete, you can grade the geometry.

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.

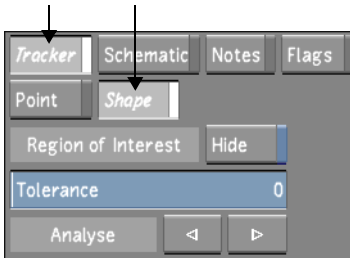
The following procedure is used if the object you want to track is small and does not contain enough trackable points.

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, enable the Secondary button.
3. Click Show.



4. Enable the Shape Tracker tool by clicking on Tracker and Shape.



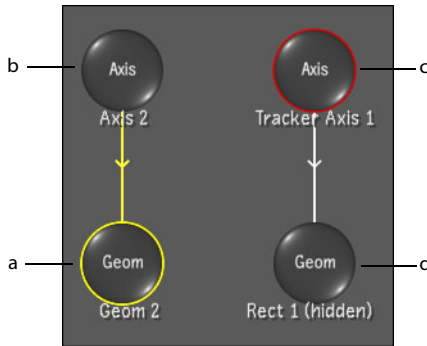
5. Create an ROI by drawing a basic geometry around a large area in the shot. See [“Drawing Basic Geometries”](#) on page 363.

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.

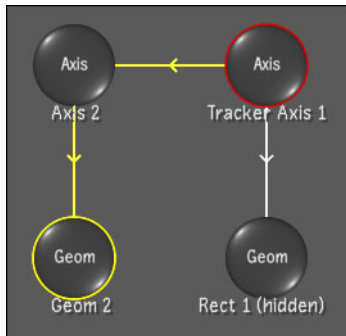
12. Press ~ (the tilde key) to enter the Schematic view.

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 Geometries with 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 [Chapter 17, “Animation,”](#) on page 423.

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

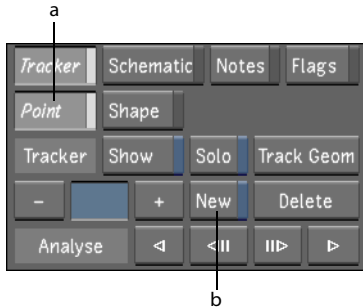
You can animate the position of geometries by defining a Point Tracker and assigning it to the axis of the geometry.

To animate the position of a geometry with the Point Tracker:

1. In the Secondaries menu, enable the Secondary button that contains the geometry you want to track.
2. Click Show.



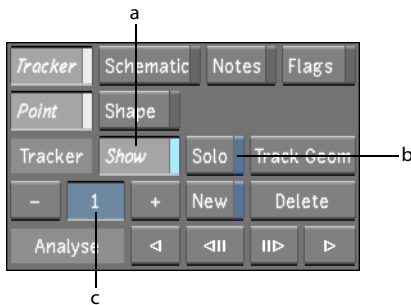
3. The Point Tracker option should already be enabled by default.



- 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.

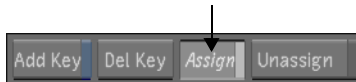


a) Analyse Backward buttons

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 (or press the right **ALT** or right **CTRL** key, respectively) to return to the last frame that was correctly tracked.

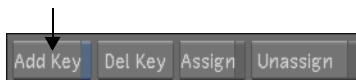
11. In the Player, do any of the following:

- 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.

HINT: You can reposition the tracker and reference point when the reference point moves off-screen.

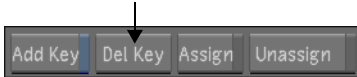
12. Continue using the Analyse buttons to analyse the shot.

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 (or press **BACKSPACE**).



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:

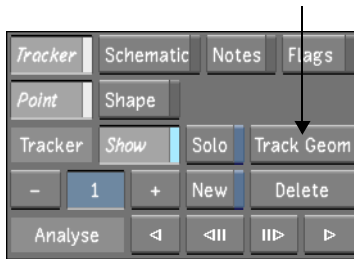
1. If working with a basic shape, right-click it in the Player to convert it to a free-form geometry.
2. In the Secondaries menu, enable the Secondary button that contains the geometry you want to track.
3. Enable Show.



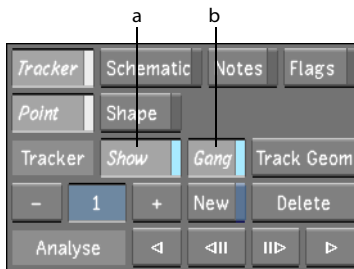
The geometries are displayed in the Player.

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.



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.

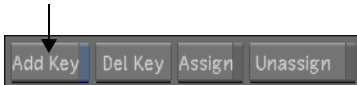


a) Analyse Backward buttons

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 (or press the right **ALT** or right **CTRL** key, respectively) 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.
- HINT:** 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 or press **BACKSPACE**.



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

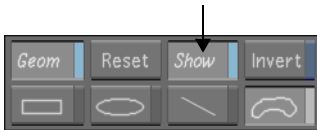
You can assign Point Trackers to, or unassign Point Trackers from, geometries at any time after you create them.

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”](#) on page 393.

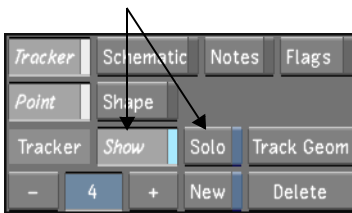
To assign a Point Tracker to a geometry:

1. In the Secondaries menu, enable the Secondary button that contains the geometry you want to track.
2. Enable Show.



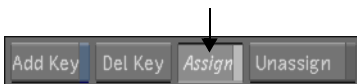
The geometry is displayed in the Player.

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.



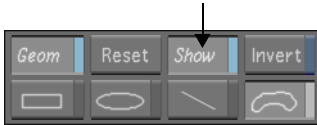
5. In the Player, do one of the following:
 - To animate the entire geometry without changing its shape, click the geometry's axis.
 - To animate a single vertex in a geometry, click the vertex.

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.

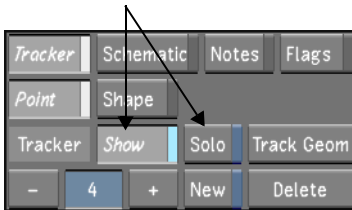
To unassign a Point Tracker from a geometry:

1. Select the shot that contains the tracking data.
2. In the Secondaries menu, enable Show.



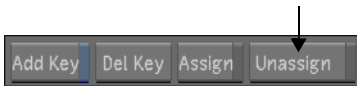
The geometry is displayed in the Player.

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.



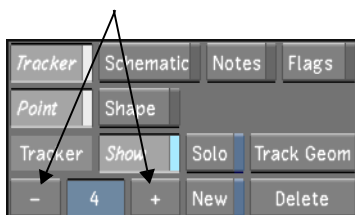
The Point Tracker is unassigned to the vertex but remains in the Tracker list.

Soloing and Ganging Point Trackers

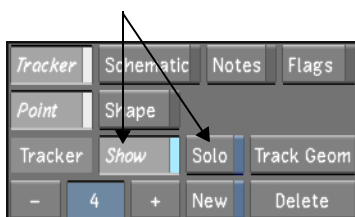
You can solo or gang Point Trackers. Solo a Point Tracker when you want to view and generate tracking data for the current tracker only. Gang Point Trackers when you want to view and generate tracking data for multiple Point Trackers at the same time.

To solo a Point Tracker:

1. Click the + or - button to select a Point Tracker from the Tracker list.



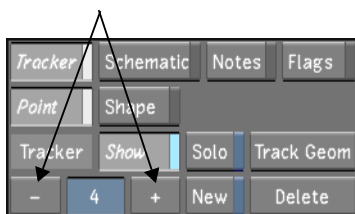
2. Make sure the Point Tracker is set to On and is in Solo mode.



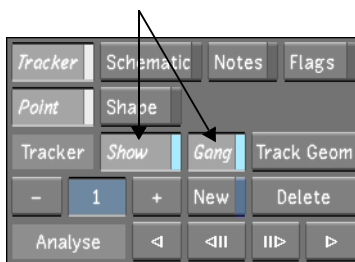
Only the current Point Tracker is displayed in the Player.

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.

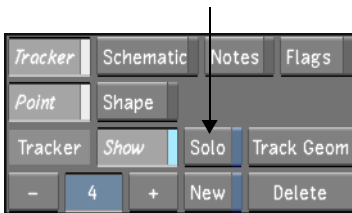
The ganged trackers that are set to On are displayed in the Player and can be analysed as a group.

Deleting Point Trackers

You can delete Point Trackers from a shot.

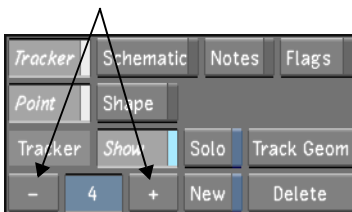
To delete a Point Tracker:

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 an ASCII file, and then save it in the `C:\Program Files\Autodesk\lustre 2009\settings\geompresets<version number>` folder (Windows), or in the `/usr/Autodesk/lustre2009/settings/geompresets` directory (Linux). 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.

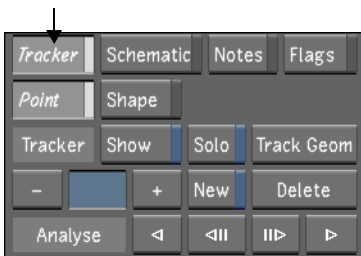
To load tracker or stabilizer data:

1. Save a tracker or stabilizer ASCII file from the applicable Autodesk product.
2. Do one of the following:
 - If using the Windows version of Lustre, store the ASCII file in the `C:\Program Files\Autodesk\lustre 2009\settings\geompresets` folder.
 - If using the Linux version of Lustre, store the ASCII file in the `/usr/Autodesk/lustre2009/settings/geompresets` directory.
3. In the Secondaries menu, enable the Secondary button that contains the geometry you want to track.
4. Enable Show.



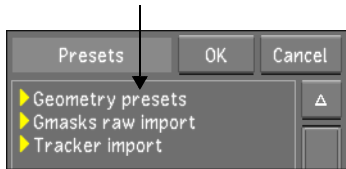
The geometries are displayed in the Player.

5. Click Tracker.



6. Click Point.

7. Right-click or middle-click the Secondaries menu 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.

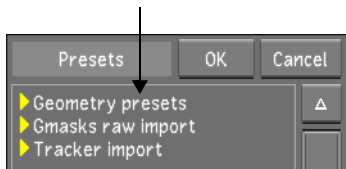
Loading Garbage Mask Setups from Autodesk Applications

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\lustre 2009\settings\geompresets* folder.
 - If using the Linux version of Lustre, store the raw setup file in the */usr/Autodesk/lustre2009/settings/geompresets* directory.
3. Right-click or middle-click the Secondaries menu to display the Presets list.



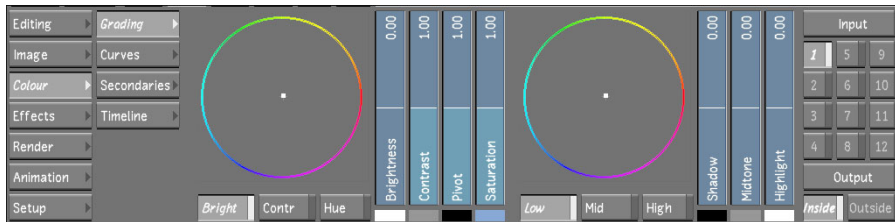
4. Expand GMask's Raw Import, and then select the raw setup file.

- Click OK to load the garbage mask setup.

The geometry appears in the Player.

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 enabled 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 in Log or Linear mode:

- In the Secondaries menu, click a Secondary button to enable it.



- 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.
- If you are colour grading geometries, enable Geom. This ensures that the colour correction will be applied selectively, rather than to the whole image.



- Click Grading to display the Grading menu.

5. Perform colour grading as you would for a primary grade. See [Chapter 13, “Primary Colour Grading,”](#) on page 313.

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. For control surface mappings, see the *Autodesk Control Surface User Guide*.

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, click a Secondary button to enable it.

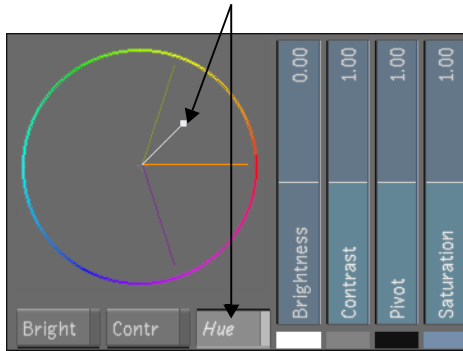


2. Define the area to be selectively colour graded by creating a key, adding one or more geometries, or combining a key with one or more geometries.
3. If you are colour grading geometries, enable Geom. This ensures that the colour correction will be applied selectively, rather than to the whole image.



4. Click Grading to display the Grading menu.

- 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

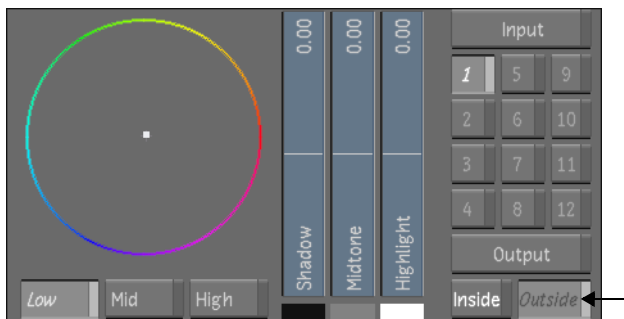
You can make printer light adjustments to secondaries just as you would to primary colour grades. See [“Adjusting Printer Lights for Primary Grading”](#) on page 317.

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:

- Colour grade the secondary.
- From either the Grading or Secondaries menu, click Outside to switch out of Inside colour grading mode.



3. Right-click Outside to enable Outside colour grading mode.

NOTE: Enabling Inside or Outside allows you to colour grade the inside or outside area of the secondary. If you subsequently disable 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:

- In the Secondaries menu, deactivate Geom and Keyer.



NOTE: To reapply the secondary colour grade, enable the Geom button again.

Turning Secondaries On or Off

Secondaries are processed in numerical order. You can toggle secondaries on and off and view the results in the Player.

To turn a secondary on or off:

- Do one of the following:
 - To turn a secondary on, click its layer number button. This allows you to manipulate shapes on the layer while keeping colour corrections hidden. To display colour corrections, right-click the layer number button to activate it.

The secondary is 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.

- To turn a secondary off, right-click the layer number button again. The number on the secondary button goes from white to gray to indicate that it is no longer enabled.

The secondary is bypassed in the processing pipeline.

16

Creating Lustre Sparks Effects

Summary

About Lustre Sparks Plugins	407
Installing Lustre Sparks Plugins	408
Licensing Lustre Sparks Plugins	408
Creating an Effect with a Lustre Sparks Plugin	408
Lustre Sparks Plugin Descriptions	411
Disabling and Re-enabling Lustre Sparks Plugins	420
Developing Lustre Sparks Plugins	421

About Lustre Sparks Plugins

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 12th colour grading secondary and the Output curves. 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. Plugin data is saved with colour grading data in the grade file when you save the grade. 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 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\lustre 2009\plugins* folder. If running the Linux version of Lustre, save your plugins in the */usr/Autodesk/lustre2009/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”](#) on page 421.

Licensing Lustre Sparks Plugins

To use your plugins, you need two separate feature licenses.

The features you need to license are:

- lustre_plug_pack1_b_3_0
- lustre_plug_pack2_b_3_0

When you receive your licenses, copy them into the *DL_license.dat* file. On Windows, this file resides in the *C:\Program Files\Autodesk\lustre 2009\license* folder. On Linux, the file is in the */usr/local/flexlm/licenses/* directory. After you have copied the plugin licenses to the license file, you can use the plugins.

For detailed information on licensing Lustre, refer to the *Autodesk Lustre Software Installation Guide* for your platform.

Creating an Effect with a Lustre Sparks Plugin

Load and use Lustre Sparks plugins from the Effects menu. You can load up to six plugins at a time.

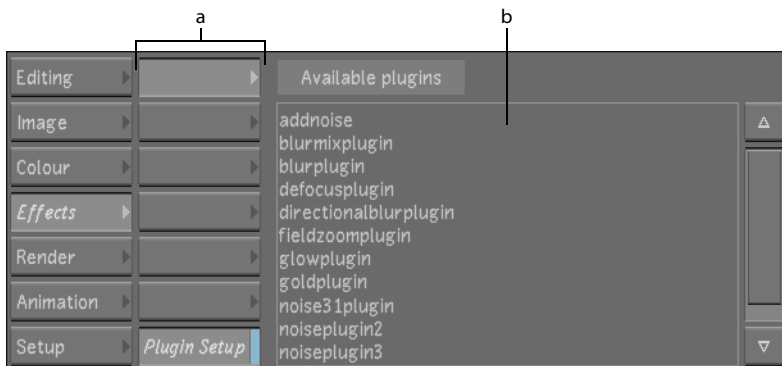
Each plugin has various parameters that you set to create the desired effect. Apply the effect to the whole shot, or an area of a shot defined by a secondary. You can animate the effect to make

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 appears.



a) Blank plugin buttons

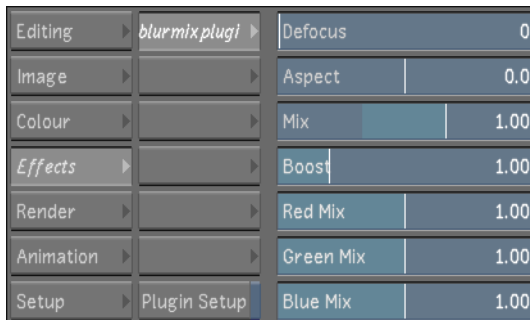
b) Available plugins

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.

HINT: 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.

- Set the parameters as needed.

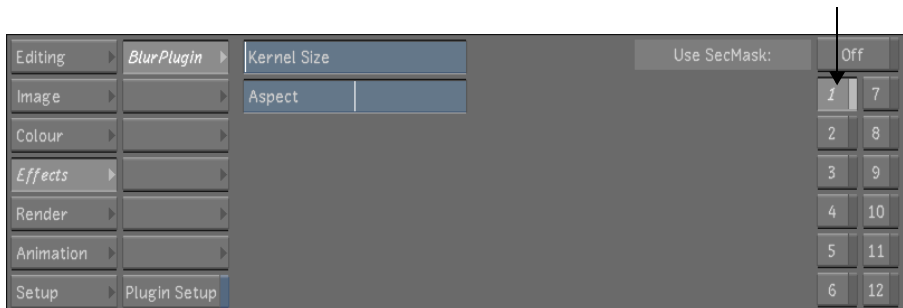


The parameters are applied to the entire shot and the image is dynamically updated to reflect the changes.

- 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:

- Create one or more secondaries for the shot to which you want to add an effect.
- In the Storyboard, select the shot.
- In the Main menu, click Effects to display the Effects menu.
- Click the layer number button that corresponds to the secondary you want to use.

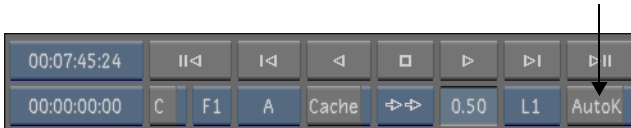


- Create the effect as described in the previous procedure.
The plugin is applied to the area defined in the secondary.

To animate an effect:

- Place the positioner on the frame where you want the effect to start.
- Load the plugin.

- Click the plugin button to select the plugin.
The plugin name on the button is highlighted and italicized and its parameters appear to the right of the buttons.
- Enable AutoK.

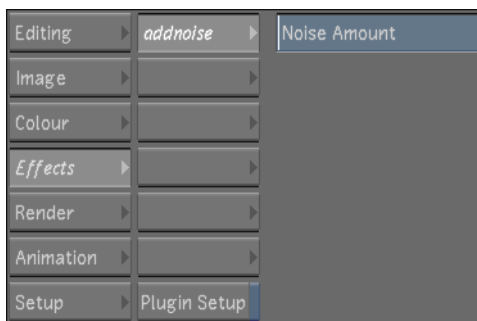


- Set one or more of the parameters.
As soon as you change a parameter, a blue line in the Shot timebar appears indicating a keyframe.
- Move the positioner to the next frame where you want to set another keyframe and adjust one or more of the parameters. Continue to add keyframes as needed.

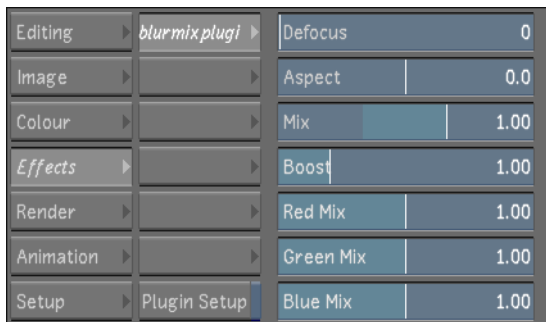
Lustre Sparks Plugin Descriptions

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.

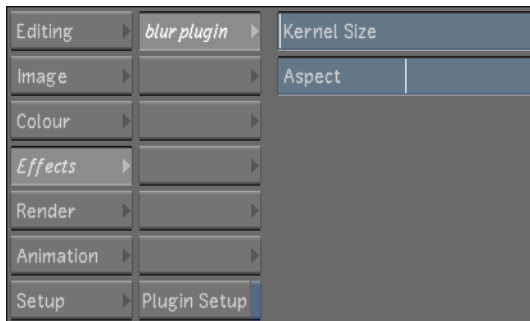


Blur Mix — Blur-type plugin that performs a multiply mix between the original image with the blurred image. You can also use this plugin to sharpen or tint an image.



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.
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.

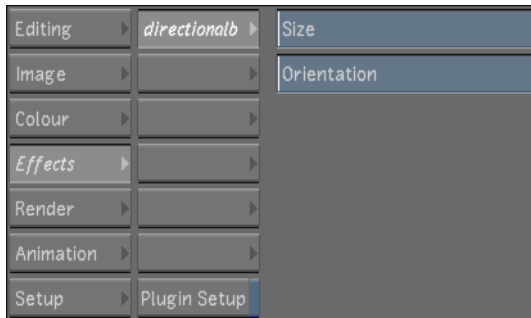
Defocus — Blur-type plugin that simulates a camera out-of-focus effect. Along with the blur parameters, this plugin includes variable iris types and additional highlight controls, as well as a control for softening the edge of a secondary.



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.

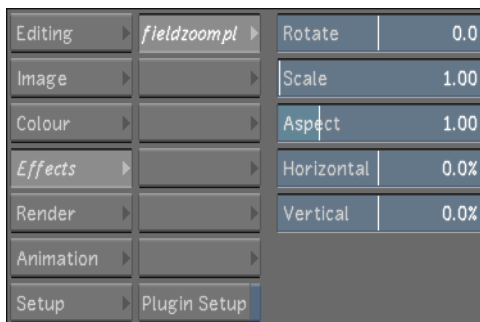
Use:	To:
Circle, Hexagon, Octagon	Specify the shape of the plugin matrix (or camera iris) used to calculate how the defocus is applied.
Rotation	Rotate the plugin matrix (or iris).

Directional Blur — Simple directional blur along a specified direction.



Use:	To:
Size	Adjust the amount of blur.
Orientation	Change the direction of the blur.

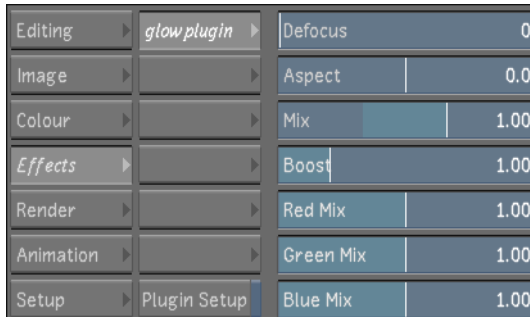
Field Zoom — Pan, scan, scale, and rotate interlaced footage. The parameters in this plugin work in the same way as the tools in the Image | Reposition menu, except that they are applied to individual fields instead of the entire frame. This yields a smooth image from interlaced footage even after repositioning.



Use:	To:
Rotate	Rotate the fields.
Scale	Resize the fields.
Aspect	Adjust the aspect ratio of the fields.

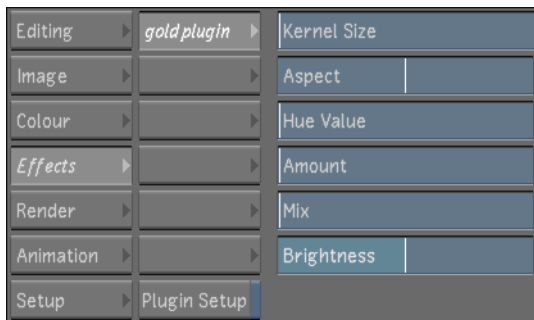
Use:	To:
Horizontal	Adjust the horizontal position of the fields.
Vertical	Adjust the vertical position of the fields.

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 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 three noise plugins available in Lustre.

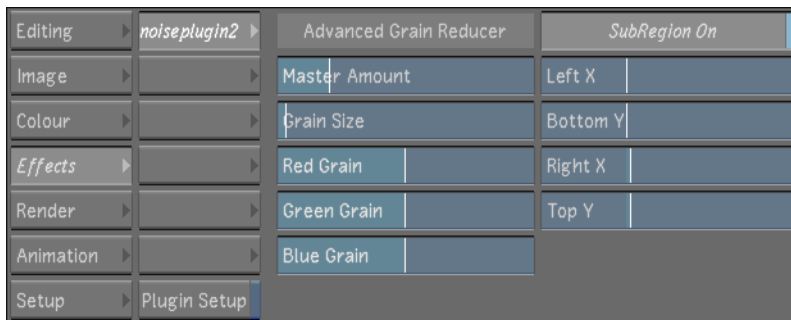
If the plugin processing slows down image playback, you can restrict grain reduction to a subregion of the frame.

Editing ▶	noise31 plugin ▶	Advanced Grain Reducer 3	SubRegion Off
Image ▶		Master Amount 0.00%	Left X 0
Colour ▶		Grain Size 0	Bottom Y 0
Effects ▶		Colour Grain 0%	Right X 0
Render ▶		Shadow Grain 0%	Top Y 0
Animation ▶		Midtone Grain 0%	
Setup ▶	Plugin Setup	Highlight Grain 0%	

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 a corner of the subregion bounding box and dragging the corner.
Left X, Bottom Y, Right X, Top Y	You can adjust the size of the subregion bounding box by adjusting these parameters.

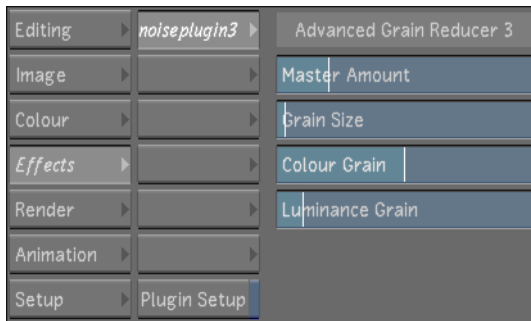
Noise Plugin 2 — This grain-type plugin is a second generation advanced noise reducer that uses a spatial filter with detail preservation control. It is useful for grain, noise, or detail reduction. During playback, this plugin updates dynamically as you adjust the values, so you can see how the grain 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.



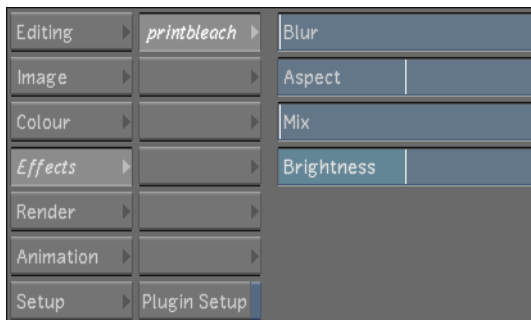
Use:	To:
Master Amount	Set how much the grain is reduced. The default value (or less) generally produces a good result for grain removal.
Grain Size	Set the size of grain that is reduced.
Red Grain	Independently adjust grain removal on the red colour channel.
Green Grain	Independently adjust grain removal 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 a corner of the subregion bounding box and dragging the corner.
Left X, Bottom X, Right X, Top X	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.

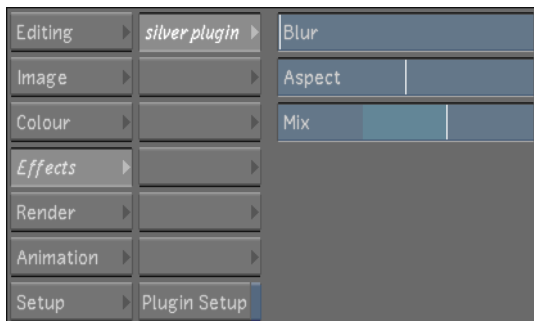
Print Bleach — Blur-type plugin that creates an effect similar to the chemical print bleach process. This plugin increases contrast for the shadows while desaturating the image. When you use this plugin, you first set a mix value so that you mix the image onto itself, and then you blur and brighten the 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.

Use:	To:
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.
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.

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 button in which the plugin is loaded.
The text on the disabled plugin button is dimmed.
2. Right-click the button again to re-enable it.

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 Lustre Sparks, you must apply to the system Sparks program. To request an application, send an e-mail to *sparks-manager@autodesk.com* and indicate your interest in developing specifically for Lustre.

17

Animation

Summary

About Animation	423
Accessing the Channel Editor	424
Animating with Keyframes	428
Editing Keyframes	438
Copying and Pasting Keyframes	449
The Track Editor	456
Animating in Other Menus	460
Specifying Setup Options	462
Undoing and Redoing Operations	464

About 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

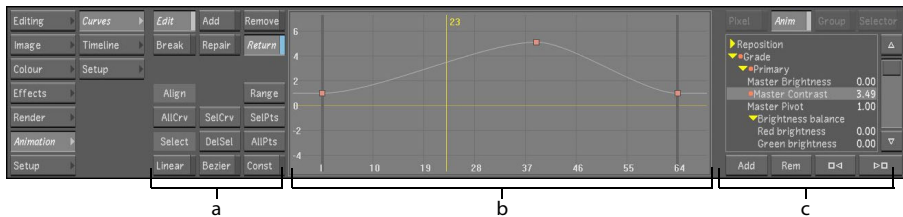
Use the animation curves in the Channel Editor to create animations.

For control surface mappings, see the *Autodesk Control Surface User Guide*.

To access the Channel Editor:

- In the Main menu, click Animation, and then click Curves.

The Channel Editor appears.



a) Animation controls

b) Animation Curves window

c) Channel hierarchy

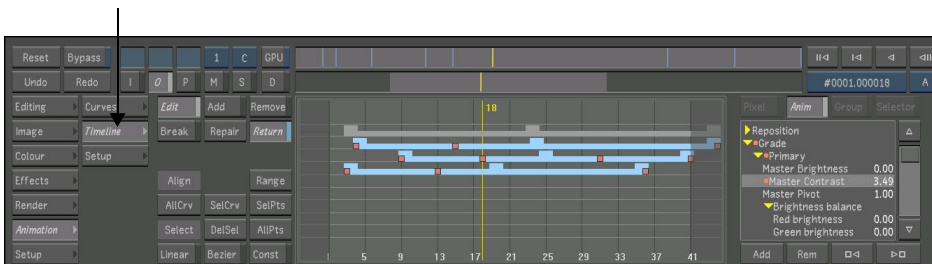
The Channel Editor is made up of the following elements.

Animation controls — Use to switch between animation modes, select and view channels, and control interpolation.

Animation Curves window — Use to create and view animation curves, adjust interpolation, and add, edit, or delete keyframes.

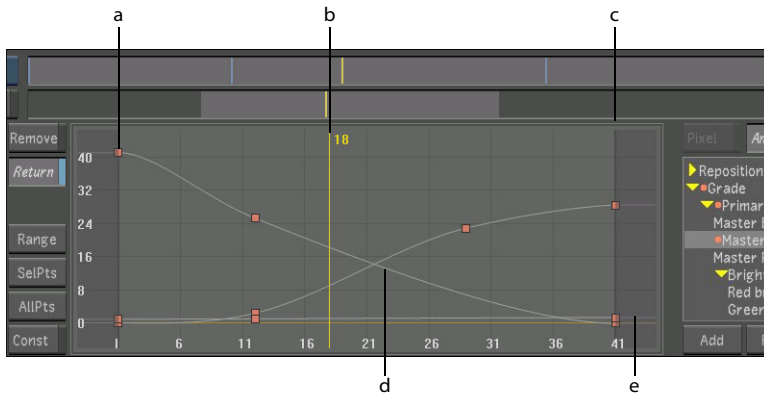
Channel hierarchy — Use to select the directories and channels to animate. You can also refer to the Channel Hierarchy to quickly see what channels contain keyframes.

Track Editor — Use to adjust animation timing. This window appears in place of the Animation Curves window when you click Timeline.



Animation Curves Window

You perform most animation operations in the Animation Curves window.



- a) Start of current shot c) End of current shot e) Trimmed frames
b) Current Frame positioner d) Animation curve

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.

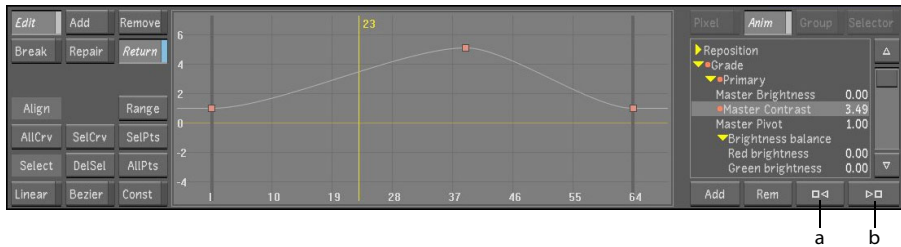
To move to any frame in the Animation Curves window:

- Drag in the Shot timebar to scrub between frames in the current shot.
The Current Frame positioner moves in sync with the timebar positioners.

HINT: You can also use the playback controls to navigate through the Animation Curves window.

To move between keyframes:

- Do one of the following:
 - Click Next to move to the next keyframe.
 - Click Prev to move to the previous keyframe.



a) Previous button

b) Next button

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:

- Middle-click and drag to pan the Animation Curves window view horizontally or vertically.

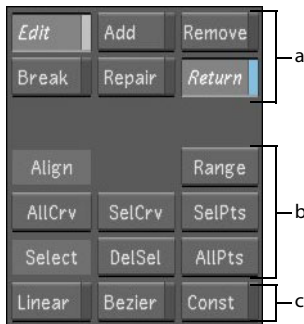
To zoom the Animation Curves window view:

- 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.

HINT: Hold down **ALT** while dragging to centre the zoom at the point where you began dragging.

Animation Controls

Use the Animation controls to switch between animation modes, select and view channels, and set keyframe interpolation type.



a) Animation modes

b) Align controls

c) Interpolation buttons

The Animation controls are made up of the following elements.

Range button — Displays the shot range horizontally (without the trimmed sections) and fits the selected curves into the Animation Curves window vertically.

AllCrv button — Fits all the existing animation curves into the Animation Curves window.

SelCrv button — Fits only the selected curves into the Animation Curves window.

SelPts button — Fits only the selected curve's selected keyframes into the Animation Curves window.

Animation modes — Allow you to manipulate keyframes in the Animation Curves window.

Interpolation buttons — Change the shape of the animation curve between keyframes.

To view the entire channel range between keyframes:

1. Click Animation, and then click Curves.
2. In the Animation controls, click Range.
All keyframes are loaded into the Animation Curves window.

To view all animated curves in the Channel Editor:

- In the Animation controls, click AllCrv.
All animated channels appear in the Animation Curves window.

To view the selected curve in the Channel Editor:

1. 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”](#) on page 439.
2. Click SelPts.

The animation curves are scaled so that the selected keyframes fill the Animation Curves window.

Channel Hierarchy

Use the Channel Hierarchy to select the channels to animate. In the following example, Master Contrast is selected. You can then set keyframes for this channel and create an animation.

In the Channel Hierarchy, channels are grouped together in directories. 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	c) Selected channel
b) Expanded folder	d) Deselected channel

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”](#) on page 429.
2. Move the cursor to the frame you wish to mark as a keyframe and then add a keyframe. See [“Adding Keyframes”](#) on page 430.

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”](#) on page 433 and [“Setting Extrapolation”](#) on page 436.
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”](#) on page 456.

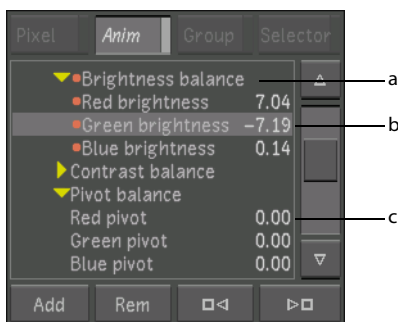
Selecting Channels

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, and then click Curves.
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 Setup menu is disabled.



a) Select folder

b) Selected channels

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.

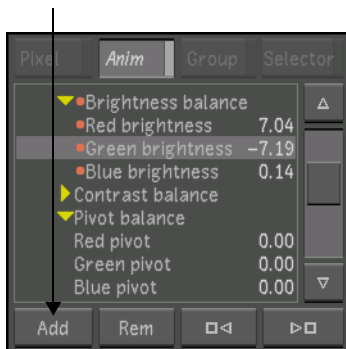
Adding Keyframes

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:

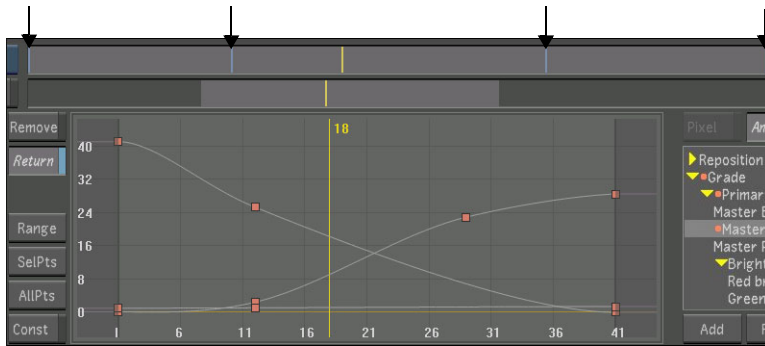
1. In the Channel Hierarchy, select the channel to which you want to add a keyframe.
2. Position the cursor at the frame you wish to mark as a keyframe.
3. Click Add.



When you add a keyframe to a channel, an orange indicator appears to the left of the channel in the Channel Hierarchy.

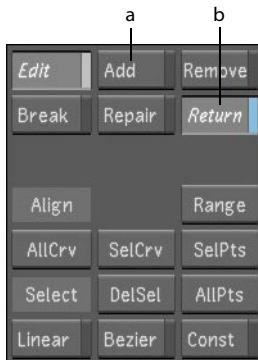
4. Set the channel value as needed by dragging the keyframe.

Once you add keyframes to a shot, 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) Add mode button

b) Return mode button

3. Enable Return.

NOTE: If Return mode is enabled, you are returned to Edit mode after adding one keyframe. For information on working in Edit mode, see ["Editing Keyframes"](#) on page 438.

4. In the Channel Hierarchy, select the channel to which you want to interactively add keyframes.
5. In the Animation controls, enable Add.

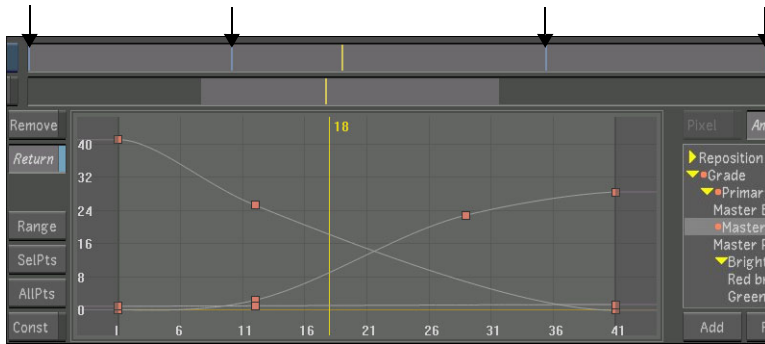
6. Move the mouse cursor over the animation curve.

Keyframes appear as you approach each frame.

7. Click the animation curve to add a keyframe at that frame.

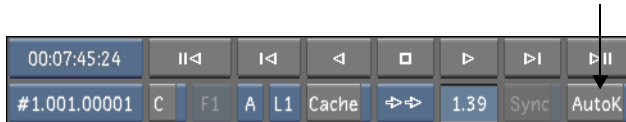
NOTE: The new keyframe is automatically set to Bezier interpolation.

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:

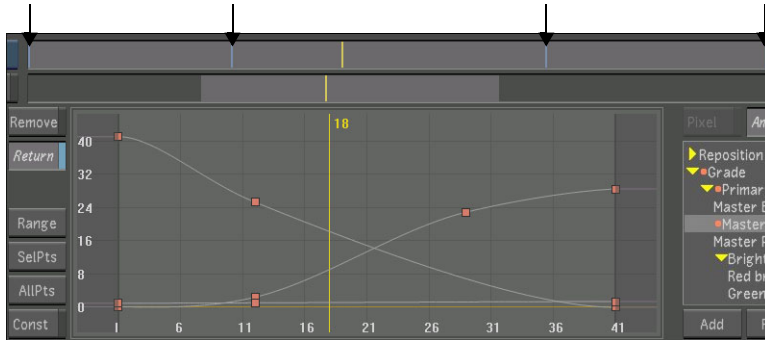
1. Enable AutoK.



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.

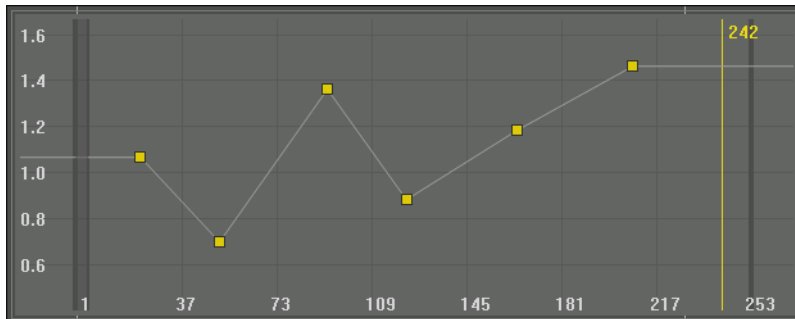
Once you have added keyframes to a shot, markers appear in the Shot timebar indicating the location of the keyframes.



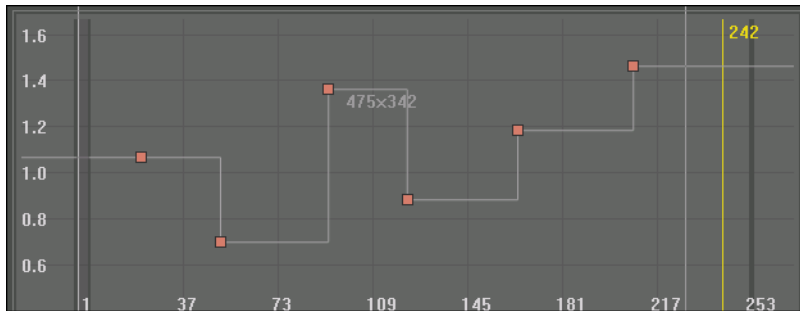
Setting Interpolation

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.

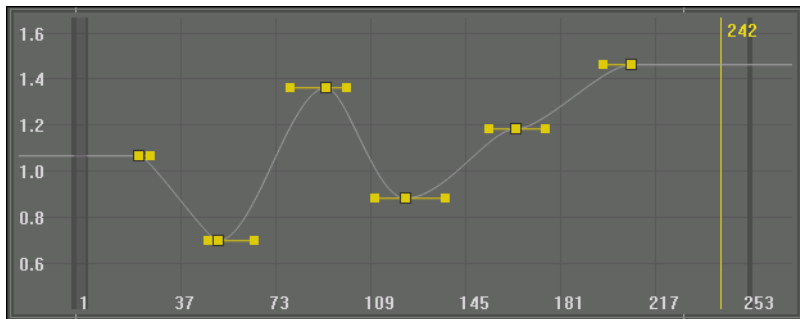
Linear — Joins keyframes with a straight line.



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.



Bezier — Produces a smooth curve with a smooth transition between keyframes. Each keyframe on the curve has tangent handles. You can change the shape of the animation curve by dragging the tangent handles.



NOTE: You can create animation curves with mixed interpolation types.

To set the interpolation type for a keyframe:

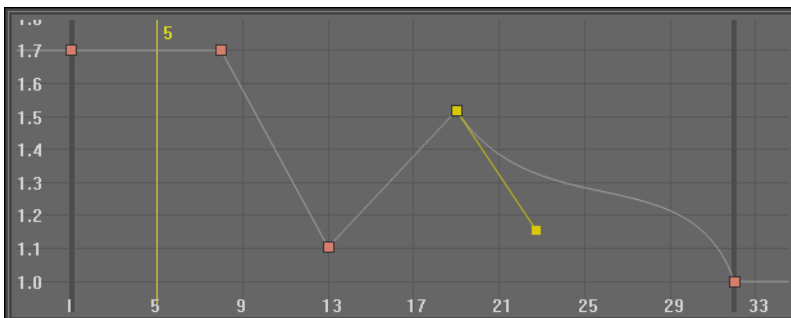
1. Select the keyframe for which you want to set the interpolation.
2. In the Animation controls, click the appropriate interpolation type button.



To create an animation curve with mixed interpolation:

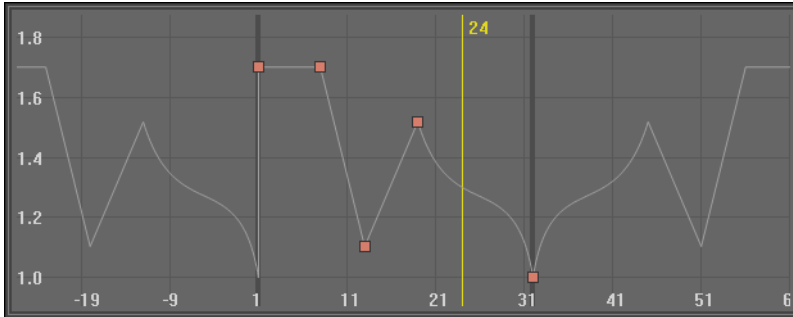
1. Create an animation with at least four keyframes. Do not worry about the interpolation setting at this point.
2. Click Animation, and then click Curves to display the Channel Editor.
3. Make sure you are in Edit mode and then select the first keyframe. See [“Editing Keyframes”](#) on page 438.
4. Click Constant to set the interpolation for the first keyframe to Constant.
5. Select the second keyframe.
6. Click Linear.
7. Select the third keyframe, and then click Bezier.

Your animation curve may resemble the following.



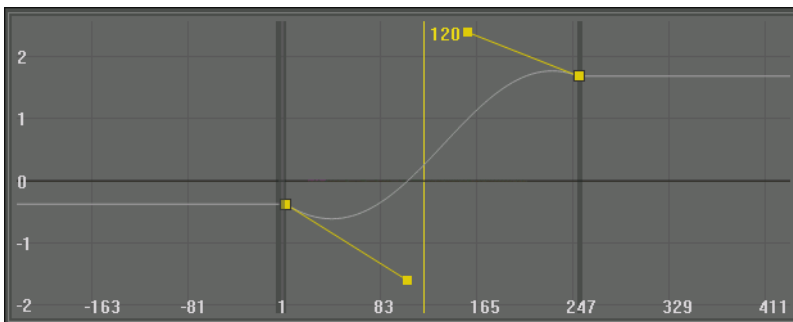
Setting Extrapolation

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.

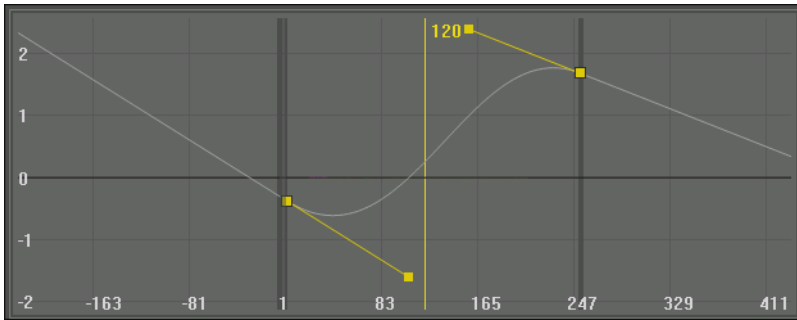


You can specify how keyframes are extrapolated using the following settings.

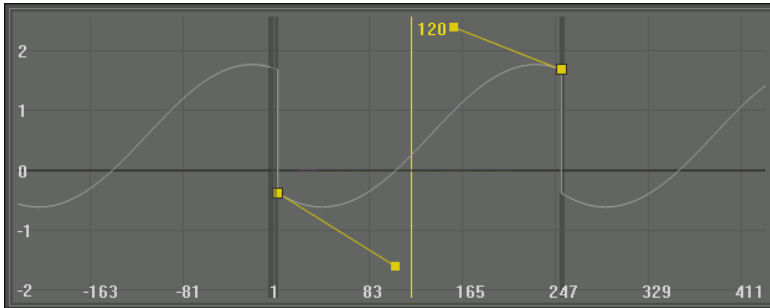
Constant — Creates an extrapolated animation curve that stays at the same value as the first or last keyframe.



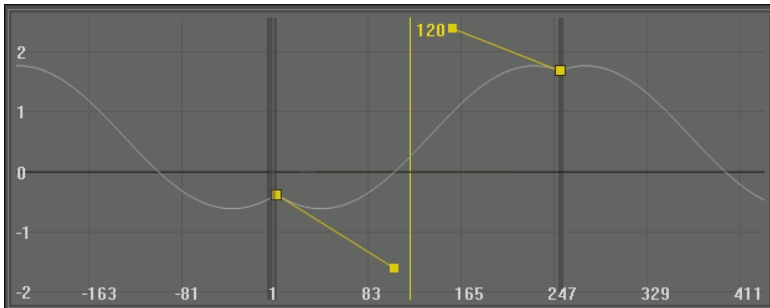
Linear — Creates an animation curve that changes in a linear fashion before the first or after the last keyframe.



Repeat — Creates a mirror image of the animation curve before the first or after the last keyframe.



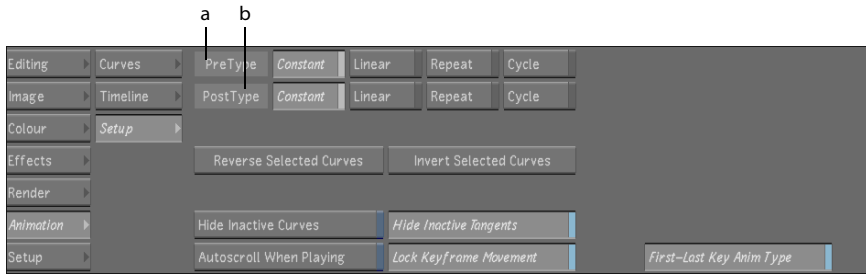
Cycle — Creates a pattern of repeating keyframes before the first of after the last keyframe.



To set the extrapolation type for a channel:

1. Select the keyframe for which you want to set the extrapolation.
2. Click Animation, and then click Setup.

The Animation Setup menu appears.



a) PreType extrapolation

b) PostType extrapolation

3. In the PreType controls, click one extrapolation type button to set the extrapolation before the first keyframe.
4. In the PostType controls, click one extrapolation type button to set the extrapolation after the last keyframe.

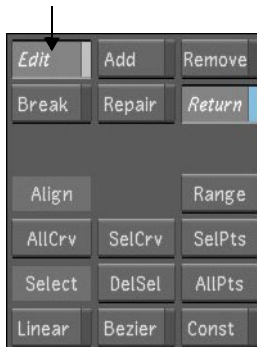
Editing Keyframes

As you create animations, you may need to edit keyframe values, modify animation curves, and manipulate keyframes. Make sure that you are in Edit mode when selecting and editing keyframes.

NOTE: You cannot edit keyframes in Add mode or Remove mode.

To enable Edit mode:

1. Click Animation, and then click Curves.
2. In the Animation controls, enable Edit.



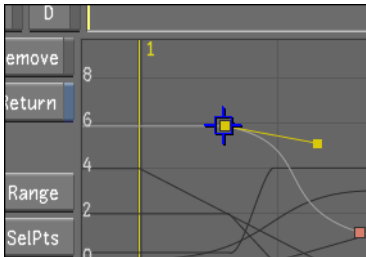
Selecting Keyframes

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:

- In the Animation > Curves menu, position the cursor over the keyframe and click.

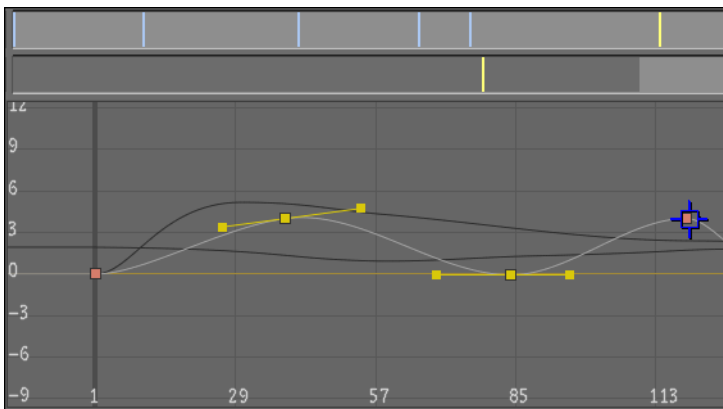


The keyframe is selected.

To select multiple keyframes:

- In the Animation > Curves menu, click a keyframe to select it and then hold down **CTRL** and click additional keyframes to add them to the selection.

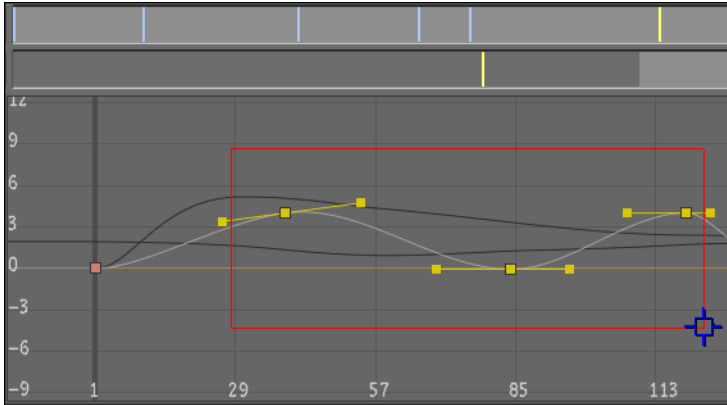
NOTE: To deselect a selected keyframe, hold down **CTRL** and click the keyframe. This is useful when you make a marquee selection but you would like not to include one or more keyframes in the selection.



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 menu, 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:

- In the Animation > Curves menu, click AllPts in the Animation Controls.

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 interpolation you have much more control over the shape of the animation curve, and therefore, the speed of the animation.

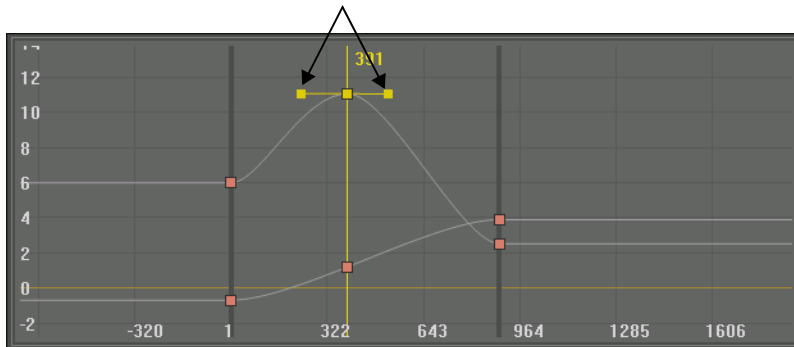
You can hide curves that are not selected. This can make it easier to manage your keyframes and modify animations.

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.

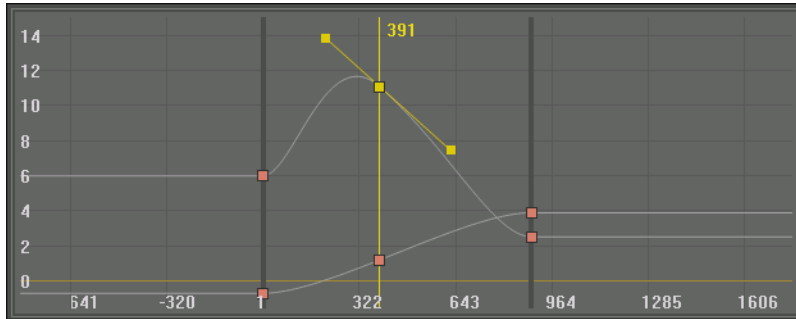


3. Select the first keyframe and set the interpolation to Bezier.
A pair of Bezier tangent handles appear for the selected keyframe.



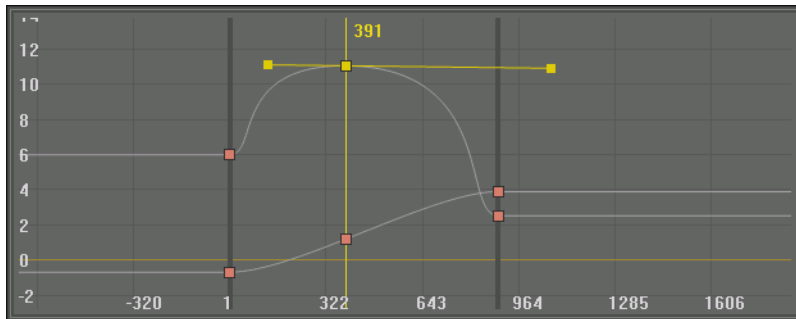
4. Drag one of the tangent handles.

The shape of the curve changes as you move the tangent handle.



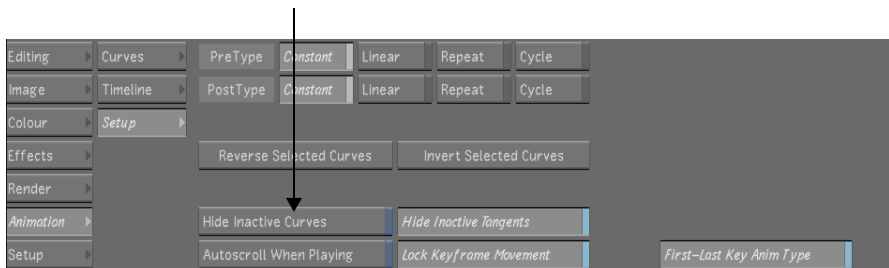
NOTE: When you drag a tangent handle, its adjacent tangent handle moves in the opposite direction. If you want to move the tangent handles independently, you need to first break them. See [“Breaking Tangent Handles”](#) on page 443.

5. You can lengthen a tangent handle independently without breaking the pair.



To hide curves that are not in use:

1. Click Animation, and then click Setup to display the Setup controls.
2. Enable Hide Inactive Curves.



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 First-Last Key Anim Type button 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.

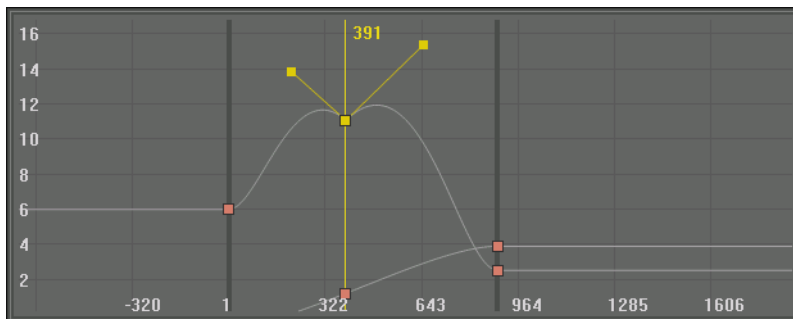
To ensure consistent keyframe functionality between the control surface and the user interface:

1. Click Animation, and then click Setup to display the Setup controls.
2. Enable First-Last Key Anim Type.



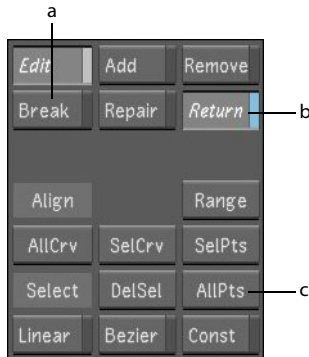
Breaking Tangent Handles

Normally when you move one Bezier 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 Edit mode, select the keyframes you want to break. You can also click AllPts to select all keyframes.



a) Break mode button

b) Return mode button

c) AllPts button

2. Enable Break.
3. 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.
4. In the Animation Curves window, click and drag the selected Bezier handle you want to break.
5. 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 Edit mode, select the broken keyframes you want to repair.
2. Click Repair.



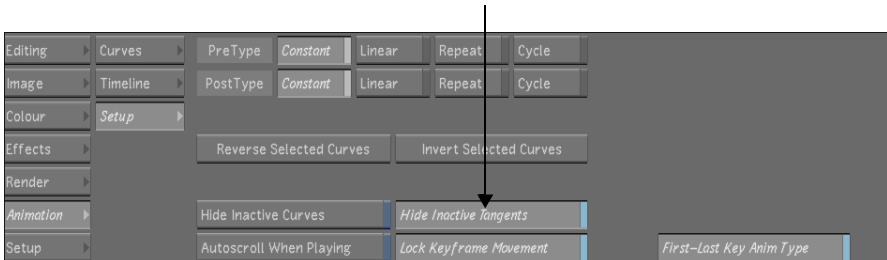
All broken tangent handles for the selected keyframes are restored to an unbroken state.

Hiding Tangent Handles

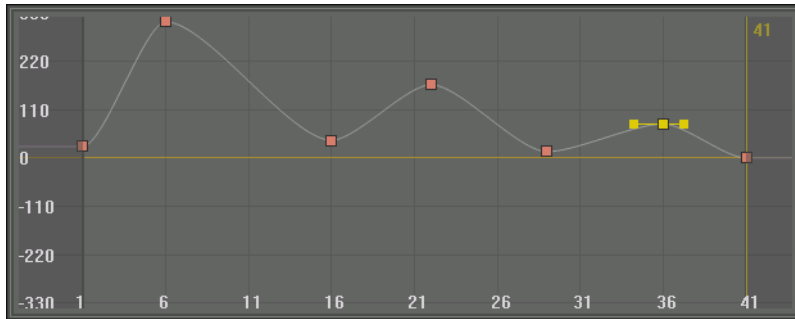
You can hide tangent handles for keyframes that are not selected.

To hide inactive tangent handles:

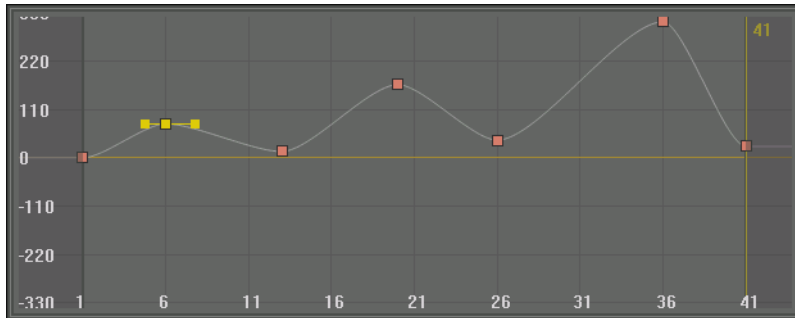
1. Click Animation, and then click Setup to display the Setup controls.
2. Enable Hide Inactive Tangents.

**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.



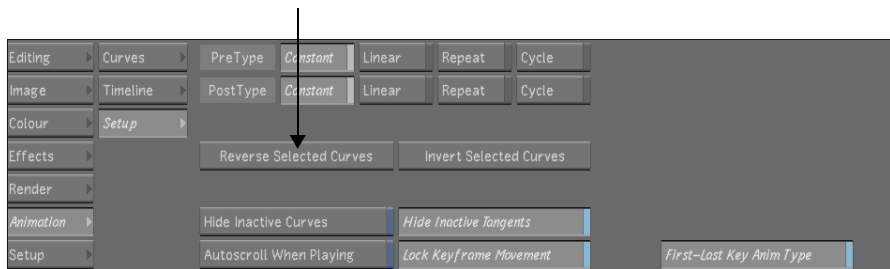
Before reversing the animation curve



After reversing the animation curve

To reverse keyframes for a channel:

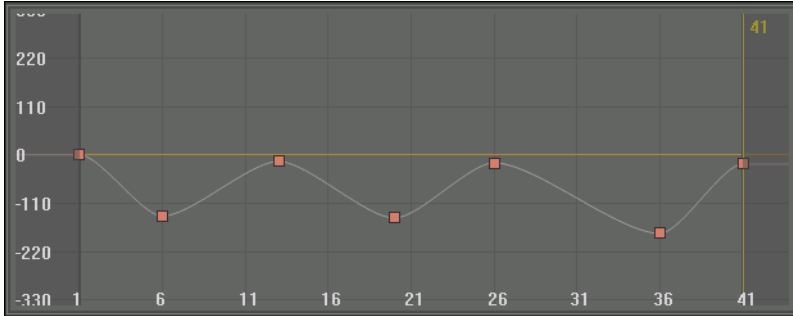
1. Click Animation, and then click Setup.
2. In the Channel Hierarchy, select the channel you want to reverse.
3. Click Reverse Selected Curves.



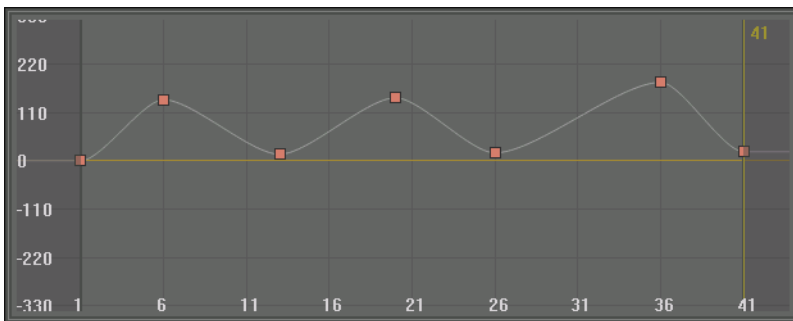
The selected channel's keyframes are reversed.

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.



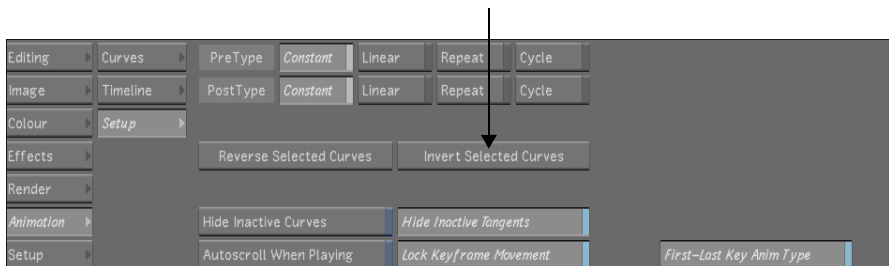
Before inverting the animation curve



After inverting the animation curve

To invert keyframes for a channel:

1. Click Animation, and then click Setup.
2. In the Channel Hierarchy, select the channel you want to invert.
3. Click Invert Selected Curves.



The selected channel's keyframes are inverted.

Deleting Keyframes

You can delete keyframes when they are no longer needed. You can:

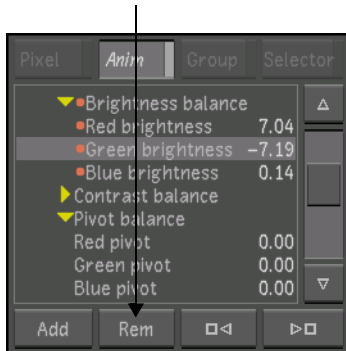
- Delete a single keyframe.
- Delete a selection of keyframes.
- Delete keyframes interactively, enabling you to click directly on keyframes in the Animation Curves window to delete them.

To delete one keyframe:

1. Make sure you are in Edit mode.
2. In the Animation Curves window, select the keyframe you want to delete.

NOTE: In menus other than the Animation menu, scrub to the frame with the keyframe you want to delete.

3. Click Rem.



The selected keyframe is deleted.

To delete a selection of keyframes:

1. Make sure you are in Edit mode.
2. In the Animation Curves window, select the keyframes you want to delete.

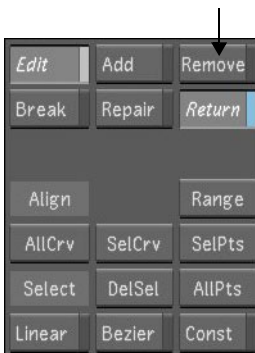
3. Click DelSel and then confirm the deletion. You can also press **DELETE**.



The selected keyframes are deleted.

To delete multiple keyframes interactively:

1. In the Animation controls, enable Remove.



2. 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

From any menu in Lustre, you can copy and paste keyframes for accessible channels in the Animation Channel Hierarchy. This includes, but is not restricted to, keyframes for grading,

geometries, plugin effects, and point trackers. There are different ways to perform a copy and paste:

- Using the user interface. See [“About the Copy Keyframe Controls”](#) on page 450.
- Using hot keys. See [“Copy / Paste Keyframe Hot Keys”](#) on page 456.
- Using the Autodesk Control Surface (ACS). See *Autodesk Control Surface User Guide*.

About the Copy Keyframe Controls

The Copy Keyframe controls are located in the Channel Hierarchy, which is visible in the user interface in all menus except the following menus:

- Editing > Timeline
- Colour > Timeline

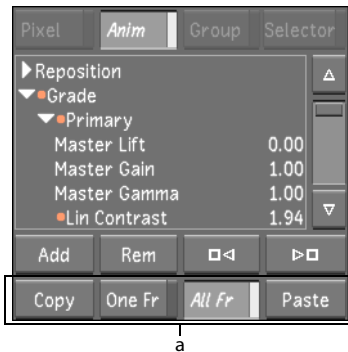
If you are in one of these menus and need to make channel selections or access the Copy Keyframe controls, navigate back to another menu that displays them. Select the channels to copy, then return to the Timeline menu; the selected channels remain active. Use the copy / paste hot keys to copy the keyframes. See [“Copy / Paste Keyframe Hot Keys”](#) on page 456.

The Channel Hierarchy and Copy Keyframe controls are not displayed, but are still active, when you are in the following menus:

- Editing > Timeline
- Colour > Timeline
- Editing > Capture
- Editing > Playout

If you are in one of these menus and need to make channel selections or access the Copy Keyframe controls, navigate back to another menu that displays them. Any channel / folder selections or Copy Keyframe actions you perform will remain active even after you leave the Channel Hierarchy.

NOTE: Hot keys and the ACS controls are available regardless of which menu is displayed in Lustre.



a) Copy keyframe controls

Copy — Click to copy one or more keyframes to memory. The Copy function is dependent on which channels are selected in the Channel Hierarchy or Animation Curve. See [“Copying Keyframes”](#) on page 451.

One Fr — When enabled, Lustre copies only the keyframes present at the current location of the Positioner for a selected channel or group of channels.

All Fr — When enabled, Lustre copies all the keyframes within a shot for a selected channel or group of channels.

Paste — Click to paste the copied keyframes. See [“Pasting Keyframes”](#) on page 453.

NOTE: If you are in the Animation > Curves menu and you select keyframes in the Animation Curves window, you do not need to use the *One Fr* and *All Fr* buttons. Lustre automatically copies and pastes the selected keyframes.

Copying Keyframes

When copying keyframes, use the Channel Hierarchy to select channels you wish to copy keyframes from. You can choose either a single channel or all channels in a parent folder. Once the channels are selected, you can choose to copy:

- Only those keyframes at the frame where the Positioner is located.
- All keyframes in the shot.
- A selection of keyframes.

To copy only keyframes at the location of the Positioner:

1. Make sure the Positioner is located directly over the keyframe.

To do this, navigate to the keyframe by clicking on the Forward and Backward buttons in the user interface or with the ACS.



2. Change the copy mode to One Fr by doing one of the following:
 - Click One Fr.
 - Press **ALT+C**.
 - Change the copy mode to One Fr using the ACS.
3. Copy the keyframes by doing one of the following:
 - Click Copy.
 - Press the left **CTRL+C**.
 - Use the ACS.

To copy all keyframes in the shot:

1. Change the copy mode to All Fr by doing one of the following:
 - Click All Fr.
 - Press **ALT+C**.
 - Change the copy mode to All Fr using the ACS.
2. Copy the keyframes by doing one of the following:
 - Click Copy.
 - Press the left **CTRL+C**.
 - Use the ACS.

To copy a selection of keyframes:

1. Navigate to the Animation > Curves menu.
2. To display the full range of the selection of curves, click Range.
3. In the Animation Curves window, select the keyframes you wish to copy. See [“Selecting Keyframes”](#) on page 439.

4. Copy the keyframes by doing one of the following:

- Click Copy.
- Press the left **CTRL+C**.
- Use the ACS.

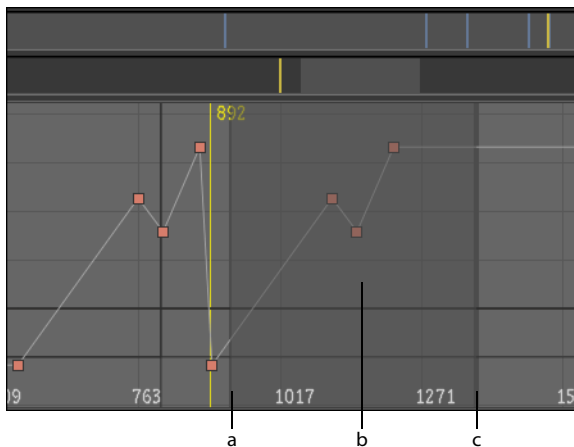
Pasting Keyframes

When pasting keyframes, use the Channel Hierarchy to select the target channels. You can select either a single channel or all channels in a parent folder.

When you copy all the channels in a sub-folder and paste into a parent folder, the keyframes are pasted into the first available sub-folder of the parent folder. For example, if you copy all the channels in the Tracker4 sub-folder and paste into the parent Trackers folder, the keyframe data are pasted into the Tracker1 sub-folder.

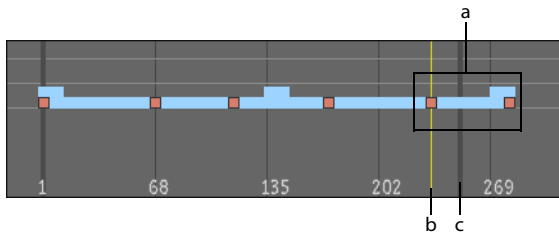
When you copy a keyframe sequence which spans more than one frame in a shot, you can paste the sequence at a location where it goes beyond the shot boundary. The part of the sequence that is cut off is still pasted after the shot boundary.

In the Animation Curves window, the cut off keyframes appear greyed out as in the following example.



a) Shot boundary b) Cut off keyframe data c) Beginning of next shot

It is also possible to see the cut off keyframes in Animation > Timeline view.



- a) Pasted keyframe sequence b) Positioner (where first of copied keyframes is pasted) c) Shot boundary

Once you have selected the target channels or parent folder, you can paste the copied keyframes:

- Over a target selection of keyframes.
- Starting at the current location of the Positioner.

If you paste keyframes and you have not selected target keyframes to paste over, the pasting operation will destroy any and all keyframe data that was originally present in a target span that starts at the location of the Positioner. This target span of the timeline is exactly the size of the span in the copy buffer.

To paste copied keyframes to the Positioner location:

1. In the Channel Hierarchy, select the target channel or parent folder.
The selected channel(s) are highlighted.
2. In the Animation Curves window, make sure no keyframes are selected.

NOTE: If keyframes remain selected the contents of the copy buffer will be pasted to this selection.

3. Do one of the following:
 - To paste the keyframes to a different shot in the same timeline, use the Scene Timebar to move the Positioner to the desired shot.
 - To paste the keyframes to a different timeline, load the required cut, then return to the Animation > Curves menu and move the Positioner to the target location. See [“Loading a Cut”](#) on page 50.

NOTE: Before loading a different cut, save the grade of the cut you are in. Failure to do so will result in a loss of any new grading information (including animations) for this cut.

4. Do one of the following:

- Click Paste.
- Press **CTRL+V**.
- Use the ACS to paste the keyframes.

The keyframe data in the copy buffer is pasted to the timeline starting at the location of the Positioner.

For rules regarding the paste behaviour of point trackers, plugins, and geometries, see [“Guidelines for Point Trackers, Plugins, and Geometries”](#) on page 456.

Pasting to a Keyframe Selection

The following are behaviours specific to pasting to a keyframe selection:

- When you paste keyframe data to a target keyframe selection, all keyframe data originally present in the span of the timeline occupied by the target keyframe selection is destroyed. This is true even if the copied keyframes occupy a timeline span that is smaller than the target span.
- If the target span is discontinuous (that is, there are one or more unselected keyframes separating the selected keyframes of the span), the unselected keyframes are not destroyed when the copy buffer is pasted there. For more information about selecting a span of keyframes, see [“Selecting Keyframes”](#) on page 439.
- If the span of the copied keyframes is larger than the span of the target selection of keyframes, the copied keyframes are pasted over the target selection as expected, but the portion of the copied keyframes that overshoots the target selection is discarded.

To paste over a selection of keyframes:

1. In the Channel Hierarchy, select the target channel or parent folder.
2. Navigate to the Animation > Curves menu.
3. Select the desired keyframes in the Animation Curves window. See [“Selecting Keyframes”](#) on page 439.
4. Do one of the following:
 - Click Paste.
 - Press **CTRL+V**.
 - Use the ACS to paste the keyframes.

The keyframe data in the copy buffer is pasted to the target keyframe selection.

Guidelines for Point Trackers, Plugins, and Geometries

When copying and pasting keyframes for point trackers, plugins, and geometries, you need to be aware of the following:

- You can only copy and paste the same channels (e.g., point tracker channel can be only be copied to another point tracker channel).
- When pasting keyframe data to a channel folder, if there are more keyframes in the copy buffer than there are appropriate channels to paste them to, Lustre will match keyframes in the copied folder (in top-to-bottom order) to channels in the target folder and ignore the remaining keyframes.
- You can only copy the keyframes for common parameters from one geometry to another.
- Geometries are not saved with absolute values. Therefore it is not possible to copy one axis to another while keeping its original position. You may have to manually modify the channel(s) value of your geometry.

Copy / Paste Keyframe Hot Keys

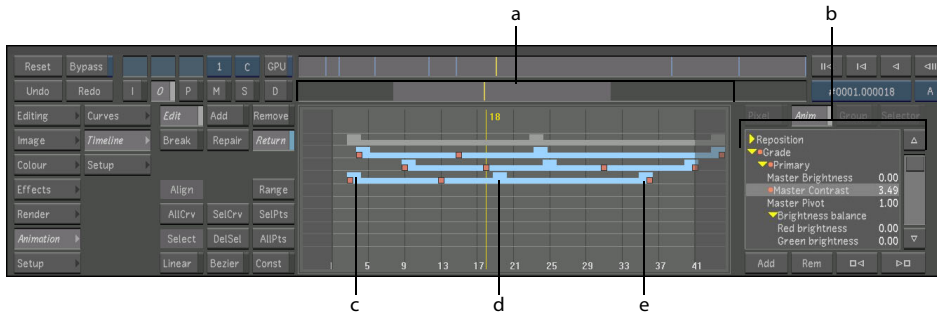
When in either the Editing > Timeline or Colour > Timeline menus, the only way to perform the copy and paste actions is by using hot keys or the ACS (see the *Autodesk Control Surface User Guide*). You can use the hot keys when you are in any of Lustre's menus.

Hot key	Action
Left-CTRL+C	Copy keyframe(s).
Left-CTRL+V	Paste keyframe(s).
CTRL+Z	Undo
CTRL+ALT+Z	Redo
ALT+C	Toggle between One Fr and All Fr copy modes.

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) Track Editor c) First keyframe in channel e) Last keyframe in channel
 b) Channel hierarchy d) A selected 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 directories 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.

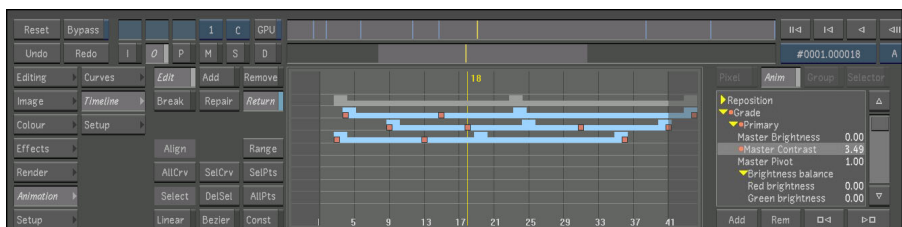
Accessing the Track Editor

Use the Track Editor to adjust animation timing for an existing animation.

To access the Track Editor:

- Click Animation, and then click Timeline.

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. Click Animation, and then click Timeline.
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

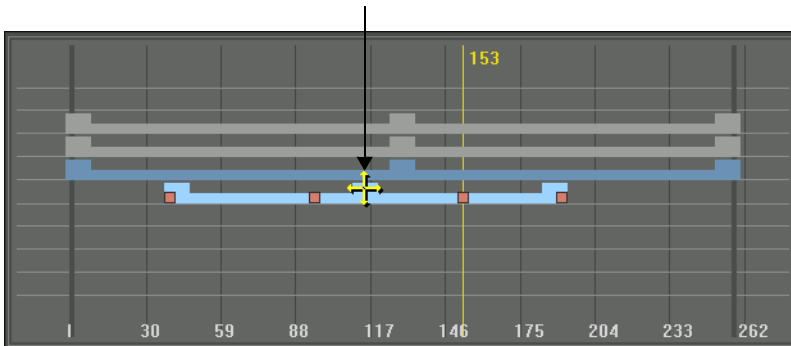
Using the Track Editor, you can adjust a keyframe's timing without adjusting its value. You can also slide the entire animation ahead or back in time, and change the duration of an animation.

To adjust the timing for an entire channel:

1. Click Animation, and then click Timeline.

HINT: Click Range to see all three handles.

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. 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.

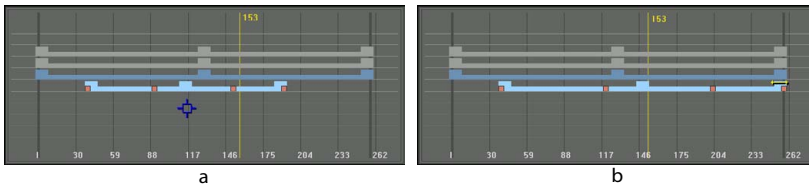
To adjust keyframe timing:

1. Click Animation, and then click Timeline.
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. 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:

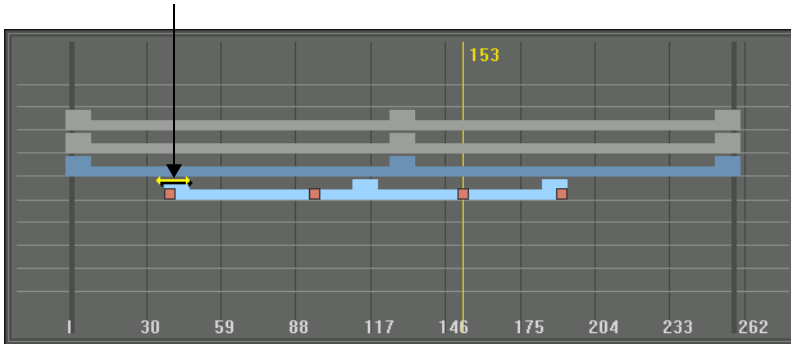
1. Click Animation, and then click Timeline.
2. In the Channel Hierarchy, select the channel you want to modify.
The corresponding track in the Track Editor turns light blue and its keyframes appear.
3. Select the first or last keyframe and drag it to change the start or end point of the animation.

NOTE: Adjusting the timing for a folder changes the timing for all channels in that folder.

To scale an animated channel's timing:

1. Click Animation, and then click Timeline.
The Track Editor appears.
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. 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.

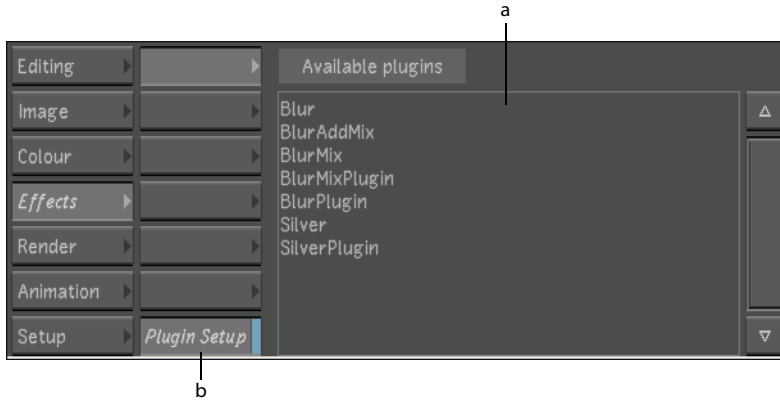
NOTE: Adjusting the timing for a folder changes the timing for all channels in that folder.

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 Effect menu:

1. Select a shot and scrub to the first frame.
Click Effects, and then enable Plugin Setup.

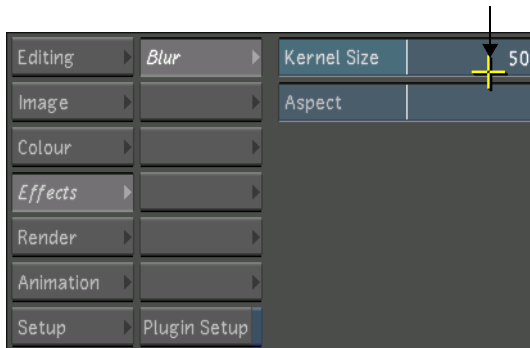


a) Available plugins list

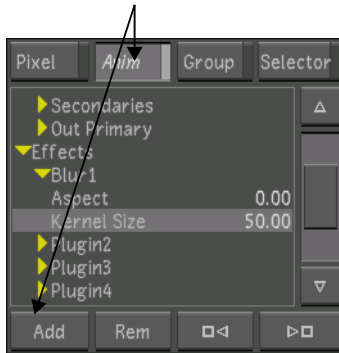
b) Plugin Setup button

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”](#) on page 440.

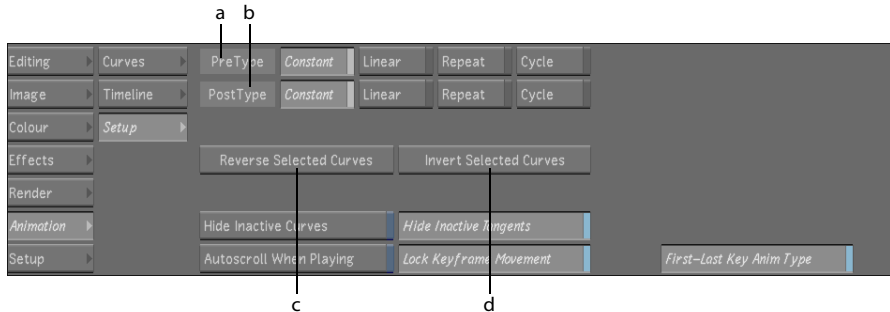
Specifying Setup Options

Use the options in the Setup menu to determine the way in which the Animation menu functions.

To access the Setup menu:

- Click Animation, and then click Setup.

The Setup menu appears.



a) PreType extrapolation options

c) Reverse Selected Curves button

b) PostType extrapolation options

d) Invert Selected Curves button

Setting Extrapolation Method

Use the buttons under PreType and PostType to set keyframe extrapolation. See [“Setting Extrapolation”](#) on page 436.

Reversing and Inverting Keyframes

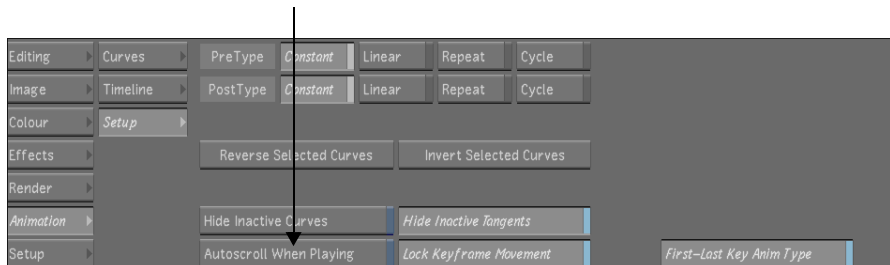
Use the Reverse Selected Curves and Invert Selected Curves options to modify your keyframes for selected curves. See [“Editing Keyframes”](#) on page 438.

Autoscrolling in the Channel Editor

You can set up the system to scroll automatically when the positioner plays off the far right edge of the Channel Editor (or the far left edge when playing backward).

To enable autoscrolling:

- In the Setup menu, enable Autoscroll When Playing.

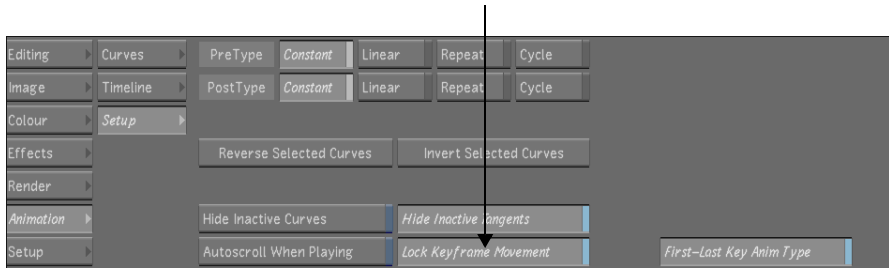


Locking Keyframe Movement

When the Lock Keyframe Movement option is on, you can drag a keyframe vertically only. By default, this option is enabled.

To lock keyframe movement:

1. In the Setup menu, enable Lock Keyframe Movement.



2. 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.

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

18

Rendering

Summary

About Rendering in Lustre	465
Rendering Shots	466
Saving and Loading Colour Mapping Presets	482
Generating and Viewing Proxies	484
Locking Colour Grading	486
Cleaning Up Render Files	487
Rendering Shots as You Work	488
Rendering with Burn for Lustre	490

About Rendering in Lustre

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, using the DVS Video I/O board, 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.

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](#)” on page 222. 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”](#) on page 249.

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 *sacc_data* 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.

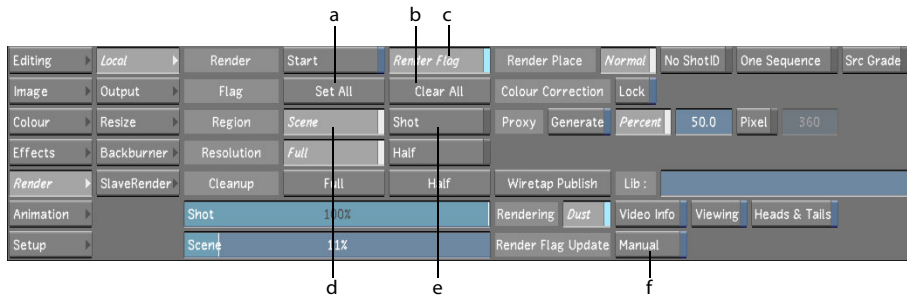


To flag shots, use the Render Local menu.

To access the Render Local menu:

- In the Main menu, click Render, and then click Local.

The Render Local menu appears.



- a) Set All button
- b) Clear All button
- c) Render Flag 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 or press].

To manually flag all shots in the cut:

- 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 or press [.

To clear all render flags:

- Click Clear All.

NOTE: Render flags are not automatically cleared after rendering.

To flag shots automatically:

- 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:

- Do one of the following:
 - Enable Scene to render all flagged shots in the cut.
 - Enable Shot to render the current flagged shot.

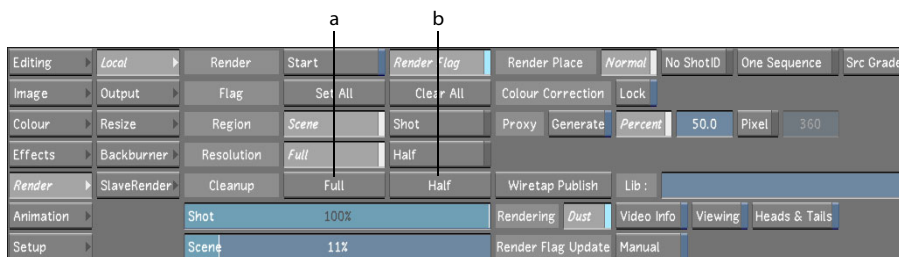
NOTE: In both cases, rendering starts from the current frame and continues forward.

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 Render Local menu.

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:

- Specify the output resolution you want to use by enabling the Full Resolution or Half Resolution button.

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 button.
2. Select a video format in the Render Resize menu. See [“Resizing Shots”](#) on page 469.

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.

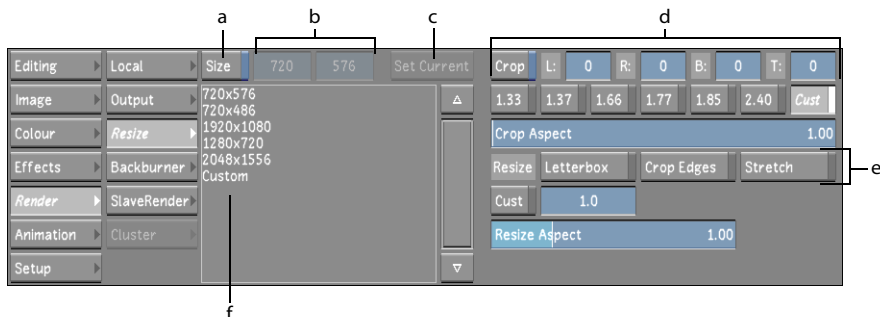
When you resize shots using the Resize menu, all shots in the timeline are resized. To resize individual shots, use the tools in the Reposition menu. See [“Repositioning an Image”](#) on page 271.

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.

To resize shots:

1. Load the grade linked to the cut containing the shots you want to resize. See [“Loading a Cut”](#) on page 50.
2. Click Render and then click Resize.

The Resize menu appears.



a) Size button	d) Crop options
b) Custom Width and Height fields	e) Resize options
c) Set Current button	f) Video and film format list

- 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.

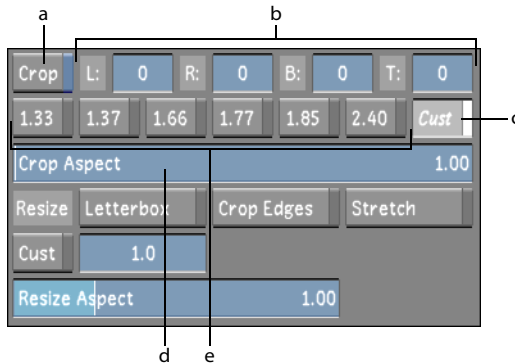
NOTE: You can check the resolution of the original scans in the Shot bin, accessible from the Editing Browse menu. Source resolution is important for quality resizing. For example, you obtain a better quality HD output with 2K full scan sources than with 1K half scan sources.

- 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.

- 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 c) Cust (Custom) button e) Crop Preset buttons
b) Left, Right, Bottom, and Top fields d) Crop Aspect slider

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.

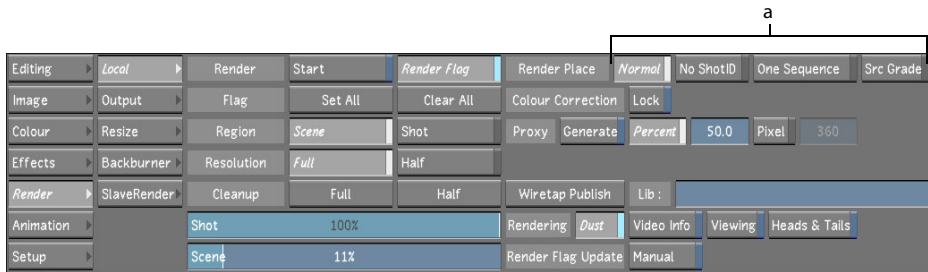
Specifying the Destination for Local Render Files

In the case of Normal and No ShotID renders, render files are saved in sub-folders of a folder called *grd*, which Lustre creates automatically when you first render shots. The location of the *grd* folder depends on your project directory structure. See [“Recommended Directory Structure for Projects”](#) on page 78.

NOTE: One Sequence renders are saved in sub-folders of the *scene* folder.

NOTE: Dust removal render files created using the Rendering controls in the Dust menu are not saved in the *grd* file. See [“Rendering the Result”](#) on page 264.

You can specify how you want render files to be organized under the *grd* folder 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. When Normal is selected, Lustre saves all the render files of each given shot into a render folder for that shot. The name for each render folder is the unique ID (UID) of the shot. When No ShotID is selected, Lustre saves all render files to a single folder called *Grd*.

To set how render files are organized:

- Enable Normal, No ShotID, One Sequence, or Src Grade.

NOTE: Examples in the following section use backward slashes. On a Linux system, these would be forward slashes.

Normal — Individually graded shots are rendered to their own UID folder in the Renders Home location.

For example, the shots are rendered to:

```
<HOME>/grd/<shot_grdXX_UID>/<resolution>/
<origfilename_grdXX_UID.xxxx>.dpx
```

Gaps are rendered out as black frames to a separate folder in the Renders Home location.

For example, the frames are rendered to:

```
<HOME>/<scene>/grd/color_<grdXX_UID>/<resolution>/
color_<grdXX.xxxx>.dpx
```

No ShotID — Individually graded shots are rendered to the grade folder as individual shots in the Renders Home location.

For example, the shots are rendered to:

```
<HOME>/grd/<shot_grdXX>/<resolution>/
<origfilename_grdXX.xxxx>.dpx
```

Gaps are rendered as black frames to a separate folder in the Renders Home location.

For example, the frames are rendered to:

```
<HOME>/<scene>/grd/color_<grdXX>/<resolution>/  
color_<grdXX.xxxx>.dpx
```

One Sequence — Individually graded shots are rendered to the scene folder as a single sequence in the Renders Home location. Black frames are rendered to the same single sequence in the same scene folder.

For example, these shots are rendered to:

```
<HOME>/<scene>/grd/<grdXX>/<resolution>/<scene_grdXX.xxxx>.dpx
```

Src Grade — Individually graded shots are rendered to their own UID folder in the Renders Home location.

For example, the shots are rendered to:

```
<HOME>/grd/<shot_grdXX_UID>/<resolution>/  
<origfilename_grdXX_UID.xxxx>.dpx
```

Gaps are rendered out as black frames to a separate folder in the Renders Home location.

For example, the frames are rendered to:

```
<HOME>/<scene>/grd/color_<grdXX_UID>/<resolution>/  
color_<grdXX.xxxx>.dpx
```

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”](#) on page 187 and [“Adding Black Media to the Multi-Layer Timeline”](#) on page 196.

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 no longer seen as transparent, and automatically render out as black frames.

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”](#) on page 205
- [“Muted Layers”](#) on page 206
- [“Shot Priority”](#) on page 207
- [“Solo Mode”](#) on page 211

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 unique ID (UID) folder in the Renders Home location. If there are no shots beneath the gap, Lustre renders the gap as a virtual black clip to a separate folder in the Renders Home location.

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. In addition, gaps are rendered out as black frames to a separate folder in the Renders Home location.

Rendering the Multi-Layer Timeline with No ShotID

When the No ShotID button is selected, Solo mode is disabled, and there are no muted layers, by default, Lustre sees gaps as 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. Lustre renders the individually graded shots with frames that are visible to the grade folder as individual shots in the Renders Home location. If there are no shots beneath the gap, the gap is rendered out as black frames to a separate folder in the Renders Home location.

You can render individually graded shots with frames that are visible and save the render files to the grade folder as individual shots in the Renders Home location. Gaps are rendered out as black frames to a separate folder in the Renders Home location.

NOTE: 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 must save a new grade (i.e., grd02).

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 UID folder under the grade directory and gaps are rendered as black frames to a separate folder in the Renders Home location. 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.

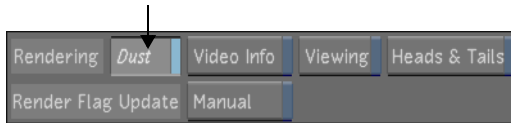
Rendering Dust Metadata

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”](#) on page 264.

To render shots with dust metadata, you must first use the dust removal tools to remove dust artefacts. See [Chapter 9, “Removing Dust,”](#) on page 253.

To render shots with or without dust metadata:

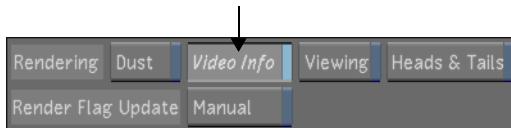
- In the Render Local menu, do one of the following:
 - To render shots without dust metadata, disable Dust.
 - To render shots with dust metadata, enable Dust.

**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:

- 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.

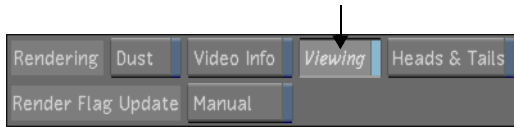
When this option is disabled, the currently applied viewing LUT is not taken into account in the output image, and the files are rendered to the file format specified in the Render Output menu rather than to 8-bit BMP files.

For information on monitor calibration, see [“Calibrating a Monitor”](#) on page 123. For information on output colour space settings, see [“Setting Colour Space Options”](#) on page 478.

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”](#) on page 478.

To set rendering with the viewing LUT:

- In the Render Local menu, enable Viewing.

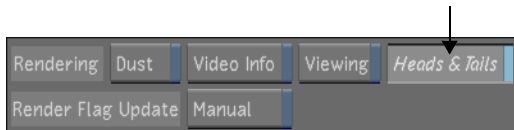
**Rendering Head and Tail Frames**

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:

- In the Render Local menu, enable Heads & Tails.

**Specifying the Output Format of the Render Files**

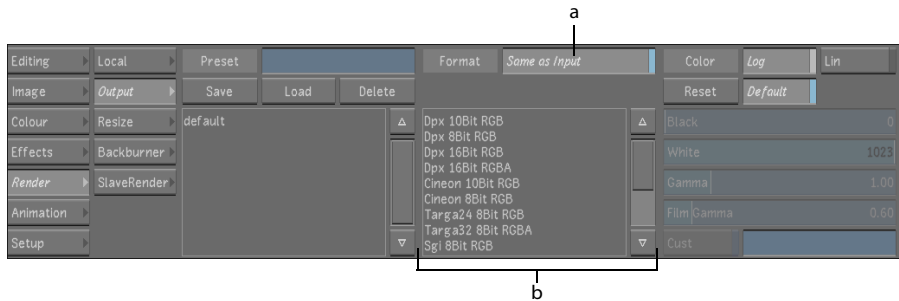
You can render your shots to the same file format as the original footage or select a different one. By default, shots are rendered to the same format as the original footage. Currently supported formats are DPX and Cineon at 10 and 8 bits per channel, TIFF at 8 and 16 bits per channel, and TARGA®, SGI, and BMP at 8 bits per channel.

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.

To specify the output format:

1. In the Main menu, click Render and then click Output.

The Render Output menu appears.



a) Same as Input button

b) List of supported output formats

2. Do one of the following:
 - To render to the same format as the originals, enable Same as Input.
 - To render to another format, disable Same as Input.
3. If you are rendering to another format, select the format in the list by highlighting it.

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 the LUT used on output. You can:

- Use the default colour mapping. The default mapping differs according to the colour architecture used on the project (as set in the Grade menu).

Colour Architecture	Output Colour Space	Colour Mapping Applied
Log	Log	Full range
Log	Lin	Standard log-to-lin (95-685)
Lin	Lin	Full range
Lin	Log	Standard lin-to-log (95-885)

- Create a LUT.

NOTE: You can save LUTs that you create in the Render Output menu as presets for use on other renders. See [“Saving and Loading Colour Mapping Presets”](#) on page 482.

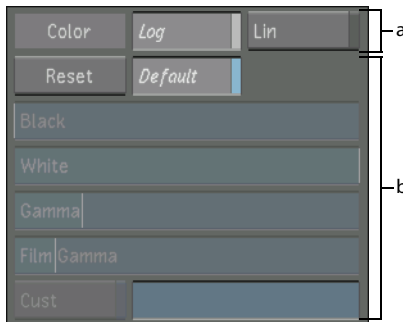
- Select an output LUT from a list of those made available in Lustre.

- Use the current viewing LUT. The colour mapping settings made in the Render Output menu are taken into account with the current viewing LUT. See [“Rendering with the Viewing LUT”](#) on page 476.

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 after the viewing LUT and *nd monitor.calib* correction (if used) are applied. For this reason, when working on shots prior to rendering, it is recommended that you set the output colour space to that set for the grade colour architecture in the Grade menu, and set the output colour mapping to Default.

To set output colour space and colour mapping options:

1. Access the Render Output menu.



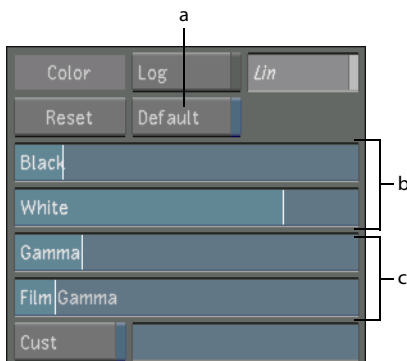
a) Output colour space Log and Lin buttons

b) Output colour mapping options

2. Enable either Log or Lin.
3. Do one of the following:
 - Enable Default to use the default colour mapping.
 - Disable Default and either create a LUT or select one, as shown in the following procedures.

To create an output LUT:

1. Access the Render Output menu.



- a) Default button b) Black and White fields c) Gamma and Film Gamma fields

2. Make sure Default is disabled.
3. Set the Black and White values.
4. Set the Gamma and Film Gamma values (for Lin colour space output only).

NOTE: If needed, you can reset these values to their default settings by clicking the Reset button and confirming.

To select an output LUT:

1. Access the Render Output menu.

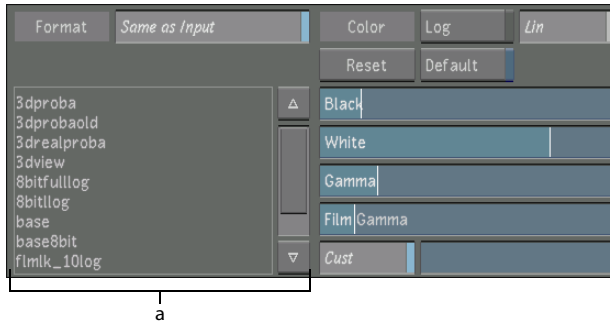


- a) Default button b) Cust button c) LUT box

2. Make sure Default is disabled.

3. Enable Cust.
4. Click in the text field next to the Cust button.

A list of available LUTs appears in the scrollable window to the left.

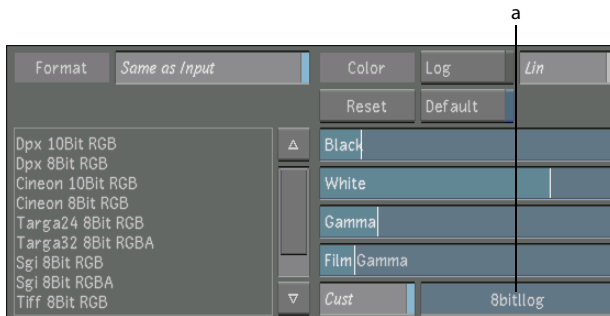


a) Available LUTs

NOTE: The LUTs displayed in the list are all located in the `C:\Program Files\Autodesk\lustre2009\luts` folder, if running the Windows version of Lustre, and in the `/usr/autodesk/lustre2009/luts` directory, if running the Linux version of Lustre.

5. Select a custom setting from the list.

The image is updated and the LUT name appears in the LUT box.



a) LUT box

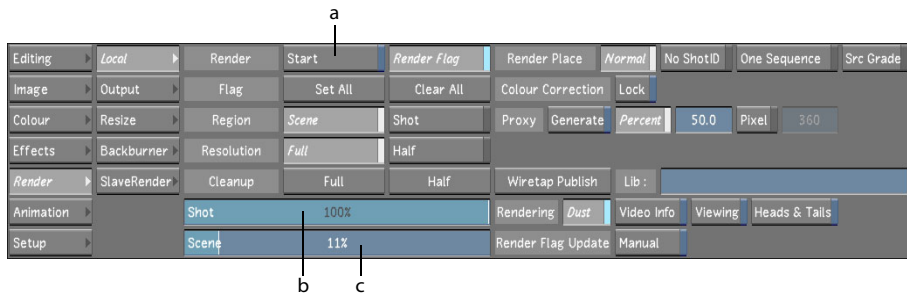
NOTE: If you are using background rendering, place the LUT on all render nodes in the `/usr/autodesk/lustre2009/lut/` directory.

Rendering the Selected Region

Once you have set all the rendering options, you can render flagged shots in the selected region using the Render Local menu.

To render the selected region:

1. Access the Render Local menu.



- a) Start button b) Shot progress bar c) Scene progress bar

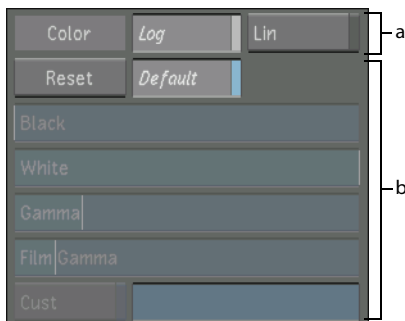
2. Click Start.

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.

Saving and Loading Colour Mapping Presets

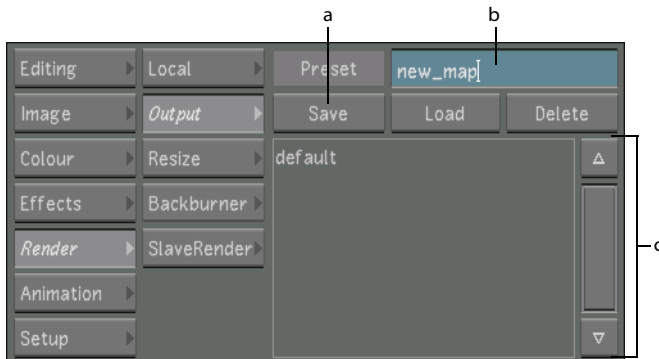
When you create output LUTs in the Render Output menu, you can save them as presets and re-use the presets on future renders. The output colour space setting (Log or Lin) and colour mapping values are saved with the presets.



- a) Output colour space Log and Lin buttons b) Output colour mapping options

To save a colour mapping preset:

1. Access the Render Output menu.



- a) Save button b) Preset field c) Preset list

2. Set the colour space and colour mapping values as needed. See [“Setting Colour Space Options”](#) on page 478.

3. In the Preset field, enter a name for the preset.

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.

4. Click Save.

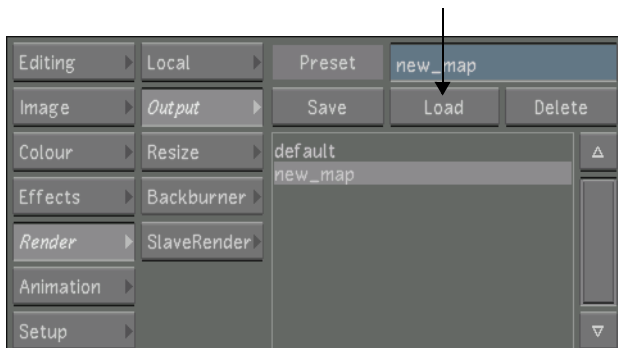
The new preset appears in the Preset list.

To load a preset:

1. In the Preset list, click the preset you want to load.

It is highlighted in light grey.

2. Click Load.



To delete a preset:

1. In the Preset list, click the preset you want to delete.
It is highlighted in light grey.
2. Click Delete and confirm.

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 menu”](#) on page 24.

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”](#) on page 78.

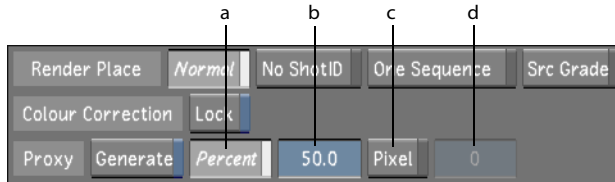
NOTE: Proxy generation differs from rendering a half-resolution render using the Half Resolution button in that grading and other effects data is not applied to proxies as it is when rendering the images.

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”](#) on page 495.
2. Load the scene and grade containing the cut for which you want to generate proxies. See [“Selecting Scenes”](#) on page 48 and [“Loading Grades for a Cut”](#) on page 59.
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.

- 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.

- 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:

- 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.

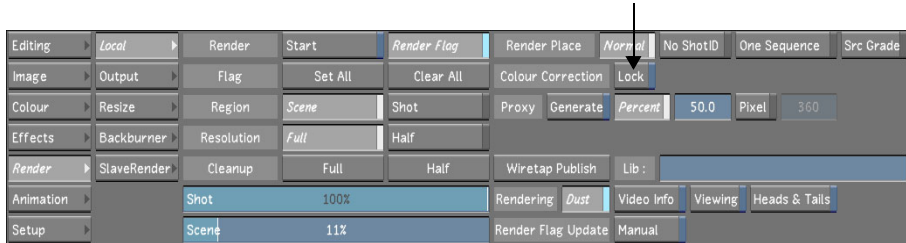
Locking Colour Grading

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”](#) on page 94.

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

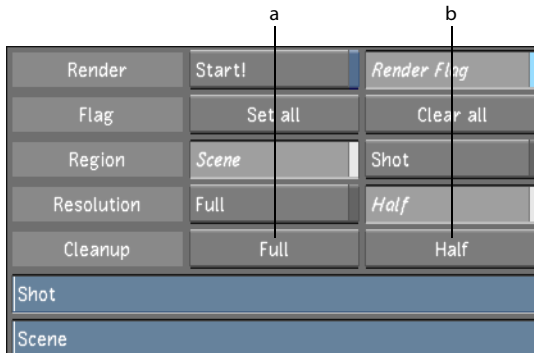
You can remove unneeded render files. Because renders are created separately for each grade (that is, grd01 to grd99), you may want to delete renders for grades that are not used anymore to free up storage.

To remove unneeded render files:

1. Load the grade that has associated unwanted render files. See [“Loading Grades for a Cut”](#) on page 59.
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.

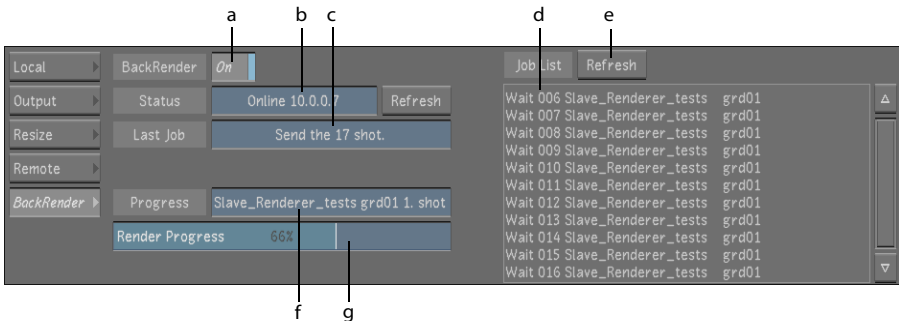
NOTE: When rendering a Multi-Layer Timeline, Slave renders the shots that are visible from a flattened timeline, or the shots from the active layer in the Multi-Layer Timeline, depending on the mode you are working in. For Solo mode, see [“Multi-Layer Timeline”](#) on page 188 and for Mute mode, see [“Muted Layers”](#) on page 206.

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”](#) on page 222.

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 menu”](#) on page 31.
2. Start Lustre.
3. Access the Render SlaveRender menu.



- a) SlaveRender On/Off button c) Last Job field e) Job List Refresh button g) Render progress bar
- b) Status field d) Job List f) Progress field

In the Status field, the status and IP address of the slave render machine are displayed.

HINT: Click Refresh to update the Status field.

4. Turn on slave rendering. Set the SlaveRender On/Off button to On, or press **X**.
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”](#) on page 466.
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.

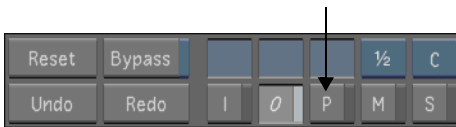
HINT: The Render progress bar shows the shot’s render progress as a percentage and updates every second. You can change the update interval using the Refresh in Seconds slider in the

Network Rendering tab of the project configuration. See [“Network Rendering menu”](#) on page 31.

8. When you have finished, you can turn off rendering with the Slave Renderer by setting the SlaveRender On/Off button to Off, or pressing **X** again.
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”](#) on page 188) or Mute (see [“Muted Layers”](#) on page 206) 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”](#) on page 492.

Example:

```
BURN_PROJECT_HOME 192.23.200.76:/L:/lustre_project
```

```
BURN_RENDERERS_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”](#) on page 495.

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.

Lustre supports Autodesk Backburner™ 2008.1. In Backburner 2008.1, 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 2008.1 User Guide*.

For details on starting and configuring Backburner Manager, see the *Autodesk Backburner 2008.1 Installation Guide*.

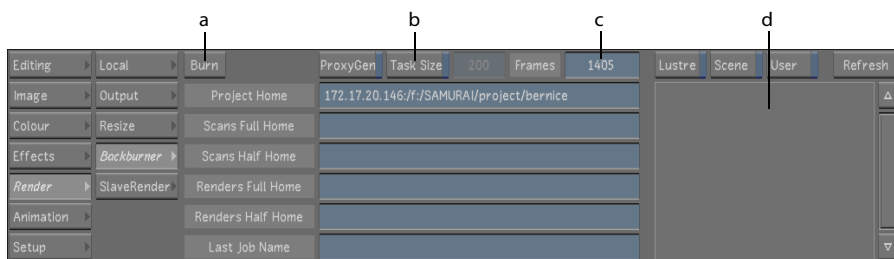
Submitting a Remote Render Job

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 [“Loading a Cut”](#) on page 50.
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”](#) on page 94.
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.

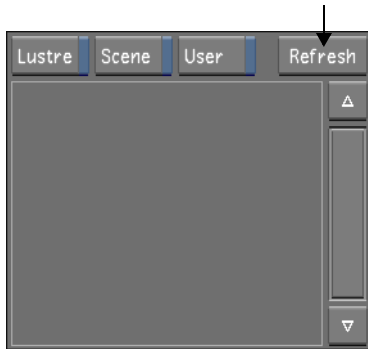
6. Click Burn.

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”](#) on page 493.
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.

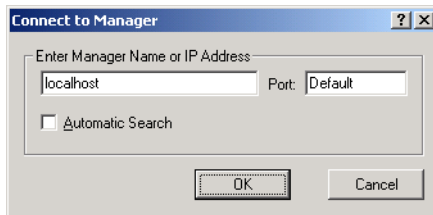
Monitoring the Render Tasks with Backburner Monitor

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:

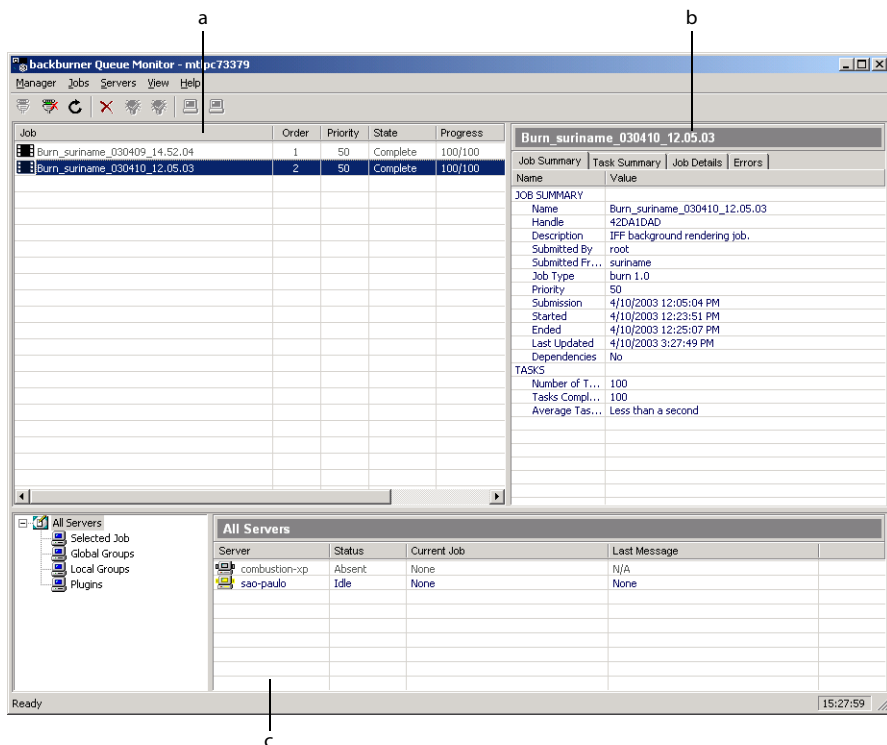
1. On the workstation designated as the render monitor, choose Start | Programs | Autodesk | Backburner | monitor.
The Monitor window appears.
2. If no servers or data appear in the Backburner Queue Monitor, the Backburner Manager was not found with the current settings. Choose Manager | Connect to connect 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 www.rdesktop.org.

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	
-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	<project_home_folder>	The path and directory name of the Project Home directory, including the drive letter. Example: G:\ProjectAB	No	Not required if set as environment variables on the local machine. See "Project Configuration" on page 15.
-2	<footage_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 rendering. Example: -f 120 340	No	Can be used to specify a new range when restarting a cancelled render.

Par.	Modifier	Description	Req'd	Comments
-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 information into account.	No	
-e	<preset>	Name of output colour space preset, if used.	No	See "Saving and Loading Colour Mapping Presets" on page 482.
-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 Destination for Local Render Files" on page 471.
-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 menu" on page 24.
-M	None	Render using the Mitchell filter for reposition. There are three parameters (s = size; B = blur; C = ringing).	No	See "Rendering menu" on page 24.
-T	None	Render using the CatmullRom filter for reposition.	No	See "Rendering menu" on page 24.

Par.	Modifier	Description	Req'd	Comments
-O	None	Render using the new type Lanczos3lobe filter for reposition.	No	See "Rendering menu" on page 24.
-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 2009
```

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/lustre2009
```

3. Enter the render command using the syntax provided in the previous table.

19

Stereoscopy

Summary

About Stereoscopy	499
Creating a Stereoscopic Project	499
Viewing From One Eye to the Other	503
Colour Grading the Left and Right Eye	504
Rendering Stereoscopic Footage	505

About Stereoscopy

Stereoscopy (Stereo) is a new feature in Lustre 2009 which allows 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 are creating a stereoscopic project, you need to do the following:

- Enable the Stereoscopy feature. See [“Enabling Stereoscopy”](#) on page 500.
- Load your left and right eye footage into your timeline. See [“Setting Up the Stereoscopic Footage”](#) on page 502.

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 Configuration Settings”](#) on page 20.
- When you select a Stereoscopy-specific raster. See [“Selecting a Raster for Stereoscopy”](#) on page 500.
- By enabling the Stereoscopy button. See [“Enabling the Stereoscopy Button”](#) on page 501.

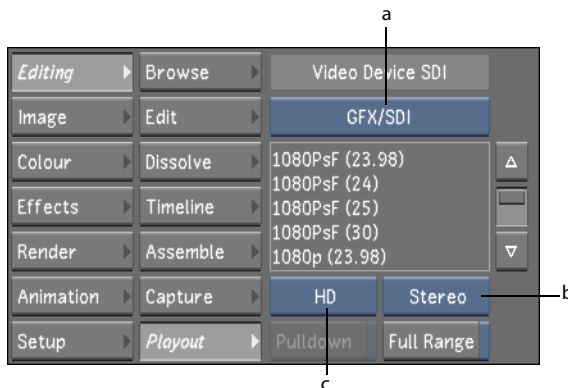
Selecting a Raster for Stereoscopy

The Stereoscopy feature can be enabled when you select a stereoscopic raster from the raster list. 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). Refer to the *Autodesk Lustre Hardware Setup Guide*.

To select a stereoscopic raster:

1. Click Editing in the main menu and then click Playback.
The Playback 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

Stereoscopy is now enabled.

Enabling the Stereoscopy Button

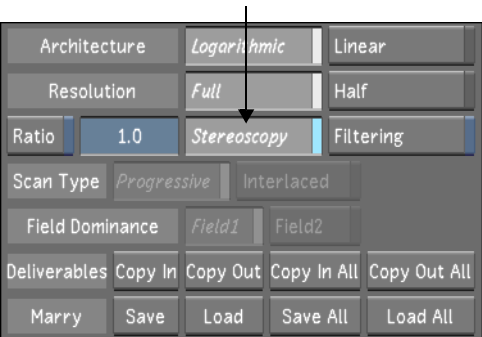
Another way to enable the Stereoscopy feature is by enabling the Stereoscopy button. You would enable the Stereoscopy button when you are grading a Stereo project and you are not outputting to a Stereo device. By enabling the Stereoscopy button, it allows you to grade the Stereo footage without having an output device to see your end results (i.e., you have a standard output).

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 Payout. Within the Payout menu, select a raster from the Video Device SDI group



a) Raster list

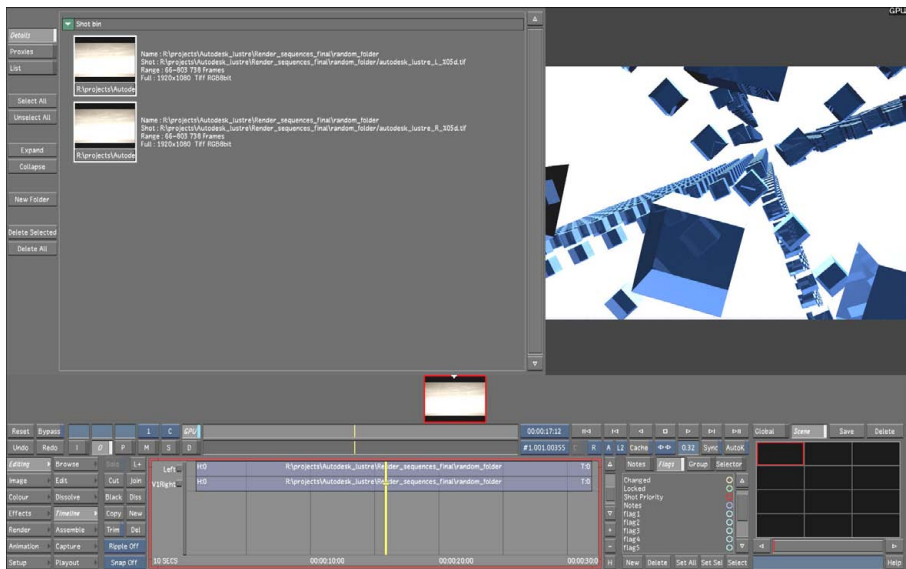
Setting Up the Stereoscopic Footage

After you have enabled the Stereoscopy feature, you need to load the stereoscopic footage and make sure that each layer is assigned to the correct eye.

To load the stereoscopic footage to the timeline:

1. Drag your stereoscopic footage from the folder location to the Shot bin. See, [“Loading Shots into the Shot Bin”](#) on page 134.
2. Click Editing in the main menu and then click Timeline.
The Timeline menu is now displayed.
3. Click L+ to add a second layer to the timeline.
4. Drag and drop your shots from the Shot bin to the timeline. Make sure to drag the Left Eye shots to one layer and the Right Eye shots to the other.

NOTE: In Stereoscopy, there is a limit of one layer for each eye.



5. Save the new cut. See [“Saving a Cut”](#) on page 50.

Assigning the Left and Right Eye Layer

Once a stereoscopic timeline is created, you need to make sure each layer is assigned to the correct eye. You must assign the layer to the correct eye for the following reasons:

- To make sure your stereoscopic footage is viewed properly when the Left and Right Eye are shown together.
- 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.

To assign the Left and Right Eye Layer:

1. Place the timeline focus point on the layer that has the Left Eye footage.

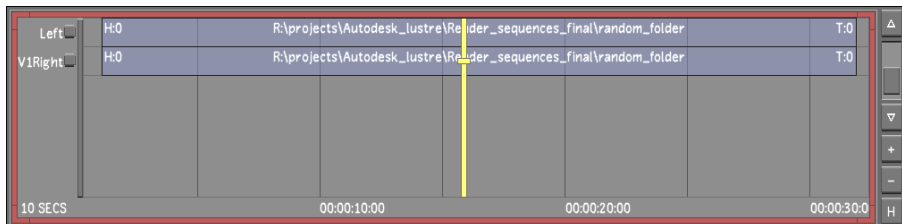
2. Press **ALT+L**.

The layer is now assigned to the Left Eye.

3. Move the focus point to the other layer.

4. Press **ALT+R**.

The layers for the Left and Right Eye are now assigned and the layer names are now V1Left and V1Right.

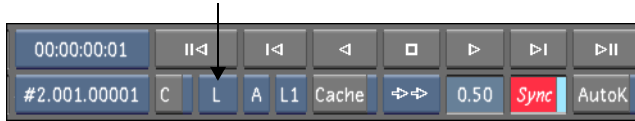


Viewing From One Eye to the Other

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:

- 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 (since interlaced is not supported).

- 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 HDSOI 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.

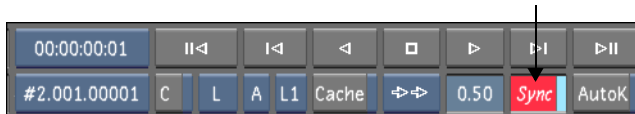
NOTE: The position of the focus point is linked to the layer output selector.

Colour Grading the Left and Right Eye

Since the Player only displays either the Left Eye or Right Eye layer, it is difficult and tedious to grade one layer and create the same grades for the other layer. An easy way for you to make sure any grading is automatically duplicated from one eye to the other is by using the Sync option. When this feature is enabled, any grading you do on one layer is duplicated on the other.

To duplicate the grading from one layer to the other:

- From the Player display controls, click Sync.



Now all grading done on one layer will be duplicated on the other.

NOTE: The Sync option is for colour grading features only and not editing features.

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 layer (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.

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”](#) on page 466.

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 menu”](#) on page 20), 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.

NOTE: Since you are rendering double the amount of shots (both the Left and Right Eye), you may experience some performance issues.

20

Video Capture and Video Playout

Summary

About Capture and Playout	507
Capture Menu Options	508
Controlling the VTR	512
Capturing Material	514
Playout Menu Options	525
Playing Out to a VTR	529
Writing Telecine-Style Tape	541
Playout and GPU Acceleration	541
About 3:2 Pulldown	548
Converting Logarithmic to Linear	549
Supported Video SDI and GFX SDI Rasters	550

About Capture and 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 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 eight audio tracks. In Lustre, audio tracks are identified as A1 through A8. You can select non-consecutive tracks (for example, you can select tracks A1, A5, and A8).

You can capture and play out audio in two different ways:

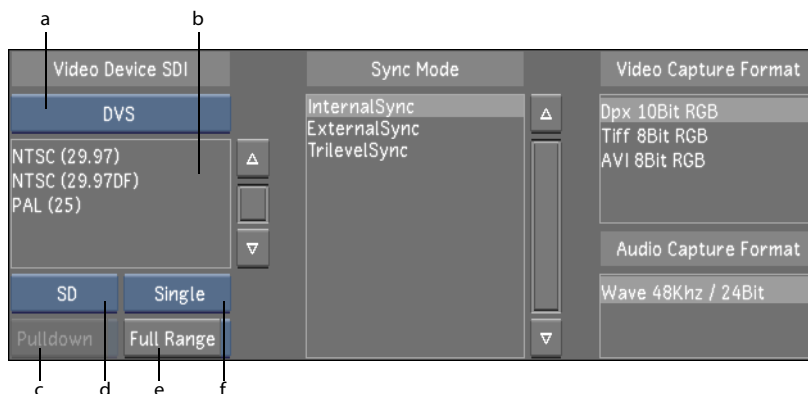
- Audio embedded with video, coming from or going to the SDI connectors ('Embedded' option)
- Audio going through XLR cables from or to the DVS or AJA break out box ('AES' option)

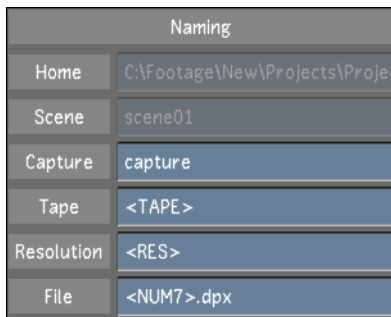
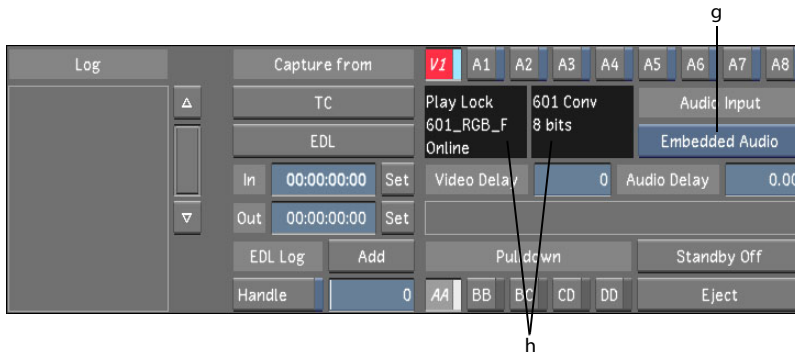
Capture Menu Options

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) Pulldown button
- d) Format option box
- e) Full Range button
- f) Link Type option box
- g) Audio Input button
- h) VTR Status panel

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”](#) on page 550.

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 break out 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).

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.

Pulldown button — Enable to remove 3:2 pulldown from Film-to-video transferred material. See [“Removing Pulldown when Capturing”](#) on page 515.

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 Capture Format list — The supported capture format is a Wave file (.WAV) with a sampling frequency of 48,000 hertz and a bit depth of 24-bit.

Log panel — Displays status messages during capture.

TC button — Initiates a capture from VTR using timecode in and out points. See [“Capturing from Timecode”](#) on page 517.

EDL button — Initiates a capture from VTR using an EDL. See [“Auto-conforming an EDL”](#) on page 522.

NOTE: EDL Capture is video only; audio events are ignored.

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”](#) on page 520.

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.

NOTE: You must enable the Handle button 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 - A8 buttons — Enable the audio tracks (A1-A8) to capture from an audio source.

VTR Status panel — Displays the VTR status messages when Lustre is connected to a VTR.

Audio Input option box — Toggle to specify the source of the captured audio.

Select:	To specify the connection to the audio source:
Embedded Audio	The SDI connection.
AES	The XLR connections on the DVS/AJA break-out 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 frame delays that may exist while transferring shots from the capture to Lustre.

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”](#) on page 548.

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`).

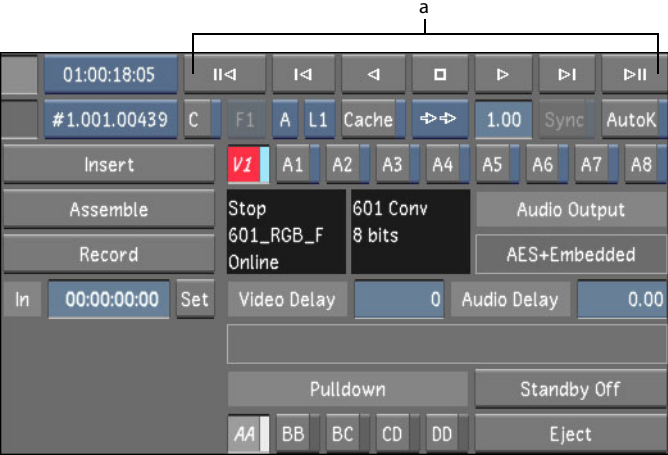
Resolution field — Defaults to the resolution of the captured files, and is used in the folder structure. Do not change this field.

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.
	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 Playback menu. See [“Playing Out to a VTR”](#) on page 529.

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”](#) on page 517.
- Batch digitize from a Log file. See [“Logging and Batch Digitizing”](#) on page 520.
- Batch digitize from an imported EDL. See [“Auto-conforming an EDL”](#) on page 522.
- Perform a live capture from non-controlled sources. See [“Performing a Live Capture from a Non-Controlled Source”](#) on page 524.

For certain video formats, you can remove pulldown when capturing. See [“Removing Pulldown when Capturing”](#) on page 515.

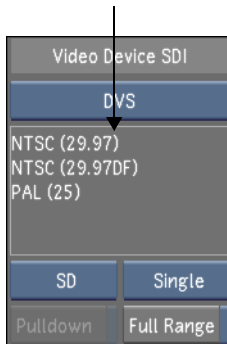
Selecting a Raster for Capture

Whenever capturing material, you must select a raster. For a list of the supported rasters, see [“Supported Video SDI and GFX SDI Rasters”](#) on page 550.

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”](#) on page 508.
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”](#) on page 515,

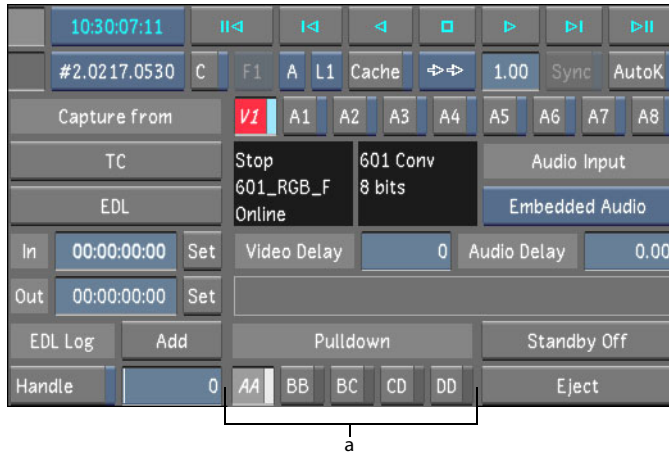
Removing Pulldown when Capturing

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”](#) on page 548.

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

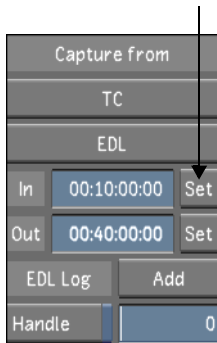
3. Enable the Pulldown Field button that matches the field composition of that frame.

Setting In and Out Points

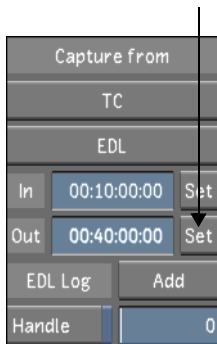
You can set in and out points to select the material to capture from an external video source.

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.

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”](#) on page 508.

Naming	
Home	C:\Footage\New\Projects\Project
Scene	scene01
Capture	capture
Tape	<TAPE>
Resolution	<RES>
File	<NUM7>.dpx

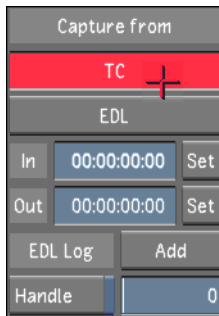
3. Select a raster for capture. See [“Selecting a Raster for Capture”](#) on page 514.
4. In the Sync Mode list, select a synchronization option. See [“Capture Menu Options”](#) on page 508.
5. In the Video Capture Format list, select a file format. See [“Capture Menu Options”](#) on page 508.

Video Capture Format
Dpx 10Bit RGB
Tiff 8Bit RGB
AVI 8Bit RGB
Audio Capture Format
Wave 48Khz / 24Bit

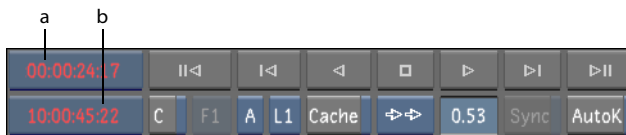
NOTE: The AVI format is only available on Windows workstations.

6. (Optional) Set video and audio capture delays. See [“Capture Menu Options”](#) on page 508.
7. Select the video and audio tracks to capture. See [“Capture Menu Options”](#) on page 508.
8. Set the in and out points. See [“Setting In and Out Points”](#) on page 516.

9. In the Capture from group, click TC. When the button turns red, click it again to confirm the action.

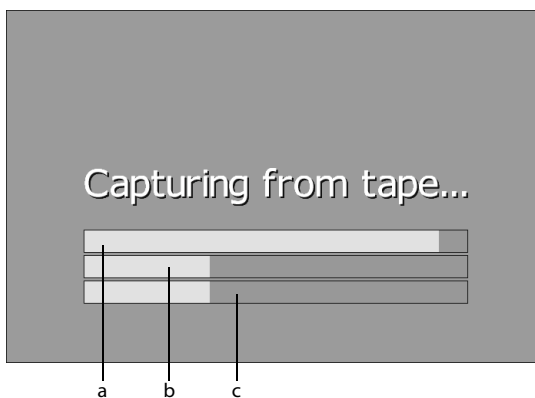


The capture starts. The VTR shuttles back to the in point and begins capturing. The VTR's current timecode and remaining timecode are displayed in a field located to the left of the playback controls. While capturing, both timecodes update and appear in red.



- 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

10. (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.

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”](#) on page 514.
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”](#) on page 508.

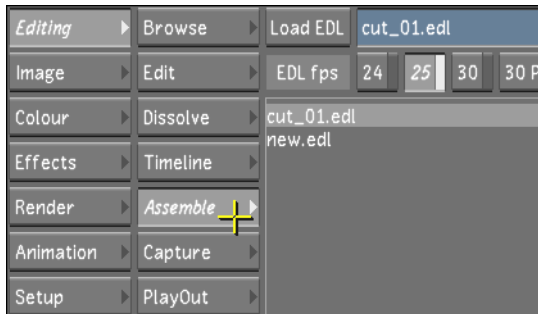


WARNING: It is important that you enter a tape name to avoid overwriting previously captured files.

5. Set the in and out points. See [“Setting In and Out Points”](#) on page 516.
6. In the EDL Log group, click Add to add the selected material to the events list.



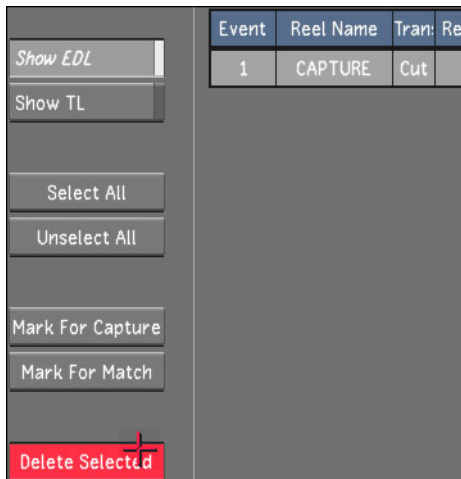
7. To view the events list, click Editing, and then Assemble.



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.

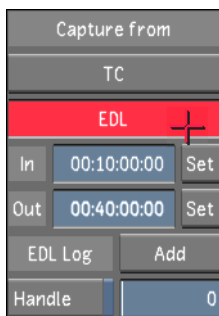
Event	Reel Name	Tran:	Retime	Source In	Source Out	Record In	Record Out
1	CAPTURE	Cut		00:00:00:00	00:00:00:00	00:00:00:00	00:00:00:00

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.



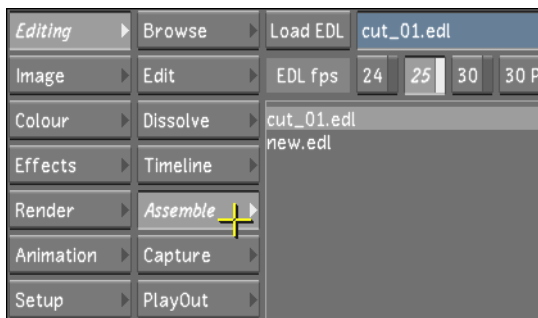
Auto-conforming an EDL

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.

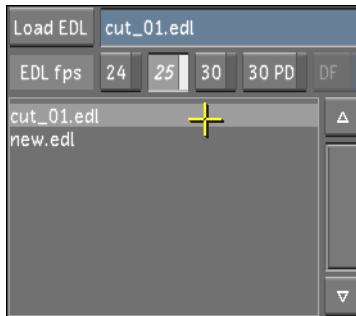
You can batch digitize from an EDL that has been placed in the current scene's *sacc_data* directory. This directory is located under *<HOME>\<scenename>*.

To capture from an EDL:

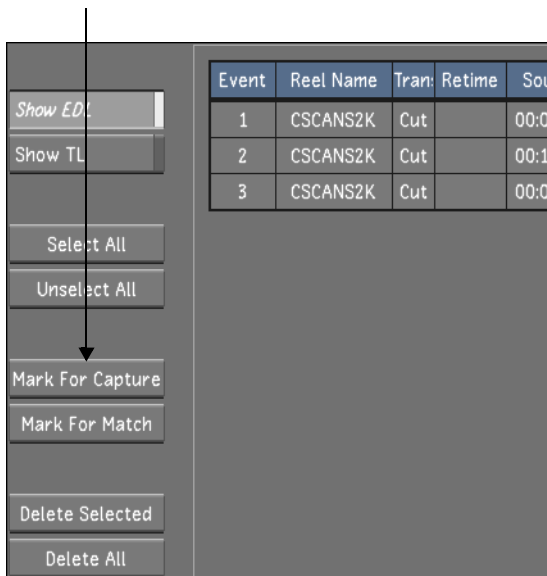
1. Make sure your EDL file is in the scene's *sacc_data* directory.
NOTE: If the file is in the wrong location, it will not show up in the EDL list.
2. Click Editing, and then click Capture.
3. Select a raster for capture. See [“Selecting a Raster for Capture”](#) on page 514.
4. Click Assemble to access the Assemble window.



5. Select an EDL file from the EDL list. **CTRL**-click the EDL list to refresh it.



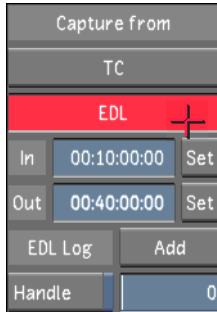
6. Click Load EDL to display the EDL contents in the Assemble menu.
7. Select individual events to capture (**CTRL**+click), a group of events (**SHIFT**+click), or click Select All.
8. Click Mark for Capture.



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”](#) on page 508.

11. In the Naming group, enter values in the Capture and Tape fields. See [“Capture Menu Options”](#) on page 508.
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 Shot bin. See [“Loading Shots into the Shot Bin”](#) on page 134.
14. Go back to the Assemble menu and assemble the EDL. See [“Assembling an EDL”](#) on page 149.
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.

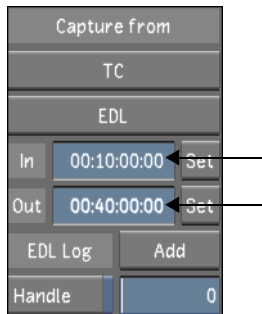
Performing a Live Capture from a Non-Controlled Source

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.

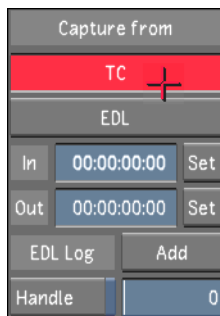
To perform a live capture from a non-controlled source:

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”](#) on page 514.
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.



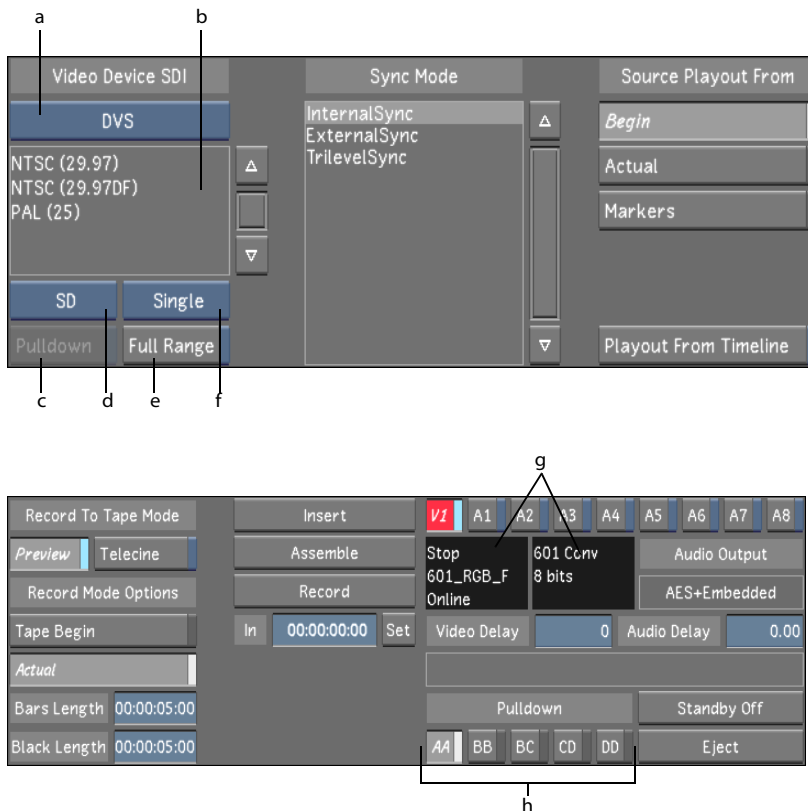
5. To start the capture, **CTRL**+double-click TC.



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) Pulldown button
- d) Format option box
- e) Full Range button
- f) Link Type option box
- g) VTR Status panel
- h) Pulldown Phase buttons

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”](#) on page 550.

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).

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).

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.

Pulldown button — Enable to configure Lustre to add a pulldown frame when playing out. See [“About 3:2 Pulldown”](#) on page 548.

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”](#) on page 531.

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” on page 541.

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”](#) on page 529.

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 - A8 buttons — Enable the audio tracks (A1-A8) to play out.

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 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.

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”](#) on page 466. There are exceptions:

- The creation of real-time deliverables using source material. See [“Creating Real-Time Deliverables from the Timeline or Rendered Material”](#) on page 542.
- GPU playout from the timeline. See [“Play Out from Timeline with GPU Acceleration”](#) on page 547.

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”](#) on page 548.

Playback 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" on page 466.
2. Use the transport controls to cue up the VTR to the in point.	"Controlling the VTR" on page 512.
3. (Optional) Cue up the timeline.	"Navigating through Shots" on page 226.
4. Select a playback mode.	"Playing Out to a VTR" on page 529.

Selecting a Raster for Playback

When playing out to an external device, you must select a raster. See ["Supported Video SDI and GFX SDI Rasters"](#) on page 550.

NOTE: When you select a drop frame (DF) video format, the timecodes are calculated in Drop Frame mode.

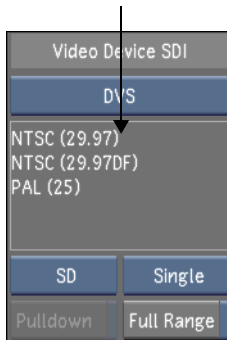
To select a raster for playback:

1. In the Video Device SDI group, toggle the Video/Graphics raster option box to display your video card (AVIO). See ["Playback Menu Options"](#) on page 525.



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 playback.



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”](#) on page 548.

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 > Payout menu that lets you select the marked section for payout 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 Payout to display the Payout menu.

3. Click Markers.



NOTE: If you enable an in point only, payout begins at that point and goes to the end of the timeline. If you enable an out point only, payout 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).

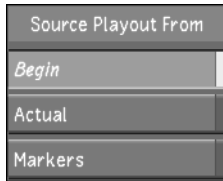
NOTE: When recording to tape, you can only configure specific video and audio tracks to play out if you are in Insert mode.

To record in Insert mode:

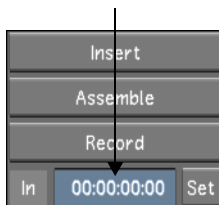
1. Set the VTR to Regen timecode.
2. Before you can play out to a VTR, you must render the timeline at the appropriate video resolution. See [“Rendering Shots”](#) on page 466 and [“Payout Workflow”](#) on page 530.
3. Click Editing, and then click Payout to display the Payout menu.
4. Select a raster for payout. See [“Selecting a Raster for Payout”](#) on page 530.
5. In the Sync Mode list, select the synchronization mode. See [“Payout Menu Options”](#) on page 525.
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”](#) on page 531.

7. In the Source Payout From group, determine where the payout operation starts.

Click:	To start the payout:
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.



HINT: To find the desired timecode more quickly, click the Source Timecode field next to the playback controls while still in the Payout 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 [“Payout Menu Options”](#) on page 525.
11. Configure Lustre to record specific video and audio tracks to tape. See [“Payout Menu Options”](#) on page 525.

- Click Insert, and then click it again to confirm the action.



HINT: 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:

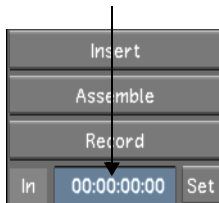
- Set the VTR to Regen timecode.
- Before you can play out to a VTR, you must render the timeline at the appropriate video resolution. See [“Rendering Shots”](#) on page 466 and [“Playout Workflow”](#) on page 530.
- Click Editing, and then click Playout to display the Playout menu.
- Select a raster for playout. See [“Selecting a Raster for Playout”](#) on page 530.
- In the Sync Mode list, select the synchronization mode. See [“Playout Menu Options”](#) on page 525.
- (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”](#) on page 531.

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.



HINT: 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”](#) on page 541.
11. (Optional) Configure video and audio record delay. See [“Playout Menu Options”](#) on page 525.

12. Click Assemble, then confirm the action.



HINT: 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.

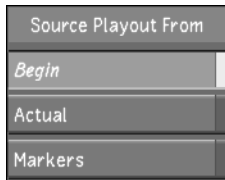
In this case, the Minimum Colourbar Length and the Black Length (which combine to add up to 30 seconds) does not fit into the gap between 00:59:40:00 and 01:00:00:00 (20 seconds). Because video must always start on the minute, additional colour bars are added so that the material starts at the next minute.

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”](#) on page 466 and [“Payout Workflow”](#) on page 530.
2. Click Editing, and then click Payout to display the Payout menu.
3. Select a raster for payout. See [“Selecting a Raster for Payout”](#) on page 530.
4. In the Sync Mode list, select the synchronization mode. See [“Payout Menu Options”](#) on page 525.
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”](#) on page 531.

6. In the Source Playback From group, determine where the playback operation starts.

Click:	To start the playback:
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.

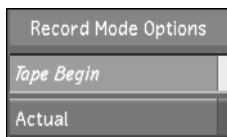


7. (Optional) In the Record To Tape Mode group, enable Preview.



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 playback.
 - Enable Actual to start the playback 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.

Bars Length	00:00:05:00
Black Length	00:00:05:00

11. (Optional) Configure the video and audio record delay. See [“Playout Menu Options”](#) on page 525.
12. Click Record, and then click it again to confirm the action.

Insert		
Assemble		
Record 		
In	00:10:00:00	Set

The playout operation begins.

VTR Emulation

Lustre to emulate a VTR for playback in real time, but not capture. 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	• Reset TC IN	• TC IN+
• Stop	• Reset TC OUT	• TC IN-
• REW	• Mark TC IN	• TC OUT+
• FF	• Mark TC OUT	• TC OUT-
• PREROLL	• Set TC IN	• CUE to (given) TC
• PREROLL TIME PRESET	• Set TC OUT	• CUE to TC OUT
• SHUTTLE	• AUTO mode ON	• EDIT channels preset
• JOG	• AUTO mode OFF	• Eject
• VARPLAY		

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.

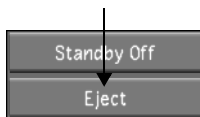


WARNING: Save your grade before editing your project, or you will lose any unsaved work.

3. Launch Lustre and navigate to the Engineering page of the Project Management settings (See [“Engineering menu”](#) on page 27).
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 Playback to display the Playback menu.
8. Select a raster for playback. See [“Selecting a Raster for Playback”](#) on page 530.
9. Prior to playback 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 Playback 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:

- Set the VTR's timecode configuration to:
 - Record run
 - Preset Timecode Generation

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”](#) on page 542.
- Play out from the timeline. See [“Play Out from Timeline with GPU Acceleration”](#) on page 547.

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. See the Lustre 2009 Release Notes for details.

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”](#) on page 546.

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”](#) on page 546.

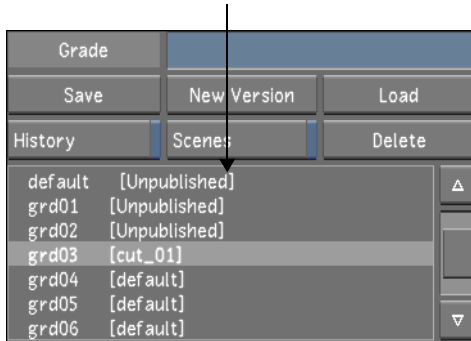
Although real-time deliverables setups are linked to the grade file, you can apply a setup to a grade linked with a different cut.

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.

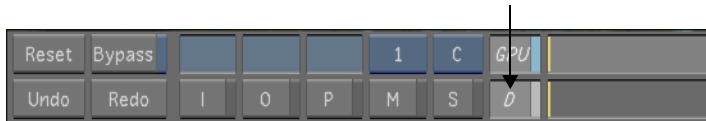
HINT: Enabling the D button also allows you to send a 10-bit buffer to the SDI output.

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.



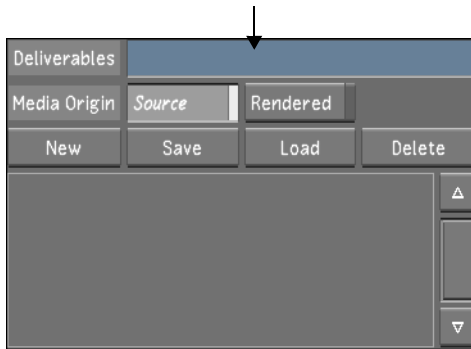
The button automatically switches you to Real-Time Deliverables view mode, and enables GPU processing. See [“GPU Acceleration”](#) on page 295.

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 [Chapter 10, “Repositioning Images,”](#) on page 267.

NOTE: Rotations cannot be included in real-time deliverables setups.

8. (Optional) Perform input primary colour grading. See [Chapter 13, “Primary Colour Grading,”](#) on page 313.

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 Print LUTs for Viewing”](#) on page 127.

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 *sacc_data* 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*.

HINT: 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:

- 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:

- 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”](#) on page 469.

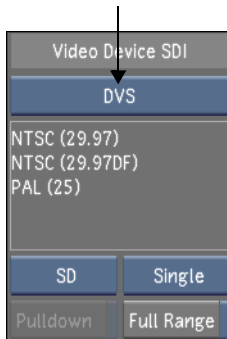
Playing Out Real-Time Deliverables to Tape

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).



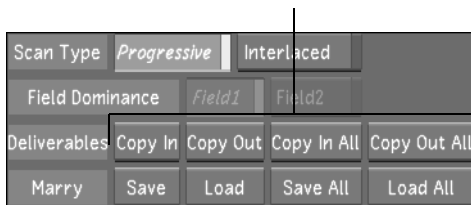
3. Toggle the Link Type option box to the required link type (Single or Dual).
4. Toggle the Format option box to the required raster format (SD, HD, Film, or Audio).
5. In the Raster list, select a raster.
6. Play out to tape in either Insert or Assemble mode. See [“Recording in Insert Mode”](#) on page 532 and [“Recording in Assemble Mode”](#) on page 534.

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:

- 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.

Play Out 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 Payout From Timeline lets you play out source material containing secondary grading without having to first perform a rendering pass.

By enabling the Payout From Timeline function in project settings or in the Payout 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 Payout From Timeline function in project settings, playing out from the Output view is the default for this project. If you enable the Payout From Timeline function from within the Payout menu, this setting applies only for the current session.

- For more information about GPU acceleration, see [“GPU Acceleration”](#) on page 295.
- For more information about Output view mode, see [“Setting Image View Options”](#) on page 222.

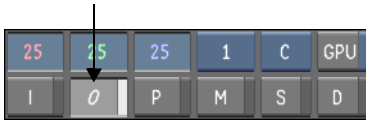
If you want to play out at a different resolution than your timeline, use the real-time deliverable solution. But if you want to play out content with complex grading, use Play out from Timeline, in which case you are limited to using the same resolution as the source.

To play out source material to tape with all features supported by GPU acceleration mode:

1. Enable the Payout From Timeline setting by doing one of the following:
 - In your project settings, enable Payout from Timeline in the Engineering menu. See [“Engineering menu”](#) on page 27.
 - In the Payout menu, enable Payout from Timeline. See [“Payout Menu Options”](#) on page 525.

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”](#) on page 529.

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.

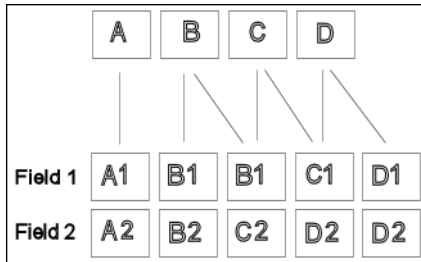
About 3:2 Pulldown

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”](#) on page 54.
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”](#) on page 466.

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”](#) on page 54.
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”](#) on page 466.
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”](#) on page 551 and [“GFX SDI Rasters”](#) on page 553.

In past releases of Lustre, you had to define a colour conversion matrix in a configuration file. Starting with Lustre 2009, 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.
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.

Video Type:	Pull-down:	Link Type:	Video Signal:
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.
1080i (29.97)	Yes	Single	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	29.97 fps, 59.94 Hz, interlaced, 1920x1080, pulldown 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, pulldown 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, pulldown 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.

Video Type:	Pulldown:	Link Type:	Video Signal:
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.

Video Type:	Pulldown:	Link Type:	Stereo:	Video Signal:
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.
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 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

Video Type:	Pulldown:	Link Type:	Stereo:	Video Signal:
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
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

21

Audio

Summary

About Audio	557
Audio Workflow	558
Importing an Audio File	558
Capturing an Audio File	559
Playing Back Audio with the Timeline	559
Playback Using the LTC Chase Option	561
Playout of Audio and Video	562

About Audio

During playback, you can have an audio track accompany the playback. There are a few ways you can hear audio during a timeline playback:

- You can import a WAV file to the system.
- You can capture audio from a video tape recorder (VTR) or audio tape recorder (ATR).
- You can have the timeline 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 break-out-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 eight tracks from the track selector. See [“Capture Menu Options”](#) on page 508 or [“Playout Menu Options”](#) on page 525.

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" on page 558 or "Capturing an Audio File" on page 559.
2. Playback the selected audio file/tracks with the video.	"Playing Back Audio with the Timeline" on page 559 or "Playback Using the LTC Chase Option" on page 561.
3. Playing out the audio file/tracks with the video.	"Playout of Audio and Video" on page 562.

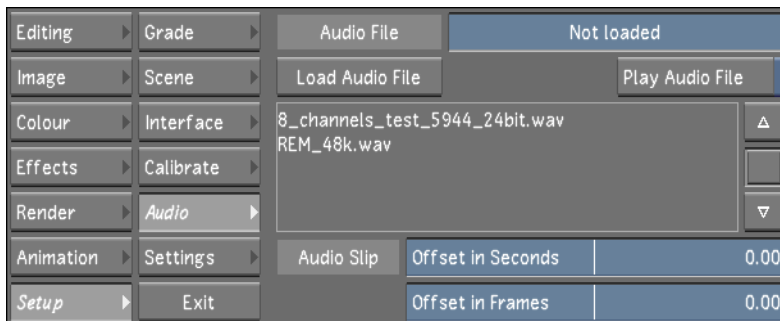
Importing an Audio File

Lustre can use the audio files that you import into the scene's *sacc_data\audio* directory. The imported WAV file can comprise of a single track or up to eight interleaved tracks. The sampling rate of the WAV file is 48 kHz.

NOTE: The audio folder is automatically created when you create a scene.

To import an audio WAV file:

- Copy the audio WAV file into the scene's *sacc_data\audio* folder. If using the Linux version of Lustre, the filename uses forward slashes.



NOTE: If you imported the audio file after launching Lustre, press **CTRL+R** to refresh the audio file list.

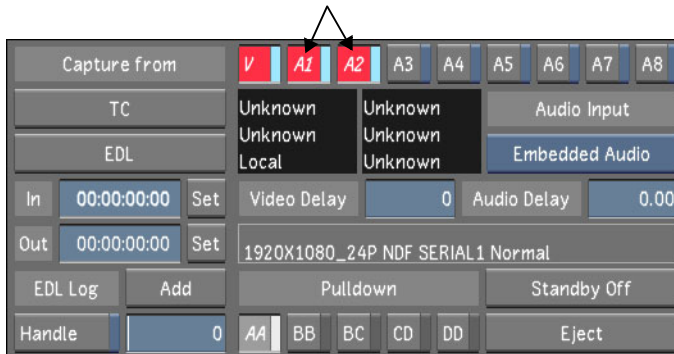
Capturing an Audio File

You can capture an audio signal of up to eight tracks as a single 48 kHz (24-bit), interleaved WAV file. The audio is saved in the scene's *sacc_data\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 file:

1. Click Editing in the main menu, and then click Capture.
2. Select the appropriate raster. See [“Selecting a Raster for Capture”](#) on page 514.
3. Enable the audio track(s) you want to capture.



4. Click the TC button to start the capture.

NOTE: When you capture a broken selection (e.g., tracks A1, A3, and A5), the resulting WAV file contains three tracks, renumbered A1, A2, and A3. Therefore in the Playout menu tracks A1, A2, and A3 will be enabled.

Playing Back Audio with the Timeline

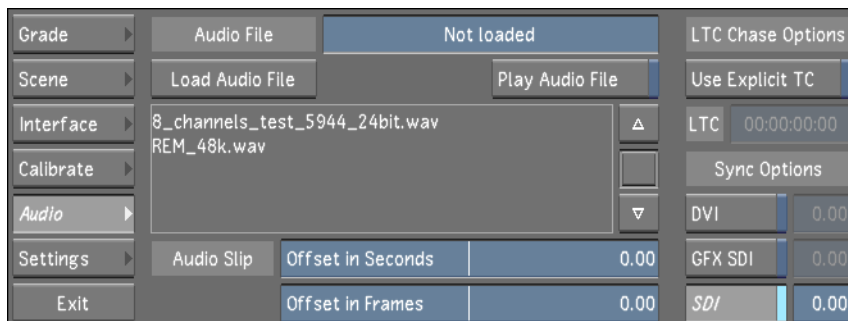
Once the audio file is in the audio folder, you can play it back at the same time as your timeline.

NOTE: The Audio menu settings are saved in the *context.config* file.

To playback an imported audio file with the timeline:

1. Click Setup in the main menu, and then click Audio.

The Audio menu is displayed.



2. Select a file from the list.

NOTE: Only one WAV file can be associated with the timeline.

3. Click Load Audio File.

The selected audio file is displayed in the Audio File field.



4. Enable Play Audio File.

Now when you play the timeline, the audio file is played as well.

NOTE: When you capture an audio file, it is automatically loaded and the Play Audio File button is enabled by default.

Sync Options

When you playback the audio with the timeline, both the audio and video begin at the same timecode. 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.



DVI — Syncs the audio and video when viewing the output on the monitor (DVI).

GFX SDI — Syncs the audio and video when viewing the output on an external device (e.g., projector, broadcast monitor, etc.)

SDI — Syncs the audio and video when viewing the output on an external device connected to the DVS / AJA BOB.

To sync the audio and video:

1. In the Audio menu enable the button corresponding to your output viewing device.
2. 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.

Offsetting the Audio File

Once you have loaded the audio file and synced the audio and video, you may 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:

- 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.

Audio Slip	Offset in Seconds	2.00
	Offset in Frames	6.66

Playback Using the LTC Chase Option

The LTC Chase Option allows you send the timeline’s timecode through the DVS 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.

NOTE: Your LTC Chase Option settings are saved in the *context.config* file.

To play back the audio track using the LTC Chase Option:

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”](#) on page 530.
3. Make sure the audio device is set to chase.

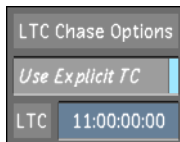
Now when you play back the timeline, the audio track is played at the same time.

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”](#) on page 530.
3. Click Setup in the main menu, and then Audio to display the Audio menu.
4. Enable Use Explicit TC.
5. 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”](#) on page 529.

NOTE: When you capture a broken selection (e.g., tracks A1, A3, and A5), the resulting WAV file contains three tracks, renumbered A1, A2, and A3. Therefore in the Playout menu, tracks A1, A2, and A3 will be enabled.

22

Working with Wiretap

Summary

<i>About Lustre and Wiretap</i>	563
<i>Before You Begin</i>	564
<i>Supported Filesystems and Media Types</i>	566
<i>Sample Workflows</i>	568
<i>Preparing the Wiretap Timeline for Lustre</i>	570
<i>Configuring Wiretap and Lustre</i>	570
<i>Setting Up the Lustre Wiretap Project</i>	574
<i>Browsing for Remote Footage</i>	576
<i>Importing the Wiretap Timeline</i>	580
<i>Working with Soft-Imported Media</i>	580
<i>Rendering Shots</i>	582
<i>Troubleshooting</i>	588

About Lustre and Wiretap

Wiretap is Autodesk's cross-platform client-server interoperability framework, providing Lustre with high-performance access to remote media and metadata. With Wiretap, there is no need to export files from the Autodesk Visual Effects and Finishing application, nor to output to a VTR, prior to grading in Lustre. You can import Visual Effects and Finishing timelines directly from the host application, and read/write clips and frames on remote Visual Effects and Finishing Stone FS and Standard FS storage devices with the same ease as you would local media.

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.

Lustre can interpret a Visual Effects and Finishing timeline as a single one-sequence clip, or as an EDL. When you reference a Wiretap EDL in Lustre, the dissolve and splice transitions are maintained. The Wiretap EDL appears on the Lustre timeline as a series of shots and transitions. Each shot comprising the EDL is added separately to the Shot bin, with DLEDL appended to the shot name. Working with remote timelines containing soft-imported clips is equally possible.

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. 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 rendering local proxies for the remote material.

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:

- **MULTI-LAYER TIMELINES:** Wiretap exposes multi-layer timelines as a single “flattened” layer. There is no need to hard-commit the timeline or merge its layers explicitly.
- **SOFT EFFECTS:** All soft effects and transitions must be processed prior to importing the timeline into Lustre.
- **SUPPORTED TIMELINE ELEMENTS:** Colour sources, timeline gaps, dissolve transitions, and fade-ins and fade-outs are supported in Lustre.
- **DISSOLVES:** These are regenerated in Lustre, but only with linear animation. If a dissolve from a Visual Effects and Finishing application has a different type of curve, or keyframes, this data will not be read in Lustre.
- **UNSUPPORTED TRANSITION TYPES:** Wipe and Axis transitions are unsupported. These are presented as single shots in Lustre.

- **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.
- **FRAME RATES:** The timeline frame rate has to be the same as the Lustre project frame rate.
- **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.
- **PROXIES OFF:** Lustre can only render to Visual Effects and Finishing projects for which proxies are disabled.

Wiretap 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”](#) on page 570 and the *Autodesk Stone and Wire Filesystem and Networking 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. 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 Stone and Wire Filesystem and Networking Guide*.

Supported Image File Types

Lustre can access the following uncompressed image files from a Stone FS or Standard FS:

- 8-bit RGB, which Lustre converts to 8-bit TIFF files
- 10-bit RGB, which Lustre converts to 10-bit DPX files
- 16-bit RGB (for 12-bit uncompressed or 16-bit soft-imported clips), which Lustre converts to 16-bit TIFF files
- Soft-imported clips in their original format

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”](#) on page 568.

Streaming Media Support

With Wiretap 2009, 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 2009. The following compressed file formats are converted on-the-fly by the Wiretap Server:

- Quicktime (no extension)
- Quicktime (.MOV/.mov)
- Media Exchange Format (.MXF/.mxf)

For a list of Lustre-supported native file formats, see [“Accessing the File Browser”](#) on page 131.

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 *stonify* or publish from within the Visual Effects and Finishing application. As a result of server-side video media conversion, this is no longer necessary. Wiretap 2009 is able to present the mixed media timeline.

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”](#) on page 575 and [“Viewing Local Proxies for Wiretap Media”](#) on page 576.

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 Shot bin. You can perform a scene detection on the clip in order to break it down to its component shots. See [“Using Scene Detection”](#) on page 215.

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”](#) on page 471. 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 Edit Decision Lists

Importing an Edit Decision List (EDL) allows you to work with a timeline that contains shots and soft transitions (such as dissolves). In this case, files are first imported into Smoke from the Stone FS and assembled into a timeline. Lustre gains access to the timeline as an EDL by way of Wiretap. This is a useful workflow for shot-to-shot grading.

NOTE: An EDL clip is a modified CMX 3600 EDL with DLEDL flags. The EDL includes information for all the segments in a clip, including the head and tail information for dissolve and splice events. When you load an EDL clip, it auto-assembles, allowing you to bypass the conform step.

In Lustre, you create local proxies and grade them in EDL context. Lustre then renders the final clip to local storage using the One Sequence option in the Render Local menu. See [“Specifying the Destination for Local Render Files”](#) on page 471.

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”](#) on page 471. 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 EDL; 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 using an EDL.
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](#)" on page 564.

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\Lustre2009* (Windows)
- */usr/autodesk/lustre2009* (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, without the need for intervention by 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 Stone and Wire Filesystem and Networking 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”](#) on page 572). 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”](#) on page 572.

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 EDLs by way of Wiretap.

HINT: 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 of the DLEDL in the Shot bin. See [“Viewing Media File Information”](#) on page 139.

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”](#) on page 580. For general considerations on setting up projects, see [Chapter 3, “Project Management,”](#) on page 13.

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
<i>Scans Full Home</i>	• To access a Stone FS or Standard FS, this field should have the same path as <i>Project Home</i>. • 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 <i>F:/SAN/Project/001</i>, the <i>Scans Full Home</i> path should be <i>F:/SAN</i>.
<i>Scans Half Home</i>	• To use proxies coming from a Stone FS or Standard FS, this field should have the same path as <i>Project Home</i>.
<i>Renders Full Home</i>	• 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” on page 582.
<i>Renders Half Home</i>	• 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”](#) on page 566.

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 menu”](#) on page 31). 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

▼ chorley	Autodesk IFFFS 2009.0.0
▶ stonefs	Free: 652.59GB Total: 1.59TB
▶ stonefs3	Free: 120.61GB Total: 144.64GB

b

c

a) Wiretap server workstation name

b) Stone FS volume

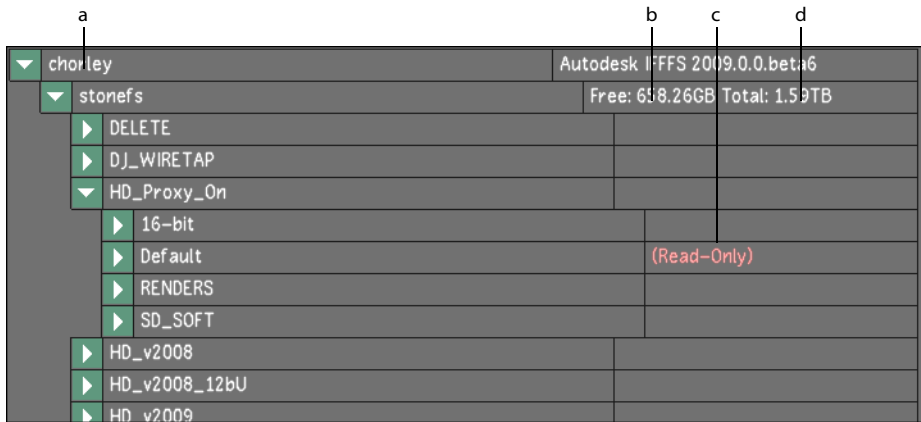
c) Standard FS volume

For details on using the file browser, see [“Accessing the File Browser”](#) on page 131.

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”](#) on page 588.

Viewing the Volume Hierarchy

Expanding a Wiretap volume presents its contents as a hierarchy.



- a) Wiretap server c) Read-Only indicator
 b) Free space on the file system d) Total space on the file system

The following information is presented:

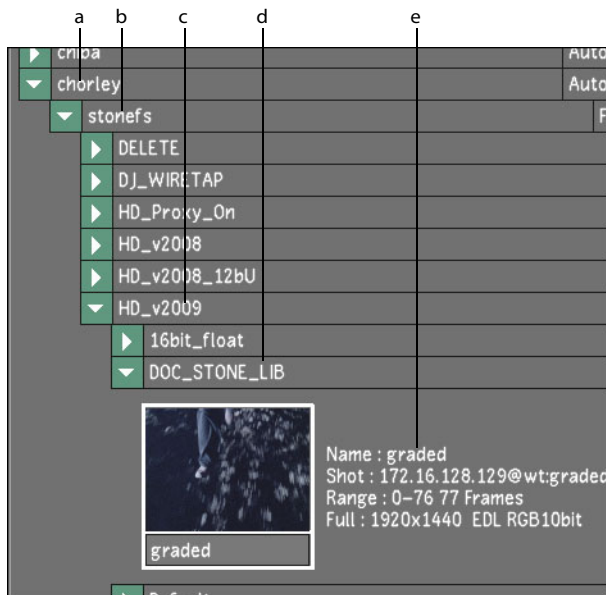
- Wiretap server
- Volume
- Project
- Library
- Reel (optional)
- Clip (in libraries and reels)

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 c) Project e) Clip and Clip Metadata
b) Volume d) Library

Image courtesy of Moviworld / UK Film & TV Company / Videolab

A detailed view of a Wiretap clip is represented in the Shot bin 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”](#) on page 139.

One-Sequence and EDL/Timeline Views

By default, the EDL button in the Browser is enabled, which allows Wiretap content to be presented in a timeline form, also known as an EDL representation. You can also choose to display the one-sequence representation of the Wiretap timeline by disabling the EDL button, in which case both the one-sequence and EDL versions are displayed at the same time.

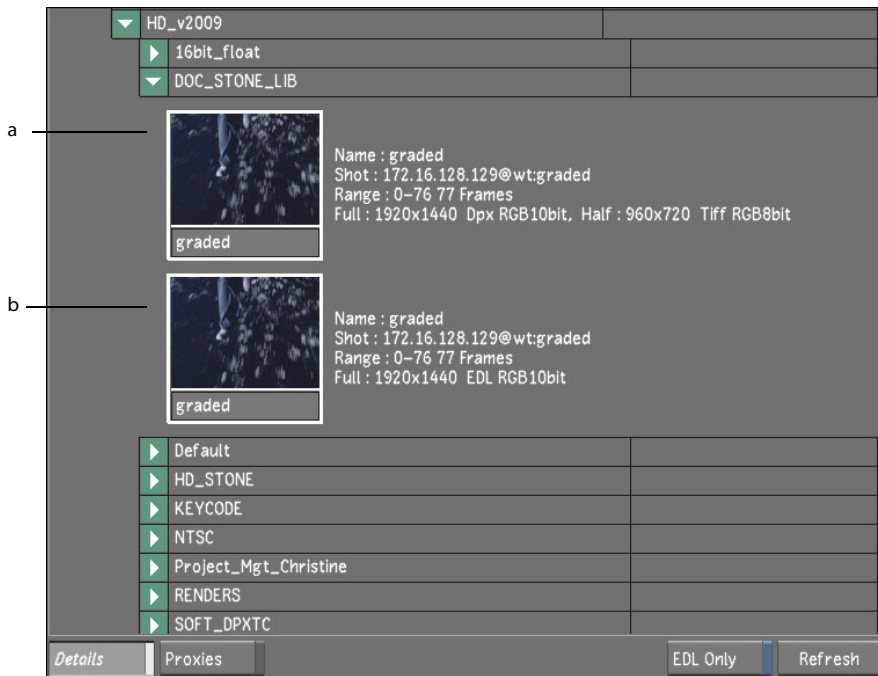


Image courtesy of Moviworld / UK Film & TV Company / Videolab

The difference between the two is as follows:

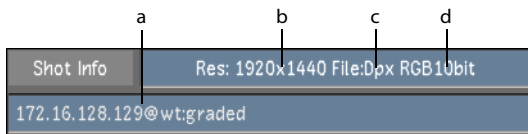
- One-sequence: a sequence of files is presented to Lustre with clip metadata as opposed to timeline metadata and you can still see transitions as 'read-only'. This is useful, for example, when manually comparing the contents of the timeline against a digital cut captured directly from an offline video tape. See [“Performing a Confidence Check”](#) on page 213.
- EDL: 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 EDL. 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”](#) on page 131.

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 EDL 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.
- For long timelines, consider creating a new cut before dragging and dropping a clip from Wiretap. See [“Creating a New Cut”](#) on page 49.
- It is possible to load multiple timelines from Wiretap into different Lustre timeline layers. 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 Shot bin.
- If you drag and drop the Wiretap EDL directly into the Shot bin (as you would for local footage), only the source clips are imported into the Shot bin. The timeline is not created in Lustre.
- 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	c) File format
b) Resolution of the clip	d) Bit depth per channel (RGB)

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:

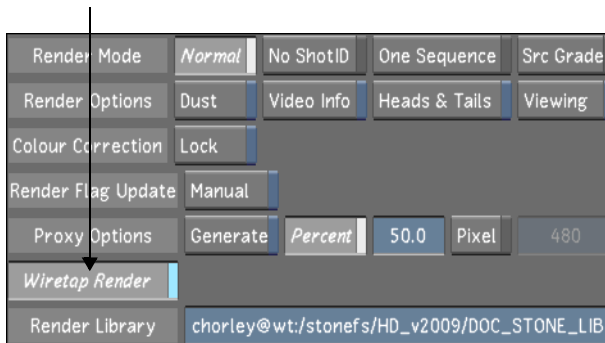
1. Click Setup, and then click Project to display the Project menu.
2. In the Renders Full Home field, enter the path for the graded files to be rendered to.
For example, if your Windows drive is mapped to the network storage as S:\, then the renders full home path should be *S:/SAN/lustre_renders*.

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.

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.

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”](#) on page 574. You render clips from a local storage the same way as you would for any other clip. See [Chapter 18, “Rendering,”](#) on page 465.

Rendering clips back to Wiretap requires that the render destination be specified in the project setup. See [“Setting Up the Lustre Wiretap Project”](#) on page 574. 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”](#) on page 583.

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 *scene_grade* reel will be created in the destination Clip Library that was defined when the project was created. See [Chapter 3, “Project Management,”](#) on page 13.

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>
```

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 (with the extension *_edl*). 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 *scene_grade* reel 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/lustregades* was entered in the Renders Full Home field, the graded rendered shots are placed in *10.145.43.31@wt:/stonefs/tutor/lustregades*. These shots contain the grade name and the shot's unique ID.

grd14_<unique_id> (example: *grd14_4323B1F4*)

An additional clip with an *_edl* suffix is also generated containing the *scene_grd* name. For example, if the grade name is *grd14* from the scene called "interior_003", the additional file is:

interior_003_grd14_edl

The EDL clip is generated based on the new rendered source files. The EDL clip allows you to rebuild the timeline in the editing application based on the new graded source files.

No ShotID Rendering Mode

As outlined in the Normal mode description above, when rendering clips to Wiretap, a *scene_grade* reel is created in the Wiretap Clip Library and all rendered material is placed there. The additional clip with the *_edl* suffix is also generated.

One-Sequence Rendering Mode

As with Normal and No ShotID mode, when rendering clips to Wiretap, a *scene_grade* reel is created in the Wiretap Clib Library and all rendered material is placed there.

One-Sequence mode generates one clip of the graded sequence containing the Lustre grade name. If, in Lustre, your scene is named *interior_003* 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_grd/

The rendered, one-sequence clip will contain the grade name: *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 *scene_grade* reel 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/lustregades* was entered in the Renders Full Home field, the graded rendered shots are placed in *10.145.43.31@wt:/stonefs/tutor/lustregades/scene_grade*. These shots contain the grade name and the shot's unique ID.

grd14_<unique_id> (example: *grd14_4323B1F4*)

An additional clip with an *_edl* suffix is also generated containing the *scene_grd* name. For example, if the grade name is *grd14* from the scene called "interior_003", the additional file is:

interior_003_grd14_edl

The EDL clip is generated based on the new rendered source files. The EDL clip allows you to rebuild the timeline in the editing application based on the new graded 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”](#) on page 574.

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:

- 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”](#) on page 471.

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>/<resolution>/  
<origfilename_grdXX.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_grd02/2046x1556/shot13_grd01.0001.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>/<resolution>/  
<origfilename_grdXX.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 UUIDs 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 UUIDs 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-ptime 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-ptime handles.

Soft-imported clips are rendered as:

```
<HOME>/grd/<sourcereel_grdXX>/<resolution>/<origfilename_grdXX.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):

- 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
<i>sw_start</i>	Starts the Wiretap server
<i>sw_stop</i>	Stops the server
<i>sw_restart</i>	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:

- On the Wiretap server, open a shell and type:

```
ps -ef |grep wiretap
```

Example system response

```
root 19579 1 0 May14 ?      00:00:00 /bin/sh /usr/discreet/sw/sw_wiretapd
root 19588 19579 0 May14 ?    00:01:51 /usr/discreet/wiretap/2009.1/
iffWsWiretapServer -c /usr/discreet/wiretap/cfg/wiretapd.cfg
root 27965 27728 0 10:02 pts/5    00:00:00 grep wiretap
```

In the above output, the line */usr/discreet/wiretap/2009.1/iffWsWiretapServer -c /usr/discreet/wiretap/cfg/wiretapd.cfg* indicates that Wiretap 2009.1 is running. If you do not see information similar to this, restart the Wiretap server.

23

Hot Keys

Summary

Storyboard Navigation	591
View Modes	592
Timeline Sort Modes	594
Notes	594
Calculators	594
Dust Removal	595
Editing	595
Colour Grading	598
Numeric Keypad	602
Stereoscopy	605
Animation	605
Miscellaneous Functions	605

Storyboard Navigation

Use these hot keys 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.

Press:	To:
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)	Toggle through 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.
. (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.
F1	Go to Playhead A.
F2	Go to Playhead B.
F	Toggle between the collapsed Storyboard and the original.

View Modes

Use these hot keys 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 hot key 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 Shot bin 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 vectorscope.
' (apostrophe)	Switch on/off monitor sub-region calculating.
F3	Switch between the number of images in multiple view (you can have 2 to 16 images).
F4	Switch between Vertical and Horizontal Wipe mode.
F5	Switch on/off split view.
F6	Refresh the images in multiple view.
F7	Put the mouse cursor and secondaries handling in the GFX SDI output. It also allows you to view two frames with split screen on the second head.
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 mask of the current secondary. Press F11 again to switch between the grayscale and the saturated mask viewing modes.
F12	Show the result.
E	Show the selected colour's numbering type (code value or F-stop).
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 hot keys 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.
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 hot keys 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 entered in 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 hot keys 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).

Editing

Certain editing hot keys are available from any menu, whereas the rest are available exclusively from the Edit menu.

Global Editing Hot Keys

When you enable the Editing While Grading button in the System & Menu page of the user configuration, you can access the following editing hot keys 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 Hot Keys

Use these hot keys in the Edit menu.

For numeric keypad hot keys that pertain to editing, see [“Editing Menu—Edit”](#) on page 604.

Press:	To:
SHIFT+D	Delete the current shot.
SHIFT+Q	Locate and select the shot in the Shot bin 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 Shot bin.
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 Hot Keys

Use these hot keys 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.

Press:	To:
SHIFT+Q	Locate and select the shot in the Shot bin 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.
.(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).
[	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 Hot Keys

Use these hot keys during the capture process.

For numeric keypad hot keys that pertain to editing, see [“Editing Menus—Capture and Playout”](#) on page 604.

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 Hot Keys

Use these hot keys during the playout process.

For numeric keypad hot keys that pertain to editing, see [“Editing Menus—Capture and Playout”](#) on page 604.

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 hot keys to perform colour grading operations carried out in the Grading, Curves, and Secondaries menus.

For numeric keypad hot keys that pertain to colour grading, see [“Colour Menus — Grading and Secondaries”](#) on page 602.

Global Colour Grading Hot Keys

Use these hot keys 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 GPU processing.
SHIFT+Y	Disable GPU processing.

Press:	To:
CTRL(left)+H	To hide or show the selected geometry.
PAGE UP (Secondaries menu)	Increase the Hue Rotator value.
PAGE UP (other menus)	Go to the Saturation curve in the Curves menu.
PAGE DOWN (Secondaries menu)	Decrease the Hue Rotator value.
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.
/ (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 Hot Keys

Use these hot keys for the Grade bin.

Press:	To:
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.

Curves Menu Hot Keys

Use these hot keys 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 Hot Keys

Use these hot keys in the Secondaries menu.

Press:	To:
1, 2, 3, 4, 5, 6, 7, 8, 9, 0, -, =	Switch between the 12 secondaries' setups.
, (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.

Press:	To:
A	Enable Add Pts, and then add vertices to a geometry.
D	Enable Del Pts, and then delete vertices from a geometry.
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 Hot Keys

Use these hot keys in the Timeline menu.

Press:	To:
SHIFT+C	To make a cut in an element.
SHIFT+J	To join/remove splice on selected element.
SHIFT+D	Delete element(s) from the Timeline.
ALT+S	Enable or disable Solo mode.
UP ARROW	Move the focus up.
DOWN ARROW	Move the focus down.
F7	Displays the Timeline, Player, and grading tools and 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 hot keys vary depending on the menu. The following section includes hot key 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 hot key 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.
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.

Press:	To:
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 hot key 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.
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 hot key 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.
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 Payout

Refer to the following table for numeric keypad hot key functions when working in the Capture and Payout 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.

Press:	To:
* (multiply)	Switch the timecode typing focus to the in timecode.
- (subtract)	Switch the timecode typing focus to the out timecode.

Stereoscopy

Use these hot keys when the Stereoscopy feature is enabled.

Do this:	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.

Animation

Use these hot keys in the Animation menu.

Do this:	To:
Press 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 hot keys are for assorted tasks such as text editing, toggling render settings, and manipulating geometries.

Press:	To:
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.

Press:	To:
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.
] (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

Summary

Lustre XML-Based Files	607
Limitations	607
CUT File Tags	608
CUT File Data Types	617
CUT File Example	618

Lustre XML-Based Files

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)
- CUTINFO (*.cutinfo.xml)
- GRADENOTES (*.desc.xml)

The CUTINFO and GRADENOTES 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”](#) on page 93.

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 *sacc_data* directory and contains the following shot information:

- Physical location of files, trims, and additional metadata information (such as EDL and keycode data)
- 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 Storyboard.

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
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 shot bin 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" on page 617.
Parameters	Type
Parents	EData
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
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
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
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
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
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" on page 617.
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" on page 617.
Parameters	Type

EDLReelName	
Parents	EData
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
EDLSrcTcln	
Description	EDL source in timecode. This tag's parameter is the timecode data type. See "CUT File Data Types" on page 617.
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" on page 617.
Parameters	Type
Parents	EData
Children	N/A
EDLRecTcln	
Description	EDL record in timecode. This tag's parameter is the timecode data type. See "CUT File Data Types" on page 617.
Parameters	Type
Parents	EData
Children	N/A
EDLRecTcOut	
Description	EDL record out timecode. This tag's parameter is the timecode data type. See "CUT File Data Types" on page 617.
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
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" on page 617.
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" on page 617.
Parameters	N/A
Parents	EData
Children	N/A

DLEDLRes	
Description	String from the DLEDL that describes the image format.
Parameters	N/A
Parents	EData
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” on page 617.
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
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
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

Data Type	Description
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.

```
<Cut Ver="3.0">
  <Shot>
    <Seq>E:/footage/my_film_scans/exports_for_lustre/0001d1/1024x778/0001d1.%04d.dpx</Seq>
    <AliasName>0001d1</AliasName>]
    <EData>
      <UniqueId Type="i64">439ED61600000C</UniqueId>
      <SrcFullStart>1</SrcFullStart>
      <SrcFullLength>68</SrcFullLength>
      <SrcTrimStart>1</SrcTrimStart>
      <SrcTrimLength>56</SrcTrimLength>
      <ResStart>0</ResStart>
      <ResLength>56</ResLength>
      <RetimeRefFrame>1</RetimeRefFrame>
      <RetimeSpeed Type="lf">1.200000</RetimeSpeed>
      <RetimeRemain Type="lf">0.000000</RetimeRemain>
      <DPXTimeCodeStart Type="tc">00:00:00:00</DPXTimeCodeStart>
      <DPXCodeRes>1</DPXCodeRes>
      <DPXTimeCodeEnd Type="tc">00:00:02:07</DPXTimeCodeEnd>
      <DPXFrameRate Type="f">30.000</DPXFrameRate>
    </EData>
  </Shot>
  <Shot>
    <Seq>E:/footage/my_film_scans/exports_for_lustre/0001f1/1024x466/0001f1.%04d.dpx*01</Seq>
    <AliasName>0001f1*1</AliasName>
    <EData>
      <UniqueId Type="i64">439ED61600000F</UniqueId>
      <SrcFullStart>1</SrcFullStart>
      <SrcFullLength>241</SrcFullLength>
      <SrcTrimStart>1</SrcTrimStart>
      <SrcTrimLength>241</SrcTrimLength>
      <ResStart>56</ResStart>
      <ResLength>241</ResLength>
      <DPXTimeCodeStart Type="tc">00:00:00:00</DPXTimeCodeStart>
      <DPXCodeRes>1</DPXCodeRes>
      <DPXTimeCodeEnd Type="tc">00:00:08:00</DPXTimeCodeEnd>
      <DPXFrameRate Type="f">30.000</DPXFrameRate>
    </EData>
  </Shot>
</Cut>
```

index

Index

Numerics

10-bit

- recording without rendering 542

3D mesh LUTs

- applying 129
- monitor calibration 118

A

ALE files

- assembling 143
- auto-conforming 522
- copying into Lustre 148
- marking shots for later capture 156

A-mode sort, see Timeline sort mode 244

Analyse tools

- using to view colour distributions 302

analysing, colours 300

animating

- colour grades 306
- dissolves 443
- geometries with the point tracker 390
- geometries with the shape tracker 390
- reposition values 276
- shape of geometries with the point tracker 393
- Sparks 410

animations

- adding from outside the Animation menu 460
- adding keyframes 430
- adjusting duration 459
- adjusting timing 456
- breaking tangent handles 443
- editing keyframes 438

- hiding inactive tangents 445

- hot keys 605

- modifying curves 440

- overview 423

- panning and zooming 426

- resetting tangents 444

- reversing keyframe order 445

- selecting channels 429

- setup options 462

- undoing and redoing 464

- using the channel hierarchy 428

- workflow 428

aspect ratio, setting for the output frame 269

assembling

- EDL files 149

- EDLs, ALE and Cutlist files 143

Audio 561

- playing out audio and video 562

- Use Explicit Timecode 562

audio, importing and offsetting 113

Autodesk control surface, using 88

autograding, shots based on sampled pixels 331

automated dust removal

- overview 254

- performing 258

axes

- adding and deleting from geometries 377

- moving within geometries 368

- viewing in schematic view 379

axes list

- creating a parent axis 378

- deleting an axis 378

- view of geometries and their axes 378

B

- Backburner Manager, starting 491
- Backburner Monitor
 - Linux workstation, accessing from 495
 - monitoring render tasks 493
- background rendering
 - overview 488
 - with the Slave Renderer 488
- batch, digitizing 520
- black clip, adding to the Storyboard 187
- black level
 - adjusting 121
 - modifying for input LUTs 286
- black media, adding to the Timeline 196
- blurring
 - geometries 375
 - keys 356
 - with Sparks 411, 415, 419
- boxes, adjusting for dust removal 261
- brightness
 - adjusting printer lights 317
 - modifying for low, mid, or high levels 318
 - modifying for the entire image 315
 - resetting 319
- building
 - change lists 49
 - cuts 49
- Burn for Lustre
 - overview 490
 - rendering locally 490
 - rendering remotely 492
- bypassing menu, in the UI 90

C

- caching, for real-time playback 249
- calculator
 - applying values to UI elements 103
 - hot keys 594
 - using to calculate timecode 229
- calibrating, see monitor calibration
- cancelling, actions 92
- capturing
 - from a batch list 520
 - from a VTR 514
 - from an EDL, ALE or Cutlist file 522
 - from timecode 517
 - hot keys 604

- performing live from non-controlled source 524
- removing pulldown 515
- shots after the initial conform 156
- supported video formats 550
- video, overview 507

CDL 158

- adding data to EDL file 146
- example workflow 145
- loading data in EDL panel 161
- Offset 159
- panel 159
- parameter in Selector 160
- Power 160
- Slope 159

CDL files

- 143

change lists

- building 49
- creating 52
- deleting 53
- loading 53
- using to track changes 52

- channel hierarchy, using to animate channels 428

channels

- displaying in colour view mode 224
- selecting 429
- selecting for a key 348
- viewing animated 427
- viewing selected animated 427

- cleaning up, render files 487

- C-mode sort, see Timeline sort mode 246

- collapsing, Storyboard 234

colour grading 291

- accessing Colour menus 294
- accessing secondary grading menus 346
- animating grades 306
- concepts 292
- creating continuity grades 294
- curves grading 293, 333
- hot keys 598, 602
- keying ranges of colours to create secondaries 347
- loading presets 305
- primary grading 293, 313
- saving presets 303
- secondary grading 294, 345, 402

- selecting Logarithmic or Linear mode 293
 - using control surfaces 299
 - using printer lights 294
 - workflows 292
 - colour space
 - rendering 478
 - setting for projects 54, 123
 - UI differences between Logarithmic and Linear mode 55
 - colourimeter, calibrating a monitor 124
 - colourist
 - Timeline 307
 - colours
 - adjusting for geometries 375
 - analysing 300
 - comparing when creating conversion LUTs 284
 - sampling values 301
 - viewing distributions with the Analyse tools 302
 - Command Line Renderer
 - example 498
 - rendering with 495
 - syntax 495
 - comparing
 - shots using a reference 299
 - shots using multi-view 299
 - confidence check, performing 213
 - confirming, actions 92
 - continuity grades, overview 294
 - continuity, matching colours between shots 330
 - contrast
 - modifying in Linear mode 328
 - modifying in Logarithmic mode 320
 - setting the pivot point 321
 - control surfaces
 - using 88
 - using for colour grading 299
 - controlling
 - VTR 512
 - converting
 - data with input LUTs 280
 - from logarithmic to linear 281
 - Copying
 - keyframes 449, 451
 - copying
 - EDL, ALE, and Cutlist files into Lustre 148
 - parameters with the Selector 101
 - shapes and grades 405
 - creating
 - change lists 52
 - cuts 49
 - projects 15
 - scenes 47
 - users 33
 - credits, adding to the timeline 113
 - cropped frames, repositioning 268
 - curves
 - adjusting saturation 339
 - colour grading 333
 - measuring for monitor calibration 125
 - modifying lightness 342
 - modifying luminance 340
 - modifying RGB channels 336
 - plotting colour values 334
 - shifting hue 338
 - curves colour grading
 - hot keys 603
 - overview 293
 - view mode 222
 - CUT files
 - data types 617
 - example 618
 - modifying 608
 - cut point, sliding between shots 179
 - Cutlist files
 - assembling 143
 - auto-conforming 522
 - copying into Lustre 148
 - marking shots for later capture 156
 - cuts
 - adding overlays 67
 - building 49
 - changing without affecting grade settings 50
 - creating new 49
 - deleting 51
 - inserting between shots 180
 - loading 50
 - managing 50
 - saving 50
 - tracking changes 52
- ## D
- deleting

- change lists 53
- cuts 51
- geometries 382
- grades 61
- keyframes 448
- shots 171
- shots from Shot bin 138
- destination, specifying render files for 583
- developing, Sparks 421
- directories, setting up for projects 78
- disabling
 - dissolves 187
 - reframing settings 272
 - Sparks 420
- dissolves
 - adding between shots 184
 - adding in the Timeline 203
 - animating 443
 - changing duration 185
 - changing orientation in the Timeline 204
 - changing speed 186
 - creating 184
 - creating additional handles for 186
 - disabling and re-enabling 187
 - editing 186
 - editing in the Timeline 204
 - in the Timeline 203
 - overview 184
 - removing in the Timeline 204
 - retrimming 186
- documentation
 - conventions 5
 - library, on the Autodesk Web site 5
 - set of guides 2
- drawing
 - basic geometries 363
 - free-form geometries 364
 - split screens 376
- dust removal
 - choosing a repair type 254
 - choosing manual or automated 254
 - dealing with problem dust artefacts 260
 - drawing boxes 261
 - hot keys 595
 - optimizing parameters 255
 - overview 253
 - performing automated 258

- performing manual 257
- rendering 264, 475
- saving metadata 263
- using the infrared channel 262

E

- edges, modifying for keys 356
- editing
 - accessing the edit menu 166, 576, 582
 - duplicating shots in the Storyboard 170
 - frames by trimming 171
 - hot keys 168, 595, 604
 - in and out points interactively 177
 - in Timeline 193
 - overview 165, 563
 - shot order in the Storyboard 170
 - shots from the Storyboard 171
 - shots from the Timeline 171
 - start timecode 212
 - with the Storyboard 168
 - XML-based files 607
- EDL files
 - assembling 143, 149
 - assembly workflow 145
 - auto-conforming 522
 - copying into Lustre 148
 - EDL display and functionality 144
 - exporting 154
 - marking shots for later capture 156
- effects, creating with Sparks 408
- Elements
 - trimming 198
- exporting, EDL files 154
- extrapolation, setting 436

F

- field-based processing, setting the active field 224
- file browser, accessing 131
- file formats, supported for playback 134
- flagging shots 94
 - automatically 468
 - for rendering 466
 - manually 467
- flipping, shots 272
- flopping, shots 272
- format, setting the output for render files 477
- full-resolution images, displaying 223

- full-screen Player
 - activating 221
 - panning and zooming 222
 - using hot keys 250

G

- gain, modifying 326
- gamma
 - adjusting for the monitor manually 127
 - measuring for monitor calibration 124
 - modifying 326
 - modifying for input LUTs 287
- garbage masks, loading setups 401
- generating, proxies 484
- geometries
 - adding and deleting axes 377
 - adjusting shapes 369
 - animating with the point tracker 390, 393
 - animating with the shape tracker 390
 - assigning tracking data 395
 - blurring 375
 - changing priority order 380
 - copying shapes and grades 405
 - creating 361
 - creating a parent axis 378
 - deleting 382
 - deleting an axis 378
 - deleting point tracker data from 399
 - drawing basic 363
 - drawing free-form 364
 - drawing split screens 376
 - grading inside and outside 404
 - inverting 381
 - linking across secondaries 383
 - modifying 368
 - moving axes 368
 - setting colour and opacity 375
 - showing and hiding 381
 - softening edges 372
 - soloing and ganging point tracker 397
 - transforming 371
- GPU preview processing
 - GPU compatibility 298
 - GpuNoAutoSwitch parameter 298
 - playing out from source with 547
 - real-time 295
- Grade bin 105

- applying intermediary grades 108
 - deleting intermediary grades 111
 - saving intermediary grades 105
- grade bin storage 307
- Grade menu, selecting a scene 48
- grades
 - applying from the Grade bin 108
 - applying to a new assembled timeline 152
 - copying unrendered changes to real-time deliverables 546
 - deleting 61
 - deleting from the Grade bin 111
 - emulating telecine-style tape 541
 - loading backup copies 60
 - locking 486
 - maintaining when swapping cuts 50
 - saving and loading for single shots 61
 - saving to the Grade bin 105
- grouping
 - applying changes to shots as a group 99
 - creating a shot group 97
 - soloing shots in a group 100

H

- head frames, rendering 477
- help, accessing online 3
- hiding, Storyboard 231
- History list, loading for a grade 60
- hot keys
 - accessing editing hot keys globally 168
 - assigning notes to shots 594
 - capturing and playing out with the numeric keypad 604
 - colour grading 598, 602
 - creating animations 605
 - curves colour grading with the numeric keypad 603
 - editing 595
 - editing with the numeric keypad 604
 - for Player and full-screen Player 250
 - miscellaneous functions 605
 - navigating the Storyboard 591
 - navigating with the playback controls 227
 - numeric keypad 602
 - removing dust 595
 - switching view modes 592
 - using calculators 594

hues

- shifting for secondaries 403
- shifting with curves 338

I

infrared channel, using for dust removal 262

input LUTs

- applying existing 280
- comparing colour values 284
- creating conversion LUTs 281
- modifying gamma of film footage 287
- modifying reference black and white levels 286
- modifying reference soft clip values 288
- overview 279
- viewing reference images 282

installing, Sparks 408

Interface menu, customizing Lustre's appearance 67

interlaced video, setting the active field 224

interpolation, setting 433

inverting

- geometries 381
- keyframe values 447
- keys 357

J

joining, shots together 181

K

keycode

- assembly 154
- assembly workflow 145

Keyframes

- copying 451
- copying and pasting 449
- geometries 456
- hot keys 456
- pasting 453, 455
- plugins 456
- point tracker 456

keyframes

- adding 430
- adding from outside the Animation menu 460
- deleting 448

editing 438

inverting values 447

locking movement 464

modifying curves 440

navigating between 425

reversing order 445

scaling to adjust animation duration 459

selecting 439

setting extrapolation 436

setting interpolation 433

viewing channel ranges between 427

viewing selected 427

keys

creating to isolate ranges of colours 347

excluding from secondaries 358

importing mattes 360

inverting 357

removing stray pixels from 357

saving and loading presets 359

selecting channels 348

setting initial softness and tolerance 349

setting softness ranges 353

setting tolerance ranges 354

shrinking edges 356

viewing results 351

L

letterbox, scaling 267

levels, selecting in the image for colour grading 323

licensing

- options for Lustre 2
- Sparks 408

lift, modifying colour values 324

lightness, modifying using the Lightness curve 342

limitations, XML-based files 607

Linear mode

defining luminance 329

improving contrast 328

modifying gain 326

modifying gamma 326

modifying lift 324

modifying saturation 327

monitor calibration 120

using in Lustre 293

Linear mode, selecting levels 323

linking, geometries across layers 383

- loading
 - change lists 53
 - cuts 50
 - projects 15
 - shots to the Shot bin 134
 - shots to the storyboard 139
 - users 33
- locking
 - colour grade 486
 - keyframe movement 464
- Logarithmic mode
 - adjusting printer lights 317
 - improving contrast 320
 - modifying brightness for low, mid, or high levels 318
 - modifying brightness for the entire image 315
 - modifying saturation 322
 - monitor calibration 120
 - resetting brightness 319
 - setting the pivot point 321
 - using in Lustre 293
- lookup tables, *see* LUTs
- LTC Chase Option 561
 - Use Explicit Timecode 562
- luminance
 - modifying in Linear mode 329
 - modifying using the Luminance curve 340
- Lustre
 - configurations 2
 - exiting 85
 - starting 85
- LUTs
 - 3D mesh 118
 - monitor calibration 118
 - See also input LUTs
 - viewing 225
- M**
 - manual dust removal
 - overview 254
 - performing 257
 - Marry controls, saving the grade for a single shot 61
 - matching
 - colours between shots 330
 - grade settings with a new assembled timeline 152
 - shots between the storyboard and Shot bin 142
 - mattes, importing 360
 - metadata, saving for dust removal 263
 - monitor calibration 117
 - 3D mesh LUTs 118, 129
 - adjusting black level and picture settings 121
 - adjusting manually 127
 - calibrating 123
 - curves 125
 - for dual-head mode 127
 - gamma 124
 - Logarithmic and Linear mode 120
 - LUTs 118
 - Print LUTs 127
 - reverting to previous settings 126
 - selecting project colour space 123
 - using a colourimeter 124
 - workflow 120
 - monitoring location of current frame 228
 - moving, geometries 371
 - Multi-Layer Timeline
 - muted layers 206
 - shot priority 207
 - solo mode 211
 - Top Vertical Priority 205
 - Multi-Layer Timeline, *See* Timeline 188
 - multi-view, viewing in the Player or SDI monitor 240
 - Muted layers, *see* Multi-Layer Timeline 206
- N**
 - navigating
 - between keyframes 425
 - the Storyboard 226, 230
 - using the timebar and Storyboard 227
 - with playback controls and hot keys 227
 - with Timeline navigation controls 191
 - noise, adding with Sparks 411, 416
 - notes
 - assigning to shots 93
 - hot keys 594
- O**
 - online help, accessing 3
 - opacity, adjusting for geometries 375
 - output LUTs

- creating 479
 - selecting 480
- output, view mode 222
- overlays, adding to cuts 67
- P**
- panning
 - animation curves 426
 - in Player and full-screen Player 222
- parameters
 - bypassing 90
 - optimizing for dust removal 255
 - resetting 89
 - setting for projects 33
- Pasting
 - keyframes 453, 455
- PDF documentation, accessing 4
- picture settings, adjusting 121
- Pixel Analyser, sampling images 301
- pixels, removing from keys 357
- playback, changing speed 182
- Player
 - activating full-screen 221
 - displaying the user interface 225
 - setting to split or multi-view 240
 - using hot keys 250
 - using Split View 238
 - viewing multiple shots 238
- playheads
 - assigning reference images 238
 - assigning user interface to active playhead 225
- playing
 - shots in the Player 219
 - shots rendered with the Slave Renderer 490
- playing out
 - defining in and out points when 531
 - from source with GPU preview processing 547
 - hot keys 604
 - in Assemble mode 534
 - in Insert mode 532
 - in Record mode 536
 - real-time deliverables 545
 - supported video formats 550
 - to a VTR 529
 - video, overview 507
 - workflow 530
- Playout
 - audio 562
- plotting, sampled colour values 334
- Plugins, see Sparks
- point tracker
 - animating geometries with 390
 - animating the shape of geometries 393
 - deleting 399
 - loading data 400
 - soloing and ganging 397
- post-production pipeline
 - integrating Lustre into 568
- presets
 - colour mapping, saving and loading 482
 - for real-time deliverables 542
 - loading colour grading parameters 305
 - saving and loading for keys 359
 - saving colour grading parameters 303
- primary colour grading
 - applying secondaries to 405
 - overview 293, 313
 - view mode 222
- Print LUTs, applying preset gamma correction 127
- print, view mode 222
- printer lights
 - accessing the calculator from 103
 - adjusting brightness 317
 - adjusting for secondaries 404
 - using in Log mode 294
- processing, image pipeline order 8
- projects
 - creating and loading 15
 - creating the directory structure 78
 - setting colour space architecture 54
 - setting parameters 33
 - setting the resolution 65
 - setting up for soft-imported clips 574
- proxies
 - displaying 223
 - generating 484
 - viewing 300
- pulldown
 - removing for capture 515

R

- racking, to reframe footage 270
- rdesktop, monitoring results 495
- real-time deliverables
 - copying changes to or from the grade file 546
 - creating presets 542
 - overview 542
 - playing out 545
 - resizing 545
- real-time playback
 - using GPU preview processing 295
 - with frame caching 249
- recording
 - Assemble mode 534
 - Insert mode 532
 - Record mode 536
- redoing
 - actions 91
 - in animation 464
- reference images
 - assigning to playheads 238
 - viewing 236
 - viewing results of logarithmic to linear conversion 282
- reference, setting the view mode 222
- reframing
 - aspect ratio output frame 269
 - disabling settings 272
 - flipping and flopping shots 272
 - racking the image 270
 - using reposition controls 271
- regions, rendering selected 481
- removing, dust and other artefacts 253
- render files, removing 487
- rendering
 - BackBurner Manager, starting 491
 - burning in timecode 476
 - colour space, setting 478
 - dust removal data 264, 475
 - file destination, specifying 583
 - flagging shots automatically 468
 - flagging single shots manually 467
 - head and tail frames 477
 - in the background 488
 - locally with Burn for Lustre 490
 - monitoring with Backburner Monitor 493
 - output format, specifying 477
 - output LUTs, creating 479
 - output LUTs, selecting 480
 - overview 465
 - presets for colour mapping, saving and loading 482
 - region, specifying 466
 - remotely with Burn for Lustre 492
 - resolution, setting 468
 - selected region 481
 - setting quality for repositioned shots 277
 - shots 466
 - Timeline 212
 - with Burn for Lustre 490
 - with the Command Line Renderer 495
 - with viewing LUTs 476
- repair type, selecting for artefact removal 254
- replacing, shots in the Storyboard 183
- repositioning
 - after adjusting the project aspect ratio 271
 - cropped frames 268
 - interlaced footage using Sparks 414
 - setting rendering quality after 277
 - using animation controls 276
 - with the tracker 273
- resetting
 - parameters 89
 - secondaries 382
- resizing, real-time deliverables 545
- resolution
 - display 223
 - project 65
 - rendering 468
- retiming, playback speed for shots 182
- reverting to previous, calibration settings 126
- RGB channels, adjusting individually with curves 336
- rotating, geometries 371

S

- sampling, pixels for autogradeing 331
- saturation
 - adjusting with curves 339
 - modifying in Linear mode 327
 - modifying in Logarithmic mode 322
- saving
 - cuts 50
- scaling

- geometries 371
- letterbox format 267
- Scene menu, selecting a scene 48
- scenes
 - creating 47
 - detecting 215
 - selecting 48
- schematic view, toggling to 379
- SDI monitor
 - viewing multiple shots 241
- SDI monitor, setting to split or multi-view 240
- secondary colour grading
 - accessing grading menus 346
 - adjusting printer lights 404
 - applying Sparks 410
 - applying to the entire image 405
 - changing layer order 380
 - creating geometries 361
 - excluding keys 358
 - grading outside the secondary 404
 - keying ranges of colours 347
 - loading garbage mask setups 401
 - loading tracker and stabilizer data 400
 - overview 294, 345
 - performing colour grading 402
 - resetting 382
 - shifting hue 403
 - showing and hiding geometries 381
 - turning on or off 406
 - view mode 222
 - viewing key results 351
- selecting, keyframes 439
- selective colour grading, see secondary colour grading
- Selector
 - CDL parameter 160
- Selector, copying parameters 101
- setup options, Animation menu 462
- shape tracker
 - animating geometries with 390
 - point cloud 385
 - region of interest 385
 - tracking through translation, rotation, scaling, partial occlusions 385
- sharing, changes between grades and real-time deliverables 546
- Shot bin
 - deleting shots from 138
 - loading shots from the file browser 131, 134
 - selecting shots for the Storyboard 169
- Shot priority, see Multi-Layer Timeline 207
- shots
 - adding dissolves 184
 - adding in Timeline 193
 - adding to the Storyboard 169
 - applying changes to a group 99
 - assigning notes 93
 - comparing multiple in the Player 299
 - comparing using a reference 299
 - deleting 171
 - deleting from Shot bin 138
 - dividing into scenes 215
 - duplicating in the Storyboard 170
 - flagging 94
 - grouping 97
 - identifying selection colours on the Storyboard 233
 - joining back together 181
 - matching between the storyboard and Shot bin 142
 - moving to the storyboard 139
 - playing in the Player 219
 - rendering 466
 - reordering in the Storyboard 170
 - replacing in the Storyboard 183
 - selecting in the Storyboard 231
 - setting to half resolution 300
 - slipping in the timeline 176
 - soloing 100
 - trimming head or tail frames 173
 - viewing information 139, 172
- size, setting the Storyboard view 231
- Slave Rendering
 - background rendering 488
 - playing shots rendered with 490
- sliders, modifying with the calculator 103
- sliding, cut point in a shot 179
- slipping, shots in the timeline 176
- softclip values, modifying for input LUTs 288
- softness
 - applying to geometries 372
 - setting initial for keys 349
 - setting ranges for keys 353
- Solo mode, see Multi-Layer Timeline 211

- Sparks
 - animating 410
 - applying to secondaries 410
 - creating effects with 408
 - descriptions 411
 - developing 421
 - disabling and re-enabling 420
 - installing 408
 - licensing 408
 - overview 407
 - using to add noise 411, 416
 - using to blur images 411, 415, 419
 - using to reposition interlaced footage 414
- speed
 - dissolves 186
 - playback 182
- splicing
 - by adding a shot to the Timeline 202
 - shots 180
- split screens, drawing 376
- split view, activating in the Player or SDI monitor 240
- Stone filesystem
 - playing clips located on 564
- Storyboard
 - adding a virtual clip 187
 - adding shots 169
 - collapsing 234
 - deleting shots from 171
 - duplicating shots in 170
 - editing shots 168
 - hot keys 591
 - inserting cuts between scenes 180
 - joining shots together 181
 - navigating 226, 230
 - navigating using Timebar and Storyboard 227
 - reordering shots 170
 - replacing shots 183
 - selecting shots 231
 - selection colours 233
 - setting view size 231
 - showing and hiding 231
 - sliding the cut point between shots 179
 - trimming frames interactively 177
 - updating thumbnails 233
 - viewing shot information 172
- supported file formats, in Lustre 134
- syntax
 - example 498
 - for the Command Line Renderer 495
- T
- tail frames, rendering 477
- Tangent CP100 control surface, using 88
- tangents
 - adjusting to reshape geometries 369
 - breaking for animations 443
 - hiding handles 445
 - resetting broken handles 444
- telecine tape
 - emulating grading with 541
- thumbnails, updating in the Storyboard 233
- timecode
 - burning into render files 476
 - calculating 229
 - modifying the start value 212
 - viewing information 172
- Timeline
 - accessing the Timeline menu 188
 - adding a dissolve in 203
 - adding black media to 196
 - adding shots to 193
 - changing orientation of a dissolve in 204
 - checking against an offline cut 213
 - colourist 307
 - copying elements in 195
 - creating 193
 - deleting elements in 197
 - deleting shots from 171
 - displaying Multi-Layer Timeline 205
 - dissolves 203
 - editing 193
 - editing a dissolve in 204
 - gaps, overview 197
 - modifying the starting timecode 212
 - moving elements in 194
 - navigation controls 191
 - removing a dissolve in 204
 - rendering 212
 - selecting elements in 193
 - sliding an element 200
 - slip-and-slide Ripple Mode 201
 - slipping an element 199

- slipping-and-sliding an element 201
- sort mode 242
- splices 202
- transitions 202
- trimming cursors 198
- trimming with Ripple Mode 198
- view in Lustre 188
- viewing element information 192
- Timeline menu, accessing 188
- Timeline sort mode 242
 - accessing Timeline sort mode 242
 - A-mode 244
 - C-mode 246
 - hot keys 245, 248
- timing, adjusting for animations 456
- tolerance
 - setting initial for keys 349
 - setting ranges for keys 354
- Top Vertical Priority, see Multi-Layer Timeline 205
- tracking
 - geometries 395
 - repositioned shots 273
- trimming
 - cursors in the Timeline 198
 - elements 198
 - frames in the timeline 171
 - head or tail frames 173
 - in and out points 178
 - interactively 177
 - shots for dissolves 186
 - with Ripple Mode in the Timeline 198
- troubleshooting, problem dust artefacts 260

U

- undoing
 - actions 91
 - in animation 464
- user interface
 - differences in Logarithmic and Linear mode 55
 - setting appearance of 67
- users, creating 33

V

- versioning, for film and video 549
- video

- capture and payout, overview 507
- capturing from timecode 517
- formats, supported 550
- outputting multiple versions 549
- view modes
 - hot keys 592
 - setting 222, 224
- viewing
 - animated curves 427
 - channel ranges between keyframes 427
 - multiple shots in the player 238
 - multiple shots on the SDI monitor 241
 - selected animated channels 427
 - selected keyframes 427
 - shot information 139
- viewing LUTs
 - rendering with 476
 - selecting 225
- VTR
 - capturing from 514
 - controlling for video I/O 512
 - emulation 539
 - playing out to 529

W

- WAV files, importing and offsetting 113
- white level, modifying for input LUTs 286
- Wiretap
 - playing clips located on Stone FS 564
 - rendering 564
 - rendering options 583
 - setting up projects for soft-imported clips 574

workflows

- creating animations 428
- EDL and keycode assembly 145
- integrating Lustre into post-production 568
- Lustre general 7
- monitor calibration 120
- payout 530
- processing pipeline 8

X

XML

- CUT file data types 617
- CUT file example 618
- limitations 607

- modifying CUT files 608
- viewing and editing files 607

Z

- zooming
 - animation curves 426
 - in Player and full-screen Player 222
 - with the calculator 103

