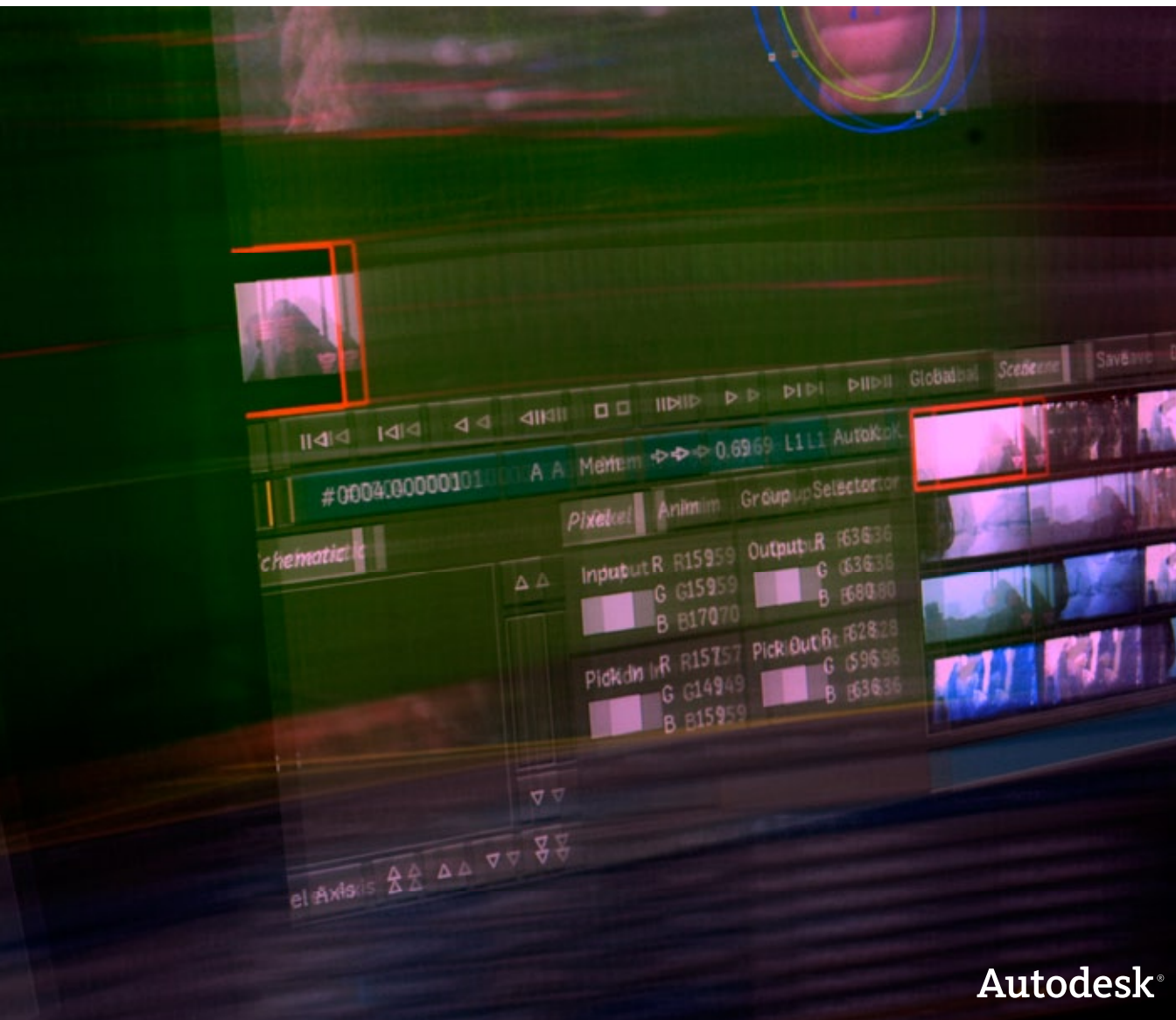


User's Guide



© 2007 Autodesk Canada Co./Autodesk, Inc. All rights reserved. Except as otherwise permitted by Autodesk Canada Co./Autodesk, Inc., this publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose.

Copyright © 2003 NetGroup, Politecnico di Torino. All rights reserved.

Neither the name of "Politecnico di Torino" nor the names of its contributors may be used to endorse or promote products derived from this software without prior written permission. THIS SOFTWARE IS PROVIDED BY THE COPYRIGHT HOLDERS AND CONTRIBUTORS "AS IS" AND ANY EXPRESS OR IMPLIED WARRANTIES, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY AND FITNESS FOR A PARTICULAR PURPOSE ARE DISCLAIMED.

IN NO EVENT SHALL THE COPYRIGHT OWNER OR CONTRIBUTORS BE LIABLE FOR ANY DIRECT, INDIRECT, INCIDENTAL, SPECIAL, EXEMPLARY, OR CONSEQUENTIAL DAMAGES (INCLUDING, BUT NOT LIMITED TO, PROCUREMENT OF SUBSTITUTE GOODS OR SERVICES; LOSS OF USE, DATA, OR PROFITS; OR BUSINESS INTERRUPTION) HOWEVER CAUSED AND OR 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. This product includes software developed by the University of California, Lawrence Berkeley Laboratory and its contributors.

Copyright 2007 Autodesk Canada Co./Autodesk, Inc. Licensed under the Apache License, Version 2.0 (the "License"); you may not use this file except in compliance with the License. You may obtain a copy of the License at <http://www.apache.org/licenses/LICENSE-2.0>. Unless required by applicable law or agreed to in writing, software distributed under the License is distributed on an "AS IS" BASIS, WITHOUT WARRANTIES OR CONDITIONS OF ANY KIND, either express or implied. See the License for the specific language governing permissions and limitations under the License.

Powered by Automatic Duck © 2006 Automatic Duck, Inc. All rights reserved.

Portions copyright 1991-2006 Compuware Corporation.

Portions of this product Copyright 2006 Glyph & Cog, LLC.

DIRAC Time Stretch/Pitch Shift technology licensed from The DSP Dimension, <http://www.dsdimension.com> Developed and (c) 2005 Stephan M. Bernsee

Trademarks

The following are registered trademarks or trademarks of Autodesk, Inc., in the USA and other countries: 3DEC (design/logo), 3December, 3December.com, 3ds Max, ActiveShapes, Actrix, ADI, Alias, Alias (swirl design/logo), AliasStudio, Alias|Wavefront (design/logo), ATC, AUGI, AutoCAD, AutoCAD Learning Assistance, AutoCAD LT, AutoCAD Simulator, AutoCAD SQL Extension, AutoCAD SQL Interface, Autodesk, Autodesk Envision, Autodesk Insight, Autodesk Intent, Autodesk Inventor, Autodesk Map, Autodesk MapGuide, Autodesk Streamline, AutoLISP, AutoSnap, AutoSketch, AutoTrack, Backdraft, Built with ObjectARX (logo), Burn, Buzzsaw, CAiCE, Can You Imagine, Character Studio, Cinestream, Civil 3D, Cleaner, Cleaner Central, ClearScale, Colour Warper, Combustion, Communication Specification, Constructware, Content Explorer, Create>what's>Next> (design/logo), Dancing Baby (image), DesignCenter, Design Doctor, Designer's Toolkit, DesignKids, DesignProf, DesignServer, DesignStudio, Design|Studio (design/logo), Design Your World, Design Your World (design/logo), DWF, DWG, DWG (logo), DWG TrueConvert, DWG TrueView, DXF, EditDV, Education by Design, Extending the Design Team, FBX, Filmbox, FMDesktop, Freewheel, GDX Driver, Gmax, Heads-up Design, Heidi, HOOPS, HumanIK, i-drop, iMOUT, Incinerator, IntroDV, Inventor, Inventor LT, Kaydara, Kaydara (design/logo), LocationLogic, Lustre, Maya, Mechanical Desktop, MotionBuilder, ObjectARX, ObjectDBX, Open Reality, PolarSnap, PortfolioWall, Powered with Autodesk Technology, Productstream, ProjectPoint, Reactor, RealDWG, Real-time Roto, Render Queue, Revit, Showcase, SketchBook, StudioTools, Topobase, Toxik, Visual, Visual Bridge, Visual Construction, Visual Drainage, Visual Hydro, Visual Landscape, Visual Roads, Visual Survey, Visual Syllabus, Visual Toolbox, Visual Tugboat, Visual LISP, Voice Reality, Volo, and Wiretap.

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, Wire.

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 CANADA CO./AUTODESK, INC. "AS IS." AUTODESK CANADA CO./AUTODESK, INC., DISCLAIMS ALL WARRANTIES, EITHER EXPRESS OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE REGARDING THESE MATERIALS.

Published by:

Autodesk Canada Co./Autodesk, Inc.

111 McInnis Parkway

San Rafael, CA 94903, USA

Images Courtesy of:

Hungarian Academy of Film & Theatre, 3rd year, Optimus, Technicolor (formerly Toybox), The House

Title:

Autodesk Lustre 2008 User's Guide

Document Version:

1

Date:

September 19, 2007

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	9
	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	Basics	13
	Summary	13
	Overview of Basic Functionality for Lustre	13
	Starting and Exiting Lustre	13
	Understanding the Lustre User Interface	14
	Working with Control Surfaces	17
	Resetting Parameters	17
	Bypassing Menu Parameters	18
	Undoing and Redoing Actions	19
	Confirming and Cancelling Actions	20
	Assigning Notes to Shots	21
	Flagging Shots	22
	Creating Shot Groups	24
	Copying Parameters with the Selector	28
	Assigning Values With the Standard Lustre Calculator	30
	Grade Bins: Saving, Applying, and Deleting Intermediary Grades	32
	Using Audio	40
	Using Text	42
4	Monitor Calibration	45
	Summary	45
	About Monitor Calibration	45
	Adjusting Black Level and Picture Settings	49
	Selecting the Project Colour Space	51
	Calibrating a Monitor	51
	Calibrating for Dual-Head Mode	55
	Applying Print LUTs for Viewing	56
	Applying 3D Mesh LUTs	57
5	Project Management	59
	Summary	59
	About Project Management	59
	Creating and Loading Users	60
	Creating and Loading Projects	62
	Wiretap-Specific Project Management Configuration	70
	Incinerator-Specific Project Management Configuration	71
	Creating Scenes	71
	Selecting Scenes	73
	Working with Scenes	74

Setting Project Parameters	74
Saving and Loading Grades for a Cut	78
Saving and Loading Grades for Single Shots	82
Saving Timeline RGB Offsets	84
Deleting Grades	85
Setting Up the User Interface	86
Reparsing the Project Structure	88
6 Browsing for Footage	91
Summary	91
About Browsing for Footage	91
Accessing the File Browser	91
Loading Shots into the Shot Bin	93
Viewing Media File Information	97
Accessing Remote Footage by Way of Wiretap	100
Matching Shots	105
Building Cuts and Change Lists	105
Working with CDL, EDL, ALE, and Cutlist Files	110
Exporting Shots to an EDL	129
7 Playing and Viewing Shots	131
Summary	131
About Playing and Viewing	131
Playing Shots	131
Panning and Zooming the Image	133
Setting Image View Options	134
Navigating through Shots	138
Storyboard Viewing Options	142
Viewing Reference Images	145
Viewing Multiple Shots in the Player	147
Caching Frames to Memory for Real-Time Playback	150
Player Hot Keys	151
8 Editing	153
Summary	153
About Editing	153
Accessing the Edit Menu	154

Editing with the Storyboard	156
Trimming	159
Changing Playback Speed	169
Replacing Shots	170
Creating Dissolves	171
Adding a Virtual Black Clip	174
About Timeline	175
Timeline View	176
Accessing the Timeline Menu	176
Editing in the Timeline	181
Trimming the Timeline	186
Transitions	190
EDL and Wiretap	193
Rendering the Timeline	193
Modifying the Timeline's Starting Timecode	194
Performing a Confidence Check	194
Using Scene Detection	195

9 Video Capture and Video Payout 199

Summary	199
About Video Capture and Payout	199
Controlling the VTR	199
Controlling the DDR	201
Capturing from a VTR	202
Defining In and Out Points when Playing Out to Tape	210
Playing Out to a VTR	211
Setting Capture and Payout Options	228
Outputting Multiple Versions	235

10 Removing Dust 237

Summary	237
About Dust Removal	237
Preparing for Dust Removal	238
Removing Dust Manually	241
Removing Dust Automatically	242
Single Frame Dust Removal	244
Working with Boxes	245
Infrared Channel Dust Detection	246
Options for Saving and Loading Dust Removal Metadata	247

Rendering the Result	248
11 Repositioning Images	251
Summary	251
About Repositioning	251
Adjusting the Frame Aspect Ratio	253
Reframing an Image	254
Flipping or Flopping a Shot	256
Tracking Repositioned Shots	257
Animating Reposition Values	260
Setting Rendering Quality	261
12 Input LUTs	263
Summary	263
About LUTs	263
Accessing the Input LUT Menu	264
Applying an Existing Input LUT	264
Linear Mode: Creating Conversion LUTs	265
13 Colour Grading: Basics	275
Summary	275
About Colour Grading	275
Accessing Colour Grading Menus	278
GPU Preview Processing	279
Control Surfaces	283
Viewing Multiple Shots	284
Reference Images	284
Viewing Proxies	284
Analysing the Image	285
Saving and Loading Presets Using the Presets Lists	288
Colour Grade Animation	291
Grade Bin Storage	291
Colourist Timeline	291
14 Primary Colour Grading	295
Summary	295

About Primary Grading	295
Accessing the Grading Menu	296
Logarithmic Mode: Modifying the Brightness	296
Logarithmic Mode: Improving Contrast	302
Linear Mode: Selecting Levels of the Image	305
Linear Mode: Modifying Lift	306
Linear Mode: Modifying Gain	308
Linear Mode: Modifying Gamma	308
Linear Mode: Modifying Saturation	309
Linear Mode: Modifying Contrast	310
Linear Mode: Setting Black Clip and White Clip	311
Matching Colours	312
Autograding Shots	313

15 Colour Grading: RGB and Hue Curves 315

Summary	315
About RGB and Hue Curves	315
Accessing the Curves Menu	316
Plotting Colour Values when Adjusting Curves	316
Modifying Red, Green, and Blue Channels	318
Modifying Hue, Saturation, Luminance, and Lightness	319

16 Secondary Colour Grading 327

Summary	327
About Secondary Colour Grading	327
Accessing Secondary Grading Menus	328
Creating a Secondary by Keying a Range of Colours	329
Saving and Loading Key Presets	340
Importing Mattes	341
About Geometries	342
Accessing the Geometry Controls	343
Drawing Basic Geometries	344
Drawing Free-Form Geometries	345
Modifying Geometries	349
Adding a Split Screen	357
Adding and Deleting Axes	358
Changing the Priority Order of Geometries	361
Showing and Hiding Geometries	362
Inverting Geometries	362

Deleting Geometries and Resetting Secondaries	363
Linking Geometries across Secondary Layers	364
Animating Geometries with the Point Tracker and Shape Tracker	366
Loading Tracker and Stabilizer Data from Autodesk Applications	380
Loading Garbage Mask Setups from Autodesk Applications	382
Colour Grading Secondaries	382
 17 Creating Lustre Sparks Effects	 387
Summary	387
About Lustre Sparks Plugins	387
Installing Lustre Sparks Plugins	388
Licensing Lustre Sparks Plugins	388
Creating an Effect with a Lustre Sparks Plugin	388
Lustre Sparks Plugin Descriptions	391
Disabling and Re-enabling Lustre Sparks Plugins	400
Developing Lustre Sparks Plugins	401
 18 Animation	 403
Summary	403
About Animation	403
Accessing the Channel Editor	404
Animating with Keyframes	408
Editing Keyframes	418
The Track Editor	428
Animating in Other Menus	432
Specifying Setup Options	434
Undoing and Redoing Operations	436
 19 Rendering	 437
Summary	437
About Rendering in Lustre	437
Rendering Shots	438
Saving and Loading Colour Mapping Presets	460
Generating and Viewing Proxies	462
Locking Colour Grading	464
Cleaning Up Render Files	465
Rendering Shots as You Work	466

	Rendering with Burn for Lustre	468
20	Hot Keys	479
	Summary	479
	Storyboard Navigation.....	479
	View Modes	480
	Notes	481
	Calculators	482
	Dust Removal	482
	Editing	483
	Colour Grading.....	485
	Numeric Keypad.....	489
	Animation	492
	Miscellaneous Functions	492
A	Software, Project, and Wiretap Configuration Files	495
	Summary	495
	Configuration File Overview.....	495
	Configuration File Default Options	496
	Configuration File Supplemental Options	502
	Configuring Wiretap to Access Remote Stone Filesystems	504
	The .lustresetup File	505
B	XML in Lustre	507
	Summary	507
	Lustre XML-Based Files.....	507
	Limitations	507
	CUT File Tags	508
	CUT File Data Types	517
	CUT File Example.....	518
	Index	519

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 2008 User's Guide</i>	Detailed instructions on using the software.
<i>Autodesk Control Surface User's Guide</i>	Detailed instructions on using the Autodesk control surface and the Tangent CP100.
<i>Autodesk Lustre 2008 What's New</i>	A complete list of the new features for this release.
<i>Autodesk Lustre 2008 Hot Keys Card</i>	A list of the most frequently used hot keys.
<i>Autodesk Lustre 2008 Release Notes</i>	A complete list of documentation and information on late-breaking features.
<i>Autodesk Lustre 2008 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> (for this release)	Provides detailed connectivity diagrams and configuration procedures for your Autodesk Stone® storage arrays.
<i>Stone and Wire Filesystem and Networking Guide</i> (for this release)	Procedures for configuring your Autodesk Wiretap™ services.
<i>Autodesk Lustre 2008 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 3 Installation and User's Guide</i>	Information on how to install, set up, and use Autodesk Backburner™.

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

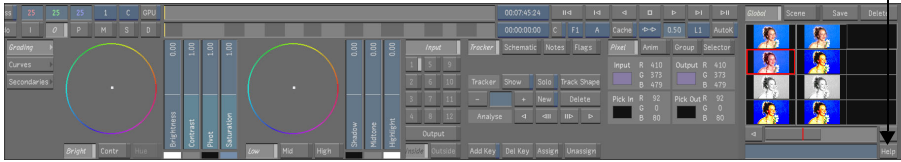
Accessing Online Help

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. Refer to the *Autodesk Lustre Software Installation Guide* for this release (see [“Related Documentation”](#) on page 2).

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_help<version number>/  
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, it is recommended that you use the Xpdf viewer. You can access the PDF files from the Lustre software CD or from www.autodesk.com/discreet-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/discreet-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.

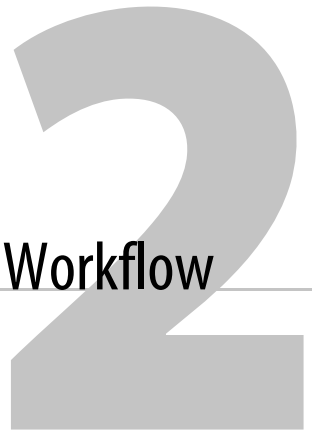
Contacting Customer Support

You can contact Autodesk Media and Entertainment Customer Support at www.autodesk.com/support or through one of the following ways:

Location:	Contact Information:
Within the Americas:	Hotline (North America): 1-800-925-6442 Direct dial: 415-507-5256 (Country code = 1) 8 AM to 8 PM EST Monday to Friday, excluding holidays me.support@autodesk.com
Within Europe, Middle-East and Africa:	Hotline (from London, UK): +44-207-851-8080 9 AM to 5:30 PM (local time) Monday to Friday, excluding holidays me.emea.support@autodesk.com
Within Asia Pacific (Excluding India, China, Australia, New Zealand and Japan)	Hotline (from Singapore): +65-6555-0399 9 AM to 6 PM (local time) Monday to Friday, excluding holidays me.support.singapore@autodesk.com
Within India:	Hotline (from Mumbai): +91-22-6695-2244 9:30 AM to 6:30 PM (local time) Monday to Friday, excluding holidays me.support.india@autodesk.com
Within Japan:	Hotline (from Tokyo): 0120-107-290 Direct dial: +81-3-6221-1810 10 AM to 6 PM (local time) Monday to Friday, excluding holidays me-sys-support@autodesk.jp
Within China:	Direct dial: +86-10-6505-6848 9 AM to 6 PM (local time) Monday to Friday, excluding holidays me.support.china@autodesk.com
Within Australia and New Zealand:	Hotline (from Melbourne): +1-300-36-8355 Direct dial: +61-3-9876-8355 8 AM to 6 PM AEST Monday to Friday, excluding holidays me.support.anz@autodesk.com

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>	9
<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 5, “Project Management,”](#) on page 59.

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 4, “Monitor Calibration,”](#) on page 45.

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 56.

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 93.
- Assembling EDLs in the timeline. See [“Working with CDL, EDL, ALE, and Cutlist Files”](#) on page 110.
- Editing shots in a multi-layer Timeline. See [“Editing in the Timeline”](#) on page 181.
- Capturing footage from a VTR. See [“Capturing from a VTR”](#) on page 202.
- Performing basic gestural edits such as trimming, slipping, and sliding shots, as well as creating dissolves. See [Chapter 8, “Editing,”](#) on page 153.

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 462.

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 10, “Removing Dust,”](#) on page 237.

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 11, “Repositioning Images,”](#) on page 251.

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 264.
- 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 265.

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 14, “Primary Colour Grading,”](#) on page 295.
- Modifying RGB and Hue curves to colour grade specific ranges of colour without using keys. See [Chapter 15, “Colour Grading: RGB and Hue Curves,”](#) on page 315.
- Performing secondary colour grading to specific ranges of colour or areas in an image through keys and geometries. See [“Colour Grading Secondaries”](#) on page 382.
- Generating keys for secondary colour grading. See [“Creating a Secondary by Keying a Range of Colours”](#) on page 329.

- Creating and tracking geometries for secondary colour grading. See [“About Geometries”](#) on page 342 and [“Animating Geometries with the Point Tracker and Shape Tracker”](#) on page 366.

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 17, “Creating Lustre Sparks Effects,”](#) on page 387.

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 18, “Animation,”](#) on page 403.

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 456.
- Rendering a print of the final grade. See [Chapter 19, “Rendering,”](#) on page 437.
- Outputting rendered files to a VTR or any other device. See [“Playing Out to a VTR”](#) on page 211.

Summary

- [*Overview of Basic Functionality for Lustre*](#) 13
- [*Starting and Exiting Lustre*](#) 13
- [*Understanding the Lustre User Interface*](#) 14
- [*Working with Control Surfaces*](#) 17
- [*Resetting Parameters*](#) 17
- [*Bypassing Menu Parameters*](#) 18
- [*Undoing and Redoing Actions*](#) 19
- [*Confirming and Cancelling Actions*](#) 20
- [*Assigning Notes to Shots*](#) 21
- [*Flagging Shots*](#) 22
- [*Creating Shot Groups*](#) 24
- [*Copying Parameters with the Selector*](#) 28
- [*Assigning Values With the Standard Lustre Calculator*](#) 30
- [*Grade Bins: Saving, Applying, and Deleting Intermediary Grades*](#) 32
- [*Using Audio*](#) 40
- [*Using Text*](#) 42

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/lustre<version number>/

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 Optimus

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 91.
- View and modify EDL details. See [“Working with CDL, EDL, ALE, and Cutlist Files”](#) on page 110.
- Display Colour menus. See [“Displaying Colour Menus in the Player”](#) on page 137.
- Monitor capture progress. See [“Capturing from a VTR”](#) on page 202.

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 138 and [“Storyboard Viewing Options”](#) on page 142.

Player Display Controls

Below the Storyboard, there are controls for:

- Resetting the current menu. See [“Resetting Parameters”](#) on page 17.
- Bypassing the current menu. See [“Bypassing Menu Parameters”](#) on page 18.
- Undoing and redoing actions. See [“Undoing and Redoing Actions”](#) on page 19.
- Setting play and view options for shots. See [Chapter 7, “Playing and Viewing Shots.”](#) on page 131.
- Adjusting printer light settings. These display only when a Colour menu is active. See [“Adjusting Printer Lights for Primary Grading”](#) on page 298.
- Setting keyframes automatically. See [“Animating Reposition Values”](#) on page 260 and [“Animating with Keyframes”](#) on page 408.

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 21.
- The Flags system. See [“Flagging Shots”](#) on page 22.
- Grouping functionality. See [“Creating Shot Groups”](#) on page 24.
- The Selector. See [“Copying Parameters with the Selector”](#) on page 28.
- The standard Lustre calculator. See [“Assigning Values With the Standard Lustre Calculator”](#) on page 30.
- The Grade bin. See [“Grade Bins: Saving, Applying, and Deleting Intermediary Grades”](#) on page 32.

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's 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's Guide*.

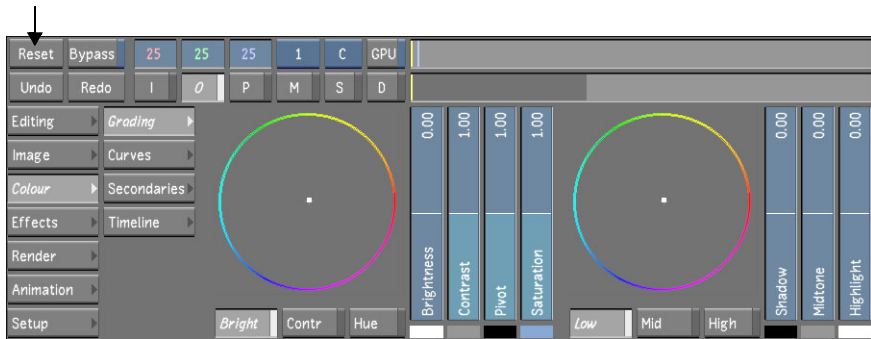
To reset a parameter:

- 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.

- Click Reset (or press **R**), and then confirm.



The menu is reset.

To reset a single shot to its original state:

- Select the shot that you want to reset.
- 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:

- Right-click to select the shots you want to reset.
- 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

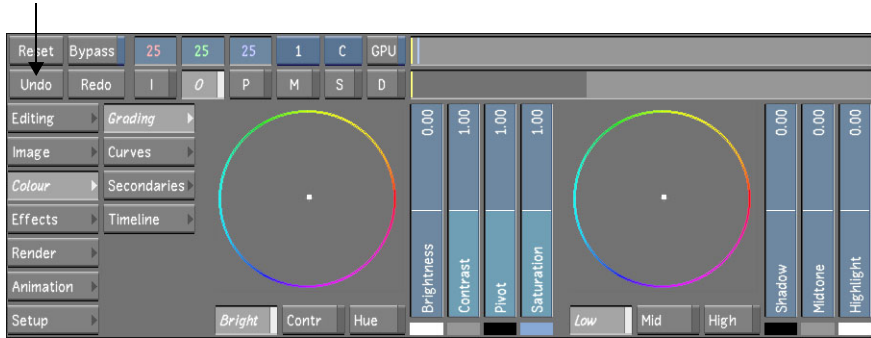
Each shot in Lustre has its own undo and redo history.

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

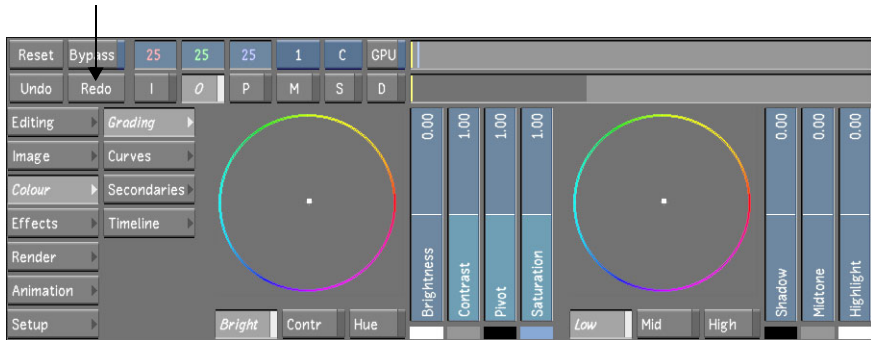
NOTE: You can disable undo functionality in the *lustre.config* file. See [“Configuration File Default Options”](#) on page 496.

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 Contro Surface User's 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

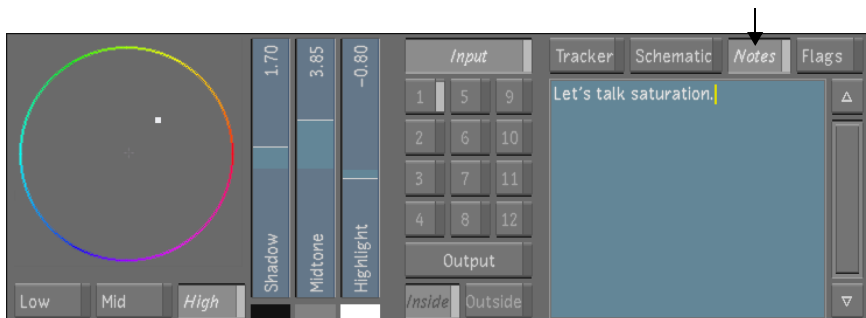
The Notes window is available from almost every menu and allows you to enter unlimited wrapping text pertaining to the current shot. Notes are retained as you jump from one shot to another, and saved when you save the grade.

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 B, “XML in Lustre,”](#) on page 507.

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 [“Saving the Grade for a Cut”](#) on page 78.

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.

System flags are available by default for all shots. There are four types:

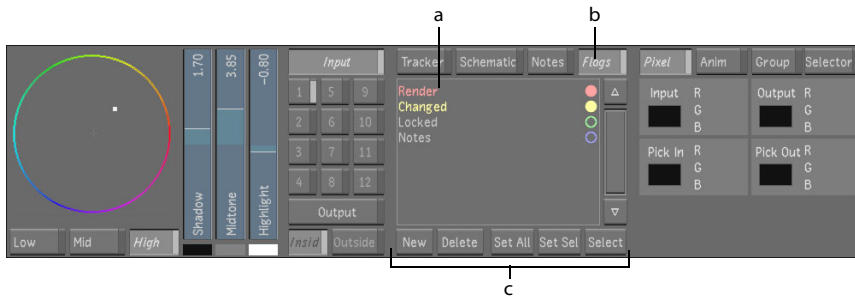
Render — Flags the shots to be rendered.

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.



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 21.



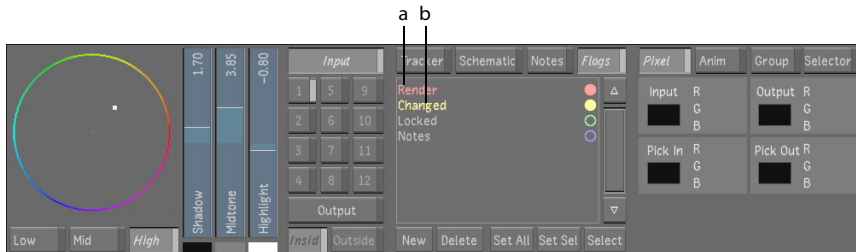
a) Available flags b) Flags button c) Flag controls

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) Enabled flag b) Disabled flag

HINT: To select a flag without enabling or disabling it, left-click the flag row.

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.

NOTE: You cannot rename system flags, nor can you assign the names of system flags to custom flags. Furthermore, spaces are not permitted in naming conventions. It is

recommended that underscores be used instead of spaces when working on either Linux or Windows.

NOTE: Custom flags are saved to the *custom.flags* file in the Lustre version folder. Consequently, custom flags are applied globally on all projects.

NOTE: After the creation of 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 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 selected flag's state to one or more shots on the Storyboard:

1. Click Set All in the Flag controls.
2. To copy the selected flag's state to some shots on the Storyboard, select the shots and click Set Sel. Select the shots by doing one of the following:
 - Right-click them in the Storyboard. The borders of the selected shots will be aqua.
 - Create a shot group, and then select the group. See [“Creating Shot Groups”](#) on page 24.

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, 15. When in group mode, values are added to all the grouped shots, even those that have already been colour corrected.

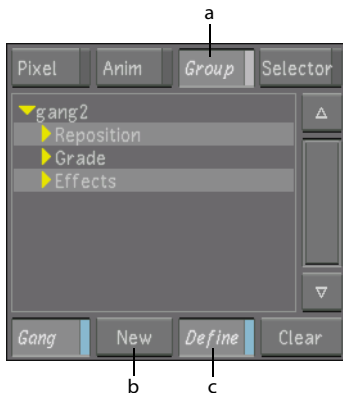
To create a shot group:

1. Click Group.



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

The Group tool appears.



a) Group button

b) New button

c) Define button

2. Click New.

A new group called gang1 is created.

3. Enable Define.

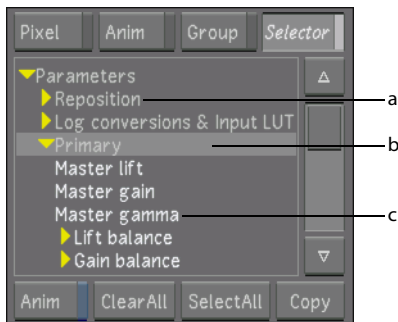
This allows you to add shots to the shot group and select the parameters that are applied to the grouping.

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.

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:

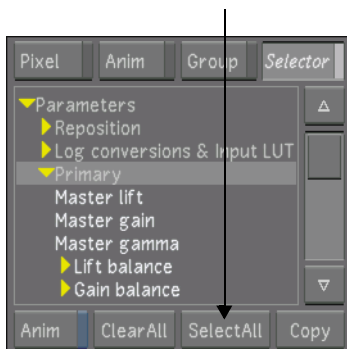
- Colour grading parameters
- Input LUT selections
- Repositioning values
- Tracker data
- Effects
- Render flag settings

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.

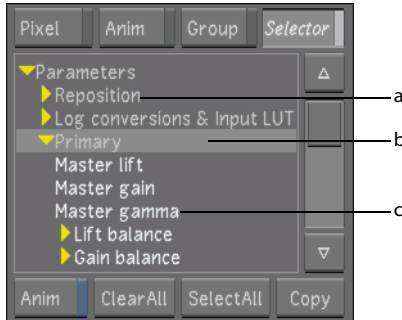


The selected shots are highlighted in aqua.

3. Click the shot to which you want to make modifications.
This shot is the current shot and it is highlighted in red.

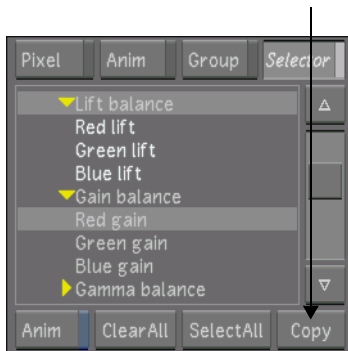
4. Specify what parameters to copy by selecting their channels in the Selector channel list:
 - To select all the channels in a directory, select the directory.
 - To select specific channels in a directory, expand the directory and select any of its channels.

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 32.
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

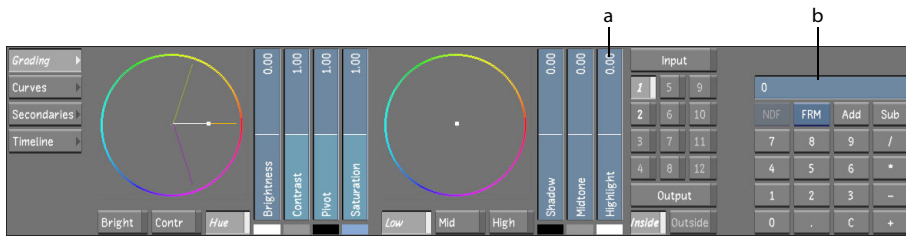
Use the standard Lustre calculator to assign values to sliders and certain fields. You can display the calculator by right-clicking any of the following user interface elements.

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 141.



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.

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.
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.

Press:	To:
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 145.

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.

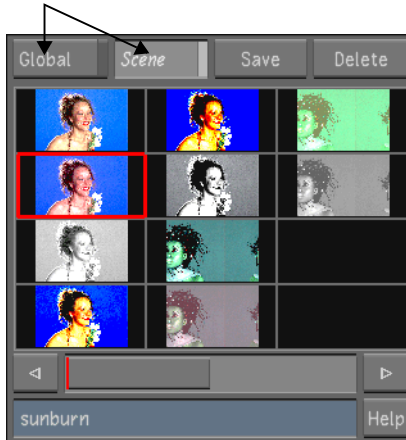
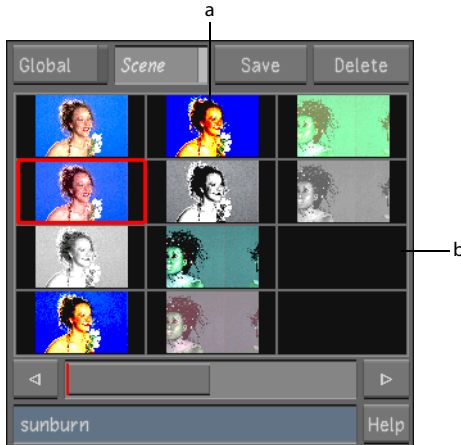


Image courtesy of The House

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 145.

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 143 and [“Creating Shot Groups”](#) on page 24.

2. Select the Grade bin you want to use.

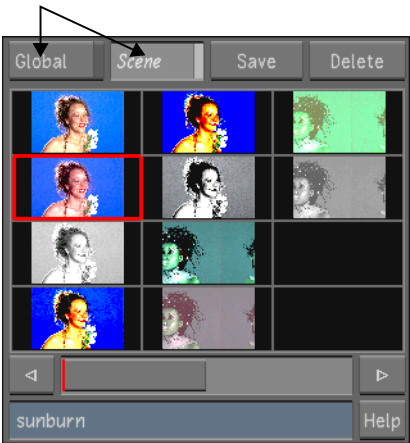


Image courtesy of The House

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.

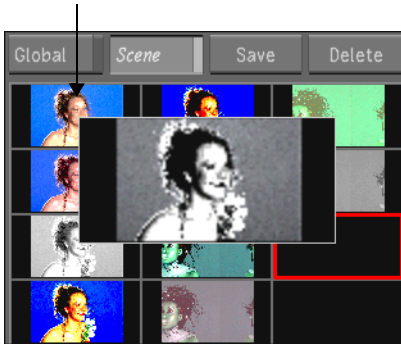


Image courtesy of The House

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.

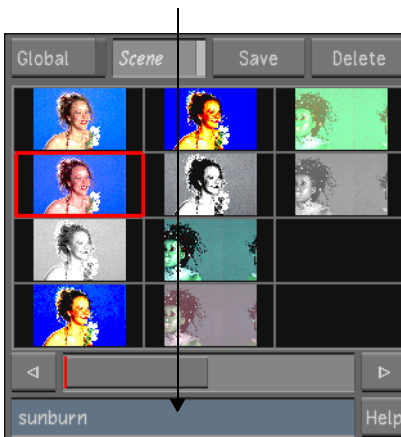


Image courtesy of The House

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 intermediate grade to selected shots, press **SHIFT** and double-click a thumbnail.
- To load the intermediary grade settings to selected shots 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 that are selected in the Selector to selected shots, press **ALT** and double-click a thumbnail. See [“Copying Parameters with the Selector”](#) on page 28.

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.

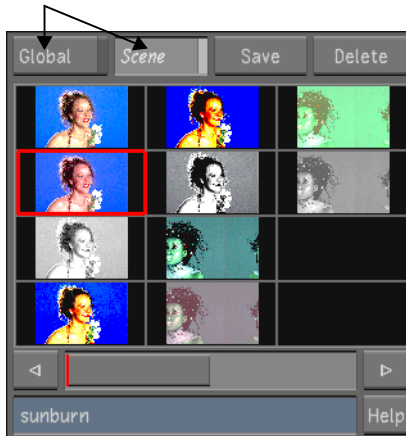


Image courtesy of The House

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.

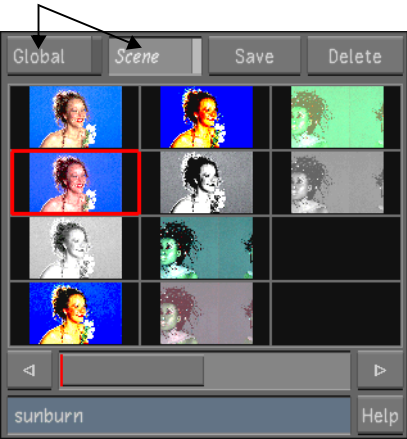


Image courtesy of The House

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.

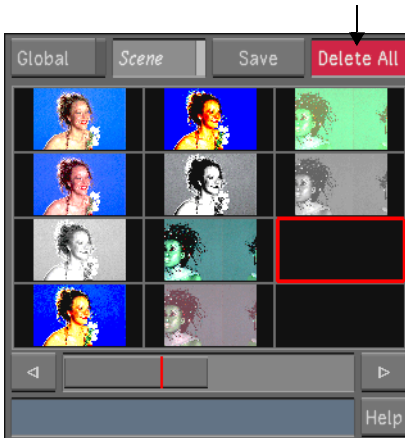


Image courtesy of The House

Using Audio

You can hear audio during timeline playback if you import a WAV file into the system. The imported file is associated with a cut and begins playback at the same time as the timeline. Only one WAV file can be associated with the timeline.

You can also offset audio in frames and seconds. Offset audio when the audio is out of sync with the video.

NOTE: You can also sync the timeline with audio residing on an external device. See [“Controlling the DDR”](#) on page 201 and [“Outputting Linear Timecode While Working on the Timeline”](#) on page 219.

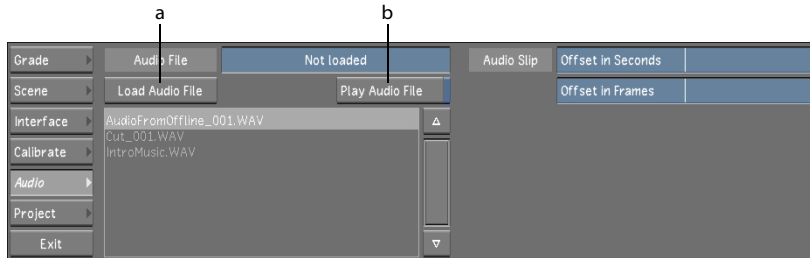
To import an audio WAV file:

1. Copy the audio WAV file into the scene’s `sacc_data\audio` directory. If using the Linux version of Lustre, the filename would use forward slashes.

NOTE: You must create the audio directory manually to store your audio files.

2. In Lustre, click Setup, and then click Audio.

The Audio menu appears.



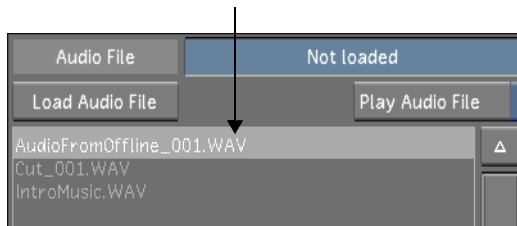
a) Load Audio File button

b) Play Audio File button

3. Select a file from the list and click Play Audio File to preview the audio.
4. When you have located the file you want to use, click Load Audio File.

To offset audio:

1. Select and load the audio file you want to use from the list.



2. Drag the Slip Audio sliders to offset the audio. You can set the offset to a positive or negative value. Drag right to set a positive offset. Drag left to set a negative offset. You can use one or both sliders to set the duration of the offset. Drag the Offset in Seconds slider to offset the audio in seconds. Drag the Offset in Frames slider to offset the audio in frames.



a) Offset in Seconds slider

b) Offset in Frames slider

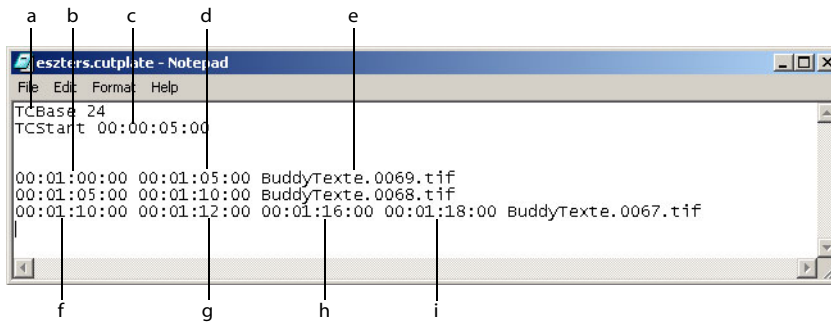
Using Text

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.

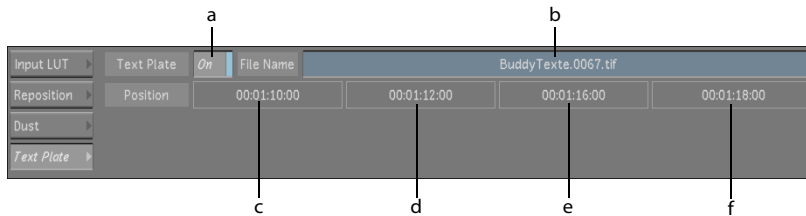
Timecode — List the record timecode for each referenced text file. Set the record in timecode and record out timecode, as well as durations for fade-ins and fade-outs.

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 106.
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.

4

Monitor Calibration

Summary

About Monitor Calibration	45
Adjusting Black Level and Picture Settings	49
Selecting the Project Colour Space	51
Calibrating a Monitor	51
Calibrating for Dual-Head Mode	55
Applying Print LUTs for Viewing	56
Applying 3D Mesh LUTs	57

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.

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<version number>\lut* folder. If running the Linux version of Lustre, save print LUTs in the */usr/Autodesk/lustre<version number>/lut* directory.

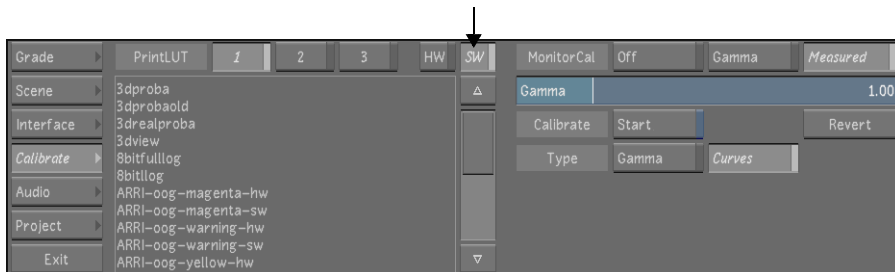
A monitor calibration LUT is applied to the monitor only and affects the display of images, not the image data itself. It is generated from one measure of gamma (white, grey, and black) or a series of progressive measures for each colour channel. The number of steps in the series is defined in the *lustre.config* file. See [Appendix A, “Software, Project, and Wiretap Configuration Files,”](#) on page 495.

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.

NOTE: Support for hardware LUTs depends on the version of the graphics card installed on your system. See your release notes for details.

Enabling HW or SW specifies whether it is a hardware or software LUT. You can then load the hardware or software LUT from the Print LUT list.



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

```

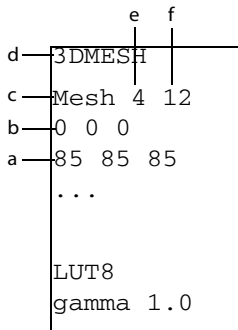
3DMESH
# 3 columns rgb
# 4913 rows
# input [0..1023]
# output [0..4095]

# 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 57.

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 265.

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 49.
2. Select the colour space you intend to use for the project—logarithmic or linear.	“Selecting the Project Colour Space” on page 51.
3. If you are using a colourimeter, automatically calibrate the monitor.	“Calibrating the Monitor Automatically” on page 52.
4. If you do not have a colourimeter, manually calibrate the monitor.	“Calibrating the Monitor Manually” on page 55.
5. If your video card is set to Dual-Head mode, set LUTs for each monitor.	“Calibrating for Dual-Head Mode” on page 55
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 56.

If you are applying a 3D LUT, see [“Applying 3D Mesh LUTs”](#) on page 57.

NOTE: Before you set up the proper display environment, make sure the graphics board is set to the supported resolution. See “Setting the Display” in the “Windows System Software” chapter of

the *Autodesk Lustre Software Installation Guide* for your platform (see [“Related Documentation”](#) on page 2).

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

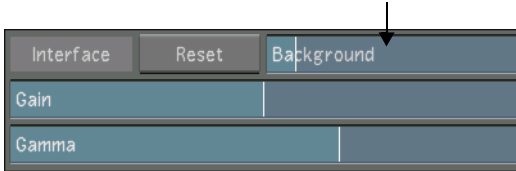
To adjust the black level setting:

1. Start Lustre.
2. Load a black frame (RGB 0, 0, 0) to the Storyboard. See [“Moving Shots to the Storyboard”](#) on page 97.

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.

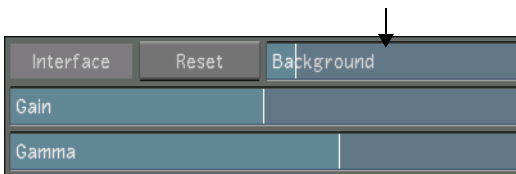
To adjust the picture setting:

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 97.

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.
6. Slowly decrease Contrast on the monitor to adjust the picture setting. When you begin to see that the white frame turns light grey, stop decreasing contrast and then increase it a little.

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.

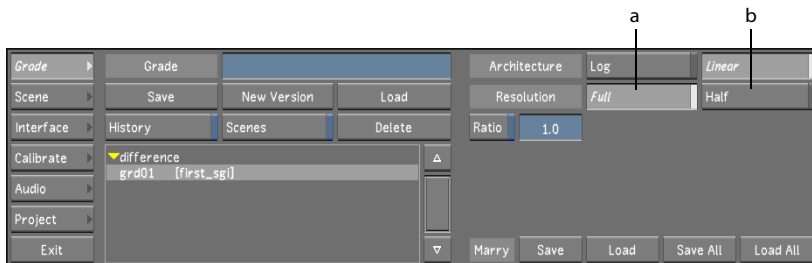
Selecting the Project Colour Space

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:

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

The Grade menu appears.



- a) Logarithmic colour space button b) Linear colour space button

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

Calibrating a Monitor

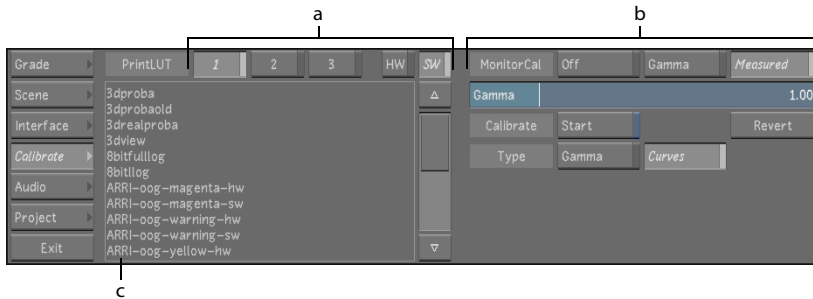
In Lustre, you can calibrate your monitor automatically or manually. Automatic monitor calibration is much more accurate than manual monitor calibration methods because manual methods hinge on perception, and your perception of the monitor can vary depending on the surrounding environment.

Access the Calibration tools in the Calibrate menu.

To access the Calibrate menu:

- 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:

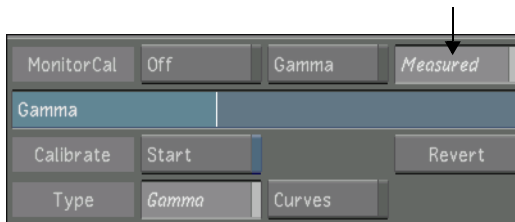
1. In the Calibrate menu, click Off to disable existing monitor calibration settings.



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.

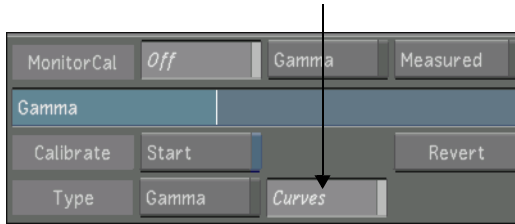
NOTE: You can set the colour steps in the *lustre.config* file. See [Appendix A, "Software Project and Wiretap Configuration Files,"](#) on page 495.

To automatically calibrate the monitor based on a curves measure:

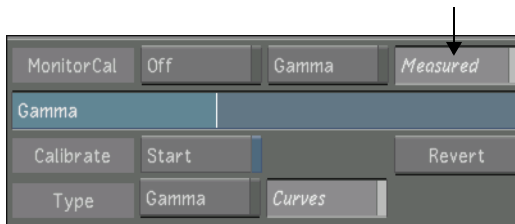
1. In the Calibrate menu, click Off to disable existing monitor calibration settings.



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.

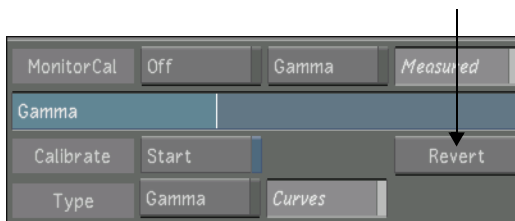


Reverting to a Previous Calibration Settings

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.



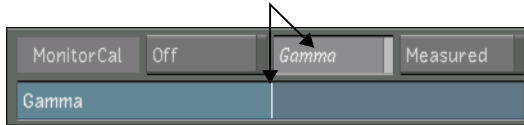
The LUT that was previously saved to the *monitor.calib* file is used to calibrate the monitor. If you click Revert again, you will apply the settings from the latest LUT.

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.



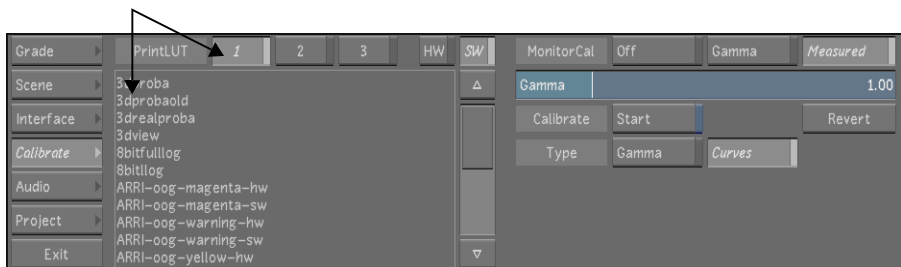
Calibrating for Dual-Head Mode

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.

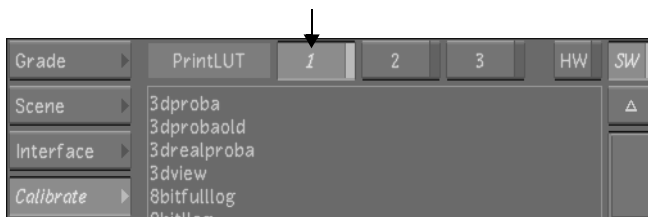
NOTE: In Lustre, LUTs used to calibrate the monitor are applied from the 16-bit result buffer and have a higher quality than those produced on the graphics card.

Selecting Print LUTs

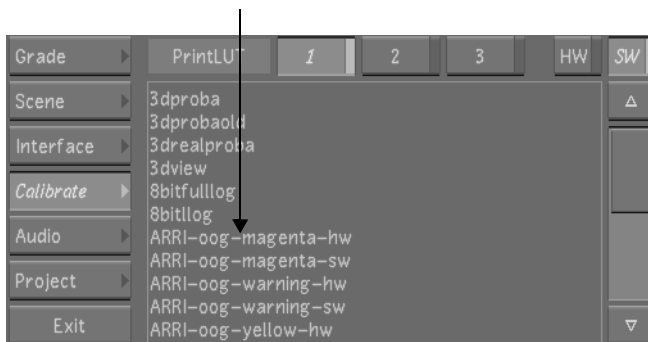
You can select up to three Print LUTs for quick comparison in a project. When your lab sends you the latest LUT, simply select it from the Calibrate menu. You can have any number of Print LUTs on the system.

To select Print LUTs for comparison:

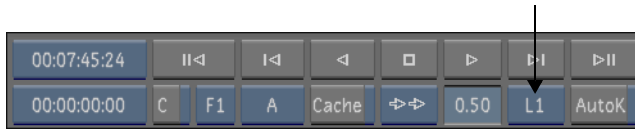
1. In the Calibrate menu, click 1.



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



The Print LUT is assigned to L1.



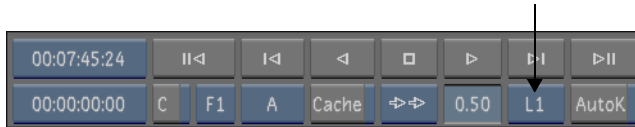
3. Repeat steps 1 and 2 to assign a Print LUT to L2 and L3.

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.

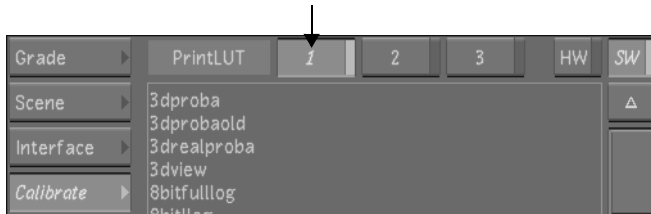
HINT: To output the colour grade with the 3D LUT to video, render the images with Render Viewing On. The 3D LUT will be included in the render without slowing down playback speed.

To apply a 3D Mesh LUT:

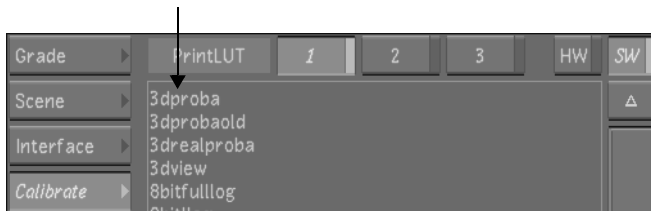
1. Save the 3D LUTs in the `C:\Program Files\Autodesk\lustre<version number>\lut` folder if running the Windows version of Lustre, or in the `/usr/Autodesk/lustre<version number>/lut` directory if running the Linux version of Lustre.
2. In the Calibrate menu, click Off to disable existing monitor calibration 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 processing by clicking the GPU button or pressing Y. See [“GPU Preview Processing”](#) on page 279.

5 Project Management

Summary

<u>About Project Management</u>	59
<u>Creating and Loading Users</u>	60
<u>Creating and Loading Projects</u>	62
<u>Wiretap-Specific Project Management Configuration</u>	70
<u>Incinerator-Specific Project Management Configuration</u>	71
<u>Creating Scenes</u>	71
<u>Selecting Scenes</u>	73
<u>Working with Scenes</u>	74
<u>Setting Project Parameters</u>	74
<u>Saving and Loading Grades for a Cut</u>	78
<u>Saving and Loading Grades for Single Shots</u>	82
<u>Saving Timeline RGB Offsets</u>	84
<u>Deleting Grades</u>	85
<u>Setting Up the User Interface</u>	86
<u>Reparsing the Project Structure</u>	88

About Project Management

In Lustre, creative work is organized in projects and scenes. A project corresponds to a large work effort such as a film. The definition for a scene is subjective, but it can correspond to a specific sequence of shots (or a scene) within the project. When you define a project, you specify a location for project data files, original footage, and rendered files.

Different users on the project are identified in Lustre so that preferences can be loaded each time a user starts a new work session. For example, each user can set the brightness and gamma of the menus and background to suit the lighting conditions of the room.

To prepare for colour grading a project in Lustre, you define the project and users, create scenes, organize your footage, and set user interface preferences.

Projects and scenes are associated with the Grade bin system. The Global grade bin corresponds with the project, and its grades are saved on the system drive. The Scene grade bin corresponds with the scene, and its grades are saved to the scene's *sacc_data* folder. See [“Grade Bins: Saving, Applying, and Deleting Intermediary Grades”](#) on page 32.

Project Setup Workflow

The recommended steps for setting up a project are as follows.

Step:	Location performed:	Refer to:
1. Create a user for each person who will be using the application.	Project menu	“Creating and Loading Users” on page 60.
2. Create a project.	Project menu	“Creating and Loading Projects” on page 62.
3. Create directories in which to store your footage and add footage to them.	Windows Explorer or Linux shell	“Recommended Directory Structure for Projects” on page 65.
4. Create scenes. Directories for each scene are created in the project directory.	Scene menu	“Creating Scenes” on page 71.
5. Set project parameters.	Grade menu	“Setting Project Parameters” on page 74.
6. Set user interface preferences.	Interface menu	“Setting Up the User Interface” on page 86.

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.

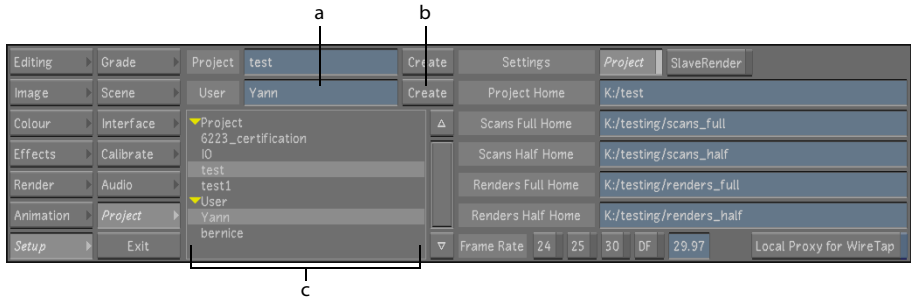
Creating and Loading Users

Lustre retains preferences and settings for each user. When you log in to Lustre with your user name, Lustre loads the last settings you used. For example, when you start Lustre, the application loads the last project, scene and grade you worked on.

To create a user:

1. In the Main menu, click Setup, and then click Project.

The Project menu appears.



a) User field

b) Create button

c) Projects and Users list

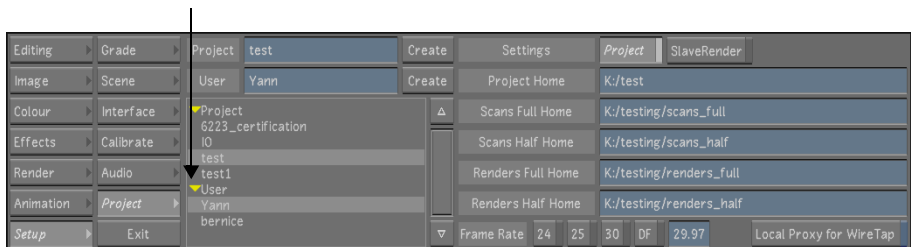
2. Enter a name in the User field. If a name appears already in the field, this means that a previously created user is loaded. You can type over the existing name with the new user name. This will not affect the settings of the other user.
3. Click Create and then click Create a second time to confirm.

The new user appears in the Projects and Users list under Users. The name is highlighted in light grey, indicating that it is the currently loaded user.

NOTE: Lustre saves user settings in the root directory on the system drive. To archive user setting files, you must manually copy them to another location.

To load a user:

1. In the Main menu, click Setup, and then click Project to display the Project menu.
2. If needed, expand the list of users by clicking the yellow arrow to the left of the User label so that it is pointing downwards.



3. Click the user that you want to load.

The name is highlighted in light grey, and it appears in the User field, indicating that it is the currently loaded user. Also, the project, scene, and grade last worked on by the user are loaded.

Creating and Loading Projects

Projects are a means of organizing work performed in Lustre, as well as the directories where 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 saved separately from the footage, as metadata, and does not get applied to the footage until you render a new version of it. When creating a project, you specify a main project directory—referred to in Lustre as the *Project Home*—in which all files containing project metadata are stored. You can store footage and render files in sub-directories of the Project Home or in another location, for example, on another drive. If you opt to store footage and renders in directories outside the Project Home, you must specify the location when you define the project.

You create projects in the Project menu in Lustre. Before creating a project, you must have decided where you want to store your metadata and footage. It is not necessary to actually create the directories prior to specifying the directories in the Project menu, but they must be in place before you start to work with the application. See [“Recommended Directory Structure for Projects”](#) on page 65.

NOTE: You have to create the project directory manually. For example: `/Lustre_Project/Supergirl_Returns`. Lustre automatically creates scene directories based on what you enter in the user interface.

To create a project:

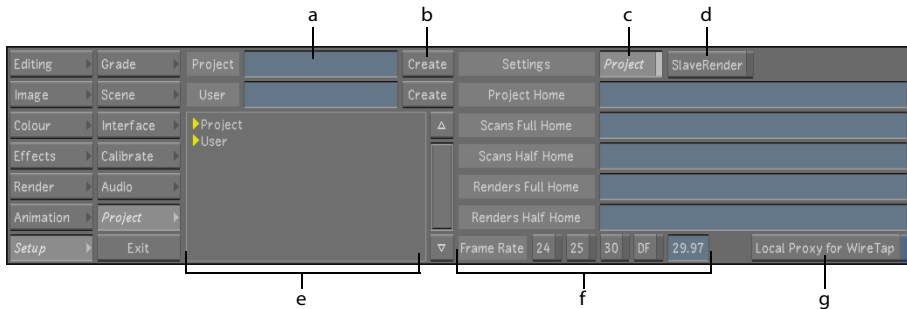
1. Start the application by double-clicking the desktop icon, or by doing one of the following:
 - If using the Windows version of Lustre, select Start | Programs | Autodesk | lustre from the Windows task bar.
 - If using the Linux version of Lustre, open a shell and navigate to the Lustre directory. Type:


```
cd /usr/autodesk/lustre<version number>/
```

 Next, start Lustre. Type:


```
./lustre
```
2. In the Main menu, click Setup, and then click Project.

The Project menu appears.



- a) Project field
- b) Create button
- c) Project Settings button
- d) SlaveRender Settings button
- e) Projects and Users list
- f) Frame Rate buttons
- g) Local Proxy for WireTap button

3. Enter a name for the project in the Project field. If there is already a name in the Project field, a previously created project is loaded. Deselect the project in the Projects and Users list by clicking the project name highlighted in light grey. The Project field will then be empty.
4. If you are working on a project using footage accessed remotely via Autodesk Wiretap, and you would like to generate local proxies from footage stored on the Autodesk Stone filesystem, enable Local Proxy for Wiretap. This ensures that Lustre will use proxies stored in the Scans Half Home field instead of those located on the Stone filesystem. For details on loading clips from a remote location, see [“Linking to Media on the Stone Filesystem Via Wiretap”](#) on page 103.

NOTE: Proxy generation on the Stone filesystem is not automatic unless you create your Stone filesystem projects with certain conditions in place. Refer to your Editing or Effects user’s guide for details.

5. Click Create and then click Create a second time to confirm.
The new project appears in the Projects and Users list under Projects. It is highlighted in light grey, indicating that it is the currently selected project.
6. Enable Project Settings.
7. With the new project still selected, enter the path and directory name of the Project Home in the Project Home field. Do one of the following:
 - If using the Windows version of Lustre, include the drive letter. For example: *E:\myproject*.
 - If using the Linux version of Lustre, include the mount point of the local storage or the remote file server. For example: */mnt/md0/myproject*.

NOTE: Each project defined in Lustre should have a unique Project Home directory.

8. If your full-resolution scans directory is not a sub-directory of Project Home, enter the path and directory name where full-resolution scans will be stored in the Scans Full Home field. Otherwise, leave this field empty.
9. If your half-resolution scans directory is not a sub-directory of Project Home, enter the path and directory name where half-resolution scans will be stored in the Scans Half Home field. Otherwise, leave this field empty.
10. If your full-resolution renders directory is not a sub-directory of Project Home, enter the path and directory name where full-resolution renders will be stored in the Renders Full Home field. Otherwise, leave this field empty.
11. If your half-resolution renders directory is not a sub-directory of Project Home, enter the path and directory name where half-resolution renders will be stored in the Renders Half Home field. Otherwise, leave this field empty.
12. If rendering to the Stone filesystem, define the Wiretap server with a Stone library path in either the Renders Full Home field, the Renders Half Home field, or both. For example, the Renders Full Home path could be defined as `<IP_address>@wt:/stonefs/project/library` or `<hostname>@wt:/stonefs/project/library`. You do not need to define a reel, because Lustre automatically generates a new reel containing the rendered clips. The IP address or hostname represents the location of the Wiretap Server.

NOTE: If using a hostname, the Wiretap host network must be configured so that Lustre can recognize it. See [“Wiretap-Specific Project Management Configuration”](#) on page 70.

13. Set the frame rate for a project’s timeline and editing tools by enabling one of the Frame Rate buttons. The following frame rates are supported.

Enable:	To:
24	Set the frame rate to 23.98 or 24 fps.
25	Set the frame rate to 25 fps.
30	Set the frame rate to 29.97 or 30 fps.
30 PD	Convert the frame rate from NTSC to 24 fps.
DF	Set the frame rate to 29.97 fps drop frame mode.

This step can be performed either at the project’s initial creation, or at any time during the grading process.

14. Enable SlaveRender to set up slave rendering.

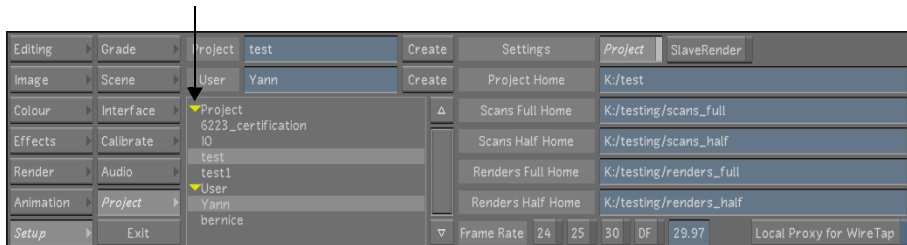
Fields for entering the project, scans, and render directories as mapped on the Slave Renderer appear. See [“Rendering Shots as You Work”](#) on page 466.

15. Set up slave rendering. Do one of the following:

- If using the Windows version of Lustre, enter the local IP address for the Slave Renderer, followed by the shared storage name, and then the local project path. For example, the Project Home could be defined for the Slave Renderer as `\\192.168.1.1\\H:\myproject`.
- If using the Linux version of Lustre, mount the storage onto the Slave Renderer node, and indicate the NFS mount point, followed by the path. For example, the Project Home could be defined for the Slave Renderer as `/<storage>/myproject`.

To load a project:

1. In the Main menu, click Setup, and then click Project to display the Project menu.
2. If needed, expand the list of projects by clicking the yellow arrow to the left of the Project label, so that it is pointing downwards.



3. Click the project you want to load.

The name is highlighted in light grey in the list, and it appears in the Project field, indicating that it is the currently loaded project.

Recommended Directory Structure for Projects

Lustre stores all project metadata in sub-directories of the main project directory, which is called the Project Home. The metadata is saved per scene, and is stored in a sub-directory of each defined scene. This 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. 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 71.

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 [“Creating Scenes”](#) on page 71.

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 443. 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 443. 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.

Wiretap-Specific Project Management Configuration

You must set up projects according to a specific workflow when accessing files by way of Wiretap. Project configurations vary depending on the Wiretap workflow used for accessing either the Stone® filesystem or soft-imported clips on the SAN or NAS system.

With your projects properly configured, you can:

- Render to the Stone filesystem.
- Publish links to soft-imported clips on the Stone filesystem.
- Generate local proxies from files on the Stone filesystem.
- Render to local storage, or to the SAN or NAS filesystem.
- Render to or from the Stone filesystem using Burn™ for Lustre.

Soft-imported DLEDs require the most configuration. In order to import and render soft-imported DLEDs, you must follow the steps below.

To configure projects to access soft-imported DLEDs by way of Wiretap:

1. On the shared network storage system, such as a SAN or NAS, ensure that files to be soft-imported into the Editing or Effects 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 use Burn for Lustre. The Background render nodes require a rigid directory structure to ensure file access from a Linux mount point. The Burn for Lustre render farm does not require an individual file system access to a SAN or NAS.

2. In the Editing or Effects application where the Stone filesystem resides, configure the Wiretap path translation file (*sw_wiretap_path_translation_db.xml*) using the platform or host/path rules explained in the “Configuring Wiretap” section in the *Stone and Wire Filesystem and Networking Guide*.

This file makes it possible for the operating system, or Lustre host, to recognize the location once files are brought in by way of Wiretap.

Below is an example of the platform rules used in the file to translate the path from Irix® (Wiretap host) to Windows (Lustre workstation).

```
<map src_os="Irix" src_path="/SAN/"
dst_os="WindowsNT" dst_path="S:\">
```

This translates the soft-imported path pulled from the directory on the shared network storage system to the following 5-folder level structure on Windows:

/SAN/lustre/scans_full/2k/images/2048x1556/

This translates on Windows as:

S:/lustre/scans_full/2k/images/2048x1556/

3. In the Lustre Setup Project menu, ensure that the Scans Full Home path points to the Windows drive configured in the Wiretap path translation file (S:\).

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 Linux/Irix host and Lustre workstation.

You are now ready to import soft-imported DLEDs 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 DLED in the Shot bin. See [“Viewing Media File Information”](#) on page 97.

HINT: You must ensure that the shared network storage is mounted and accessible from both the Wiretap Linux/Irix host and the Lustre workstation.

Incinerator-Specific Project Management Configuration

If running Lustre with Incinerator in-line processing, the same project management concepts apply as described above. 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 GigE IP address of the Lustre Media Server and leave the Scans Full Home path empty. When the Scans Full Home path is empty, Lustre uses the Project Home path to locate the Scans Full Home. You access storage remotely using a rapid network connection. For further information on project management configuration in Incinerator, refer to the *Autodesk Incinerator Installation and User's Guide*.

Creating Scenes

A scene is a means of organizing metadata within a project, for example, cuts and grades. You can create as many cuts per scene as needed. Typically, though, a scene is created for each

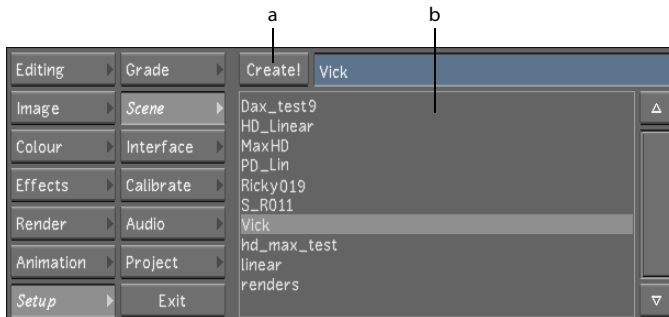
discrete sequence of shots (or scene) in a film project. An alternative approach is to create one scene for each reel, in which case a series of cuts may be set up within the scene.

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) Create Scene field

b) 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

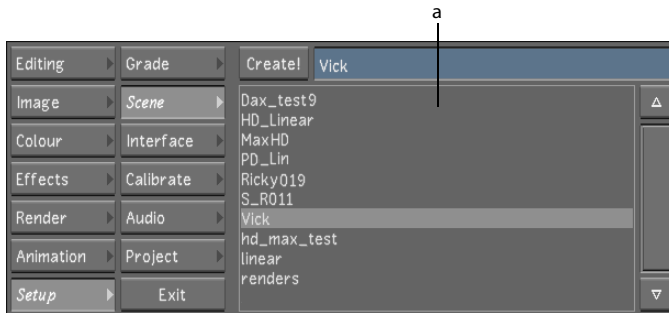
When you select a scene, you gain access to the grade files associated with the scene. Nothing changes in the application or gets loaded when you select a scene except the contents of the Grade bin, which is associated with the scene. Once you select a scene, you then select the grade you want to work 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.



a) Scene list

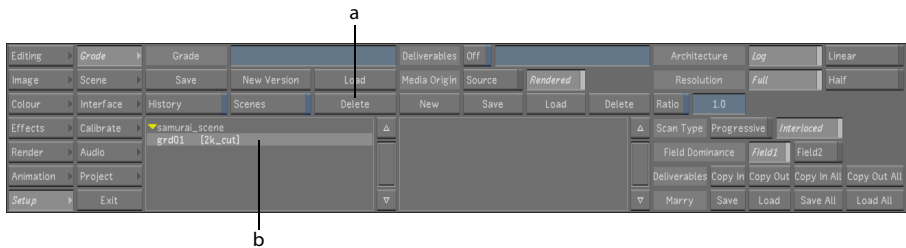
2. In the Scene list, click a scene to select it.

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) Scenes button

b) 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 Scenes

Once you have completed all the steps for setting up a project (see [“Project Setup Workflow”](#) on page 60), refer to the following table to learn how to set up a scene for use.

Step:	Refer to:
1. Select a scene.	“Selecting Scenes” on page 73.
2. Build or load a cut.	“Building Cuts and Change Lists” on page 105.
3. Save a default grade. The grade is saved and the cut is associated with it.	“Saving and Loading Grades for a Cut” on page 78.

Once a scene is set up, refer to the following table for steps on how to start a session to work on the scene.

Step:	Refer to:
1. Start Lustre if not already started.	“Starting and Exiting Lustre” on page 13.
2. Select a user.	“Creating and Loading Users” on page 60.
3. Switch projects if needed.	“Creating and Loading Projects” on page 62.
4. Select the scene.	“Selecting Scenes” on page 73.
5. Load the grade. This loads the grade and its associated cut.	“Saving and Loading Grades for a Cut” on page 78.

Setting Project Parameters

Once you have created at least one new scene for a new project, set up project-level parameters that define the working environment. For example, specify the grading architecture — that is, whether you will be working in log or linear colour space — and whether you will be working on full-resolution footage or half-resolution proxies.

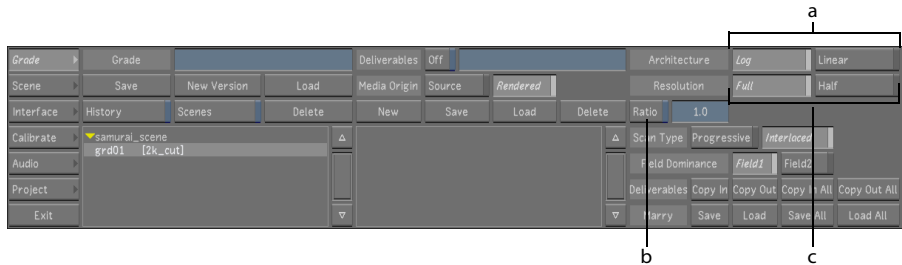
You specify project-level settings in the Grade menu.

NOTE: It is recommended that you verify the parameters in the project configuration file, *lustre.config*, prior to starting a project. See [Appendix A, “Software, Project, and Wiretap Configuration Files.”](#) on page 495.

To access the Grade menu:

1. In the Main menu, click Setup.
2. Click Grade.

The Grade menu appears.



a) Colour Space Architecture options

b) Ratio button

c) Resolution options

NOTE: For instructions on using the Real-Time Deliverables section, see [“Creating Real-Time Deliverables From Source or Rendered Material”](#) on page 223.

Setting Log 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 12, “Input LUTs,”](#) on page 263.

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 456.

User Interface Differences

The user interface 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 14, “Primary Colour Grading,”](#) on page 295.
- Printer lights only appear in Log mode. See [“Adjusting Printer Lights for Primary Grading”](#) on page 298.
- 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 265.

To set the colour space architecture for the project:

- In the Grade menu, enable Log or Linear.

HINT: To ensure that the Linear setting is kept after a video capture, add the `KeepSettingsAfterCapture` keyword to the configuration file. See [“Configuration File Supplemental Options”](#) on page 502.

Setting the Resolution

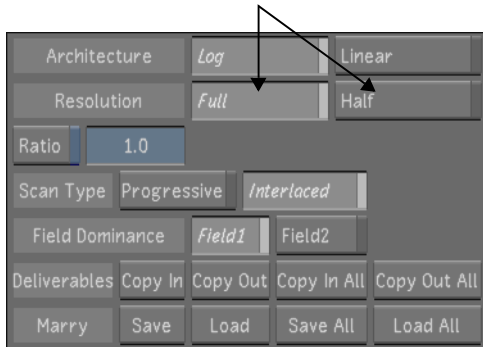
You can work on the full-resolution version of shots or proxies. Using proxies speeds up interaction and playback of shots containing many effects. Proxies are also useful for playing to projectors.

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 462 and [“Recommended Directory Structure for Projects”](#) on page 65.

To set the resolution:

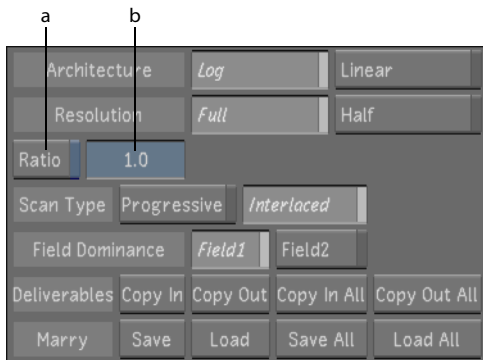
- In the Grade menu, enable Full or Half resolution.

**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.



- a) Ratio button b) Ratio field

2. Enable Ratio.

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

Saving and Loading Grades for a Cut

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.

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.

NOTE: Do not save a grade until you have created at least one scene.

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 82.

Saving the Grade for a Cut

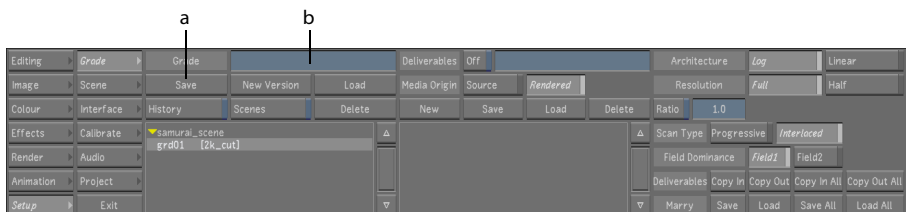
Save the grade for the cut you are working on to save all your work on the cut. You can save up to 100 grades per cut. Grades are named using sequential numbers, but you can optionally add a text name to the number.

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 107.

To save a grade for a cut:

1. Create or load a cut for which you want to save a grade. See [“Building Cuts and Change Lists”](#) on page 105.
2. Work on the grade as needed.
3. In the Main menu, click Setup, and then click Grade.

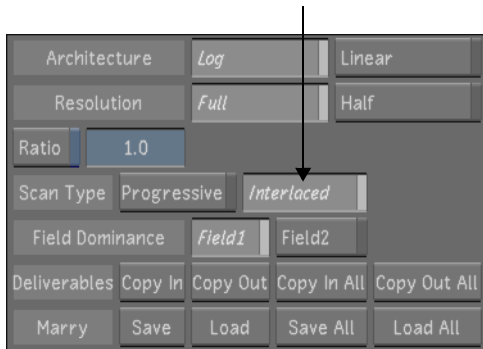
The Grade menu appears.



a) Save button

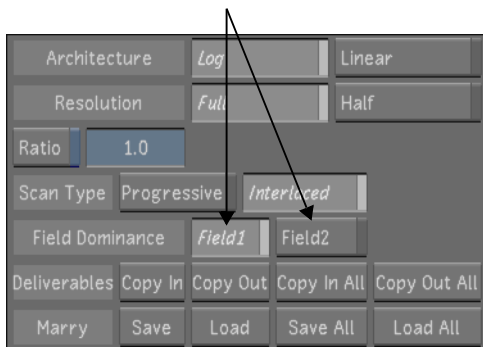
b) Grade field

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.

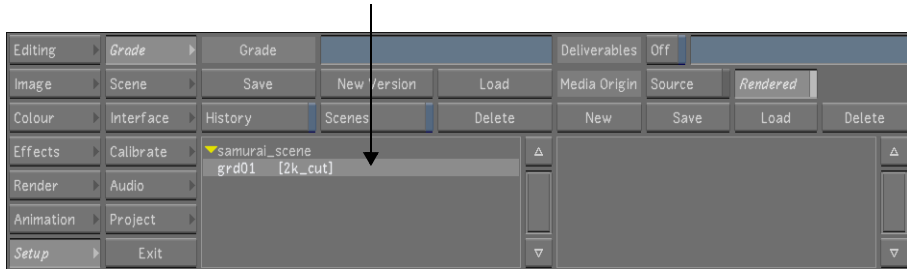


The dominant field is rendered first, and is the first field to which animations are applied. By correctly setting the dominant field, you can avoid introducing noticeable jitter that would result if the field recorded earlier in time is displayed after the field that was recorded later in time.

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.

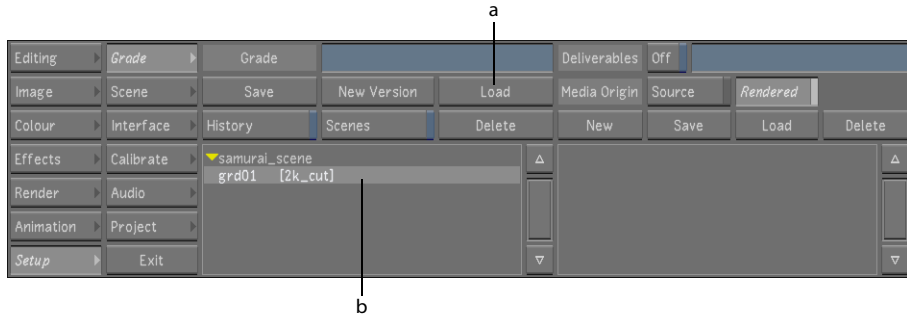
Loading Grades for a Cut

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.



- a) Load button b) Grade list

2. If it is not already selected, select the scene containing the grade you want to load. See [“Selecting Scenes”](#) on page 73.
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.

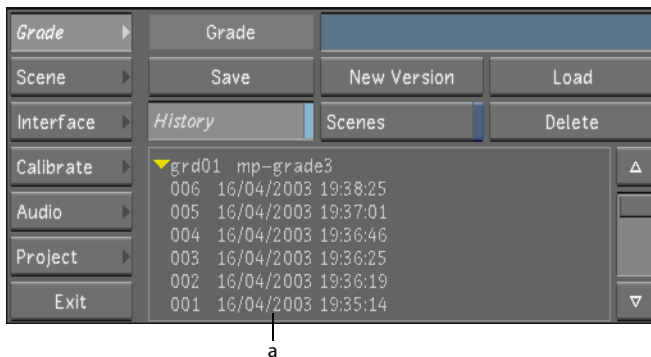
NOTE: To use this feature, you need to first enable the History line in the *lustre.config* file. See [Appendix A. “Software, Project, and Wiretap Configuration Files.”](#) on page 495.

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 80.
2. In the Grade menu, click History.



The Grade History list appears.



a) Grade History list

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.

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

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 in a file named *marry.grade*. 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.

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 32.

As an alternative to the *marry.grade* file, you can save to a text file that uses a *.dsc* extension. While the *marry.grade* file saves colour grading metadata across all channels, the *.dsc* file is limited to the primary colour correction data. This data is saved as offset in code values for red, green, and blue. This can be useful, for example, when exchanging colour correction data with a third party application.

The text file will only be created and read if you use the MarryType PrimText keyword in the *lustre.config* file. See [Appendix A. “Software, Project, and Wiretap Configuration Files.”](#) on page 495. When you add the MarryType PrimText keyword, you use the Marry controls in the Grade menu the same as if you were creating a *marry.grade* file. The resulting *.dsc* file is saved with the grade, and the filename is based on the grade you are currently working on. For example, if you are working on Grade 1, the colour grading metadata is saved to *grade01.dsc*. In order to load the *.dsc* file, its associated grade file must be loaded already. You can have only one *.dsc* file per grade.

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 138.
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 Scans Half Home directory. 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<version number>\export*.

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<version number>\export* directory.
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.

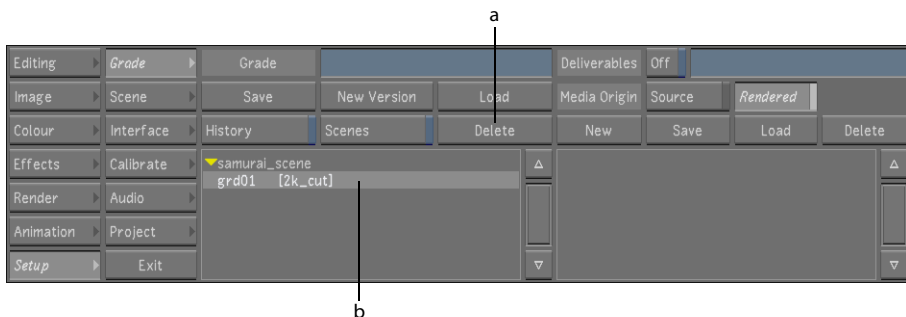
Deleting Grades

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) Delete button

b) 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.

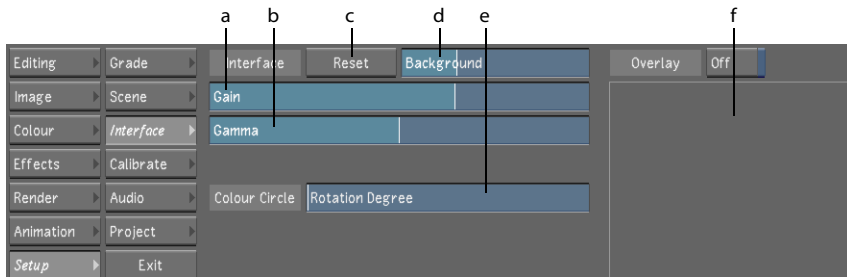
Setting Up the User Interface

Before starting to work in Lustre, 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 field	c) Reset button	e) Colour Circle field
b) Gamma field	d) Background field	f) Overlays list

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 87.

Applying Overlays

Use this feature to apply an overlay to your cut. For example, you can add a logo to every frame. An overlay consists of a single uncompressed RGBA, TIFF, or SGI® file containing alpha channel information. The white areas of the file are viewable in the cut.

To make files available for use as overlays:

1. Exit Lustre.
2. Navigate to the Lustre directory by doing one of the following:
 - If using the Windows version of Lustre, in Windows Explorer go to *C:\Program Files\Autodesk\lustre<version number>*, where C is the drive where Lustre is installed.
 - If using the Linux version of Lustre, type:


```
cd /usr/autodesk/lustre<version number>
```
3. Create a directory called *overlay*. To do this in Linux, type:

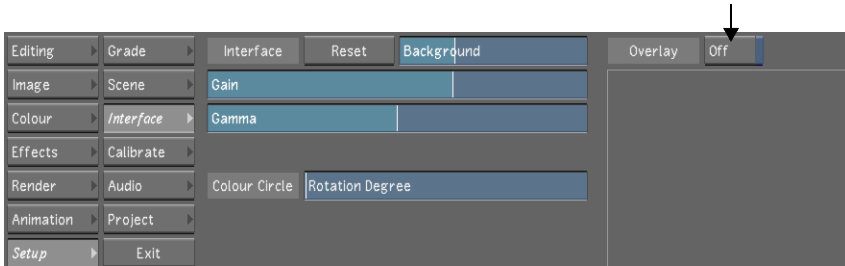

```
mkdir overlay
```
4. Place any files you want to use as overlays in this directory. To do this in Linux, type:


```
cp /<path>/<filename> /usr/autodesk/lustre<version number>/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 106.
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 the On/Off button.



The overlay is applied to the cut.

Reparsing the Project Structure

If you move metadata or material used in your project outside of Lustre, the project structure changes. For example, importing complete scenes from outside the project changes the structure. When you change the project structure, you will need to reparse the project structure at start-up to ensure that the colour grading metadata is applied properly to the cut. If you do not reparse the project, images may not be updated with the changes made in Lustre.

To reparse the project structure at start-up:

1. Exit Lustre.
2. On the Master Station or the Lustre HD Station, navigate to the Lustre directory. Do one of the following:
 - If using the Windows version of Lustre, go to *C:\Program Files\Autodesk\lustre<version number>*.
 - If using the Linux version of Lustre, type:


```
cd /usr/autodesk/lustre<version number>
```
3. Open the configuration file. Do one of the following:
 - If using the Windows version of Lustre, use a text editor such as *Notepad* to open the *init.config* file.
 - If using the Linux version of Lustre, using a text editor such as *nedit* to open the *init.config* file. Type:


```
nedit init.config
```
4. Uncomment the line (remove the # at the start of the line) `#MustReparse`
5. Save the *lustre.config* file.

6. Restart Lustre.

The project is reparsed and the application starts.

7. Go back into the *lustre.config* file and re-comment the same line when you no longer need to reparse the project at start-up.

HINT: To reparse only when you specify, as opposed to every time the project structure changes, you can leave the `MustReparse` keyword commented out and use hot keys instead. You can reparse the entire project by pressing **SHIFT+P**, or the currently loaded timeline only by holding down **SHIFT** while clicking the Load button.

Browsing for Footage

Summary

<i>About Browsing for Footage</i>	91
<i>Accessing the File Browser</i>	91
<i>Loading Shots into the Shot Bin</i>	93
<i>Viewing Media File Information</i>	97
<i>Accessing Remote Footage by Way of Wiretap</i>	100
<i>Matching Shots</i>	105
<i>Building Cuts and Change Lists</i>	105
<i>Working with CDL, EDL, ALE, and Cutlist Files</i>	110
<i>Exporting Shots to an EDL</i>	129

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

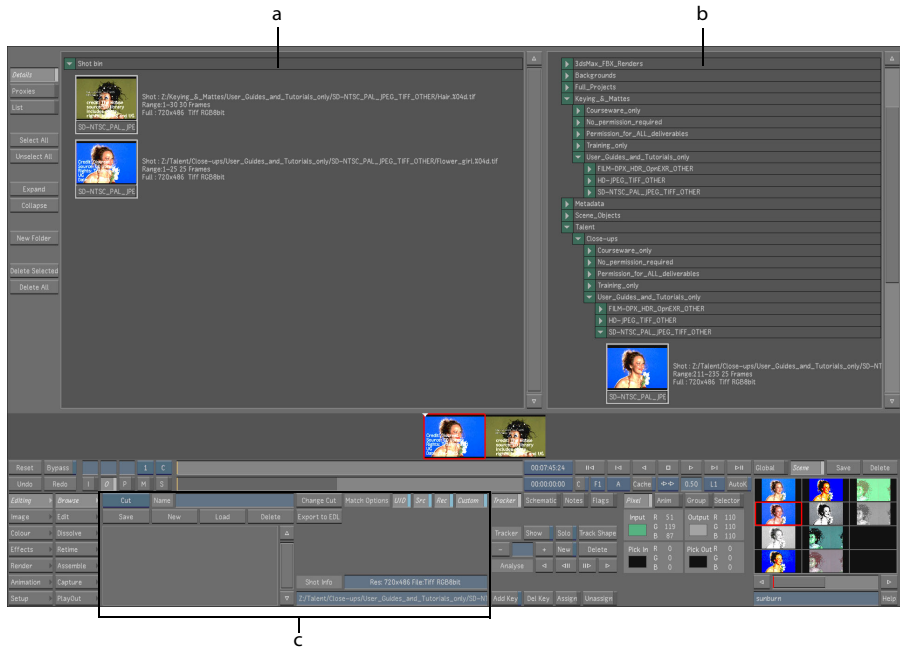
Use the file browser to locate shots and load them into the Shot bin.

6 Browsing for Footage

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 The House

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:
List	Display shots in Text view.
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.
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:
Delete Selected	Delete selected shots.
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 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 project home directory. In the file browser, you can see if the files are correctly located in the project home directory. If they are, the path to the media file starts with the word <HOME>.

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)

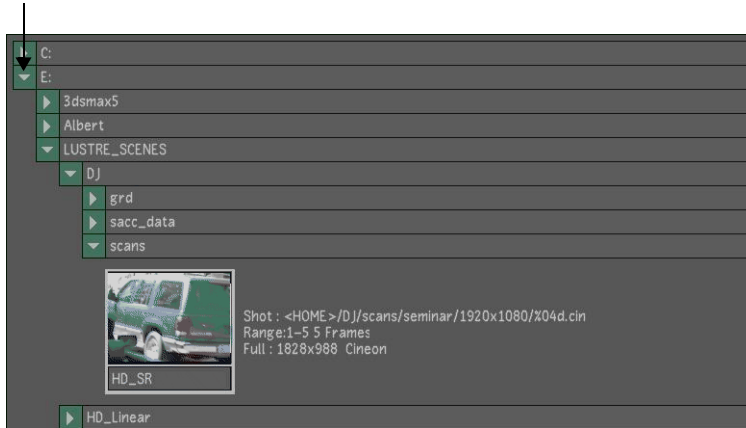
NOTE: You cannot work with compressed files in Lustre.

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.



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.

2. In the file browser, 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.

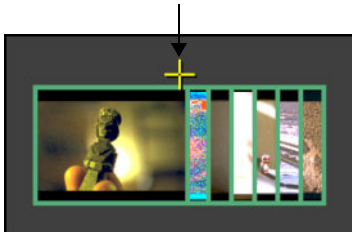


Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

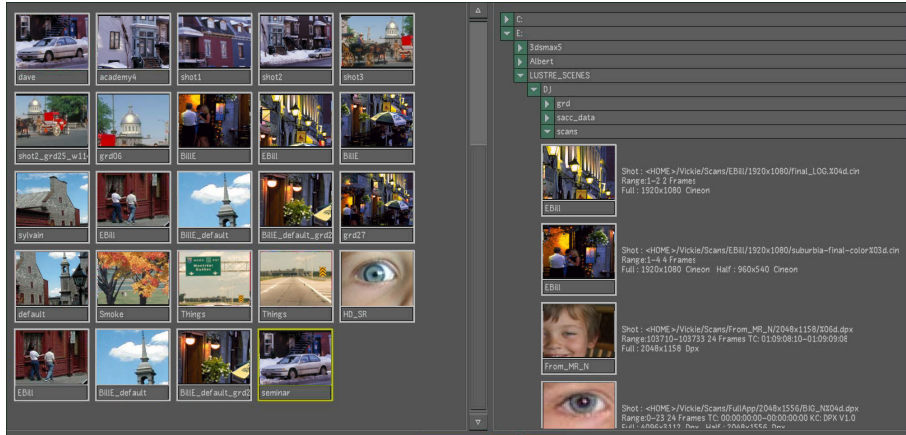
HINT: To select all the shots from a folder, select the folder.

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

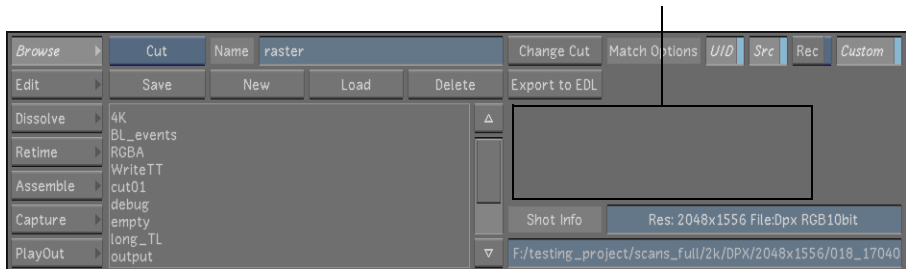
HINT: You can drag shots directly from the file browser to the Storyboard. When you do this, the shots are also placed in the Shot bin.

Loading Shots into the Shot Bin

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

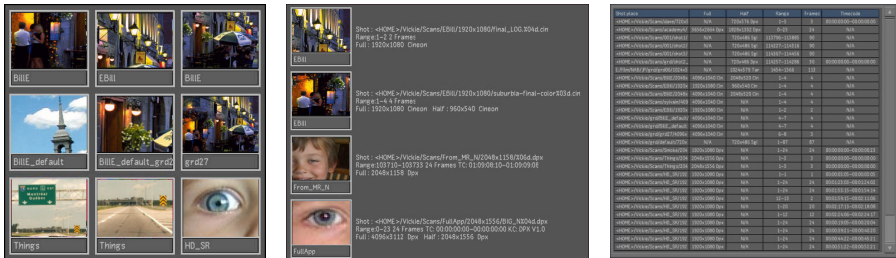


4. If you pick up the wrong shots, click the grey area in the centre of the Browse menu.



5. 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.



c) 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.

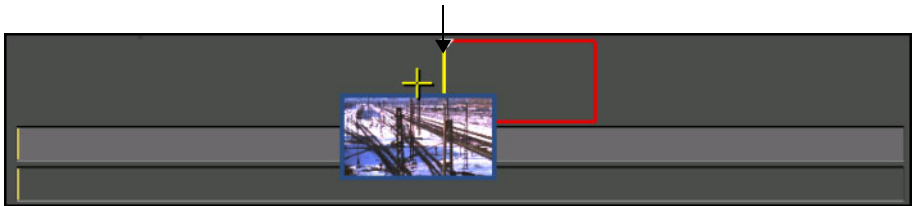


Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

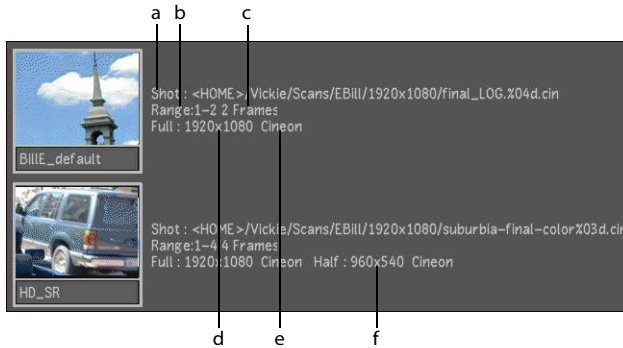
3. Click to drop the shot at the insertion point.

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 place c) Frames e) File format
- b) Range d) Full resolution f) Half resolution

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.

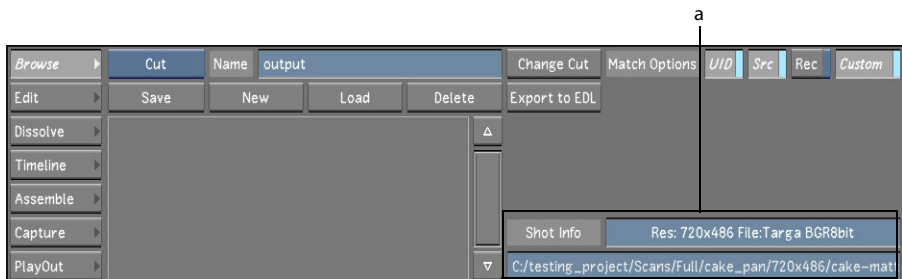
To view information for shots in the Shot bin or file browser:

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

Accessing Remote Footage by Way of Wiretap

With Autodesk Stone® and Autodesk Wire® 2.5 and later, Lustre can access media and metadata stored in Stone filesystems (that is, file system arrays) across a TCP/IP network by way of Wiretap.

Stone and Wire is found on Autodesk IRIX®- and Linux-based systems. It provides high performance playback for Autodesk Flint®, Autodesk Flame®, Autodesk Inferno®, Autodesk Fire®, Autodesk Smoke®, Autodesk Backdraft®, and Autodesk Backdraft® Conform.

Wiretap is a client server architecture that makes it possible for applications that do not have a Stone filesystem, such as Lustre, to access clips on a remote Stone filesystem.

Stone filesystems are displayed in the Lustre file browser. They appear as IP addresses or host names followed by 'Wiretap server' and the originating software name and version. For example: *helsinki -- Wiretap server, Autodesk IFFFS 2008.0.0beta4*. You can navigate these directories as with any NTFS or FAT directories.

NOTE: When To Layer is enabled, DLEDLs are imported per layer according to the record TC.

▶ F:
▶ I:
▶ V:
▶ W:
▶ X:
▶ Y:
▶ Z:
▶ ball -- Wiretap server, Autodesk IFFS 2007.0.0
▶ belfast -- Wiretap server, Discreet IFFS © mprel – Product_2770 1.7.0
▶ belgium -- Wiretap server, Autodesk IFFS 2007.0.0
▶ belize -- Wiretap server, Autodesk IFFS 2008.0.0beta4
▶ bochum -- Wiretap server, Autodesk IFFS 2008.0.0beta4
▶ boston -- Wiretap server, Autodesk IFFS 2007.0.0
▶ boulogne -- Wiretap server, Autodesk Backburner 2008.0.0 (True Lies)
▶ boulogne -- Wiretap server, Autodesk IFFS 2008.0.0beta4
▶ briland -- Wiretap server, Autodesk Backburner 2008.0.0 (True Lies)
▼ bristol -- Wiretap server, Autodesk IFFS 2008.0.0beta4
▶ stonefs
▶ stonefs6
▶ stonefs7
▶ cabarete -- Wiretap server, Autodesk Backburner 2008.0.0 (True Lies)
▶ cabarete -- Wiretap server, Autodesk IFFS 2008.0.0beta4
▶ caen -- Wiretap server, Autodesk Backburner 2008.0.0 (True Lies)
▶ cairo -- Wiretap server, Autodesk IFFS 2008.0.0beta4
▶ caplan -- Wiretap server, Autodesk IFFS 2007.0.0
▶ chambly -- Wiretap server, Autodesk IFFS 2008.0.0beta4
▶ chamonix -- Wiretap server, Autodesk IFFS 2007.0.0
▶ chiba -- Wiretap server, Autodesk IFFS 2008.0.0beta4
▶ chorley -- Wiretap server, Autodesk IFFS 2008.0.0beta3
▶ cotonou -- Wiretap server, Autodesk IFFS 2007.0.0

Integrating Lustre Into the Post-Production Pipeline

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 Fire would be equally suitable.

Workflow #1

Files are first imported into Smoke from the Stone filesystem and assembled into a multi-layer timeline. Lustre then accesses the timeline as a one-sequence clip by way of Wiretap.

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 195.

In Lustre, you create local proxies, grade them, and then render the source clips back to the Stone filesystem using any of the standard Render Place controls in the Render Local menu. See [“Specifying the Destination for Local Render Files”](#) on page 443. 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

Files are first imported into Smoke from the Stone filesystem and assembled into a multi-layer Timeline. Lustre is used to access the Timeline as an Edit Decision List (EDL) clip by way of Wiretap.

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 443.

Workflow #3

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 443. 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 filesystem in order to allow Smoke users to bypass the soft-import step.

Workflow #4

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.

Detecting Wiretap Servers

You can allow Lustre to automatically detect all the Wiretap servers running on Stone filesystems in the network and display them in the browser after startup. If your facility has numerous Stone filesystems, you may notice performance slowdown at application startup while Lustre detects each one.

To set up automatic Wiretap server detection:

1. In the *init.config* or *lustre.config* file, comment out the `WtServerNoAuto` keyword. See [“Configuration File Supplemental Options”](#) on page 502.
2. Save the file.

To specify the Wiretap hosts that you would like to access:

1. In the *wt.config* file, enter the IP address of each Stone filesystem that you would like Lustre to detect. See [“Configuring Wiretap to Access Remote Stone Filesystems”](#) on page 504.
2. Save the file.

Linking to Media on the Stone Filesystem Via Wiretap

Lustre can link to clips, soft-imported clips, and soft timeline structures from Stone filesystems. Media residing on a Stone filesystem is dragged from the Lustre file browser to the timeline or Shot bin, as with any other media. This creates a reference to the full image on the filesystem instead of a copy of the image file in Lustre storage. The clip can then be used like any other media.

When referencing timelines, Lustre can interpret a Smoke or Fire 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.

Playing Clips Located on the Stone Filesystem

Clip playback speed is dictated by network bandwidth availability and by demands made on the host application, for example, Flame. To ensure maximum interactivity on high-performance IRIX- or Linux-based systems, access to Stone filesystems is assigned a lower priority function on the host machine. In a tuned Gigabit Ethernet environment, you can expect up to 80 MB/sec throughput. However, due to the uncontrolled nature of networking environments, this is not guaranteed. This speed will provide real-time SD playback, or about 1/4 real time for 2K scans. Rendering a local proxy in Lustre is recommended. See [“Generating and Viewing Proxies”](#) on page 462.

Rendering

When you have finished grading the imagery from the filesystem, you can render to the local storage array, or write back to the Stone filesystem 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 [“Wiretap-Specific Project Management Configuration”](#) on page 70. You render clips from a Stone filesystem to the local storage the same way as you would for any other clip. See [Chapter 19, “Rendering,”](#) on page 437.

Rendering clips back to the Stone filesystem via Wiretap requires that the render destination be specified in the project setup. See [“Creating and Loading Projects”](#) on page 62. 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 [“Specifying the Destination for Local Render Files”](#) on page 443.

Frames rendered to the local storage array can be accessed in the IRIX/Linux environment either by soft importing or linking them using shared storage in the Autodesk IRIX- or Linux-based product. See the “Soft Import and Publish Overview” chapter in the user’s guide for the relevant IRIX- or Linux-based system.

Supported Image Files

Lustre can access the following uncompressed image files from Stone filesystems:

- 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 clips or 16-bit soft-imported clips), which Lustre converts to 16-bit TIFF files
- Lustre can access soft-imported clips in their original format.

Although Autodesk’s IRIX products support the 16-bit RGB compressed format, Lustre does not. If directories containing this format are browsed, no media will appear.

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.

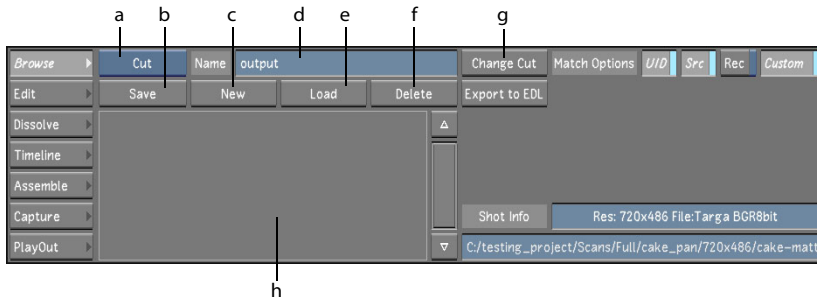
The selected shot is highlighted in the Storyboard and the positioner is updated in the Timeline.

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.

Building Cuts and Change Lists

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.

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	d) Cut name field	g) Change Cut button
b) Save button	e) Load button	h) Cut Name list
c) New button	f) Delete button	

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. Although you can set up a regular autosave interval in the Lustre configuration file, you explicitly save your cut in the Browse menu.

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 128.

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.

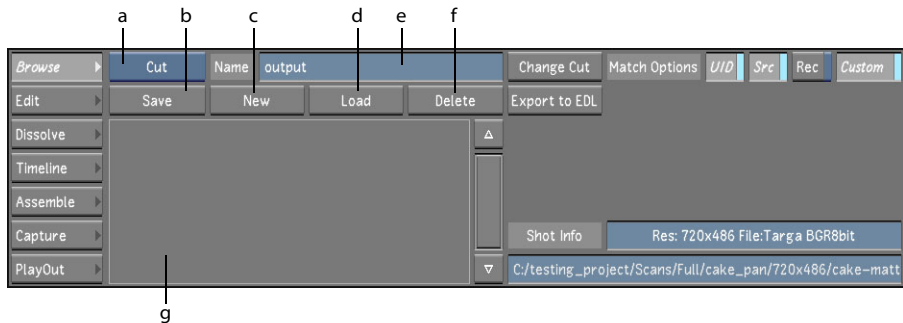
Creating a Change List

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 d) Load button g) Change List Name list
b) Save button e) Change List Name field
c) New button f) Delete button

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 CDL, 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

CDL and Lustre

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).

Grade Data

CDL data is saved in the same way as any other grade data, in a grade file.

NOTE: Grade files created in this version of Lustre are not be loadable in prior releases of Lustre.



WARNING: Users are able to import grade files from previous releases of Lustre into Lustre 2008. However, even if a grade file created in Lustre 2008 does not contain CDL data, this grade file does contain an empty section for CDL data. Therefore, it is impossible to load grade data from Lustre 2008 into previous versions.

GPU Mode

CDL data can be used in GPU mode.

CDL and Real-Time Playback

Users can perform real-time playback of media with CDL data applied, just as they are able to perform real-time playback of media with Input LUT data.

CDL and Lin/Log Architectures

CDL data can be used when working in Lin or Log architecture.

Rendering

CDL data can be processing when processing locally, using Slave rendering, or Burn rendering.

EDL Export

The Export EDL function does not support CDL data.

Playing out to VTR

CDL data can be played out to VTR as processed media in real time if using the **PlayOutFromTimeline** keyword.

Write Telecine Tape

Rendered CDL data can be output with Write Telecine tape.

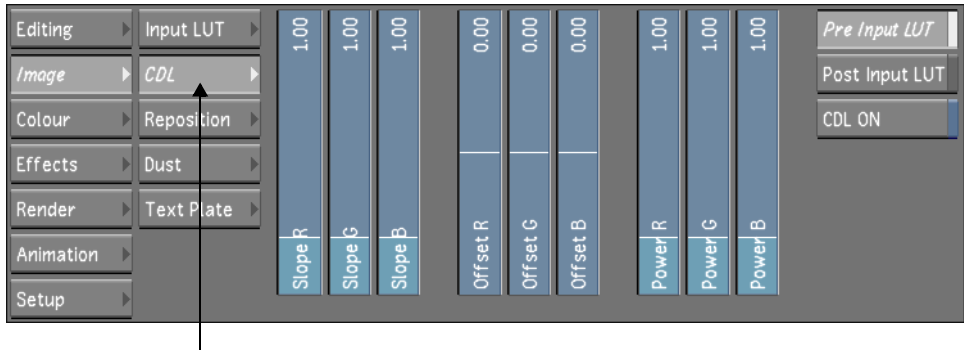
Wiretap

CDL data is not part of the Wiretap Metadata stream. There is no support for CDL in Wiretap.

NOTE: Users can apply CDL data from an EDL to a timeline loaded from Wiretap.

CDL Panel

The CDL panel is located in the Image menu, just under the Input LUT button.



The panel contains the following widgets:

- Slope (RGB sliders)
- Offset (RGB sliders)
- Power (RGB sliders)
- Pre/Post Input LUT
- CDL On/Off

NOTE: The CDL panel is not accessible from the ACS or the CP-100.

Slope

Slope (RGB) controls the following:

- Default value = 1.00 (i.e. no value)
- Minimum value is -5.00
- Maximum value is 5.00

Offset

Offset (RGB) controls the following:

- Default value = 0.00 (i.e. no value)
- Minimum value is -5.00
- Maximum value is 5.00

Power

Power (RGB) controls the following:

- Default value = 1.00 (i.e. no value)
- Minimum value is 0.0001
- Maximum value is 5.00

Pre/Post Input LUT

Users decide whether to apply CDL data 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.

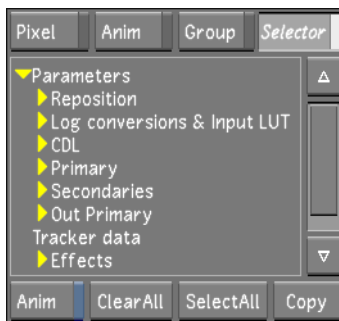
CDL On/Off

The CDL On/Off toggle allows users to either enable or disable CDL data. Since the Bypass All function does not bypass both the Input LUT and CDL, users 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



Users can use the Selector to:

- select the CDL section they 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 users 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

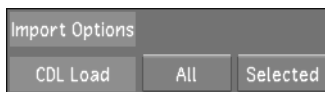
The EDL panel contains a new interface to support loading CDL data.

When users load an EDL into Lustre, the CDL data is not converted at that time. Users can conform and assemble the EDL as they normally would do. However, if users want to use the CDL data part of the currently loaded EDL, they would need to use the CDL Load options to apply the CDL data to the currently opened Cut file.

CDL Load Options

When users are ready to use CDL data, they 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 Time-Code (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.



NOTE: Loading CDL data does not erase the current grade applied to shots. Loading CDL data only loads the data into the CDL panel. If 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.

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 118.

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. In Lustre 2008, 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 118.

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>
```

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.
Mark For Capture	Flag the currently selected event for capture. See “Metadata Maintained for Missing Shots” on page 126.
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 91.
2. Load shots into the Shot bin.	"Loading Shots into the Shot Bin" on page 93.
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 118.
4. Copy the EDL into Lustre.	"Copying an EDL, ALE, or Cutlist File into Lustre" on page 120.
5. Assemble the EDL to automatically build your footage into a timeline.	"Assembling an EDL" on page 121.
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 8, "Editing," on page 153, and "Exporting Shots to an EDL" on page 129.
7. Save the assembled EDL as a cut.	"Creating a New Cut" on page 106.

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.

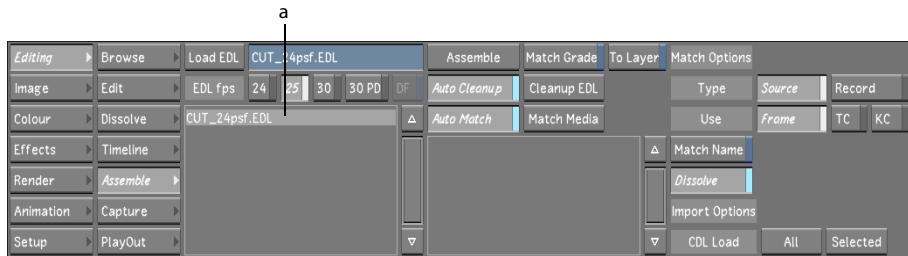
NOTE: The file must have a *.edl, *.ale, or *.ctl file extension or it will not appear in the EDL list.

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/lustre<version number>/
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.

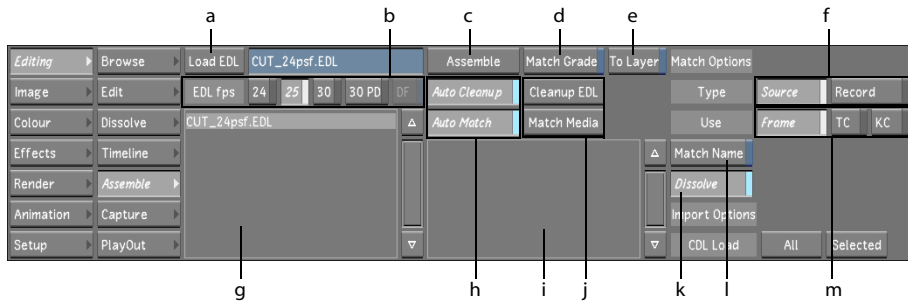
Assembling an EDL

Use an EDL to rebuild a timeline using original source footage that matches the edited timeline from the offline editing stage of post-production.

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) Load EDL button
- b) Frame Rate buttons
- c) Assemble button
- d) Match Grade button
- e) To Layer button
- f) Base of assembly buttons
- g) EDL list
- h) Assemble options
- i) Log window
- j) Manual options
- k) Dissolve button
- l) Match Name button
- m) 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 422.
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.
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, enter the DPXReelname keyword in the configuration file. See [Appendix A, “Software, Project, and Wiretap Configuration Files,”](#) on page 495.

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 124.

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 124.

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

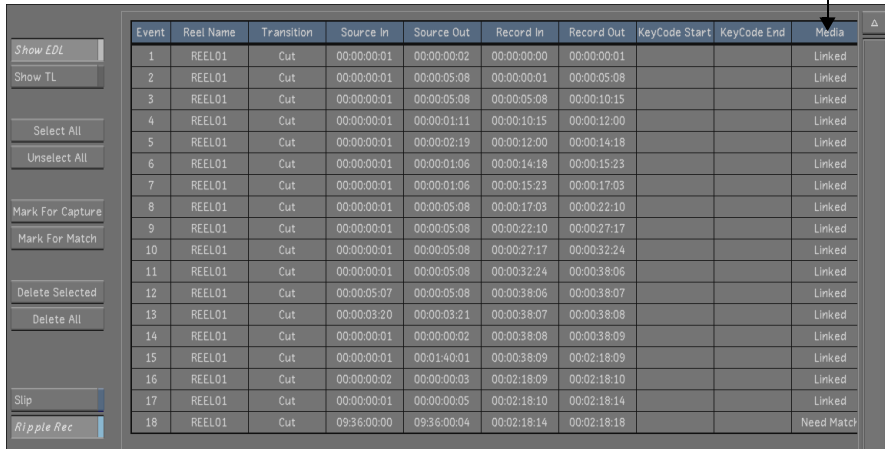
- Click EDL Cleanup to perform cleanup prior to assembling.
- Enable Auto Cleanup to automatically perform cleanup during assembly.

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	Δ
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 126.

12. 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 170.
13. Enable or disable the To Layer depending on how you want to assemble the new cut.
 - Enable To Layer to assemble a cut to a new layer above the existing layer(s) in the currently loaded cut.
 - Disable To Layer to overwrite the currently loaded cut with the newly assembled cut.
14. Click Assemble to conform the EDL and build the cut.

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

Keycode Assembly

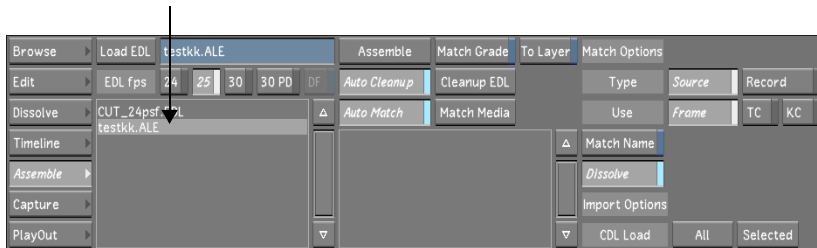
You can load ALE (*.ale) and Cutlist files (*.ctl) as EDLs in the Assemble menu. Once loaded, you can use their keycode information for media assembly.

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<version number>* on the Windows version of Lustre, or in */usr/autodesk/lustre <version number>* on the Linux version of Lustre. For example, if the missing type is FN and the keycode value is 123456, you would enter the following in the *filmtype.config* file:

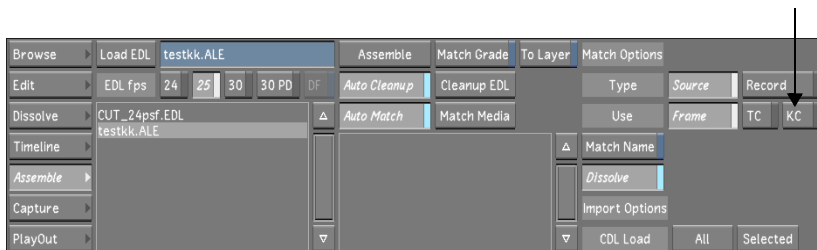
FN 123456

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.

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 126.

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 170.
11. Click Assemble to conform the ALE or CTL file and build the cut.

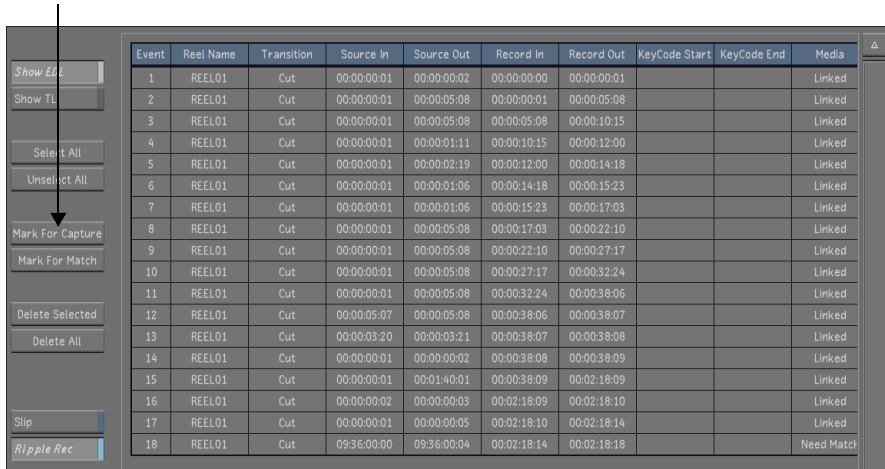
Metadata Maintained for Missing Shots

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:

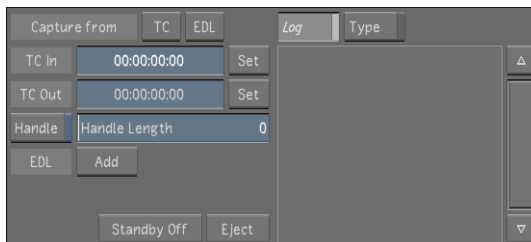
1. In the Editing menu, click Assemble to display the Assemble menu.
2. In the Assembly window, **CTRL**-click each event to be marked for capture.

- 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

- In Editing, click Capture to display the Capture menu.



Capture from TC EDL Log Type

TC In 00:00:00:00 Set

TC Out 00:00:00:00 Set

Handle Handle Length 0

EDL Add

Standby Off Eject

- Click Type to display the Type list.
- Select the file format that you want to create during the capture operation.
- Enter values in the Capture, Tape, and Resolution fields. See [“Specifying Capture Path, File Type, and Filename”](#) on page 228.
- 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.

9. In Editing, click Assemble.

The status of the newly-captured events is automatically updated to Linked in the Assembly window.

10. To assemble the EDL to include the newly-captured shots, click Assemble.

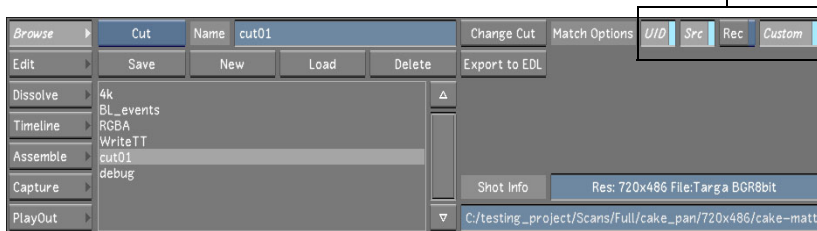
Match Grade

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. Define the metadata using the MatchCustom keyword in the configuration file. See [Appendix A, “Software, Project, and Wiretap Configuration Files,”](#) on page 495.

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.

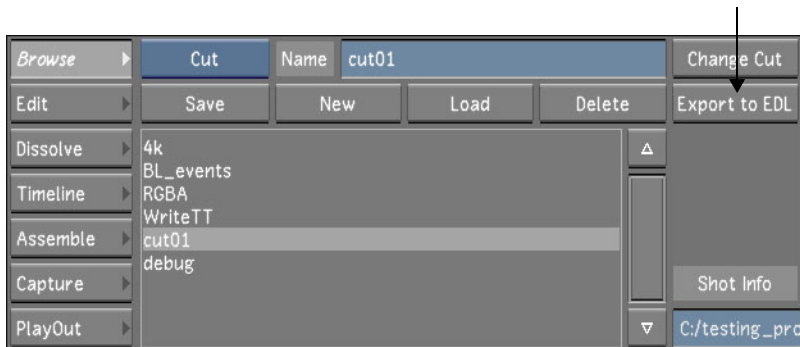
Exporting Shots to an EDL

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 106.
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.

7

Playing and Viewing Shots

Summary

About Playing and Viewing	131
Playing Shots	131
Panning and Zooming the Image	133
Setting Image View Options	134
Navigating through Shots	138
Storyboard Viewing Options	142
Viewing Reference Images	145
Viewing Multiple Shots in the Player	147
Caching Frames to Memory for Real-Time Playback	150
Player Hot Keys	151

About Playing and Viewing

Lustre offers multiple options for playing and viewing 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. 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 97.

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.

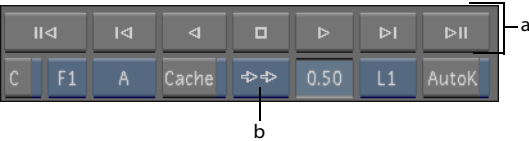
When you play shots in the Player, you can do the following:

- 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 40.

To play shots in a cut:

1. Load your shots into the timeline. See [“Loading a Cut”](#) on page 106.
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’s 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 138.

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.

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 151 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's 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 19, “Rendering,” on page 437. 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 329.
S	L	The stored reference image. See “Viewing Reference Images” on page 145.

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 462.

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 greyscale. The greyscale versions of colour channels represent in values of grey the amount of that colour channel found in each part of the image. The darker the grey, the greater the amount of colour present.

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 56.

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 145.

To display Colour menus in the Player:

- Press **F7**.

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

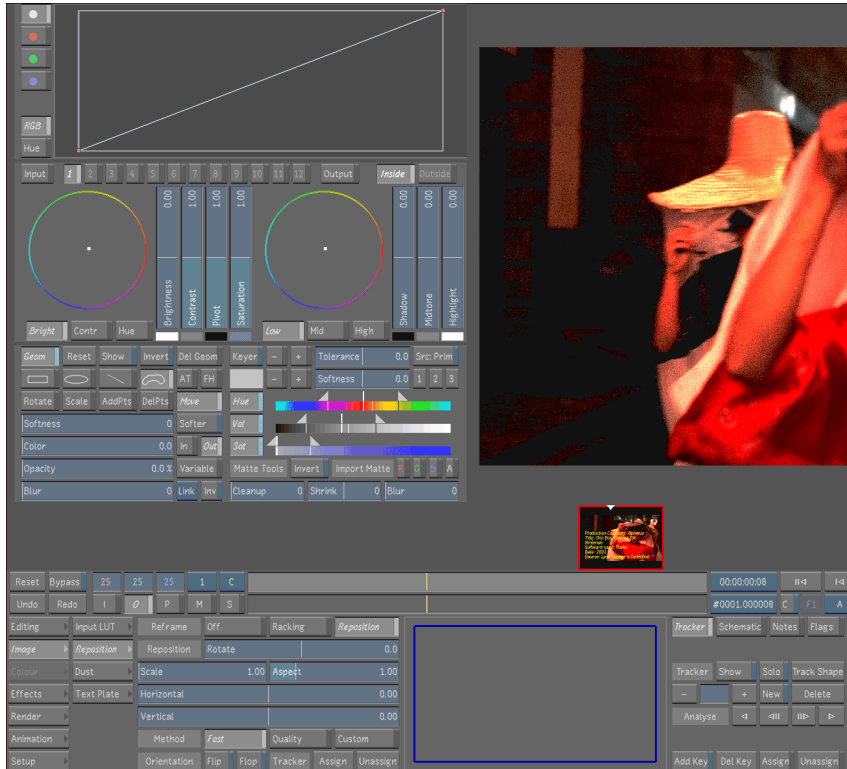


Image courtesy of Optimus

Navigating through Shots

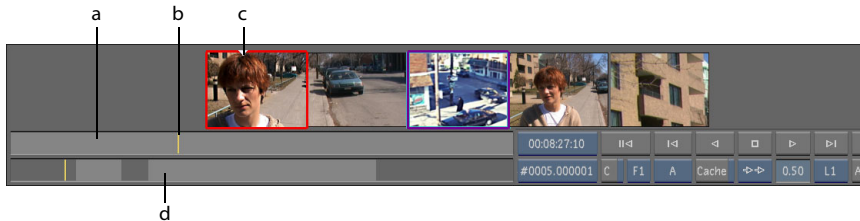
Various methods are available for navigating through the shots in your cut. You can navigate using:

- 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's 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:

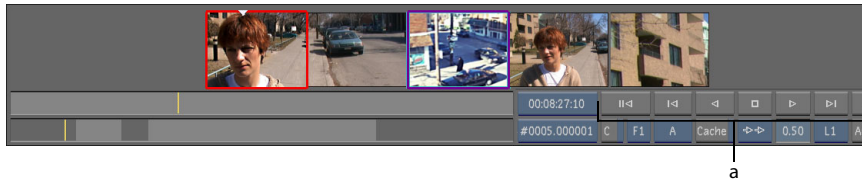
- Place the cursor over any thumbnail, press the middle mouse button, and drag to the left or right.

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.

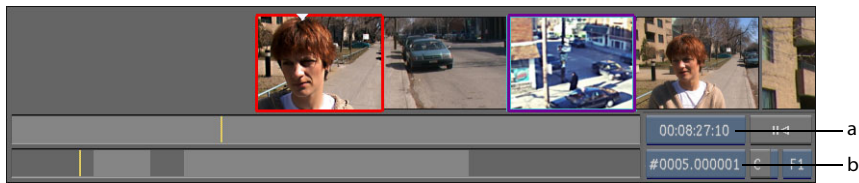


a) Playback controls

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

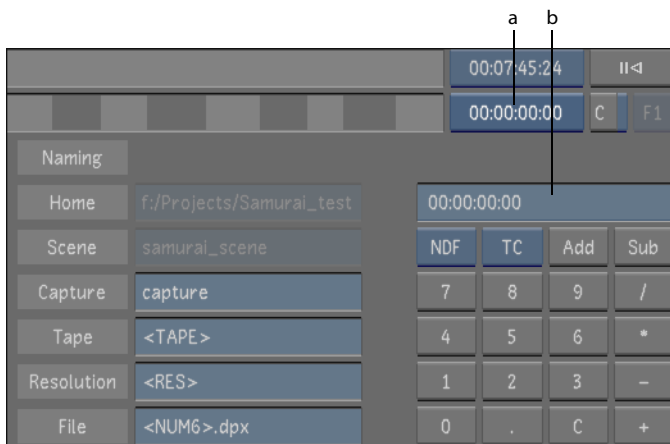
b) Timeline Information field

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 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.

The Storyboard was previously called the ‘timeline’. For more information about nomenclature, see [“‘timeline’ vs ‘Timeline’ vs ‘Storyboard’”](#) on page 175.

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 or multiple shots in the Storyboard. Multi-select shots to copy parameters, delete multiple shots, and create groups.

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 shot.

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.

NOTE: Storyboard thumbnails are stored in a sub-directory of the scans directory. The thumbnail directory is named *128x96*.

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**.

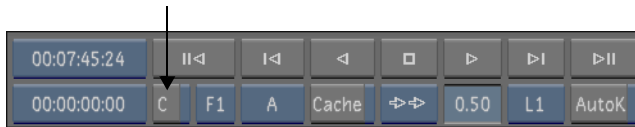
Collapsing the Storyboard

You can collapse the Storyboard so that only the selected shots are visible. This is useful to isolate certain types of shots for grading, such as outdoor scenes.

You can toggle back to the original Storyboard at any time.

To collapse the Storyboard in order to include only the selected shots:

1. Select the desired shots.
2. Press **F** on the keyboard, or enable **C** to toggle between the collapsed Storyboard and the original.



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.

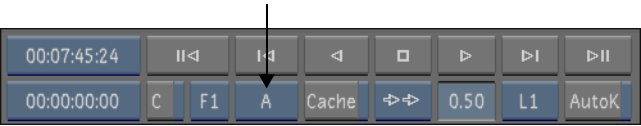
It is recommended that you work on your shot in Playhead A and load the reference images into Playhead B. For example, in Playhead B, you can view a reference image from the shot adjacent to the one you are currently working on in Playhead A. Visually compare the two images to match the saturation, whites 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's 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 147.

To view a reference image loaded from a cut:

1. Press **F1** or select A from the Playhead button and then select the shot in the Storyboard.



The shot you are working on is assigned to Playhead A.

2. In the Storyboard, go to the frame you want to use for the reference image.
3. Press **K** to store the frame in the reference buffer.

The current frame overwrites the last frame loaded into the reference buffer.

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

The shot is assigned to Playhead B.

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



The reference image appears in the Player.

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).

6. To work on the shot you assigned to Playhead A, press **F1** or select A from the Playhead button.

The shot you are working on is selected. You can move back and forth between the shots in Playhead A and Playhead B by pressing **F1** and **F2**, respectively.

To view a reference image loaded from a Grade bin:

1. Press **F1** or select A from the Playhead button and then select the shot in the Storyboard.
The shot you are working on is assigned to Playhead A.
2. In the Grade bin, right-click the frame you want to use for the reference image.
The frame from the Grade bin overwrites the last frame you loaded into the frame buffer.
3. Press **F2** or select B from the Playhead button.
The shot is assigned to Playhead B.
4. Press **L** or click the S button to display the reference image in Playhead B.
The reference image appears in the Player.

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

5. To work on the shot you assigned to Playhead A, press **F1** or select A from the Playhead button.
The shot you are working on is selected. You can move back and forth between the shots in Playhead A and Playhead B by pressing **F1** and **F2**, respectively.

To switch between a reference image and the shot you are working on:

1. Load the reference image to the frame buffer. Do one of the following:
 - Press **K** to load the current frame in the Storyboard to the frame buffer.
 - Right-click a Grade bin thumbnail.
2. In the Storyboard, select the shot you are working on.
The shot appears in the Player.
3. To toggle between the reference image and the shot you are working on, press **L**.

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.

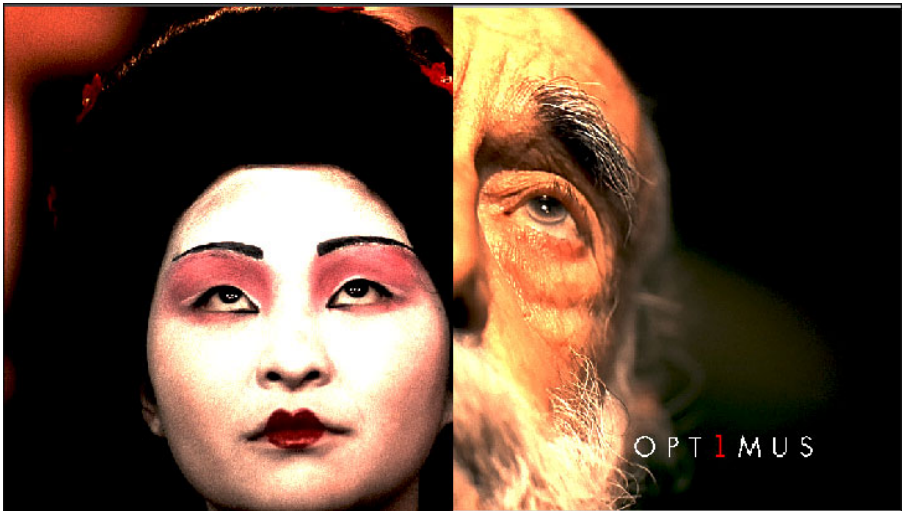
To view a reference image loaded from a cut:

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.



Images courtesy of Optimus

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

The images assigned to Playhead A and Playhead B appear.

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 143.

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

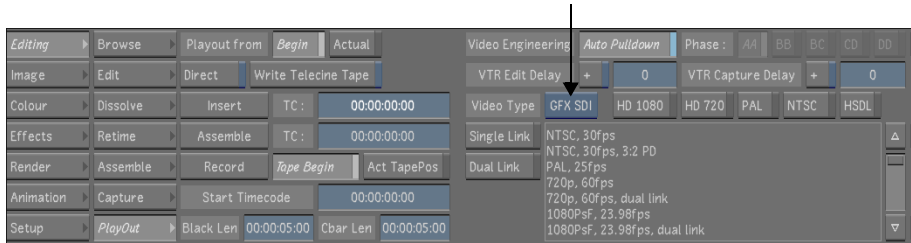
You can use split view and multi-view in the SDI monitor.

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.

NOTE: The way the frame displays depends on the parameter specified by the DualType keyword in the configuration file. See [“Configuration File Default Options”](#) on page 496.

6. To view multiple shots on the SDI monitor, perform the procedure for viewing multiple shots in the Player.

Caching Frames to Memory for Real-Time Playback

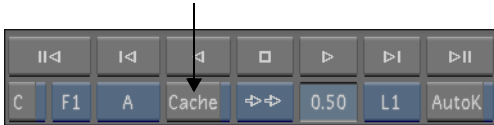
After applying many effects to a shot, you may no longer get real-time playback. To play shots with many effects in real time, you can render and temporarily cache the rendered frames into memory.

You can also navigate through shots with the control surface. See the *Autodesk Control Surface User's 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 19, “Rendering,”](#) on page 437 and [“Setting the View Mode”](#) on page 134.

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 131.
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.

Press:	To:
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.
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 R, G, and B colour channels together in the waveform monitor.
ALT+3	Display the histogram.
ALT+4	Display the vectorscope.

8

Editing

Summary

About Editing	153
Accessing the Edit Menu	154
Editing with the Storyboard	156
Trimming	159
Changing Playback Speed	169
Replacing Shots	170
Creating Dissolves	171
Adding a Virtual Black Clip	174
About Timeline	175
Timeline View	176
Accessing the Timeline Menu	176
Editing in the Timeline	181
Trimming the Timeline	186
Transitions	190
EDL and Wiretap	193
Rendering the Timeline	193
Modifying the Timeline's Starting Timecode	194
Performing a Confidence Check	194
Using Scene Detection	195

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 will appear 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.

Timeline editing is useful for 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 will be used in the final master. For a more detailed description of timeline, the multi-layered Timeline, and the Storyboard, see [“‘timeline’ vs ‘Timeline’ vs ‘Storyboard’”](#) on page 175.

Accessing the Edit Menu

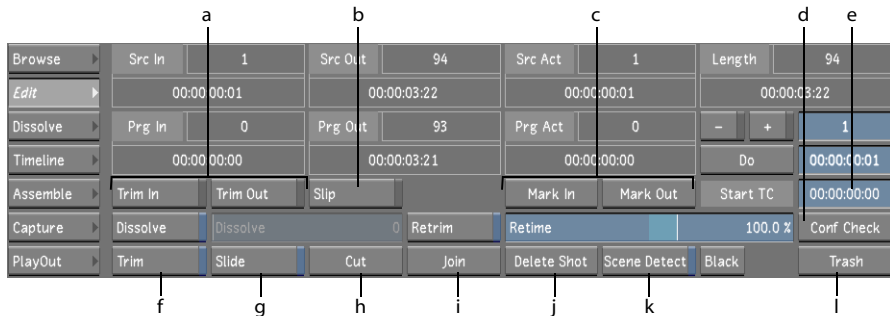
To modify shots in the timeline, you use the Edit menu.

NOTE: If a layer has been soloed, the Edit tools (Trash, Delete, Trim In/Trim Out, and Mark In/Out) will 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 tab. For more information about Ripple Modes, see [“Trimming Ripple Mode”](#) on page 187.

To access the Edit menu:

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

The Edit menu appears.



- a) Trim In/Trim 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/Mark 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 explicitly enter a 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 160.

Slip — Moves the source frames within a shot without adjusting the shot’s position in the timeline. See [“Slipping a Shot”](#) on page 164.

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 166.

Start TC — Sets the frame number at which you would like the timeline to begin. The frame number is converted into the corresponding timecode value. See [“Modifying the Timeline’s Starting Timecode”](#) on page 194.

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 165.

Slide — Slides a cut point forward or backward. See [“Sliding the Cut Point between Shots”](#) on page 166.

Cut — Cuts one shot into two at the Current Frame positioner. See [“Inserting Cuts”](#) on page 167.

Join — Joins two shots coming from the same source clip with consecutive source timecode. See [“Joining Shots”](#) on page 168.

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 195.

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 194.

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 enter the EditingEverywhere keyword in the configuration file, 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 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.

See [Appendix A, “Software, Project, and Wiretap Configuration Files.”](#) on page 495.

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 top-most 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. See [“Accessing the Timeline Menu”](#) on page 176.

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.

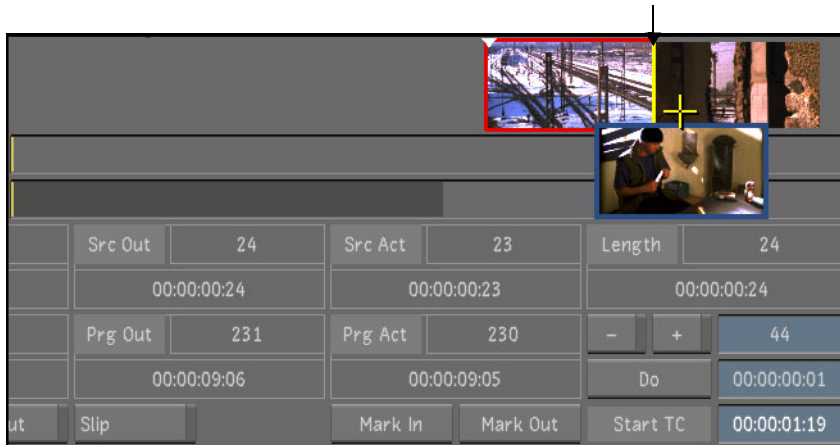


Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

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.

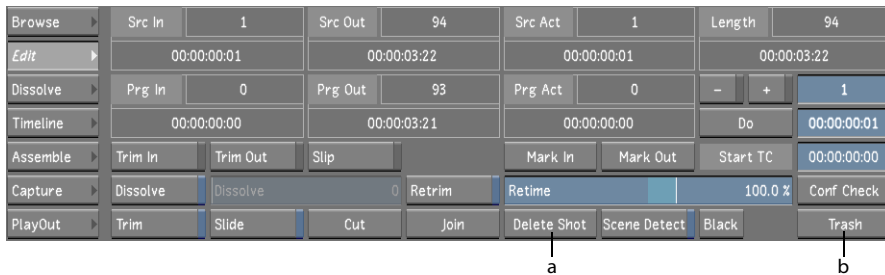
HINT: If you select the wrong shots, click the grey area to the left of the Trash button. This releases the selected shots from the cursor.

HINT: To insert a duplicated shot to the insertion marker location of the Storyboard, you must enable Solo mode in the Timeline menu. See [“Accessing the Timeline Menu”](#) on page 176. Otherwise, the duplicated shot will be inserted at the end of the Storyboard.

Deleting Shots

You can delete shots from the Shot bin, Storyboard, or Timeline (see [“Deleting Elements in the Timeline”](#) on page 184). Deleting shots from the Shot bin does not delete them from the Storyboard or 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 will 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 187) and will 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

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 will 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 187 and [“Slip & Slide Ripple Mode”](#) on page 189) and will 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.

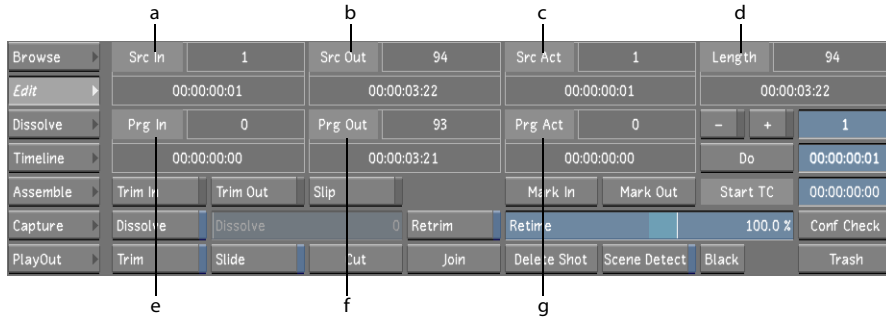
Viewing Shot Information

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 d) Length field g) Prg Act field
 b) Src Out field e) Prg In field
 c) Src Act field f) Prg Out 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 will 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 158.

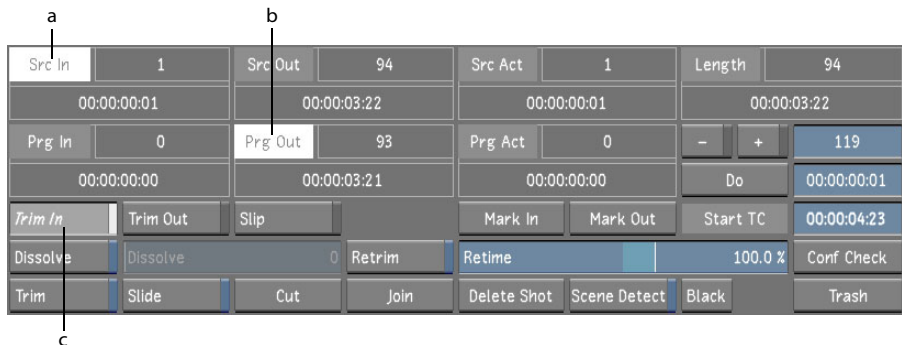
To trim frames at the in point for one or more selected shots:

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, and then click Edit.

The Edit menu appears.



a) Src In field

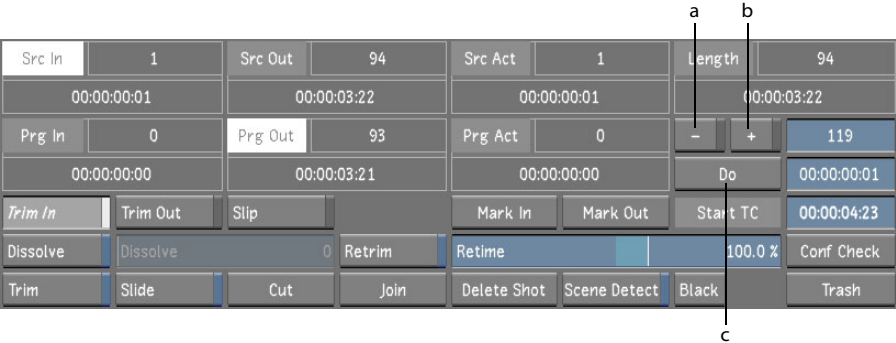
b) Prg Out field

c) Trim In button

3. Click Trim In to go into Trim In mode.

The Src In and Prg Out fields are highlighted because these values are changed by the Trim In operation.

- Add frames to or remove frames from the mark in (or head):
 - Click + to add frames, and then enter the number of frames to add using the numeric keypad.
 - Click - to subtract frames, and then enter the number of frames to subtract using the numeric keypad.



- a) - (minus) button
- b) + (plus) button
- c) Do button

NOTE: If you do not click either + or -, the trim operation removes an absolute number of frames.

- Click Do.

The specified number of frames is added to or subtracted from the shot or shots. The Length field is updated to display the number of frames now used.

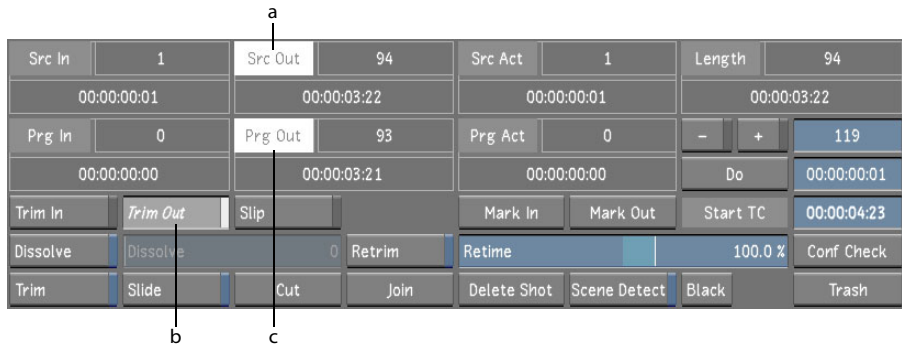
To trim frames at the out point for one or more selected shots:

- 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.

- Click Editing, and then click Edit.

The Edit menu appears.



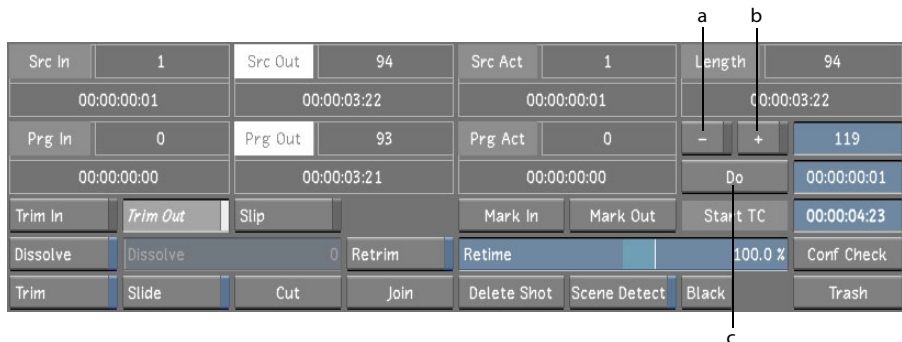
- a) Src Out field b) Trim Out button c) Prg Out field

3. Click Trim Out to go into Trim Out mode.

The Src Out and Prg Out fields 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, and then enter the number of frames to add using the numeric keypad.
- Click - to subtract frames, and then enter the number of frames to subtract using the numeric keypad.



- a) - (minus) button b) + (plus) button c) Do button

NOTE: If you do not click either + or -, the trim operation removes an absolute number of frames.

5. Click Do.

The specified number of frames are added to or subtracted from the shot. 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.

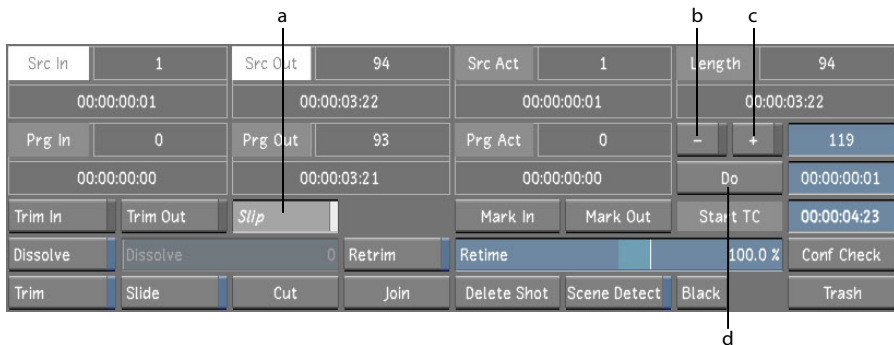
To slip one or more shots:

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, and then click Edit.

The Edit menu appears.



- a) Slip button
- b) - button
- c) + button
- d) Do button

3. Click Slip to go into Slip mode.

The Src In and Src Out fields are highlighted.

4. Slip forward or backward:

- Click + to slip forward, and then enter the number of frames by which you want to offset the shot.
- Click - to slip backward, and then enter the number of frames by which you want to offset the shot.

5. Click Do.

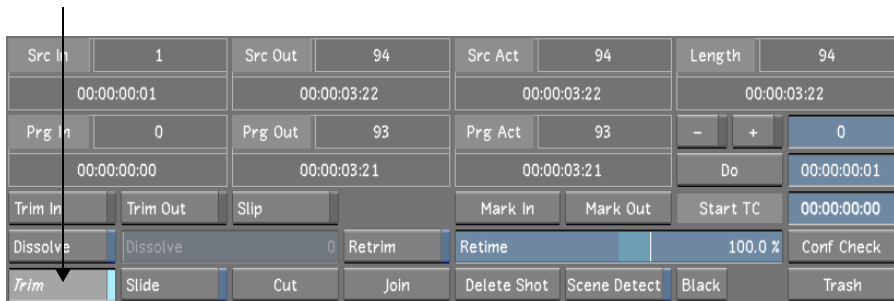
The shot or shots are offset by the specified number of frames. The Src In and Src Out fields are updated to display the frames used.

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, and then click Edit to display the Edit menu.
3. Enable Trim.



HINT: If the `EditingEverywhere` keyword is entered in the configuration file, you can start interactively trimming shots from any menu by pressing **SHIFT+T**. See [Appendix A, “Software, Project, and Wiretap Configuration Files,”](#) on page 495.

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. Click Trim again to exit Trim mode and complete the trim operation.

Changing In and Out Points for a Shot

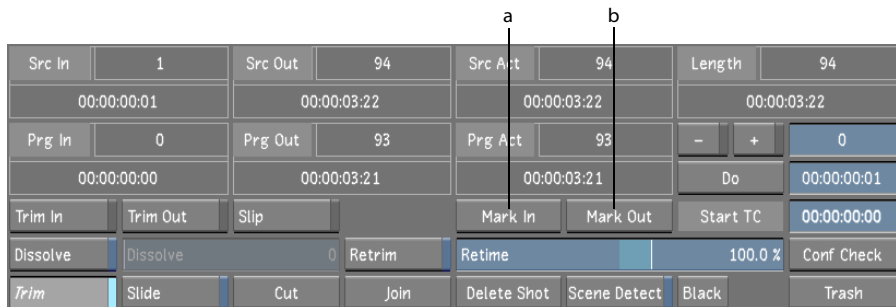
You can change the in and out points for a shot. This is like trimming, except you can only shorten a shot by changing the in or out points.

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.

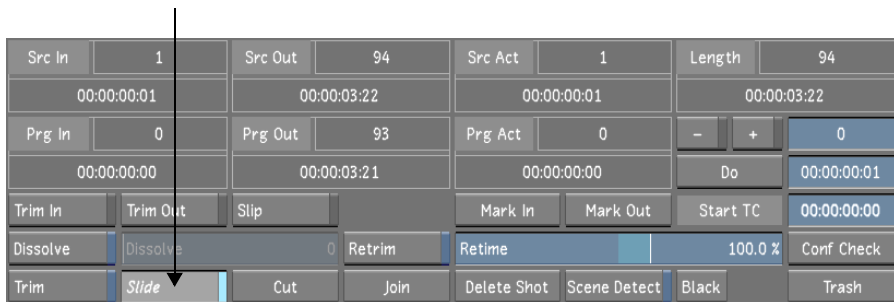
Sliding the Cut Point between Shots

You can interactively move the cut point between two shots forward or backward, if handles are available. Using Slide will 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 EditingEverywhere keyword is entered in the configuration file, you can start interactive visual sliding between the current and the previous shot from any menu by pressing **SHIFT+R**. See [Appendix A, “Software, Project, and Wiretap Configuration Files,”](#) on page 495.

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. Click Slide again to exit Slide mode and complete the operation.
The timeline is updated when you exit Slide mode.

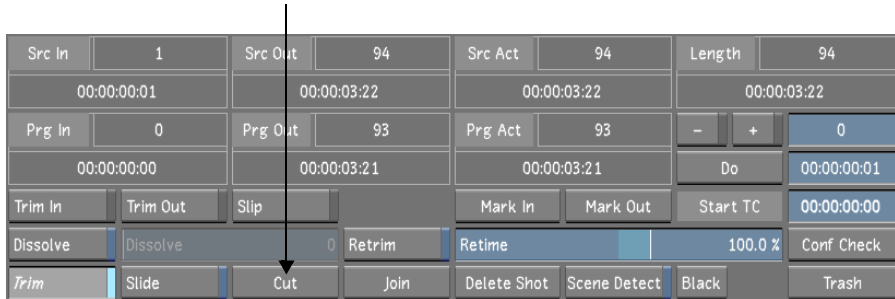
Inserting Cuts

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 `EditingEverywhere` keyword is entered in the configuration file, you can insert a cut from any menu by pressing **SHIFT+C**. See [Appendix A, “Software, Project, and Wiretap Configuration Files,”](#) on page 495.

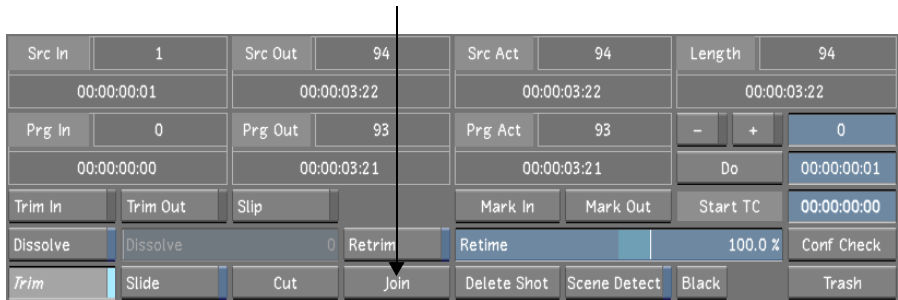
Joining Shots

You can rejoin a shot that has been 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.

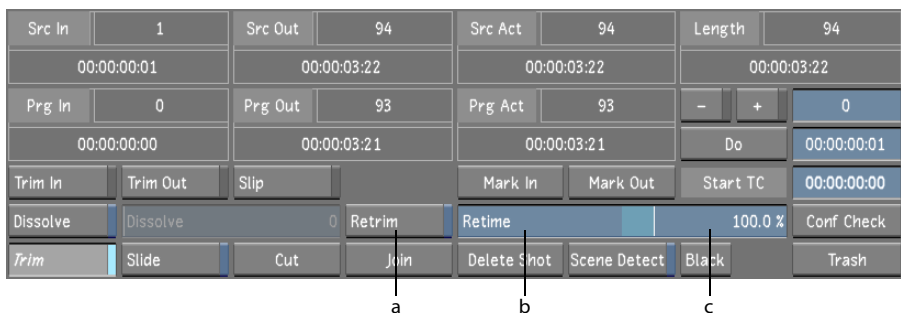
HINT: If the EditingEverywhere keyword is entered in the configuration file, you can join shots from any menu (except the Colourist Timeline) by pressing **SHIFT+J**. See [Appendix A, "Software, Project, and Wiretap Configuration Files,"](#) on page 495.

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 will change shots. This will be 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”](#) on page 159.

4. Drag the Retime slider to the new playback speed or enter the new playback speed 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.

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 will appear in the viewer for these frames.

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.

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

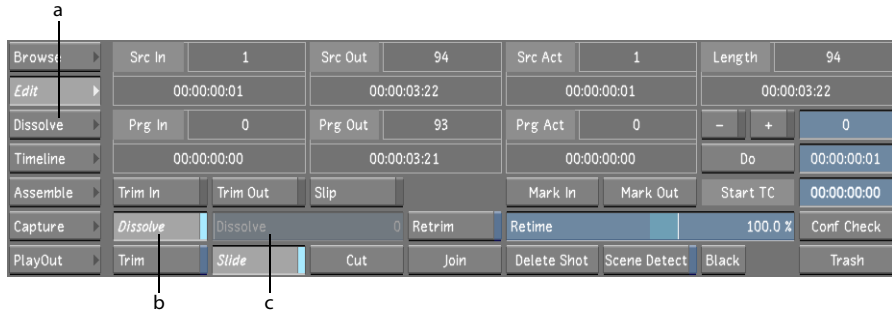
You can add a dissolve between two shots provided that there are a sufficient number of underlying handles.

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 173.

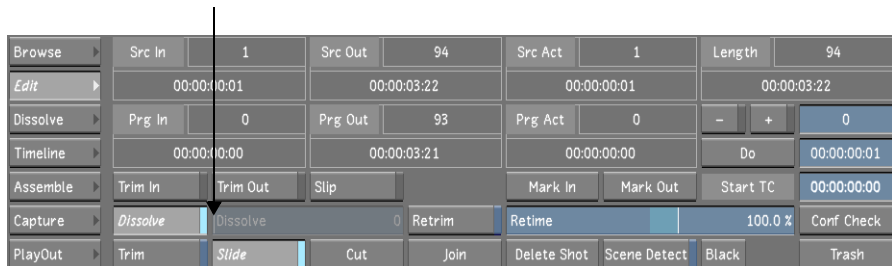
3. Enable Dissolve, and then drag the Dissolve slider to set the dissolve duration.
4. To edit the shape of the Dissolve curve, see [“Editing Dissolves”](#) on page 173.

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.

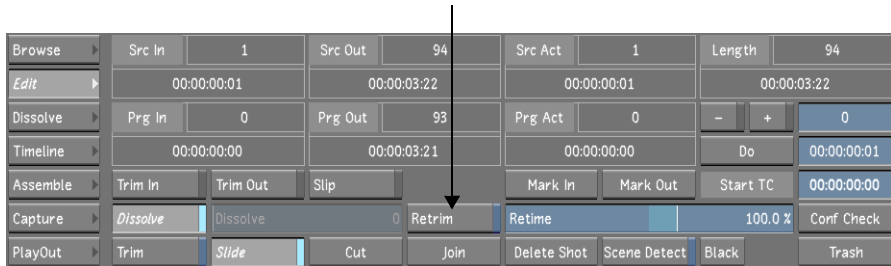


Creating Additional Handles for Dissolves

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.

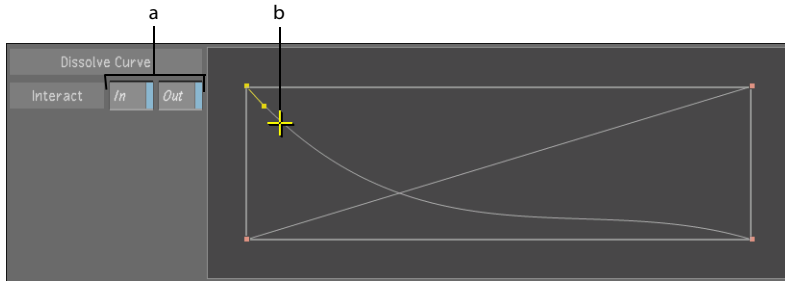
Editing Dissolves

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

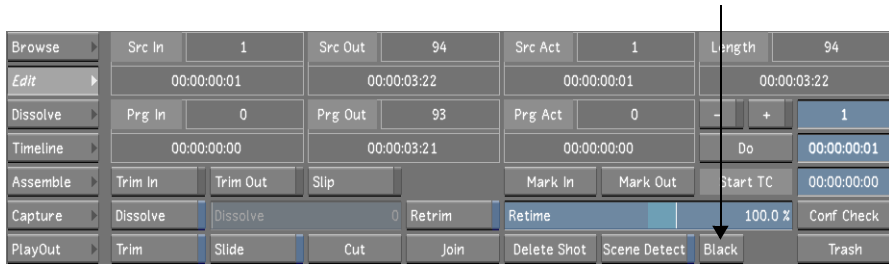
Virtual black clips can be added anywhere on the timeline. The default duration of each clip is one second according to the current project frame rate.

To add a virtual black clip:

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 [“Accessing the Timeline Menu”](#) on page 176. If Solo mode is disabled, the black clip is inserted at the end of the Storyboard or Timeline.

2. In the Edit menu, click Black.



3. Move the clip to the Storyboard.
A yellow insertion marker appears at the cut where the black clip will be entered.
4. Insert the clip on the Storyboard.

NOTE: You can change the virtual clip's colour through the configuration file. See [Appendix A, "Software, Project, and Wiretap Configuration Files,"](#) on page 495.

About Timeline

The Timeline menu organizes the cut or sequence into a mutiple 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. You can scroll, pan, and zoom the timeline, as well as rearrange elements on the timeline by dragging and dropping. Timeline editing is useful for roughing out edit sequences and for multi-layer editing. There are some editing features that are specific to Timeline. There is also a Colourist's Timeline in the Grading menu with limited editing function. See [Chapter 13, "Colour Grading: Basics,"](#) on page 275.

'timeline' vs 'Timeline' vs 'Storyboard'

In past versions of the User's Guide, the term 'timeline' was used to describe both the sequence of shots in a cut file, as well as the thumbnail view of the shots. Beginning in Lustre 2008, there are now two visual representations of the sequence of shots in a cut file:

- the multi-layer Timeline (or Timeline)
- the Storyboard

The Timeline view (note the upper-case 'T') displays multiple layers within the cut. See ["Timeline View"](#) on page 176.

The timeline (note the lower-case 't') continues to represent the concept of a sequence of shots, otherwise known as a 'cut' or 'cut file'.

The Storyboard is the new name for the thumbnail view of shots that represent a single-layer view of the timeline. That is, the Storyboard displays either the active Timeline layer or the flattened view of the multi-layer Timeline (see [“Storyboard Viewing Options”](#) on page 142). The single layer of the Storyboard can represent either one of the following:

- the active layer of a 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

Timeline View

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

- Storyboard. See [“Storyboard Viewing Options”](#) on page 142.
- Timeline. See [“Accessing the Timeline Menu”](#) on page 176.
- File browser. See [“Accessing the File Browser”](#) on page 91.
- Shot bin. See [“Accessing the File Browser”](#) on page 91.
- Player and full-screen Player. See [“The Player”](#) on page 15.

Accessing the Timeline Menu

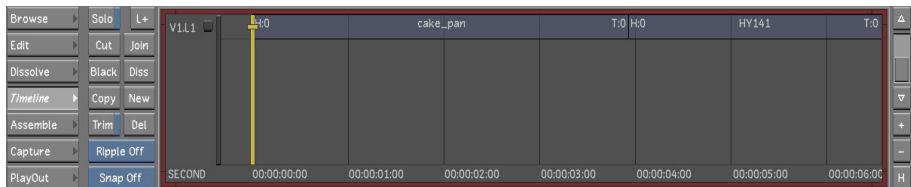
To edit shots in the Timeline view, you need the Timeline menu.

NOTE: You can also access some of the Timeline features through the Colourist’s Timeline. See [Chapter 13, “Colour Grading Basics,”](#) on page 275.

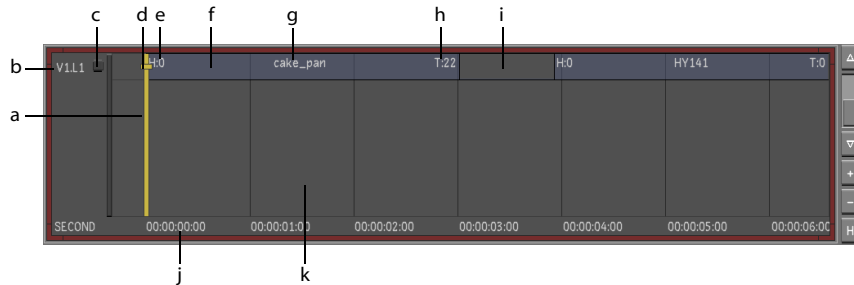
To access the Timeline menu:

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

The Timeline menu appears.



The Timeline is made up of the following components.



a) Positioner	d) Focus point	g) Element name	j) Time scale
b) Layer name	e) Head	h) Tail	k) Canvas
c) Layer selector	f) Element	i) Gap	

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

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.

Layer selector — Selects the entire layer in preparation for a delete, copy, move, slip, or trim 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.

Head — Number of available frames for trimming, preceding the transition.

Element — The media in the timeline.

Element name — The name of the element based on the clip name.

Tail — Number of available frames for trimming, following the transition.

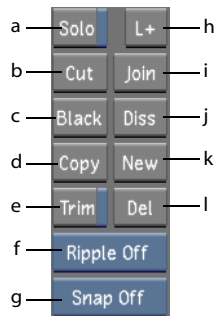
Gap — The absence of media. In multi-layer context and when Solo mode is disabled, gaps are also transparent.

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

Canvas — Click the canvas to zoom in/out and move the timeline.

For the hot keys to these functions, see [Chapter 20, “Hot Keys.”](#) on page 479.

The Timeline menu is made up of the following buttons.



a) Solo button	d) Copy button	g) Snap mode button	j) Diss button
b) Cut button	e) Trim button	h) L+ button	k) New button
c) Black button	f) Ripple mode button	i) Join button	l) Del button

Solo — Within a multi-layered 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.

NOTE: If a layer has been soloed, the Edit tools (Trash, Delete, Trim In/Trim Out, and Mark In/Out) will 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 tab. For more information about Ripple Modes, see [“Trimming Ripple Mode”](#) on page 187.

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 — Enables/disables the Trim mode.

Ripple mode — Determines whether the length of the edit sequence is affected when new material is gesturally 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.

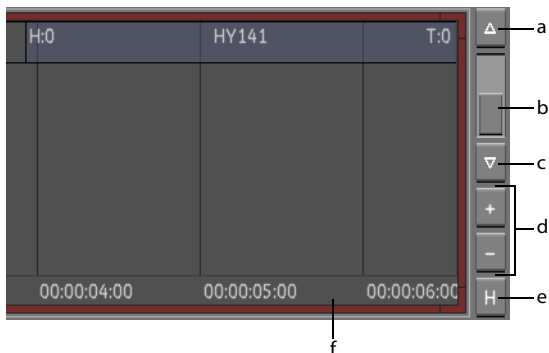
Del — Deletes the current selection.

Timeline Navigation Controls

The following is a list of hot keys and mouse gestures commonly used to navigate within the Timeline menu. See [Chapter 20, “Hot Keys.”](#) on page 479.

Press:	To:
RIGHT ALT (right hand side of keyboard)	Go to previous frame.
RIGHT CTRL (right hand side of keyboard)	Go to next frame.
Left arrow	Go to previous element.
Right arrow	Go to 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 Timeline.
Right mouse button and drag	Zoom into or out of the Timeline.

Additional Timeline navigation controls are shown below.



- a) Move Timeline up
- b) Vertical scroll bar
- c) Move Timeline down
- d) Vertical zoom control
- e) Timeline home view
- f) Scrub area

Move Timeline up — Displays the top layers located outside of the Timeline view.

Vertical scroll bar — Quickly moves the Timeline view up or down.

Move Timeline down — Displays the bottom layers located outside of the Timeline view.

Vertical zoom control — Enlarges or reduces the size of the layers.

Timeline home view — Resets the vertical/horizontal zoom and position of the Timeline view to display the entire Timeline.

Scrub area — Click or drag in this area to quickly move to any point in the Timeline.

Viewing Element Information

It is possible to expand the size of a layer to see more information about the element.

To expand the layer to see more information:

- Click +.

More information about the element is revealed.

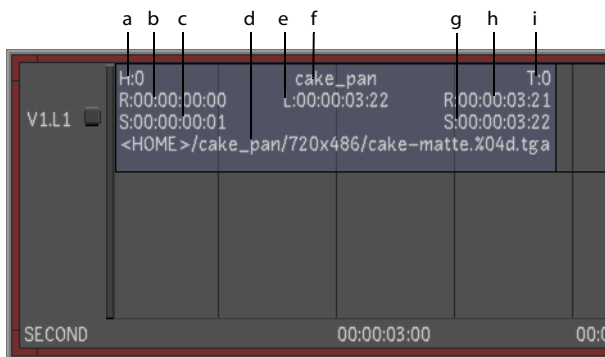
NOTE: Each time you press +, more information is revealed.

To minimize the layer of information:

- Click -.

Information about the element is concealed by the layer so other layers can be visible.

The following information is available through the layer.



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, assemble an EDL, or create or load a cut, 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 Timeline or to the Storyboard view. See [“Adding Shots to the Storyboard”](#) on page 156.

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 in the Timeline

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. Select the element you want to move.
The selected element becomes yellow.
2. Drag the element and drop it at the destination.

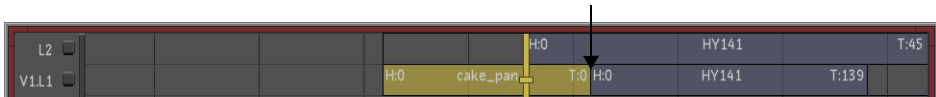
NOTE: The status of Ripple and Snap modes are taken into account when dropping elements.

Snap mode

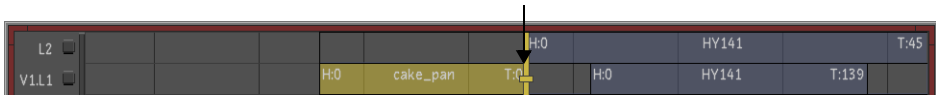
Snap mode affects how elements are placed when gesturally moved or copied to the Timeline. Snap mode consists of the following three options.

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

Snap Trn — Snaps the element to the closest transition



Snap Pos — Snaps the element to the positioner.



Copying Elements in the Timeline

You can copy one or more elements in the Timeline 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 in the Timeline within a layer:

1. Select the Timeline element(s) you want to copy.
The copied element(s) become red.
2. Click the Copy button.
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. Select the Timeline element you want to copy.
2. Click the Copy button.
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 on the Timeline.
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. 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 the Copy button.
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 182.

Adding Black Media to the Timeline

You can add black media to the Timeline to create a space between elements. By default, a black clip is one second in duration, based on the project’s frame rate.

To create black media:

1. Click Black.

A black shot appears.

2. Drag the black shot to either the Storyboard or Timeline and position the cursor where you want the black media to appear.

NOTE: If you are inserting it into the Storyboard, the yellow insertion line tells you where the black media will appear (Solo mode enabled only). If Solo mode is disabled, the black media will appear at the end of the sequence.

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 in the Timeline

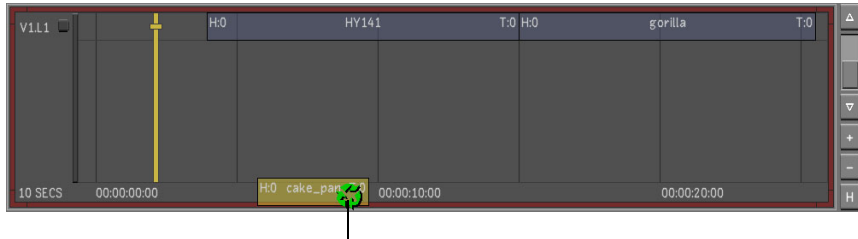
You can delete selected elements in the Timeline in numerous ways.

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 181), click Del, and then confirm the deletion.
 - Select the element(s) (see [“Selecting Elements”](#) on page 181), 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 181), 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 158. The Timeline immediately reflects the changes.

Muting Layers Within the Timeline

There are a couple of ways you can mute a layer within the Timeline.

NOTE: If you are using the mute 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.

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.

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 Timeline. A red X is displayed in the Player when the positioner is located on a gap.

When working with a multi-layer Timeline, a gap located on the top layer shows the elements located below (Solo mode disabled only). This is called vertical editing.

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 187). Their length can also be increased or decreased by modifying the clips around them. Gaps can be selected, copied, or deleted.

Trimming the Timeline

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 view. 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.

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 185). 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 189.

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 has been soloed, the Edit tools (Trash, Delete, Trim In/Trim Out, and Mark In/Out) will 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 tab. For more information about Ripple Modes, see [“Trimming Ripple Mode”](#) on page 187.

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 187.
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

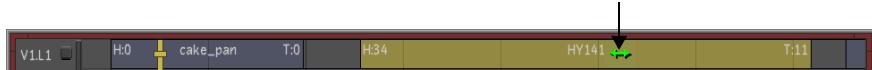
Slipping simultaneously trims the head and tail of the element without changing its position in the sequence. The duration of the shot is not altered. Other elements in the edit sequence are not affected and the total duration of the sequence does not change. You can slip a single element or a selection.

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 190.

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.

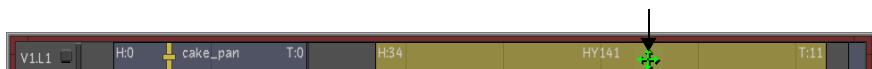
To slide an element:

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.

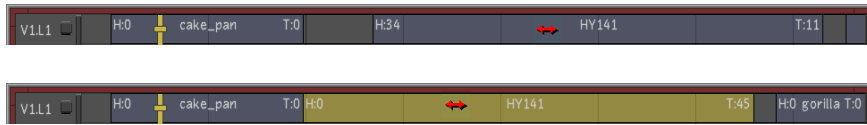
Ripple Off — With Ripple Off, only the first and last shot of the Timeline and any shot surrounded by a gap, can slip and slide. The slip & slide operation adds a gap and changes the length of the edit sequence.



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.

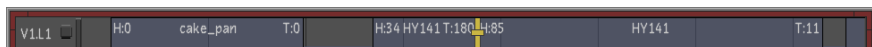
To add a splice:

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.

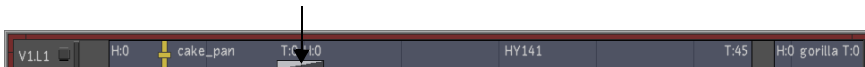
Adding a Dissolve

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 will display 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 172.

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 192.

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 will go from the outgoing element to black and from black to the upcoming element. Trimming or slipping the elements may help solve the problem.

EDL and Wiretap

Many grading projects involve working with an EDL or shots created by Inferno, Flame, Flint, Fire, or Smoke through Wiretap.

EDL

Previously, when you assembled an EDL, it replaced the current cut with the newly assembled EDL. Now, it is possible to add the assembled EDL to a new layer of the current Timeline. See [“Working with CDL, EDL, ALE, and Cutlist Files”](#) on page 110.

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. 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.

Wiretap

Previously, loading a DLEDL from Wiretap replaced the currently loaded cut. Now, it is possible to add a DLEDL to a new layer of the current Timeline. When you drag and drop the DLEDL to the Timeline or the Storyboard, a layer is created above the existing layer(s) at its record timecode position (not at the drop point), and with the content of the DLEDL. See [“Accessing Remote Footage by Way of Wiretap”](#) on page 100.

Rendering the Timeline

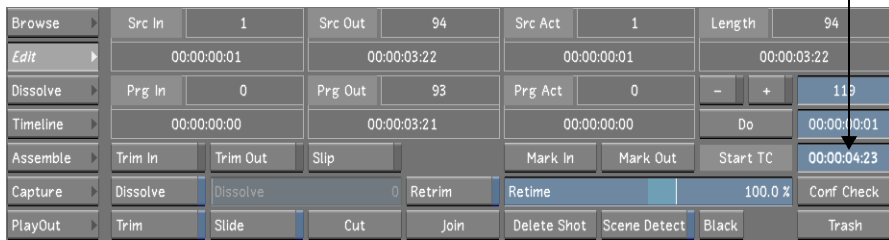
There are numerous scenarios when you render a multi-layer Timeline. For more information on rendering a multi-layer Timeline, see [Chapter 19, “Rendering,”](#) on page 437.

Modifying the Timeline's Starting Timecode

You can change the Timeline's start timecode. Modifying the start timecode does not delete frames that precede the new start frame.

To modify a shot's starting timecode:

1. Click Editing, and then click Edit to display the Edit menu.
2. Click in the Start TC 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.



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 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, incorrect frames are used in the Timeline, your picture could end up out of sync with the sound.

During a confidence check, if you determine that there are shots using incorrect frames, you can use the Editing tools (trim, slip, and slide) to adjust the shots to the correct frames.

To perform a confidence check:

1. If you have not already done so, load the scanned frames into the footage directory.



WARNING: If you do not save the current Storyboard in the following step, it is replaced by the material captured for the confidence check.

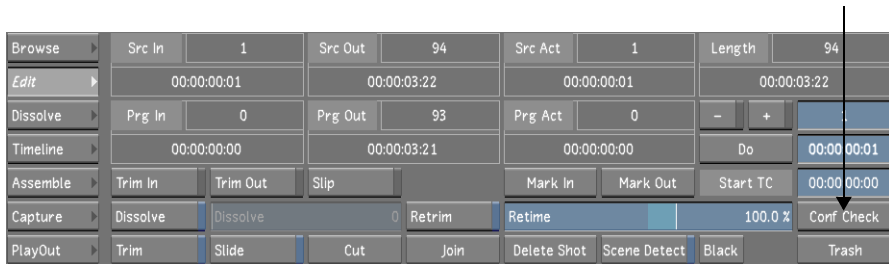
2. Save the current cut, create a new cut, and load it.

When you capture the material, it is placed in the current Storyboard.

3. Capture the footage you want to use as a point of comparison (this is the offline digital cut). See [“Capturing from a VTR”](#) on page 202.

After completing the capture operation, the digital cut appears in the Storyboard and in the Shot bin.

4. Create another new cut and assemble the EDL. See [“Working with CDL, EDL, ALE, and Cutlist Files”](#) on page 110.
5. Click Editing, and then click Edit to display the Edit menu.
6. Drag the digital cut from the Shot bin to the Confidence Check button.



A split screen display appears. The current Storyboard is loaded in the left viewer and the digital cut is loaded in the right viewer.

7. Scrub the Scene timebar (or navigate between shots using the playback controls) to make sure that each shot starts and ends on the exact same frame as in the digital cut.

HINT: You can also scrub the Shot timebar to view the frames with greater granularity.

8. Continue to step through the cut points frame-by-frame.

You must check the two versions by eye in search of discrepancies. If you find a problem, you can offset the shot by dragging the left side of the image. You can also fine-tune a shot's frame offset by clicking the middle mouse button (to offset a shot by -1 frame at a time) or the right mouse button (to offset by +1 frame). See [“Trimming”](#) on page 159.

Using Scene Detection

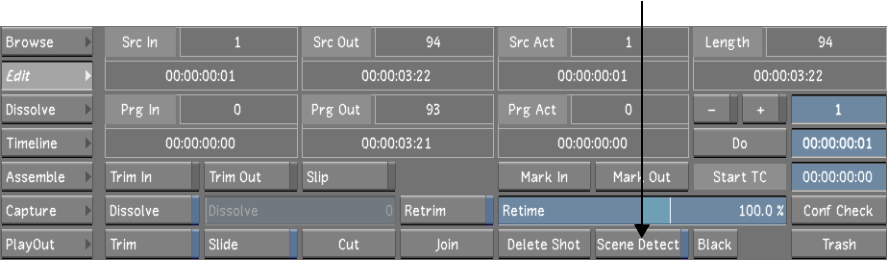
Use scene detection when you are working on a single shot made up of multiple scenes. Scene detection locates the frames when scenes change and splices the shot at those points. When you apply scene detection to a shot in the Storyboard, it is broken up into numerous smaller pieces that you can then 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 will have 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, will be maintained.

To use scene detection:

1. Capture from an edited video tape. See [“Capturing from a VTR”](#) on page 202.
2. Create a new empty cut. See [“Building Cuts and Change Lists”](#) on page 105.
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.

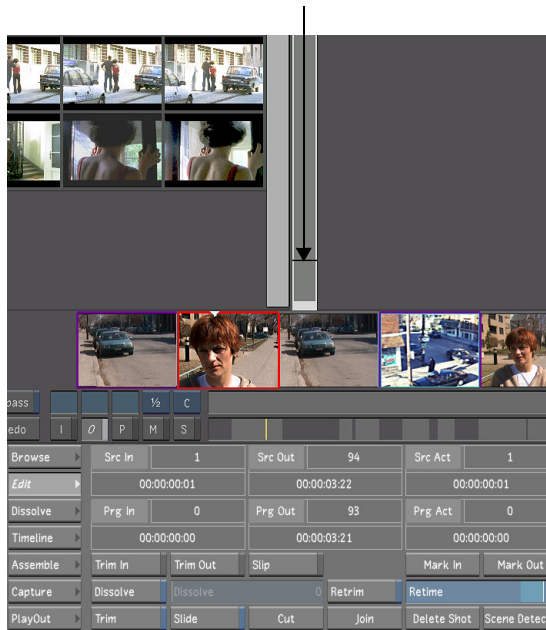


HINT: If the EditingEverywhere keyword is entered in the configuration file, you can start a scene detection from any menu by pressing **SHIFT+S**. See [Appendix A, “Software, Project, and Wiretap Configuration Files,”](#) on page 495.

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 will 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 stop the scene detection process.
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 167.

Video Capture and Video Playout

Summary

About Video Capture and Playout	199
Controlling the VTR	199
Controlling the DDR	201
Capturing from a VTR	202
Defining In and Out Points when Playing Out to Tape	210
Playing Out to a VTR	211
Setting Capture and Playout Options	228
Outputting Multiple Versions	235

About Video Capture and Playout

During the colour grading process, you may need to capture video from, or record video to, a VTR.

For example, you may receive an edited HD or SD video tape that needs to be brought into Lustre and colour graded. Or, you may be given raw footage on video tapes and an EDL, which you auto-conform in Lustre and then colour grade.

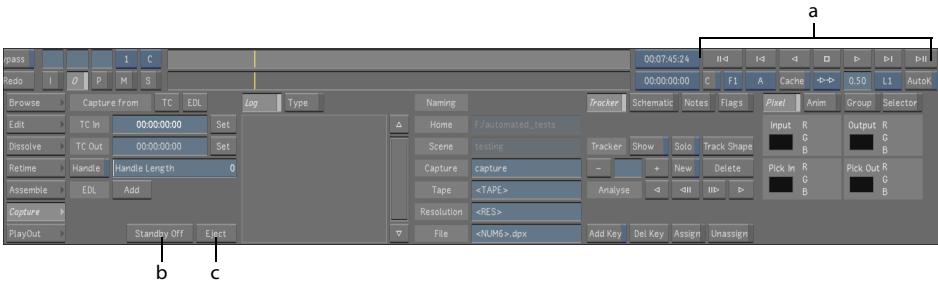
When you are finished grading a scene, you can record it out to a VTR for client approval or for final delivery on HD or SD tapes.

NOTE: The availability of DVS rasters, which are required to control a VTR, depends on the type of license you have purchased. Contact Customer Support for details.

Controlling the VTR

When working with the 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.

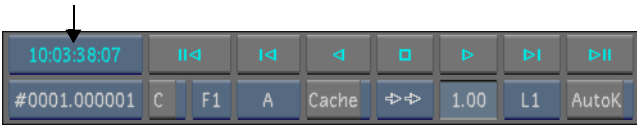


a) VTR transport controls b) Standby Off button c) Eject button

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 (see [“Related Documentation”](#) on page 2).

To control the VTR:

- Click Editing, and then click Capture.
The Capture menu appears.
- Do one of the following:
 - Use the VTR transport controls in the Capture menu.
 - 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.

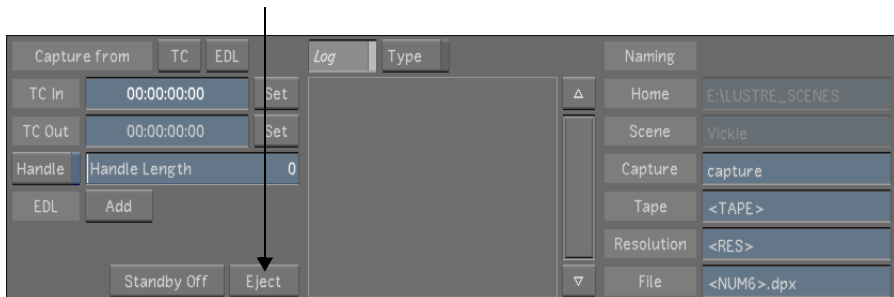


The transport controls are as follows.

Click:	To:
	Fast rewind.
	Cue to TC In. Right-click to jog back one frame.
	Play in reverse.

Click:	To:
	Stop the VTR.
	Play forward.
	Cue to TC Out. Right-click to jog ahead one frame.
	Fast forward.

- To eject the tape in the connected VTR, click the Eject button in the Capture menu.



NOTE: You can also control the VTR from the Playout menu. See [“Playing Out to a VTR”](#) on page 211.

Controlling the DDR

If your timeline has a corresponding soundtrack stored on a digital disk recorder (DDR), you can configure Lustre to synchronize the soundtrack with the timeline.

The DDR can be controlled via the RS422 in LTC Chase mode. If you are controlling the DDR via LTC, the Record timecode and audio track will always match. However, with the RS422, you can offset the Record timecode from the audio track. In either case, you play the soundtrack using the same transport controls that are used for the VTR. See [“Controlling the VTR”](#) on page 199.

To configure Lustre to control the DDR:

- Open the configuration file. Do one of the following:
 - If using the Windows version of Lustre, use a text editor such as *Notepad* to open the *init.config* file, located in *C:\Program Files\Autodesk\lustre<version number>*.
 - If using the Linux version of Lustre, use a text editor such as *nedit* to open the *init.config* file, located in */usr/autodesk/lustre<version number>/*. Type:

nedit init.config

2. Add the DDRSlaveDevice keyword to the configuration file. See [Appendix A, “Software, Project, and Wiretap Configuration Files,”](#) on page 495.
3. Load a cut to the Storyboard.
4. Select the appropriate video format from among the LTC out only types. See [“Selecting the Video Resolution and Scan Mode”](#) on page 230.
5. To offset the timeline from the start of the soundtrack, scrub the Storyboard positioner forward the desired number of frames.
6. To set the synchronization point, press **T**.

NOTE: You can set the synchronization point from all menus except the Colour menus. The **T** hot key is used in the Colour menus to enable the Move button on the Secondaries menu.

7. To ensure that the synchronization was set, switch to the Lustre console window and check to see that the following line has appeared:

DDR synchronized! TCact: 10:00:00:00, set offset: XXXX

In the Capture menu, the Timecode field displays the picture timecode, whereas the field above it displays the audio timecode.

Capturing from a VTR

Use the Capure menu to digitize from a VTR. 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 202.
- Batch digitize from a Log file, or from an imported EDL, ALE, or Cutlist file. See [“Logging and Batch Digitizing”](#) on page 205 and [“Auto-conforming an EDL File”](#) on page 207.
- Perform a live capture from non-controlled sources. See [“Performing a Live Capture from a Non-Controlled Source”](#) on page 209.

For certain video formats, you can remove pulldown when capturing. See [“Removing Pulldown when Capturing”](#) on page 209.

In Lustre, you cannot digitize audio.

Capturing from Timecode

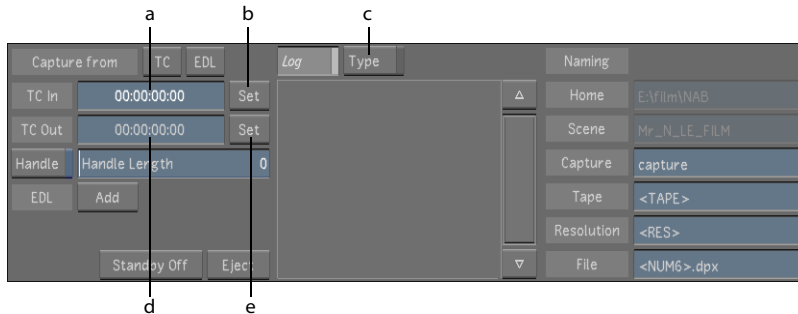
Capturing from timecode is the simplest form of capturing. In this mode, you capture one shot by marking the limits of the shot with in and out timecode values. You can either enter the

timecode values manually or scrub the VTR to the in and out points, and then enter the current timecode as the in or out point.

To capture based on timecode:

1. Select the video format that you want to capture. See [“Selecting the Video Resolution and Scan Mode”](#) on page 230.
2. Click Editing, and then click Capture.

The Capture menu appears.



- a) TC In field c) Type button e) Set TC Out button
 b) Set TC In button d) TC Out field

3. Click Type to display the Type list.
4. Select the file format that you want to create during the capture operation.
5. Enter values in the Capture, Tape, and Resolution fields. See [“Specifying Capture Path, File Type, and Filename”](#) on page 228.
6. Use the VTR transport controls to navigate to the in point. See [“Controlling the VTR”](#) on page 199.
7. Click Set to the right of the TC In field.

The current VTR timecode is entered into the TC In field.

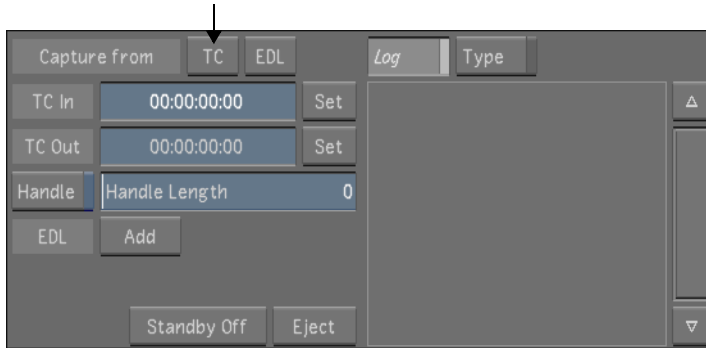
HINT: You can also enter the in point timecode by typing the value directly in the TC In field, or by right-clicking in the TC In field to access the timecode calculator. See [“Navigating With the Timecode Calculator”](#) on page 141.

8. Navigate to the out point, and then click Set to the right of the TC Out field.

The current VTR timecode is entered into the TC Out field.

HINT: You can also enter the out point timecode by typing the value directly in the TC Out field, or by right-clicking in the TC Out field to access the timecode calculator. See [“Navigating With the Timecode Calculator”](#) on page 141.

- Click TC. When the button turns red, click it again to confirm the action.



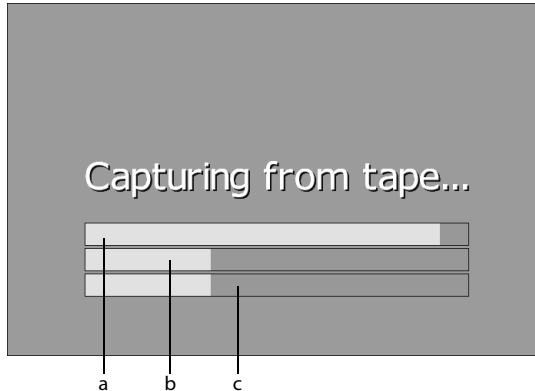
Capture starts. The VTR shuttles back to the in point and begins capturing. The VTR's current timecode and remaining timecode are displayed in a field located to the left of the playback controls. While capturing, both timecodes update and the current timecode appears in red.



a) VTR timecode remaining

b) Current VTR timecode

During capture, the words “Capturing from tape...” appear 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** to pause, 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.

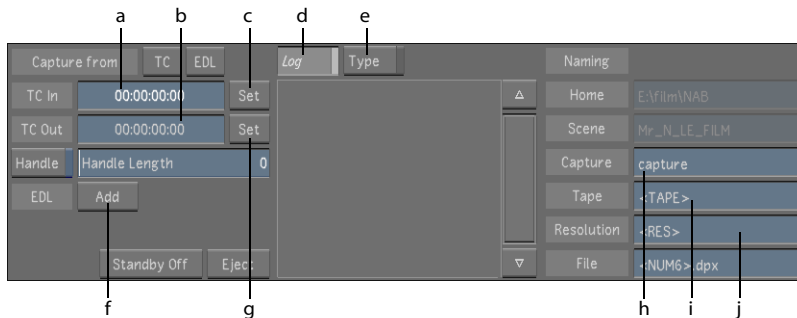
Logging and Batch Digitizing

You can log shots and batch digitize them simultaneously. Shuttle through the tape, mark in and out points that correspond to the sections you want to digitize, and then add the sections to a list of shots that you digitize all at once.

To batch digitize shots from a video tape:

1. Select the video format that you want to capture. See [“Selecting the Video Resolution and Scan Mode”](#) on page 230.
2. Click Editing, and then click Capture.

The Capture menu appears.



- a) TC In field c) Set TC In button e) Type view button g) Set TC Out button i) Tape field
- b) TC Out field d) Log view button f) EDL Add button h) Capture field j) Resolution field

3. Click Type to display the Type list.
4. Select the file format that you want to create during the capture operation.
5. Enter values in the Capture, Tape, and Resolution fields. See [“Specifying Capture Path, File Type, and Filename”](#) on page 228.



WARNING: It is important that you enter a tape name to avoid overwriting previously captured files.

6. Use the VTR transport controls to navigate to the in point. See [“Controlling the VTR”](#) on page 199.

NOTE: You can also type the timecode value directly into the TC In field.

7. Click Set to the right of the TC In field.

This marks an in point at the current timecode value.

HINT: You can also enter the in point timecode by typing the value directly in the TC In field, or by right-clicking in the TC In field to access the timecode calculator. See [“Navigating With the Timecode Calculator”](#) on page 141.

8. Navigate to the out point, and then click Set to the right of the TC Out field.

The current VTR timecode is entered into the TC Out field.

HINT: You can also enter the out point timecode by typing the value directly in the TC Out field, or by right-clicking in the TC Out field to access the timecode calculator. See [“Navigating With the Timecode Calculator”](#) on page 141.

9. Click Set to the right of the TC Out button.

This marks an out point at the current timecode value.

10. Click EDL Add to add the entry to the events list.
11. To view the events list, click Assemble in the Editing Menu. The marked entries for batch digitization appear in the Assemble window.
12. Continue adding entries to the events list as needed.
13. To delete any entry from the events list, select it from the Assemble window and click Delete Selected. To clear the list, click Delete All.
14. When you are ready to digitize, return to the Capture menu.
15. Click EDL, and then confirm the action.

Auto-conforming an EDL File

You can automatically digitize material based on the instructions contained in an EDL and then assemble the digitized events into a cut which is displayed in the Storyboard and Timeline. When digitizing from one of these file types, the process is optimized so that you are prompted for each tape in the file only once.

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>\.*

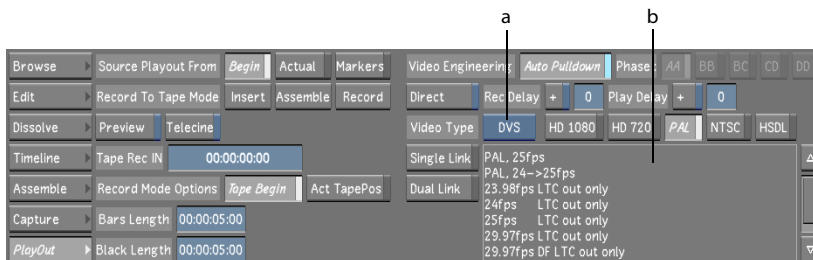
To capture from an EDL:

1. Make sure your 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 PlayOut.

The PlayOut menu appears.



- a) DVS button b) Video Resolution list

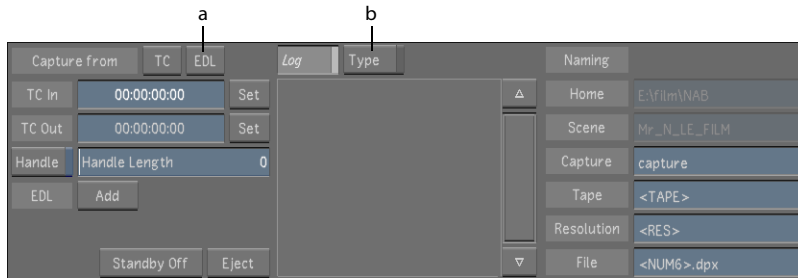
3. Enable the DVS button.
4. Select the video resolution that you want to capture. See [“Selecting the Video Resolution and Scan Mode”](#) on page 230.

5. Click Assemble to access the Assemble menu.
6. Select an EDL file from the EDL list.
7. Click Load EDL to display the EDL contents in the Assemble menu.
8. Select individual events to capture (**CTRL**+click), a group of events (**SHIFT**+click), or click the Select All button.
9. Click Mark for Capture.

The selected events are marked as Need Capture under the Media column of the EDL events.

10. Click Capture to access the Capture menu.

The Capture menu appears.



a) EDL button b) Type button

11. Click Type to display the file format list in the View window.
12. Select the file format that you want to create during the capture operation: either DPX, AVI, or TIFF. See [“Specifying Capture Path, File Type, and Filename”](#) on page 228.
13. Enter values in the Capture, Tape, and Resolution fields. See [“Specifying Capture Path, File Type, and Filename”](#) on page 228.
14. Click EDL. When the button turns red, click it again to confirm the action.

Capture starts.

The tape currently loaded in the VTR is ejected. You are prompted to load the first tape needed in the EDL. After you load the tape, Lustre begins capturing automatically. The words “Capturing from tape...” appear in the Player, and progress bars display shot, tape, and overall progress. 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 load the next tape.

HINT: If the requested tape does not exist, or if you would like to skip to another tape, press **S** to proceed to the next available tape specified in the EDL, ALE, or Cutlist file.

15. When capture is completed, add the new shots to the Shot bin. See [“Loading Shots into the Shot Bin”](#) on page 93.
16. Go back to the Assemble menu and assemble the EDL. See [“Assembling an EDL”](#) on page 121.
17. Click Browse to display the Browse menu.
18. Enter the name for the new cut in the Cut name field.
19. 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 that has an SDI Out, such as a VTR, another Lustre machine, or a Smoke or Flame workstation. For example, if you have footage residing on another Lustre machine that you would like to capture without having to capture again from tape, you can connect to that machine and capture the footage live.

To perform a live capture from a non-controlled source:

1. Connect the SDI Out of the non-controlled source to the DVS SDI In of the Lustre machine.
2. Click Editing, and then click PlayOut.
The PlayOut menu appears.
3. Select the appropriate DVS raster.
4. Click Capture to display the Capture menu.
5. In the TC In and TC Out fields, enter fake in and out points.
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.
6. Hold down **CTRL** while double-clicking the TC button with the left mouse key.

Removing Pulldown when Capturing

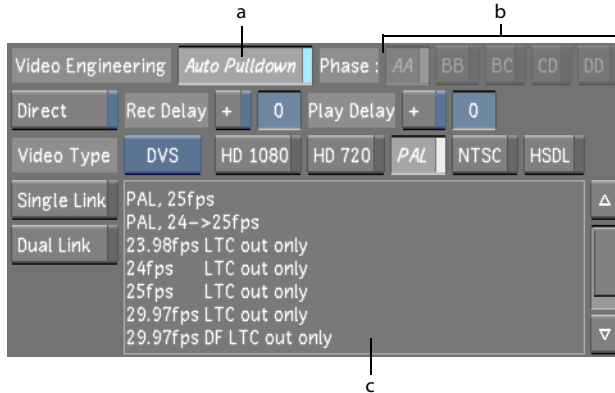
Depending on the type of source material you are working from, you may need to remove 3:2 pulldown at capture time. For example, you may be working with an NTSC tape at 29.97 fps that was a straight Telecine transfer from film at 24p. You can then capture this NTSC tape in Lustre and remove its added 3:2 pulldown to end up with a progressive 24p NTSC clip in Lustre.

NOTE: Inconstant cadence material (i.e. re-edited material) may produce jittery results.

To remove 3:2 pulldown during capture:

1. Click Editing, and then click PlayOut.

The PlayOut menu appears.



a) Auto Pulldown button

b) Pulldown Phase buttons

c) Video Type list

2. In the Video Type list, select the appropriate video format for removing 3:2 pulldown. See [“Selecting the Video Resolution and Scan Mode”](#) on page 230.
3. Do one of the following:
 - Enable Auto Pulldown. When this option is enabled, Lustre automatically detects the pulldown sequence so that the appropriate fields are removed.
 - Use the VTR to shuttle to the first frame you want to capture, and then enable the Pulldown Phase button that matches the field composition of that frame.
4. Capture from the VTR using one of the capture methods.

Defining In and Out Points when Playing Out to Tape

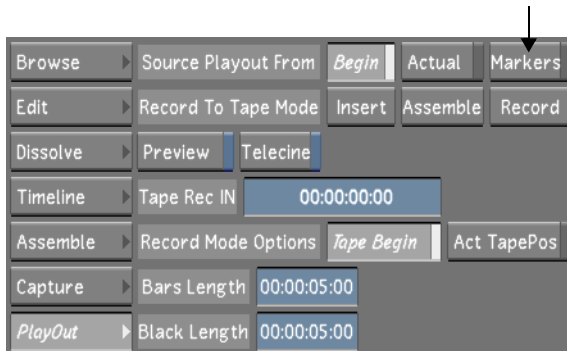
Users can create a videotape from a specific segment of the timeline by setting in and out points for playing out to tape. This is done using the same hot keys as for setting in and out points for playing shots in the Player. A new option has been added to the Editing Playout menu that lets you select the marked section for playout to tape.

To define in and out points when playing out to tape:

1. Go to the start and end frame and mark the in and out points.

Press:	To:
SHIFT+I	Mark an in point.
SHIFT+O	Mark an out point.
SHIFT+L	Clear in and out points.

2. Click Editing, and then click PlayOut to display the PlayOut menu.
3. Click Markers.



NOTE: If you enable an in point only, playout begins at that point and goes to the end of the timeline. If you enable an out point only, playout begins at the start of the timeline and ends at the out point.

Playing Out to a VTR

Using the Playout tool, you can play out rendered frames from Lustre so that they can be recorded to a VTR.

You almost always play out rendered frames to a VTR. See [“Rendering Shots”](#) on page 438. The only exception is the creation of real-time deliverables using source material or GPU playout from timeline. See [“Creating Real-Time Deliverables From Source or Rendered Material”](#) on page 223 and [“Play Out from Source with GPU Preview Processing”](#) on page 218.

There are three modes for playing out to a VTR:

- In Record mode, you can play out video to a blank tape or a tape that contains existing material. Anything on the tape is overwritten when recording.
- In Insert mode, you record video only 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.

For some video formats, you can add 3:2 pulldown during the playback operation. You can also add pulldown when converting film to PAL video.

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 438.
2. Use the transport controls to cue up the deck to the in point.	"Controlling the VTR" on page 199.
3. If necessary, cue up the timeline.	"Navigating through Shots" on page 138.
4. Select a playback mode.	"Playing Out to a VTR" on page 211.

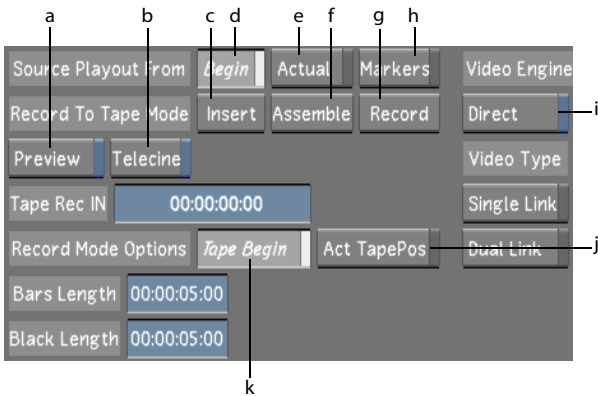
Accessing the PlayOut Menu

Use the PlayOut menu to record rendered frames to a VTR.

To access the PlayOut menu:

- In the Main menu, click Editing, and then click PlayOut.

The PlayOut menu appears.



a) Preview button	e) Actual button	i) Direct button
b) Telecine button	f) Assemble mode button	j) Act (actual) TapePos button
c) Insert mode button	g) Record button	k) Tape Begin button
d) Begin button	h) Markers button	

Recording in Record Mode

Use Record mode to record an entire cut to a VTR. Record mode replaces the entire contents of the tape (including timecode). Therefore, to use Record mode, you must set the VTR timecode to Preset or Auto timecode. You do not need 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 at the start of your timeline.

At least 10 seconds of black is always recorded to the video tape before the video. Also, video always begins 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 438.
2. Click Edit, and then click PlayOut to display the PlayOut menu.
3. Determine where you want the playout operation to start:
 - Click Begin to start the playout operation from the start of the current timeline.
 - Click Actual to start the playout operation at the current frame.
4. Determine where on the video tape you want to record to:
 - Click Tape Begin so that the playout operation rewinds to the start of the tape and then begins playing out.
 - Click Act TapePos to start the playout operation at the current frame of the tape.
5. Enter the start timecode in the Tape Rec IN field.
6. Enter the Min Black Length and the Min Bars (colour bar) Length.
7. Click Record, and then confirm the action.
The record operation begins.

Recording in Insert Mode

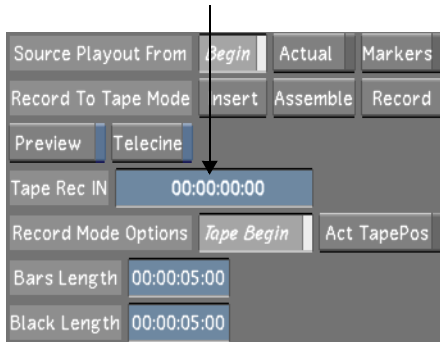
Use Insert mode when you want to insert new video onto a tape that already contains material. The entire area to which you are recording must have a valid, continuous control track as this mode only plays out the video (existing timecode and audio are not affected).

NOTE: In Insert mode, make sure the deck is set to Regen timecode.

To record in Insert 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 438.
2. Click Editing, and then click PlayOut to display the PlayOut menu.
3. Determine where you want the playout operation to start:
 - Click Begin to start the playout operation from the start of the current timeline.
 - Click Actual to start the playout operation at the current frame.

4. In the Tape Rec 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 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.

5. (Optional) Enable Preview.

If Preview is enabled, payout is still the same but nothing is written to tape because VTR is in Preview mode.

6. Click Insert, and then confirm the action.

HINT: To record a single shot as opposed to the entire timeline, hold down **SHIFT** while clicking Insert.

NOTE: If Preview is enabled, the Insert button turns green after the first click to show that the action is non-destructive.

The payout operation begins.

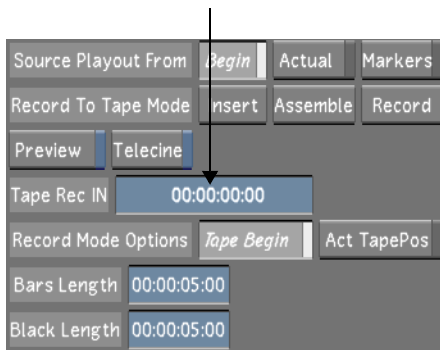
Recording in Assemble Mode

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 new video 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: In Assemble mode, make sure to set the deck to Regen timecode.

To record in Assemble 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 438.
2. Click Editing, and then click PlayOut to display the PlayOut menu.
3. Determine where you want the playout operation to start:
 - Click Begin to start the playout operation from the start of the current timeline.
 - Click Actual to start the playout operation at the current frame.
4. In the Tape Rec 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.

5. (Optional) Enable Preview.
If Preview is enabled, playout is still the same but nothing is written to tape because VTR is in Preview mode.
6. Click Assemble, then confirm the action.

HINT: To record a single shot as opposed to the entire timeline, hold down **SHIFT** while clicking Assemble.

NOTE: If Preview is enabled, the Assemble button turns green after the first click to show that the action is non-destructive.

The playout operation begins.

Adding Pulldown when Playing Out

In order to produce output video at various common formats, you may need to add 3:2 pulldown to original 24p material.

If converting film to PAL video, you can use the PAL 24>25fps conversion option in the Video Types list. This allows you to convert 24 fps to 25 fps when playing out. The PAL24/25 option adds an extra field every 12 frames. The following is an example of the resulting pulldown sequence:

A1A2 B1B2 C1C2 D1D2 E1E2 F1F2 G1G2 H1H2 I1I2 J1J2 K1K2 L1L2L1 M1M2 N1N2 O1O2
P1P2 Q1Q2 R1R2 S1S2 T1T2 U1U2 V1V2 W1W2 X1X2X1 A1A2

NOTE: When playing out to a tape with existing material, make sure that the pulldown phase is not damaged by the addition of the new material. If there is a problem, adjust the Playout Start timecode so that it starts on an AA phase image.

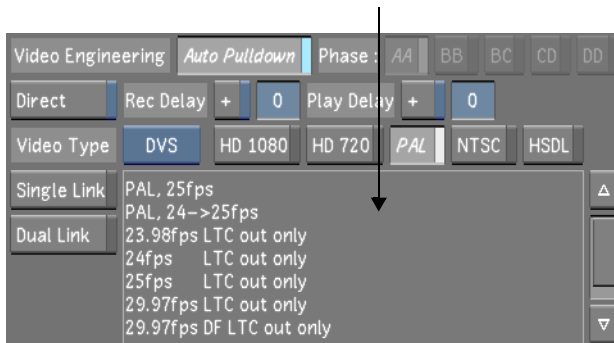
To add pulldown during payout:

1. Click Editing, and then click PlayOut.

The PlayOut menu appears.

2. In the Video Type list, select one of the following:

- The appropriate video format for adding 3:2 pulldown. See [“Selecting the Video Resolution and Scan Mode”](#) on page 230.
- The PAL 24>25fps conversion option.



3. Use one of the previously described playout methods to play out from Lustre to a VTR.
The pulldown is added to the outputted video. The first frame played out becomes the AA frame in the pulldown sequence.

Play Out from Source with GPU Preview Processing

You can now use GPU processing to play out source material in real time with both primary and secondary grading 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 latest feature enhancement lets you play out source material containing secondary grading without having to first perform a rendering pass.

By adding the `PlayoutFromTimeline` keyword to the project's configuration file and by enabling GPU preview processing, you can play out from Output view mode and use all features currently supported by enhanced GPU preview processing. See [“GPU Preview Processing”](#) on page 279. However, playback performance may be impacted depending on the grading applied to the current shot. If frames are dropped during play out, the application recues the VTR to the last known frame laid down to tape.

Prior to playing out, you must select the appropriate DVS raster. The `lanczos2Lobe` mastering quality rescaler that is applied when in Real-Time Deliverables mode is not used here. Instead, the default graphics card rescaling algorithm is applied.

To play out source material to tape with all features supported by GPU preview processing mode:

1. In the *lustre.config* file for your project, enter the `PlayoutFromTimeline` keyword.
2. In Lustre, enable GPU preview processing by clicking the GPU button or by pressing **Y**.
3. Play out to the VTR. See [“Playing Out to a VTR”](#) on page 211.

Expected Performance

Real-time playback is not guaranteed when you play out directly to tape from GPU preview processing. 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 Storyboard 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 on the Editing | PlayOut menu.

Regardless of whether or not you conform to all conditions stated above, some animations will not play back in real time.

NOTE: Support for direct playout to tape from GPU preview processing depends on the version of the graphics card installed on your system.

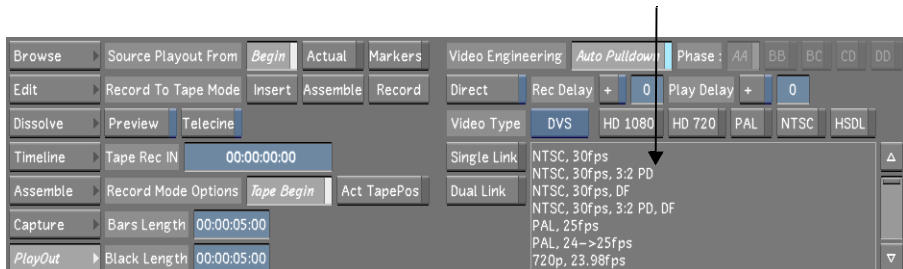
Outputting Linear Timecode While Working on the Timeline

You can send the Timeline's record timecode to an external device such as a VTR or audio device. This ensures interoperability between Lustre and external devices. Specifically, it allows Lustre to manage Linear Timecode (LTC) audio by matching the timeline's record timecode to that of the audio track on the audio device.

NOTE: Another option for matching the timeline to a soundtrack is to configure Lustre to control a DDR. See [“Controlling the DDR”](#) on page 201.

To output LTC while working on the timeline:

1. Connect the DVS breakout box LTC output to the TC input on the external device.
2. Set the device's configuration to external.
3. In Lustre, click Editing and then click PlayOut to display the PlayOut menu.
4. Select one of the DVS rasters marked “LTC out only”.



These rasters ensure that only the LTC timecode is output, thereby ensuring faster playback rates. If any other DVS raster type is used, both the timecode and the video are outputted.

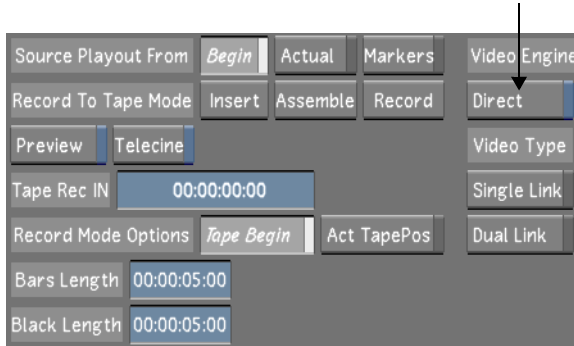
5. Ensure that the frame rates for the Lustre timeline, the external device, and the DVS raster all match.

Outputting to 10-bit Colour Space

Use the Direct playout option to output the rendered timeline to 10-bit colour space on tape without reducing the file dynamic range, thereby ensuring better quality images on output. The 10-bit buffer is sent from the DVS SDI out. To use Direct play out, all rendered files must be 10-bit DPX or CIN images. If they are not, you must render the files to a single resolution and image type prior to enabling Direct playout.

To output to 10-bit colour space:

1. Click Editing, and then click PlayOut to display the PlayOut menu.
2. Enable Direct.



Once Direct is enabled, you can play out the selected Storyboard.

3. Play the timeline out to a VTR in either Record, Insert, or Assemble mode.

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 five to six seconds long. For specific information about enhancements to Writing Telecine-Style Tape workflows that use NTSC, see [“Enhancements to Writing Telecine-Style Tape for Workflows Using NTSC”](#) on page 221.

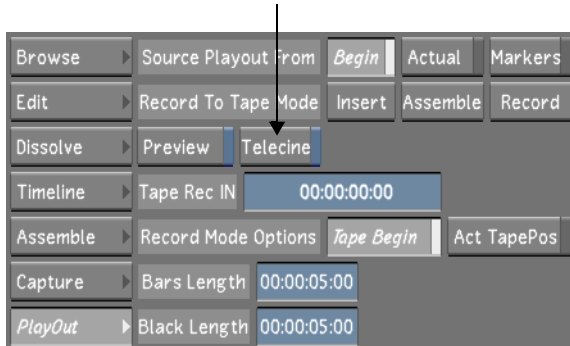
Before you can use the Telecine feature, you must configure the VTR and the DVS breakout box.

To configure the VTR and DVS breakout box:

1. Connect the DVS breakout box LTC output to the LTC input on the VTR.
2. Set the VTR's timecode configuration to external, record run, and regen.

To output telecine-style tape:

- In either Record, Insert, or Assemble mode, enable Telecine prior to starting the playback operation.



NOTE: The crash record Tape Rec IN timecode feature does not work when Telecine is enabled. Although the timecode starts at zero, it will be modified by the first shot's timecode a few seconds before it starts. This is a known limitation.

Enhancements to Writing Telecine-Style Tape for Workflows Using NTSC

Enhancements have been made to the Telecine feature that specifically benefit commercial workflows using NTSC.

Lustre allows you to write a telecine-style tape. Lustre converts 24 fps to 29.97 fps and writes the converted clip's source timecode to the tape's LTC tract. So if, for example, you are working in Lustre with 2K footage at 24 fps and you render a resized 24p timeline for the NTSC format, you can convert the frame rate and the output timecode in order to use the tape with other systems. In a nutshell, both the video and the clip's SRC TC gets converted on the fly on the way out to tape. The original source timecode that is written to tape as LTC SMPTE timecode is the one defined by the numbering of the file on the Lustre file system. For example, the source timecode for myfile.0000.dpx is 00:00:00:00.

NOTE: These enhancements work on rendered material only and cannot be used for real-time deliverables. The video, VTR control, and LTC output are all performed using the DVS board.

The conversion workflow is as follows:

- Configure the VTR to receive LTC timecode from an external source.
- If required, add the appropriate Write Telecine Tape pull-down timecode (WTT PD TC) conversion keyword to the project configuration file. The default mode is a straight 2/3 conversion.

- Perform the procedure for outputting telecine-style tape, making sure that you use a DVS video output resolution, either NTSC 30 fps 3:2 PD, or 1080i 29.97 3/2 PD. These convert the original 24 fps to 29.97 fps. See “To output telecine-style tape” in [“Writing Telecine-Style Tape”](#) on page 220.

Configuring the VTR to Receive Source Timecode

A menu change may be required to configure the VTR to receive source timecode and write it to the LTC track.

The following table shows an example of HDCAM timecode settings.

Value	Setting
TCR SEL	auto
TCG SRC	ext-L
TCG Mode	regen
RUN MODE	rec
VITC	TCG

To configure the VTR to allow the LTC track to receive timecode:

1. On the Digital Betacam VTR, search for the menu with the name regen control (usually #610).
2. If the menu is set to auto (the default), change it to manual.

Adding WTT PD TC Conversion Keywords

By default, the timecode conversion is locked to the pulldown sequence described in the following table.

Original 24 fps TC	Converted 30 fps TC	Field	Cadence
01:10:10+00	01:10:10+00	F1	A
01:10:10+00	01:10:10+00	F2	A
01:10:10+01	01:10:10+01	F1	B
01:10:10+01	01:10:10+01	F2	B
01:10:10+01	01:10:10+02	F1	B
01:10:10+02	01:10:10+02	F2	C
01:10:10+02	01:10:10+03	F1	C
01:10:10+03	01:10:10+03	F2	D
01:10:10+03	01:10:10+04	F1	D
01:10:10+03	01:10:10+04	F2	D
01:10:10+04	01:10:10+05	F1	A
01:10:10+04	01:10:10+05	F2	A

NOTE: The table does not describe the entire sequence. However, the pattern repeats throughout the sequence, as indicated by the last two rows.

To add a WTT PD TC conversion keyword to the project configuration file:

1. Open the *lustre.config* file for the project, located in *C:\Program Files\Autodesk\lustre<version number>\project\<project name>*, and add one of the keywords described in the following table.

Add:	To convert:
WTT_PD_phase_A	2-3 starting on F1
WTT_PD_phase_B	3-2 starting on F1
WTT_PD_phase_C	2-3 starting on F2
WTT_PD_phase_D	3-2 starting on F2

2. Save and close the file, launch Lustre, and perform the Telecine procedure using the appropriate DVS raster.

Creating Real-Time Deliverables From Source 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 your 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 Print 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 226.

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 226.

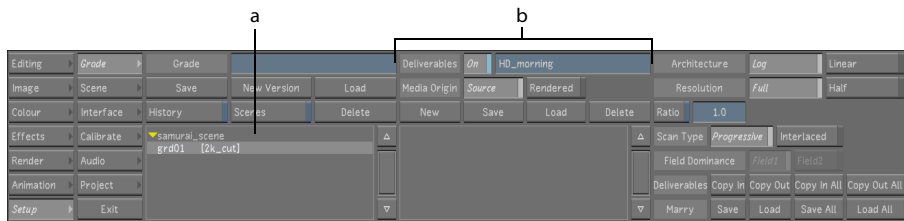
Although real-time deliverables setups are linked to the grade file, you can apply a setup to a grade linked with a different cut.

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.

To create a real-time deliverables setup:

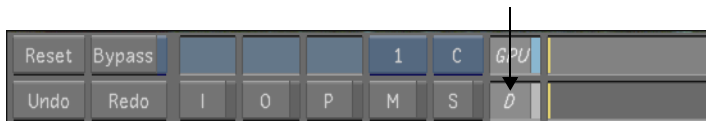
1. In the Main menu, click Setup and then click Grade.

The Grade menu appears.



- a) Grade list b) Deliverables section

2. From the Grade list, load the grade from which you want to create a real-time deliverables setup.
3. Click the D button.



The button automatically switches you to Real-Time Deliverables view mode, and enables GPU processing. See [“GPU Preview Processing”](#) on page 279.

HINT: Enabling the D button also allows you to send a 10-bit buffer to the SDI output.

4. In the Deliverables field, enter a name for the setup.

5. 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.

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 11, “Repositioning Images.”](#) on page 251.

NOTE: Rotations cannot be included in real-time deliverables setups.

8. (Optional) Perform input primary colour grading. See [Chapter 14, “Primary Colour Grading.”](#) on page 295.

NOTE: Only input primary grading is permitted for real-time deliverables setups.

9. (Optional) Apply a HW LUT. See [“Applying Print LUTs for Viewing”](#) on page 56.

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 delete a setup:

- Select a setup from the list, and click Delete.

Resizing Shots Prior to Playing Out

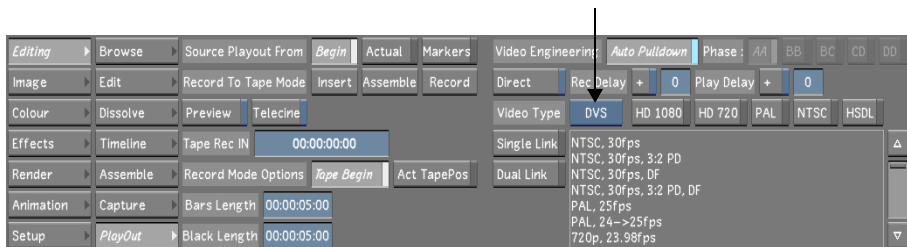
Before you play out real-time deliverables, resize your shots to ensure that they properly conform to the destination output format. See [“Resizing Shots”](#) on page 441.

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. Make sure DVS is enabled.



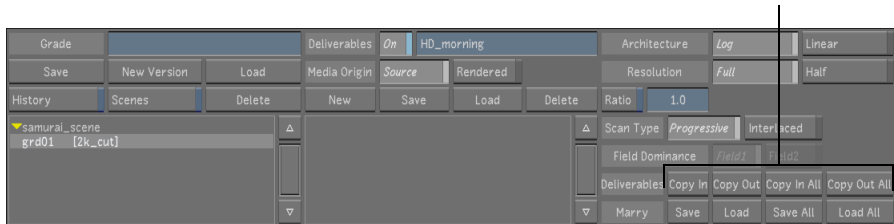
3. Select a video resolution.
4. Play out to tape in either Insert or Assemble mode. See [“Recording in Insert Mode”](#) on page 214 and [“Recording in Assemble Mode”](#) on page 215.

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 menus, 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.

VTR Emulation

Users can use Lustre to emulate a VTR for playout in real time. 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 preroll time on the Lustre VTR emulator defaults to seven seconds with each video initialization.

To configure Lustre to function as a slave 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 IBM IntelliStation Z Pro 6221, connect to the RMT IN port on the DVS HD station 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.
3. Launch Lustre and enter the following keywords in the *init.config* or *lustre.config* file.

EnablePlayerSlave — Enables Lustre to act as a VTR for a master machine.

PlayerSlaveDelay 0 — Compensates for frame delays that may exist while transferring shots from the controlling device to Lustre. Specify the number of frames the shot should be shifted ahead as it is transferred from the controlling device to Lustre. Enter a negative number of frames to shift the clip backwards.
4. Click Editing, and then click PlayOut to display the PlayOut menu.
5. Enable the DVS button and select the appropriate DVS raster playout format. This must match the selected video timing on the controlling device.
6. Prior to playout from the VTR emulator, ensure that a header of at least five seconds has been added to the start of the Storyboard, and that the Start Timecode begins at the end of the header and not at 00:00:00:00. This is necessary due to the fact that Lustre VTR emulation does not provide a pre-roll outside of the boundaries of the existing Storyboard.
7. To disable VTR emulation, use the Eject command. To re-initialize, either switch between the Editing Capture and PlayOut menus, or re-select the DVS raster.

Setting Capture and Playout Options

Before you can capture from, or play out to, a VTR or other device, you need to 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.

Specifying Capture Path, File Type, and Filename

Before digitizing video, you need to specify the file format of the captured files, a path to the files, and a filename.

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.

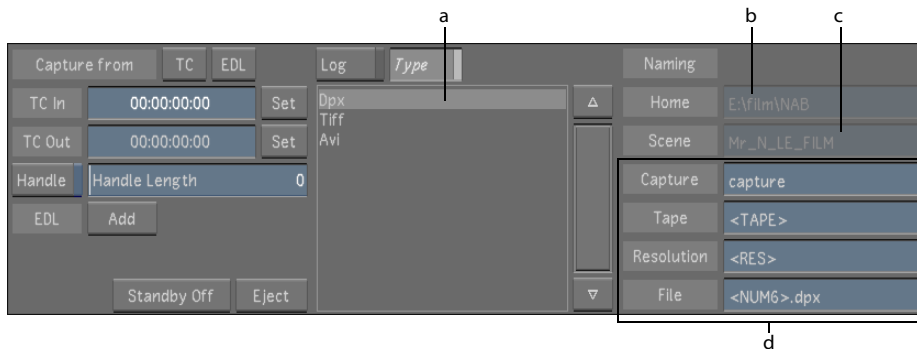
When digitizing media, you can select to store the media on your system in any of the following file types:

- DPX
- TIFF
- AVI

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 will appear as part of the DPX file information and be available for use in EDL assembly.

When you capture, the files are created on-the-fly and are available after capturing is completed. There are, however, limitations. TIFF and AVI files are created faster than DPX files, but they are restricted to 8-bit colour depth.

When setting up a path and filename, you use the following elements of the Capture menu.



- a) Type list
- b) Home field
- c) Scene field
- d) Path and filename fields

Type list — Select the capture file format.

Home field — <Home> is the project home directory. You cannot change this value without modifying your project settings.

Scene field — <Scene> shows the current scene you are working in during capture. You cannot change this value without modifying your project settings.

Capture field — Defaults to “Capture.”

Tape field — Displays the tape name. This field defaults to “Tape” in TC and List mode. If you capture from an EDL, the default tape name is determined by the tape name as specified in the

EDL. For example, if you capture an EDL with three tapes, by default three directories are created at the Tape level.

Resolution field — Defaults to the resolution of the captured files. Generally, you should not change this field.

File field — Displays the filename. You can add an extra name before the numbers, but to ensure consistent frame counting, you should not erase the <NUM6>. For example, enter Mymovietitle<NUM6>.dpx. NUM6 corresponds to the number of digits that are used to increment the frame number. In rare cases, when you are working from many different reels and high numbers in timecode, the frame numbers may exceed 6 digits. In such cases, for consistency in the counting, change <NUM6> to <NUM7>.

Selecting the Video Resolution and Scan Mode

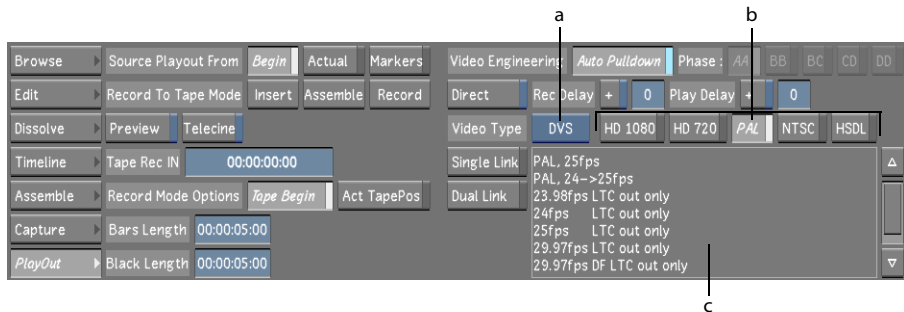
From the PlayOut menu, use the Video Type list to select the required video resolution and scan mode for the VTR.

NOTE: You can change the video type without changing projects or exiting the application.

To set the video resolution and scan mode:

1. Click Editing, and then click PlayOut.

The PlayOut menu appears.



- a) DVS button b) Video Type buttons c) Video Resolution list

2. Make sure DVS is enabled.
3. Enable the Video Type buttons to filter the contents of the Video Type list. For instance, if you enable HD 1080, only formats that support 1080i or 1080PsF appear in the Video Type list.

You can sort using the following filters.

Enable:	To display:
HD 1080	1080i or 1080PsF formats.
HD 720	720p formats.
PAL	Formats at a resolution of 720x576.
NTSC	Formats at a resolution of 720x486.
HSDL	Formats that allow HSDL transfer.
Single Link	Non-dual link formats.
Dual Link	Formats that support dual link.

4. Select one of the video resolutions and scan modes from the Video Type list. See [“Supported Video Formats”](#) on page 231.

NOTE: If clicking an item in the Video Type list does not select the item, that format is not available for capture or playback on that system.

NOTE: When you select a drop frame (DF) video format, the timecodes for both capture and playback are calculated in Drop Frame mode. To ensure that your timeline plays out to the correct duration, be sure to match its timecode count with that of the destination tape.

Supported Video Formats

The following table lists the formats that are available for capture and playback 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	Video Signal
NTSC, 30fps	29.97 fps, 59.94 Hz, interlaced, 720x486 resolution, NDF timecode, YUV 4:2:2 data.
NTSC, 30fps, 3:2 PD	29.97 fps, 59.94 Hz, interlaced, 720x486 resolution, pulldown insert on playback and removal on capture, NDF timecode, YUV 4:2:2 data.
NTSC, 30fps, DF	29.97 fps, 59.94 Hz, interlaced, 720x486 resolution, DF timecode, YUV 4:2:2 data.
NTSC, 30fps, 3:2 PD, DF	29.97 fps, 59.94 Hz, interlaced, 720x486 resolution, pulldown insert on playback and removal on capture, DF timecode, YUV 4:2:2 data.
PAL, 25fps	25 fps, 50 Hz, interlaced, 720x576 resolution, YUV 4:2:2 data.
PAL, 24->25fps	25 fps, 50 Hz, interlaced, 720x576 resolution, PAL 24->25fps frame rate conversion (1 hybrid field frame inserted every 12 frames), YUV 4:2:2 data.

Video Type	Video Signal
720p, 23.98fps	23.98 fps, progressive, 1280x720 resolution, YUV 4:2:2 data.
720p, 24fps	24 fps, progressive, 1280x720 resolution, YUV 4:2:2 data.
720p, 25fps	25 fps, progressive, 1280x720 resolution, YUV 4:2:2 data.
720p, 30fps	30 fps, progressive, 1280x720 resolution, YUV 4:2:2 data.
720p, 50fps	50 fps, progressive, 1280x720 resolution, YUV 4:2:2 data.
720p, 59.94fps	59.94 fps, progressive, 1280x720 resolution, NDF timecode, YUV 4:2:2 data.
720p, 59.94fps, DF	59.94 fps, progressive, 1280x720 resolution, DF timecode, YUV 4:2:2 data.
720p, 60fps	60 fps, progressive, 1280x720 resolution, YUV 4:2:2 data.
720p, 60fps, dual link	60 fps, progressive, 1280x720 resolution, RGB 4:4:4 data.
720p, 60p, 30fps	30 fps, 60 progressive frames, every second frame is captured, 1280x720 resolution, YUV 4:2:2 data.
720p, 60p, 30fps, dual link	30 fps, 60 progressive frames, every second frame is captured, 1280x720 resolution, RGB 4:4:4 data.
720p, 72fps	72 fps, progressive, 1280x720 resolution, YUV 4:2:2 data.
720p, 72fps, dual link	72 fps, progressive, 1280x720 resolution, RGB 4:4:4 data.
1080PsF, 23.98fps	23.98 fps, 72 Hz, segmented frames, 1920x1080 resolution, YUV 4:2:2 data.
1080PsF, 23.98fps, dual link	23.98 fps, 72 Hz, segmented frames, 1920x1080 resolution, RGB 4:4:4 data.
1080PsF, 24fps	24 fps, 48 Hz, segmented frames, 1920x1080 resolution, YUV 4:2:2 data.
1080PsF, 24fps, dual link	24 fps, 48 Hz, segmented frames, 1920x1080 resolution, RGB 4:4:4 data.
1080PsF, 25fps	25 fps, 50 Hz, segmented frames, 1920x1080 resolution, YUV 4:2:2 data.
1080PsF, 25fps, dual link	25 fps, 50 Hz, segmented frames, 1920x1080 resolution, RGB 4:4:4 data.
1080i, 25fps	25 fps, 50 Hz, interlaced, 1920x1080 resolution, YUV 4:2:2 data.
1080i, 25fps, dual link	25 fps, 50 Hz, interlaced, 1920x1080 resolution, RGB 4:4:4 data, RGB 4:4:4 data.
1080i, 29.97fps	29.97 fps, 59.94 Hz, interlaced, 1920x1080 resolution, NDF timecode, YUV 4:2:2 data.
1080i, 29.97fps, dual link	29.97 fps, 59.94 Hz, interlaced, 1920x1080 resolution, NDF timecode, RGB 4:4:4 data.
1080i, 29.97fps, DF	29.97 fps, 59.94 Hz, interlaced, 1920x1080 resolution, DF timecode, YUV 4:2:2 data.
1080i, 29.97fps, DF dual link	29.97 fps, 59.94 Hz, interlaced, 1920x1080 resolution, DF timecode, RGB 4:4:4 data.

Video Type	Video Signal
1080i, 29.97fps, 3:2 PD	29.97 fps, 59.94 Hz, interlaced, 1920x1080 resolution, pulldown insert on playout and removal on capture, NDF timecode, YUV 4:2:2 data.
1080i, 29.97fps, 3:2 PD, dual link	29.97 fps, 59.94 Hz, interlaced, 1920x1080 resolution, pulldown insert on playout and removal on capture, NDF timecode, RGB 4:4:4 data.
1080i, 29.97fps, 3:2 PD, DF	29.97 fps, 59.94 Hz, interlaced, 1920x1080 resolution, pulldown insert on playout and removal on capture, DF timecode, YUV 4:2:2 data.
1080i, 29.97fps, 3:2 PD, DF dual link	29.97 fps, 59.94 Hz, interlaced, 1920x1080 resolution, pulldown insert on playout and removal on capture, DF timecode, RGB 4:4:4 data.
1080PsF, 29.97fps	29.97 fps, segmented frame, 1920x1080 resolution, NDF timecode, YUV 4:2:2 data.
1080PsF, 29.97fps, dual link	29.97 fps, segmented frame, 1920x1080 resolution, NDF timecode, RGB 4:4:4 data.
1080PsF, 29.97fps, DF	29.97 fps, segmented frame, 1920x1080 resolution, DF timecode, YUV 4:2:2 data.
1080PsF, 29.97fps, DF dual link	29.97 fps, segmented frame, 1920x1080 resolution, DF timecode, RGB 4:4:4 data.
1080i, 30fps	30 fps, 60 Hz, interlaced, 1920x1080 resolution, NDF timecode, YUV 4:2:2 data.
1080i, 30fps, dual link	30 fps, 60 Hz, interlaced, 1920x1080 resolution, NDF timecode, RGB 4:4:4 data.
1080i, 30fps, DF	30 fps, 60 Hz, interlaced, 1920x1080 resolution, DF timecode, YUV 4:2:2 data.
1080i, 30fps, DF dual link	30 fps, 60 Hz, interlaced, 1920x1080 resolution, DF timecode, RGB 4:4:4 data.
1080i, 30fps, 3:2 PD	30 fps, 60 Hz, interlaced, 1920x1080 resolution, pulldown insert on playout and removal on capture, NDF timecode, YUV 4:2:2 data.
1080i, 30fps, 3:2 PD, dual link	30 fps, 60 Hz, interlaced, 1920x1080 resolution, pulldown insert on playout and removal on capture, NDF timecode, RGB 4:4:4 data.
1080i, 30fps, 3:2 PD, DF	30 fps, 60 Hz, interlaced, 1920x1080 resolution, pulldown insert on playout and removal on capture, DF timecode, YUV 4:2:2 data.
1080i, 30fps, 3:2 PD, DF dual link	30 fps, 60 Hz, interlaced, 1920x1080 resolution, pulldown insert on playout and removal on capture, DF timecode, RGB 4:4:4 data.
1080p, 23.98fps, HSDL	23.98 fps, progressive, 1920x1080 resolution, allows HSDL transfer.
1080p, 24fps, HSDL	24 fps, progressive, 1920x1080 resolution, allows HSDL transfer.
1556PsF, 14fps, HSDL	14 fps, segmented frame, 2048x1556 resolution, allows HSDL transfer, YUV 4:2:2 data.

Video Type	Video Signal
1556PsF, 14fps, dual link, HSDL	14 fps, segmented frame, 2048x1556 resolution, allows HSDL transfer, RGB 4:4:4 data.
1556PsF, 15fps, HSDL	15 fps, segmented frame, 2048x1556 resolution, allows HSDL transfer, YUV 4:2:2 data.
1556PsF, 15fps, dual link, HSDL	15 fps, segmented frame, 2048x1556 resolution, allows HSDL transfer, RGB 4:4:4 data.
23.98fps LTC out only	23.98 fps, 72 Hz.
24fps LTC out only	24 fps, 48 Hz.
25fps LTC out only	25 fps, 50 Hz.
29.97fps LTC out only	29.97 fps, 59.94 Hz, NDF timecode.
29.97fps DF LTC out only	29.97 fps, 59.94 Hz, DF timecode.
30fps LTC out only	30fps, 60 Hz, NDF timecode.

Setting Video Engineering Options

Use the Video Engineering fields in the PlayOut menu to select options for transferring shots between Lustre and the VTR. You can adjust the VTR edit delay and the VTR capture delay.

The VTR edit delay compensates for frame delays that may exist while transferring shots from Lustre to the output device.

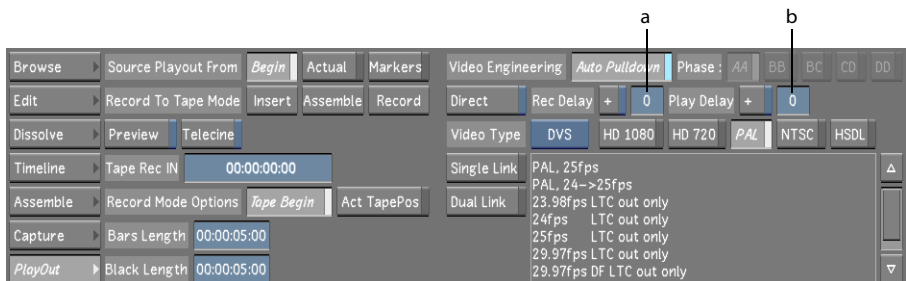
The VTR capture delay compensates for frame delays that may exist while transferring shots from the input device to Lustre.

NOTE: You must enter the required number of frames in the Video Engineering fields for the VTR you are using. Change the values in the Video Engineering fields as required.

To set video engineering options:

1. Click Editing, and then click PlayOut.

The PlayOut menu appears.



a) Rec Delay field

b) Play Delay field

2. In the Rec Delay field, specify the number of frames the shot should be shifted ahead as it is transferred from Lustre to the VTR. Enter a negative number of frames to shift the clip back by clicking the +/- button next to the field.
3. In the Play Delay field, specify the number of frames the shot should be shifted ahead as it is transferred from the VTR to Lustre. Enter a negative number of frames to shift the clip back by clicking the +/- button next to the field.

Outputting Multiple Versions

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 or manually creating 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 Log or Linear Mode”](#) on page 75.
3. Colour grade the footage for video.
4. In the Render Local menu, set Render Viewing to ON, and then adjust the black, white, and gamma as needed.
5. Render the files in the correct video resolution. See [“Rendering Shots”](#) on page 438.

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 Log or Linear Mode”](#) on page 75.
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 438.
8. Play out the rendered frames to record them on an appropriate VTR.

10

Removing Dust

Summary

<i>About Dust Removal</i>	237
<i>Preparing for Dust Removal</i>	238
<i>Removing Dust Manually</i>	241
<i>Removing Dust Automatically</i>	242
<i>Single Frame Dust Removal</i>	244
<i>Working with Boxes</i>	245
<i>Infrared Channel Dust Detection</i>	246
<i>Options for Saving and Loading Dust Removal Metadata</i>	247
<i>Rendering the Result</i>	248

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

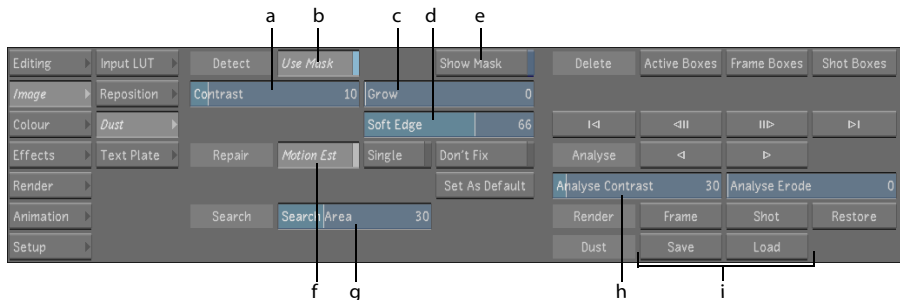
For most shots, when you set dust removal parameters for one dust artefact, the settings will work well for other artefacts in the shot. Prior to removing dust either manually or automatically, set parameters in the Dust menu to the best settings for the amount of motion and the colour composition of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

3. Enable Use Mask.

Lustre will create masks for any artefacts it finds inside dust removal boxes, based on the Contrast and Grow parameters.

4. Enable Motion Estimation.

This repair type uses a motion estimation calculation to identify motion in successive frames.

5. Set the Search Area.

This parameter controls the size of the area (in pixels) around each pixel in which the motion estimation calculation searches for motion. This enables Lustre to differentiate between motion in the shot and dust artefacts. Estimate the approximate search area required for the amount of motion-caused colour displacement between frames. Set a large area for shots with fast motion, and a small area for shots with slower motion.

NOTE: If the area set is too small for the speed of the motion, the calculation may mistakenly define colour value differences between frames caused by motion as dust artefacts. However, larger search areas take longer to analyse.

6. Adjust the Analyse Contrast slider to set the amount of contrast used to find dust during automatic analysis.

Low contrast values cause the algorithm to accept low colour value differences between corresponding pixels on the current, previous, and next frames. Therefore, low values result in more complete removal of the artefact. However, if the contrast is set lower than actually needed, adjacent pixels are modified more than actually required to remove the dust.

7. On the shot you are working on, find a frame containing an artefact that is typical of the artefacts on the footage. Use the Next Frame and Previous Frame buttons or **CTRL** and **ALT** 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 245.

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 238.
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 245.
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 244.

Removing Dust Automatically

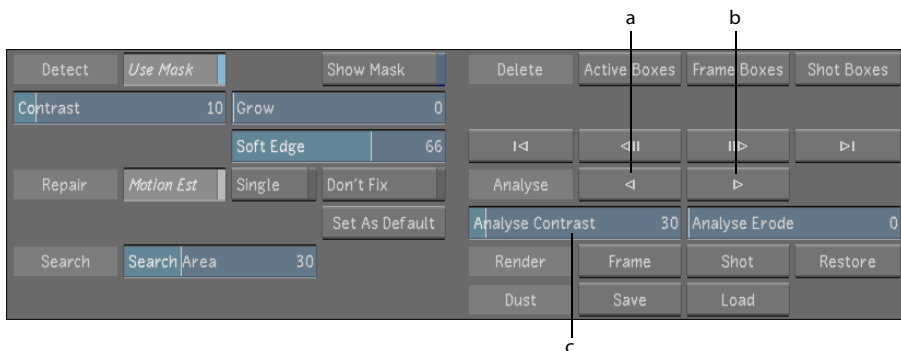
You can have Lustre automatically remove dust from each frame in the sequence by analysing the frames based on specified parameter values. Once the analysis is complete, you will need to verify that each box created is around a legitimate dust artefact, and you may need to tweak the settings for some 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 239.
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:

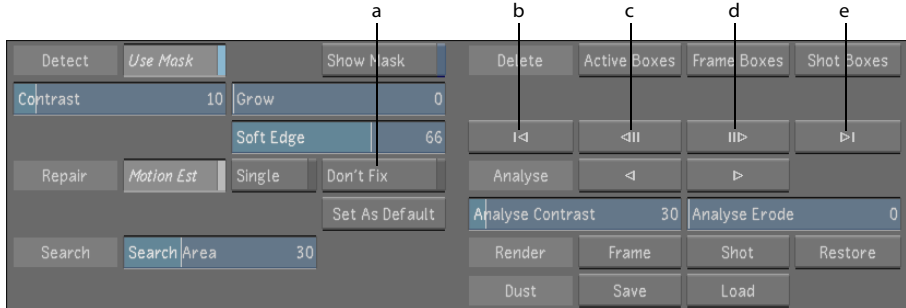
1. Visually scan the first analysed frame.

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 244.

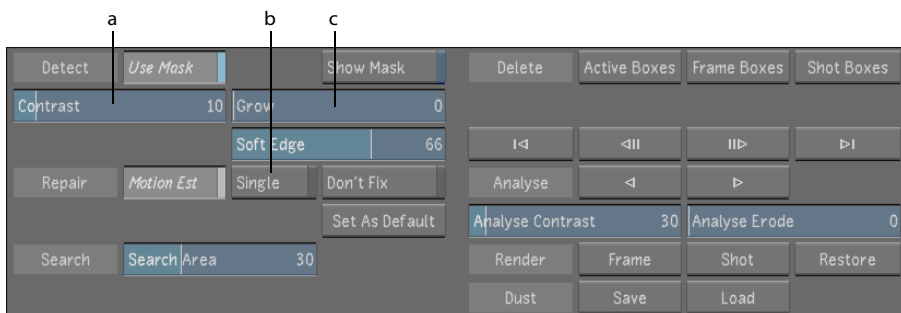
Single Frame Dust Removal

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.

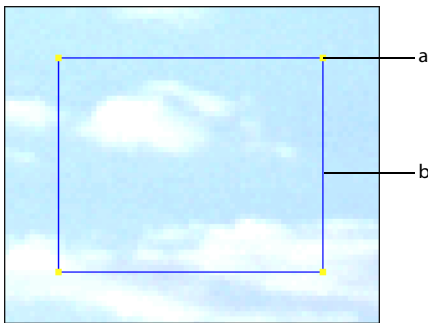
6. (Optional) If the artefact is still visible, set the repair type to Don't Fix.

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.

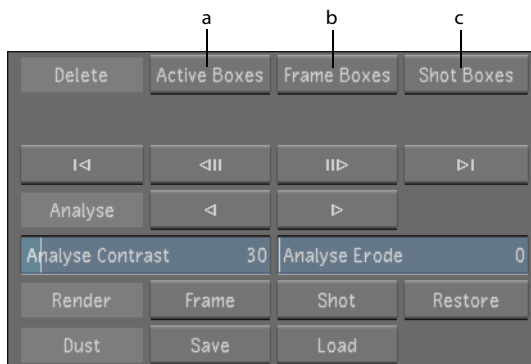
To re-size a box:

- 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.

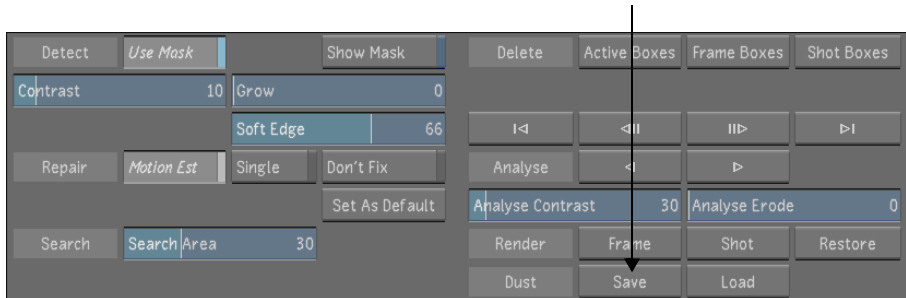
Options for Saving and Loading Dust Removal Metadata

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 [“Saving and Loading Grades for a Cut”](#) on page 78.

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:

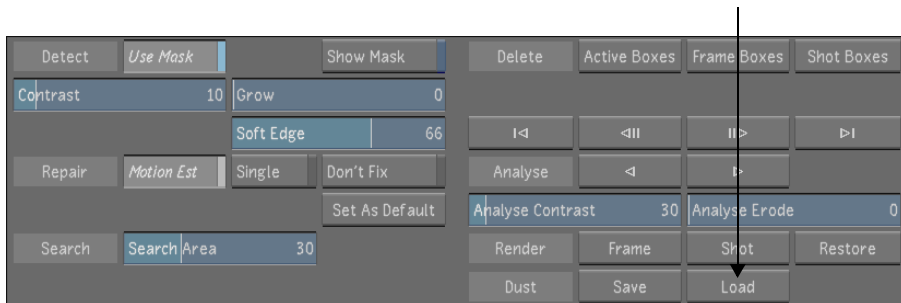
1. Add the NoAutoDustLoad keyword to the project configuration file. See [“Configuration File Supplemental Options”](#) on page 502.

Adding this keyword 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.

2. Start Lustre. If Lustre is already running, you must restart it in order for the keyword to take effect.

To load dust removal metadata after you have already loaded the grade:

- In the Dust menu, click Load.



Rendering the Result

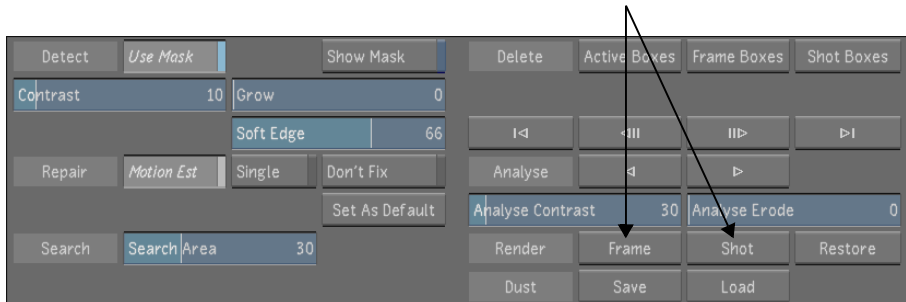
When you are satisfied with the dust removal results, you can render your material. Two rendering options are available depending on whether you render using the Render menu or the Dust menu:

- 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 438 and [“Rendering Dust Metadata”](#) on page 453.
- 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.

11

Repositioning Images

Summary

About Repositioning	251
Adjusting the Frame Aspect Ratio	253
Reframing an Image	254
Flipping or Flopping a Shot	256
Tracking Repositioned Shots	257
Animating Reposition Values	260
Setting Rendering Quality	261

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's Guide*.

HINT: For real-time playback of repositioned shots, enable GPU processing by clicking the GPU button or pressing **Y**. See [“GPU Preview Processing”](#) on page 279.

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 77.

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.

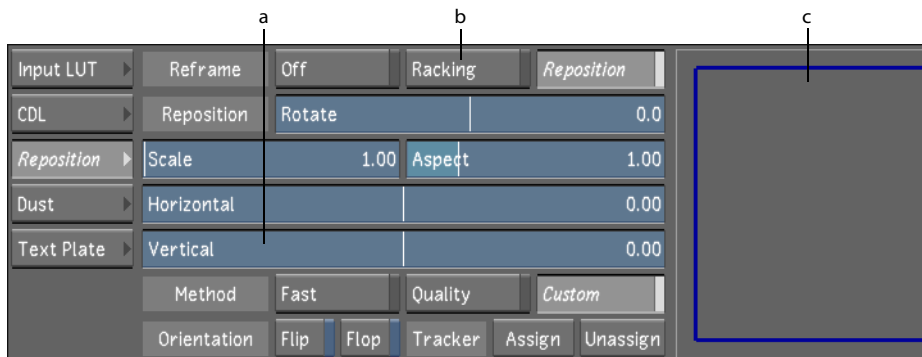
Racking an Image

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 77.
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 260.

Repositioning an Image

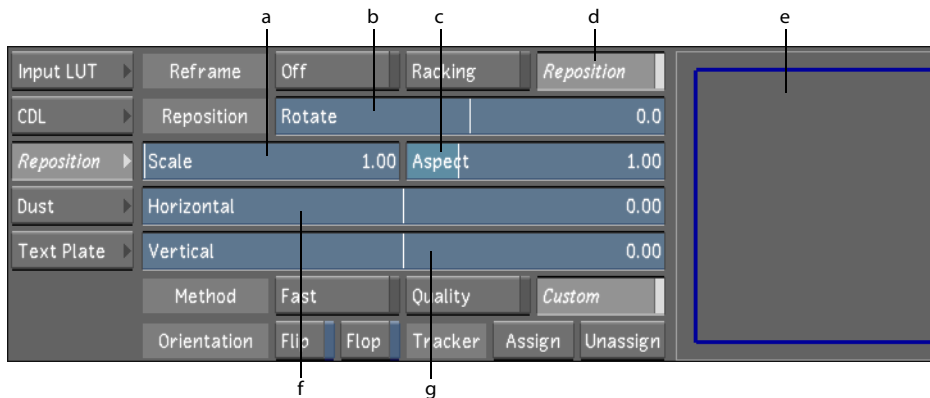
Repositioning involves scaling, rotating, and horizontally and vertically moving images to ensure that the correct elements are visible in the frame. You need to do this after adjusting the project aspect ratio. If you do not, key elements of the shot composition may be cropped or even completely off 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 77.
2. Select the shot to adjust.
3. Click Image, and then click Reposition.

The Reposition menu appears.



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

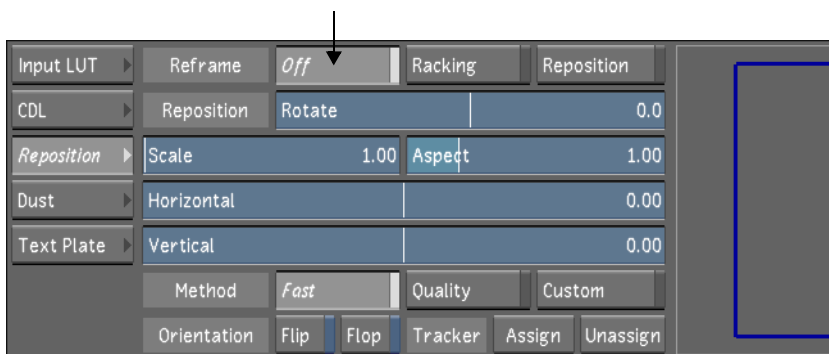
- You can animate Reposition parameters using the Animation controls. See [“Animating Reposition Values”](#) on page 260.

Turning Off Reframing

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

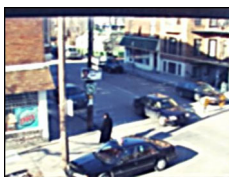
To turn off reframing:

- 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

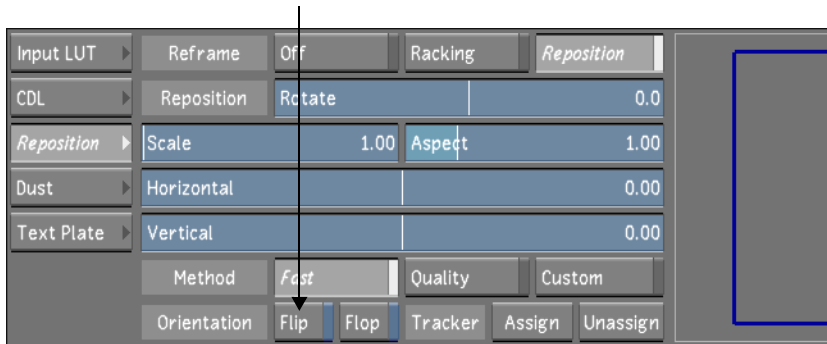


Flopped image

To flip a shot:

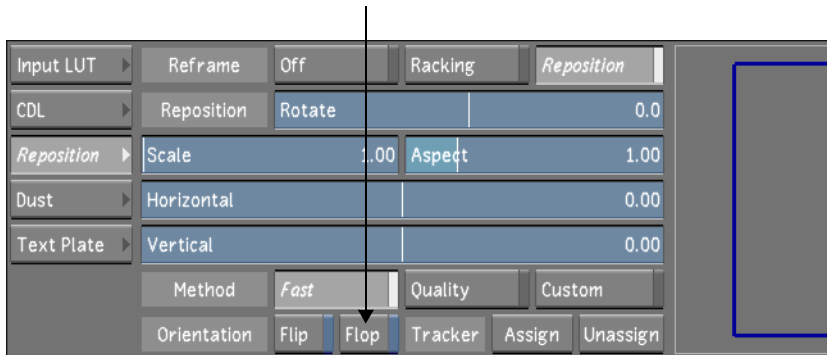
- Select the shot.
- Click Image, and then click Reposition.

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 380.

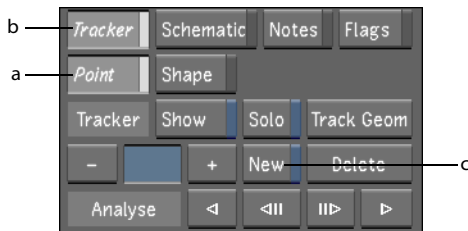
This section is intended to provide an overview of the stabilizing process. See [“Animating Geometries with the Point Tracker and Shape Tracker”](#) on page 366.

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 77.
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.



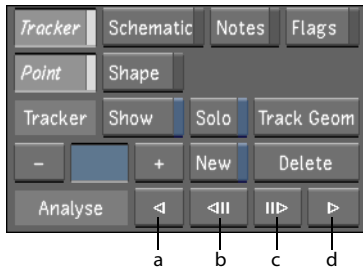
a) Point button

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 and Shape Tracker”](#) on page 366.



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

10. Enable either Racking or Reposition.

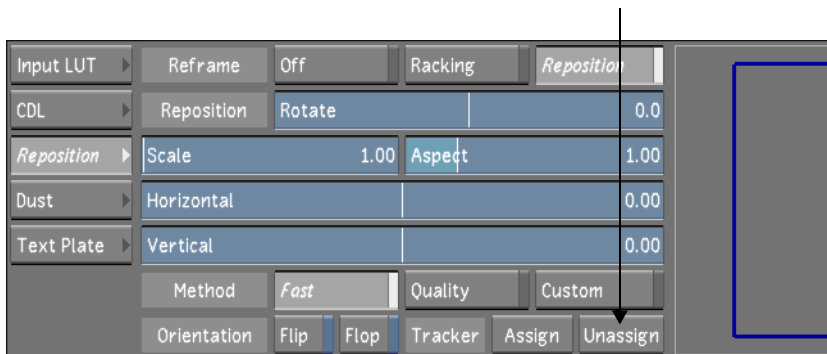
11. If you enabled Reposition, scale the image as needed.

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.

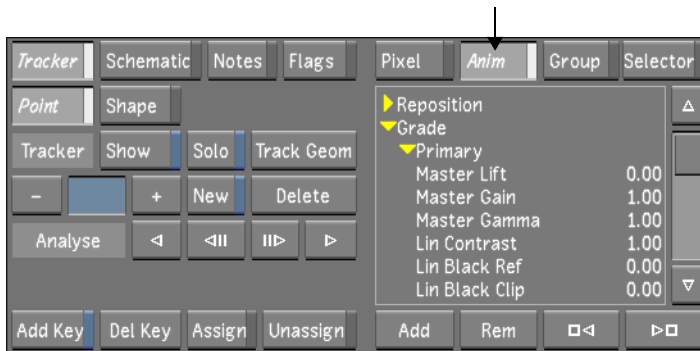


Animating Reposition Values

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:

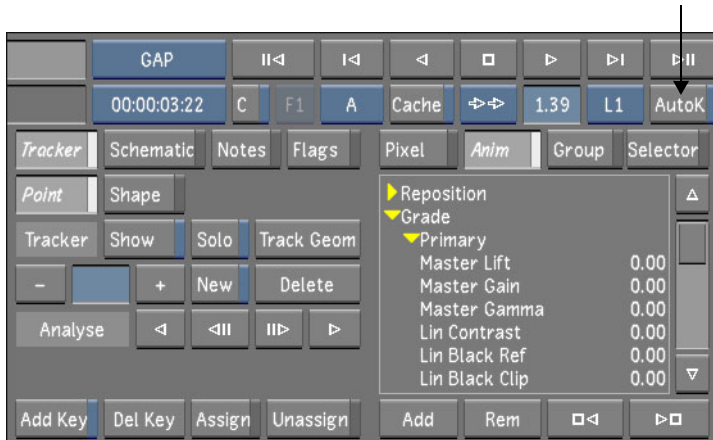
1. Select the shot to modify.
2. Click Image, and then click Reposition to display the Reposition menu.
3. Enable Racking or Reposition.
4. Click Anim to display the Animation controls.



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 404.

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).



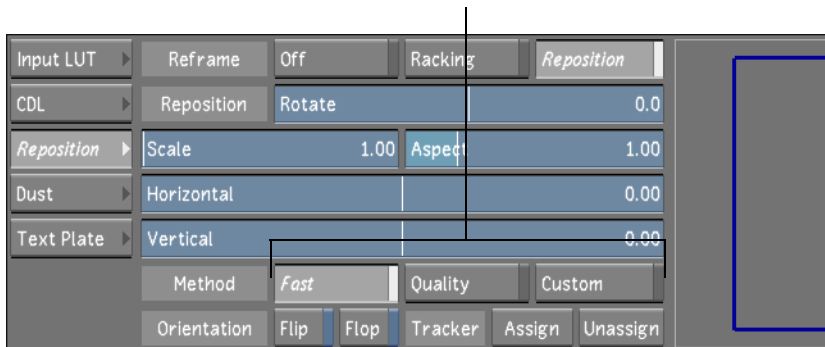
5. Move to the first frame of the shot and then adjust the scaling or the positioning as needed. Keyframes are automatically added at the current frame to the channel parameter you change.
6. Move to the last frame of the animation.
7. Set the scaling and the position. Keyframes are added to the adjusted 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 404.

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.

12

Input LUTs

Summary

About LUTs	263
Accessing the Input LUT Menu	264
Applying an Existing Input LUT	264
Linear Mode: Creating Conversion LUTs	265

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 456.

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 4, “Monitor Calibration,”](#) on page 45.

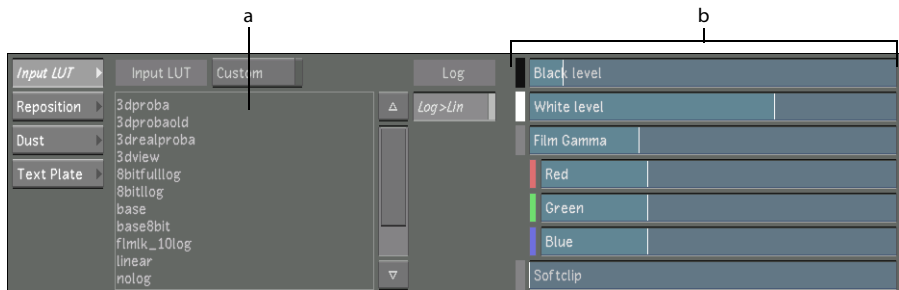
Accessing the Input LUT Menu

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)

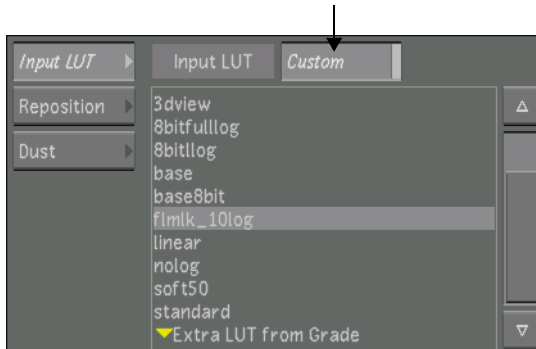
Applying an Existing Input LUT

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.

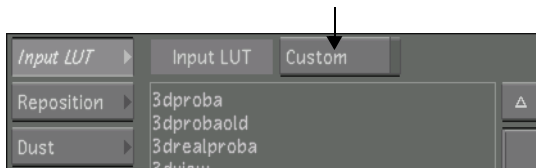


The input LUT is applied to the current shot.

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.

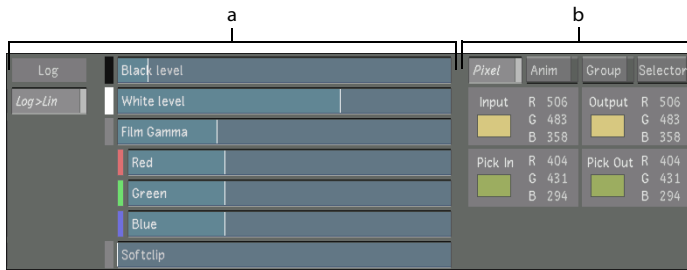


The input LUT is no longer applied to the shot.

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

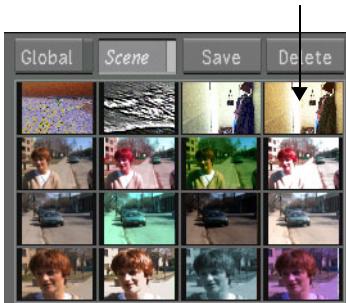
While creating Log to Lin conversion LUTs, it is important to view reference images. For example, view other shots in the cut to ensure continuity.

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 32.

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:

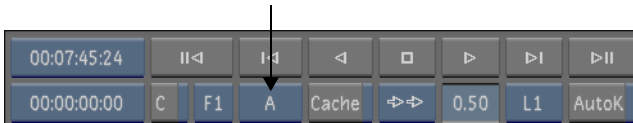
1. Load the reference image to the frame buffer by doing one of the following:
 - Go to the frame you want to use in the Storyboard and then press **K** to load it to the frame buffer.
 - Right-click a Grade bin thumbnail.



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.

- 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.

- Press **F5** to enable split view.



- 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.

- 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.

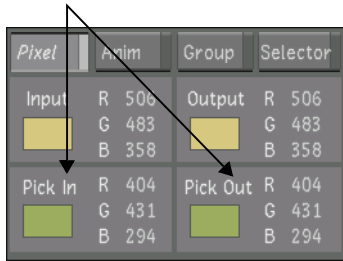
Comparing Colour Values

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:

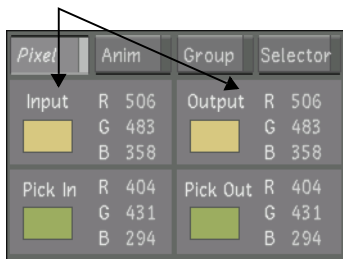
1. Click a pixel in the image.

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.

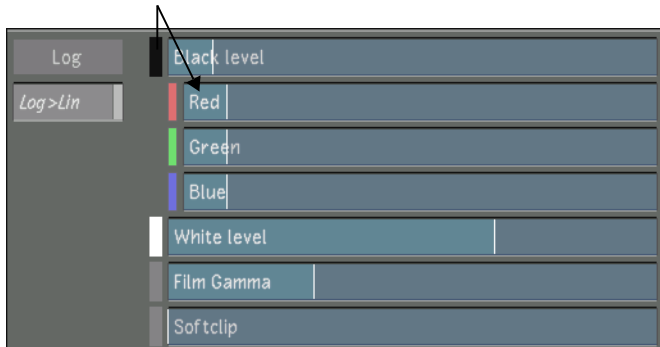


2. To modify the red, green, and blue channels in the reference white or black levels proportionally, drag the White Level or Black Level slider.



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.



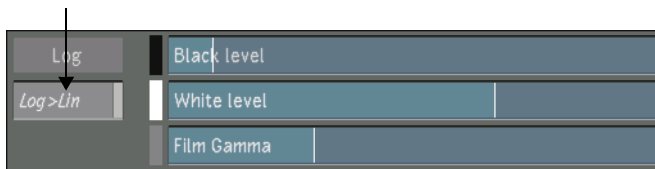
While dragging, the reference white or black level of the selected channel is displayed in the slider.

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.

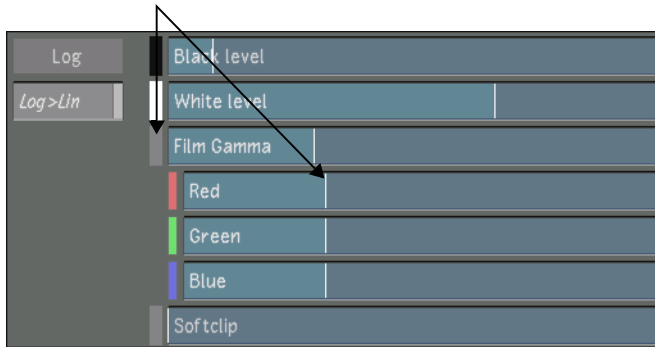


2. To modify the red, green, and blue channels for the film gamma levels proportionally, drag the Film Gamma slider.



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.



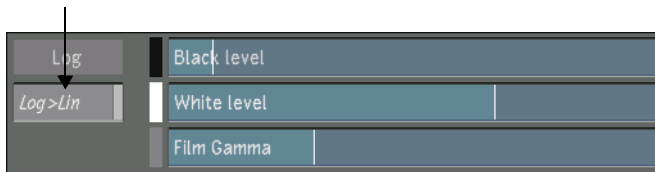
While dragging, the film gamma level for the selected channel is displayed in the slider.

Adjusting Softclip Values

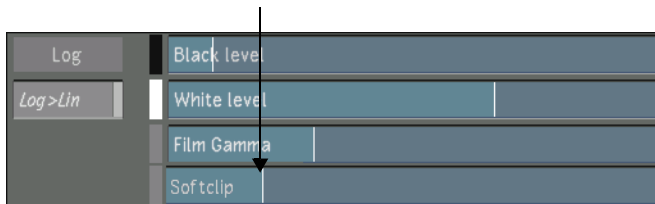
You can soften the shoulder of the gamma correction curve by adjusting the softclip levels. When you soften the shoulder of the conversion curve, you soften the transition of colours toward the reference white value. This results in softer highlights in your shot.

To soften the shoulder of the gamma correction curve:

1. In the Input LUT menu, enable the Log>Lin button.

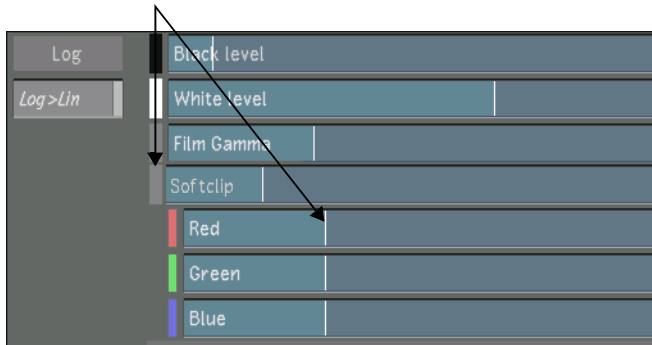


2. To modify the red, green, and blue channels for the softclip levels proportionally, drag the Softclip slider.



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.

13

Colour Grading: Basics

Summary

About Colour Grading	275
Accessing Colour Grading Menus	278
GPU Preview Processing	279
Control Surfaces	283
Viewing Multiple Shots	284
Reference Images	284
Viewing Proxies	284
Analysing the Image	285
Saving and Loading Presets Using the Presets Lists	288
Colour Grade Animation	291
Grade Bin Storage	291
Colourist Timeline	291

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 Log or Linear Mode" on page 75.
2. With primary colour grading, balance colours in the shot and develop a continuity grade to create a consistent colour look.	Chapter 14, "Primary Colour Grading," on page 295.
3. Further develop the colour look using the RGB and Hue curves.	Chapter 15, "Colour Grading: RGB and Hue Curves," on page 315.
4. Perform secondary colour grading to colour grade objects and colour ranges in the shot.	Chapter 16, "Secondary Colour Grading," on page 327.
5. Use Sparks plugins to apply image processing effects such as grain reduction.	Chapter 17, "Creating Lustre Sparks Effects," on page 387.
6. Adjust the colour look using the RGB curves.	Chapter 15, "Colour Grading: RGB and Hue Curves," on page 315.
7. Perform any necessary primary colour grading to adjust the colour look and complete the colour grade.	Chapter 14, "Primary Colour Grading," on page 295.

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 *lustre.config* file. See [Appendix A, “Software, Project, and Wiretap Configuration Files,”](#) on page 495.

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 Preview Processing

Processing with the Graphics Processing Unit (GPU) is available for certain features and is more efficient than processing with the Central Processing Unit (CPU). GPU processing allows for

higher performance for up to 2K resolution while previewing and real-time playback (24 fps) for HD resolution under certain scenarios.

NOTE: Support for GPU preview processing depends on the version of the NVIDIA graphics card installed on your system. The following configurations are supported for GPU preview processing.

Configuration	Feature available?
FX5500	Yes
FX4000 with HD/2K	Yes, but only when GFX SDI out is disabled
FX4000 with SD	Yes

Though GPU preview processing can be enabled, real-time playback, while applying certain colour grading effects, is not guaranteed for all scenarios. On a system with the NVIDIA FX5500, HD (1920x1080) real-time playback performance (24 fps) is expected for up to four secondaries with one geometry per secondary, and with the supported GPU parameters. On a system with the NVIDIA FX4000, SD (780x486 and 720x576) real-time playback performance (NTSC 30 fps/PAL 25 fps) is expected for up to four secondaries with one geometry per secondary, and with the supported GPU parameters. For 2K full AP (2048x1556), improved performance is expected, but real-time playback is not guaranteed. On all systems with more than four (up to 12) secondaries, improved performance is expected, but real-time playback is not guaranteed.

NOTE: When GPU preview processing is enabled, the DVS should only be used as an output device when playing out to VTR. When using the DVS as a preview, only the GPU features are displayed.

GPU preview processing is available for the following features:

- 1D or 3D on calibration LUTs
- NVIDIA GFX SDI video resolutions for play out
- 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 black clip/reference and white clip/reference in Linear mode
- Gamma and contrast adjustments to secondaries in Linear mode
- Low, mid, and high adjustments to secondaries
- Input and render/output LUT
- Output primary saturation

- Lustre Sparks plugins
 - Blur mix
 - Glow
 - Printbleach

NOTE: A performance hit can occur when using the Lustre Sparks plugins.

For more information about these features, see:

- [Chapter 14, “Primary Colour Grading,”](#) on page 295
- [Chapter 15, “Colour Grading: RGB and Hue Curves,”](#) on page 315
- [Chapter 17, “Creating Lustre Sparks Effects,”](#) on page 387

GPU preview processing is also available for certain secondary grading features with up to 12 layers enabled. When a certain feature that is not supported on GPU is enabled, Lustre automatically toggles back to CPU processing mode. You can apply, and optionally animate, each of the following secondary grading features and continue to use GPU preview processing:

- 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 16, “Secondary Colour Grading,”](#) on page 327.

The following parameters cannot be processed by the GPU. These parameters are processed by the CPU instead:

- Secondary grading with hue curves
- Imported mattes for secondaries
- Secondary key cleanup and shrink

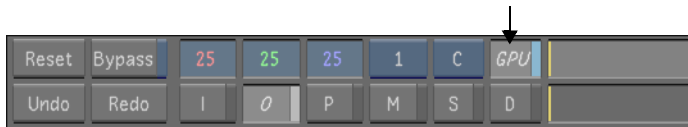
- Lustre Sparks plugins

- Add noise
- Blur
- Defocus
- Directional blur
- Field zoom
- Gold
- Noise plugin 2
- Noise plugin 3
- Noise plugin 3.1
- Silver

NOTE: If GPU processing is enabled and Lustre encounters a shot with features that cannot be processed by the GPU, CPU processing 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 preview processing.

To enable GPU preview processing:

- Click the GPU button or press **Y**.



A GPU flag appears in the upper-right section of the Player.



To disable GPU preview processing:

- Press **SHIFT+Y**.
- GPU processing is disabled.

GpuNoAutoSwitch Parameter

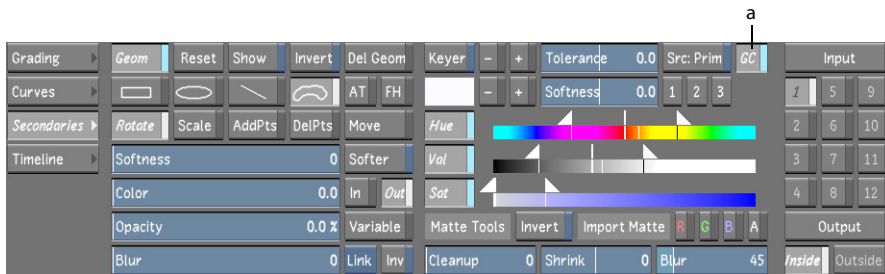
By default, when you use a feature that is not supported by GPU preview processing while in GPU preview processing mode, you are automatically switched to CPU processing mode. You can prevent being switched to CPU mode by enabling the GpuNoAutoSwitch parameter in the configuration file. See [Appendix A, “Software, Project, and Wiretap Configuration Files,”](#) on page 495. This parameter disables all features that are unsupported by GPU preview processing and makes them unavailable (greyed-out) in the user interface.

GPU Compatibility

Key blurs and geometry blurs are processed differently depending on whether CPU or GPU preview processing 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.

NOTE: The GPU compatibility button is automatically enabled when GPU preview processing is enabled.

NOTE: Enabling the GC button does not impact final renders, nor does it jeopardize preview performance. It ensures that GPU previewing is performed in such a way as to closely match how keys, key blurs, and geometry blurs are displayed in CPU grading mode.



a) GC button

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's 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 147.

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 145.

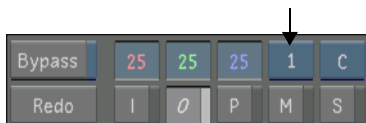
Viewing Proxies

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 462.

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

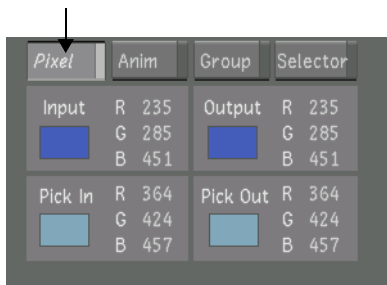
When you generate mattes, modify shots, or match colours between shots, you can sample the current image in the Player. When you sample the image, the input and output colour values are displayed in the Pixel Analyser.

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 329.

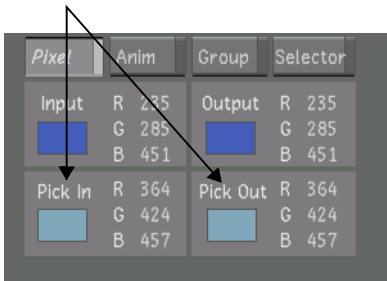
To sample colour values:

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.

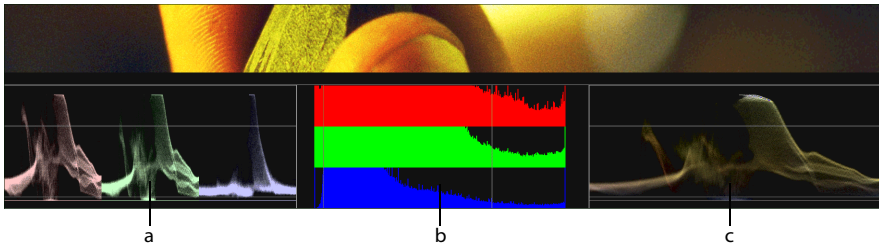
Vectorscope — Displays the distribution of image pixels across luma and hue ranges.

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.

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 340.

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.

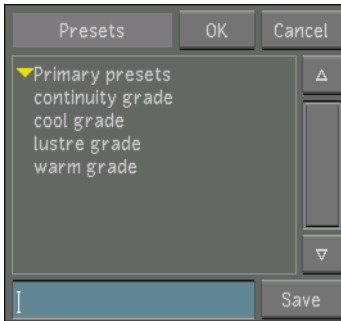
NOTE: Saving to a preset saves the settings currently displayed in the Player. For example, if you save to a preset in the middle of a colour dissolve or animation, this timeline position is saved to the preset file. Saving to a preset is a good way to save and load animation keyframes.

To save a preset:

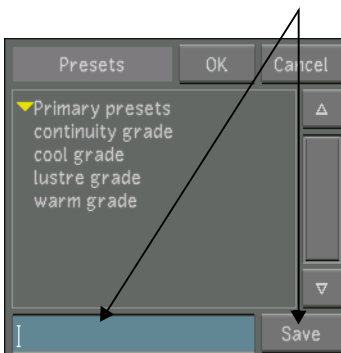
1. Do one of the following:
 - Right-click or middle-click the Colour menu where you want to set parameters for a preset.
 - In the Secondaries menu, right-click the Keyer button to set parameters for a Key preset.

HINT: You can also save a Key preset directly to the Secondaries menu. See [“Saving and Loading Key Presets”](#) on page 340.

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

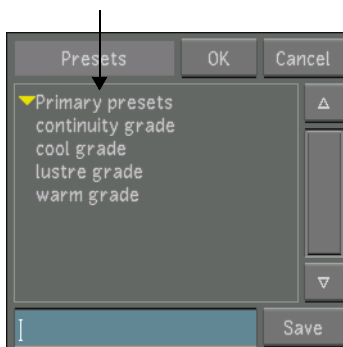
Increase efficiency by loading presets to the current shot when you want to reuse Colour menu settings. Preview the effect a preset will have on a shot before you apply the settings.

To load a preset from a Presets list:

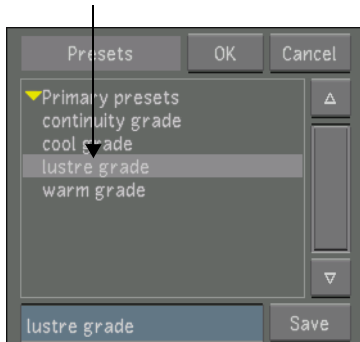
1. Do one of the following:
 - Right-click the Colour menu that contains the preset you want to load.
 - In the Secondaries menu, right-click the Keyer button to set parameters for a Key preset.

HINT: You can also load a Key preset directly from the Secondaries menu. See [“Saving and Loading Key Presets”](#) on page 340.

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.



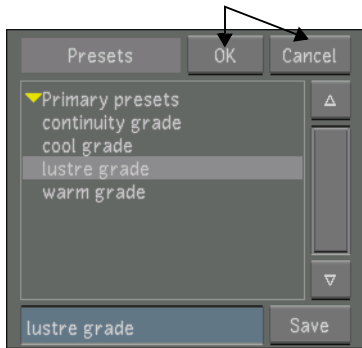
The image in the Player is updated with the settings from the preset so that you can preview the effect.

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 18, “Animation,”](#) on page 403.

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 32.

Colourist Timeline

The Colourist Timeline allows you to perform limited Timeline editing functions within the Colour menu.

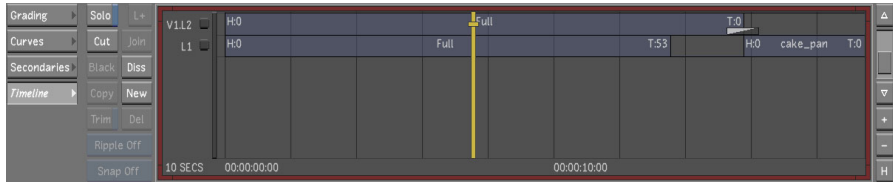
Accessing the Colourist Timeline Menu

To edit shots in the Colourist Timeline view, you need the Timeline menu.

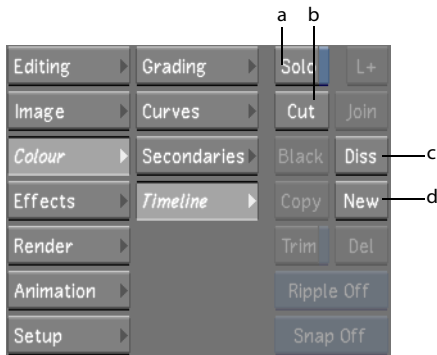
To access the Colourist Timeline menu:

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
- b) Cut button
- c) Diss button
- d) New button

Solo — Within a multi-layered 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. To see the Timeline user interface and the grading toolset at the same time, press **F7**.

For more information about the multi-layer Timeline, see [“About Timeline”](#) on page 175.

14

Primary Colour Grading

Summary

<i>About Primary Grading</i>	295
<i>Accessing the Grading Menu</i>	296
<i>Logarithmic Mode: Modifying the Brightness</i>	296
<i>Logarithmic Mode: Improving Contrast</i>	302
<i>Linear Mode: Selecting Levels of the Image</i>	305
<i>Linear Mode: Modifying Lift</i>	306
<i>Linear Mode: Modifying Gain</i>	308
<i>Linear Mode: Modifying Gamma</i>	308
<i>Linear Mode: Modifying Saturation</i>	309
<i>Linear Mode: Modifying Contrast</i>	310
<i>Linear Mode: Setting Black Clip and White Clip</i>	311
<i>Matching Colours</i>	312
<i>Autograding Shots</i>	313

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 Preview Processing”](#) on page 279.

Accessing the Grading Menu

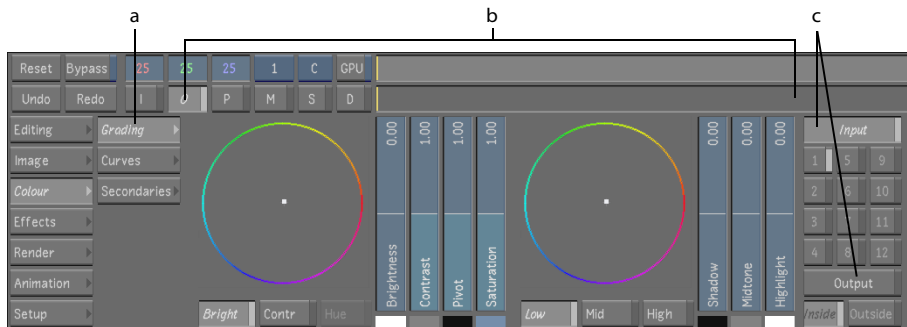
Primary colour grading is done from the Grading menu.

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

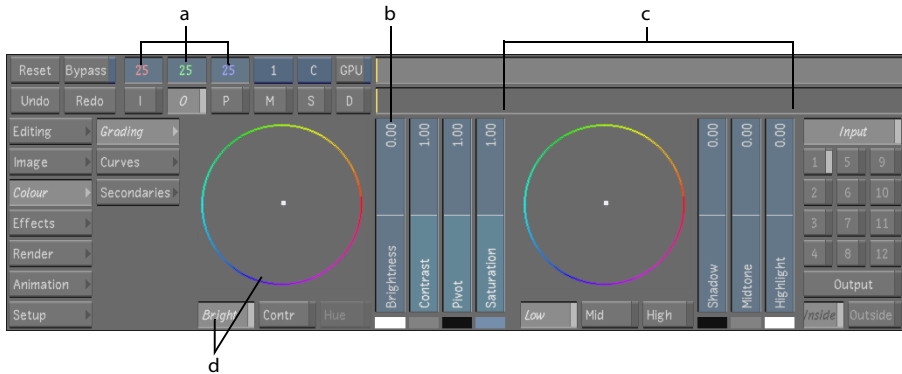
Output — Enables output primary grading. Use it to perform primary colour grading at the end of the colour grading process, that is, after RGB and Hue modifications, as well as selective grading have been performed.

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 318.

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:

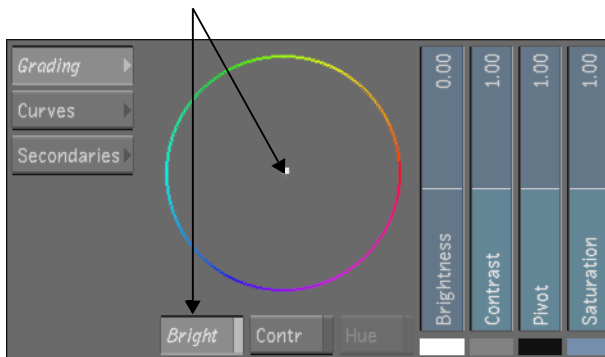
- To increase the overall brightness in the image, drag the Brightness slider up.

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 *lustre.config* file and are acquired from your film lab. See [Appendix A, “Software, Project, and Wiretap Configuration Files,”](#) on page 495.

NOTE: Hot keys for printer light adjustments only work in Log mode.

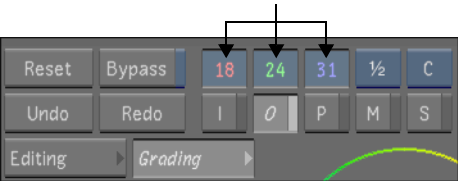
To modify brightness by printer lights:

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.
6	Increase yellow brightness by one step. Press SHIFT+6 to increase yellow brightness by a half step.

Press:	To:
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 300.

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:

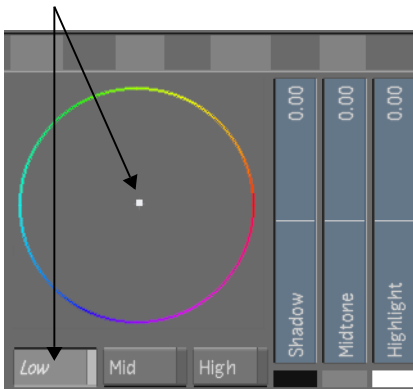
1. Indicate whether this is the initial or final primary grade:
 - To work on the initial primary colour grade, click Input.
 - To make final adjustments to the primary colour grade, click Output.

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:

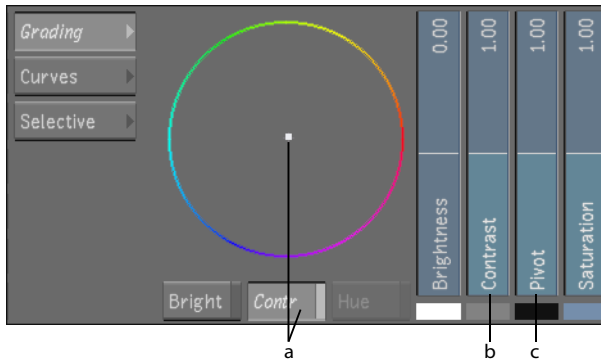
- Press - (minus sign) on the keypad.

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 17.

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 303 and [“Modifying Saturation”](#) on page 304.



- 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.

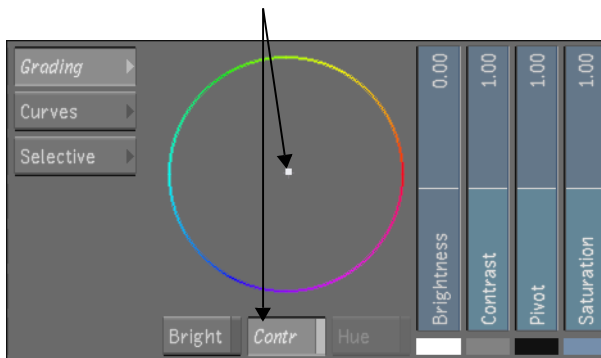
Modifying Image Contrast

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.

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 303.

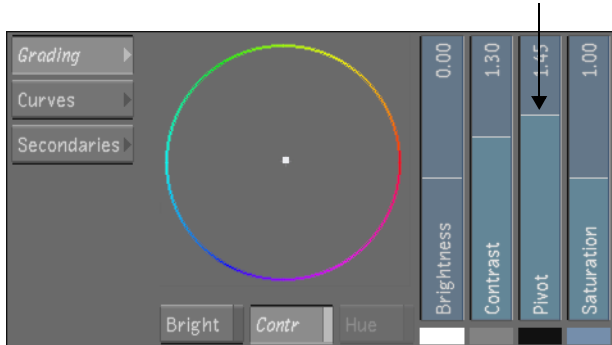
4. Make adjustments to the image using the Contrast slider and Contrast Balance wheel until you are satisfied with the results.

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**

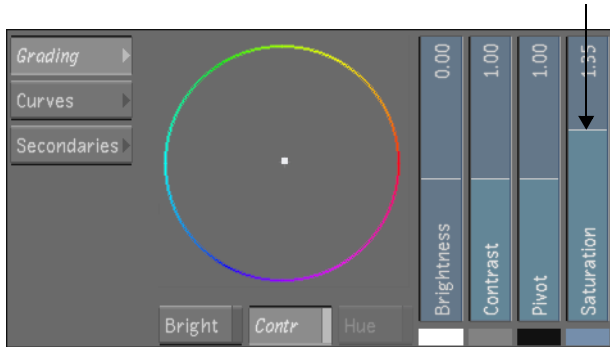
Control the image's colour intensity by modifying saturation levels.

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

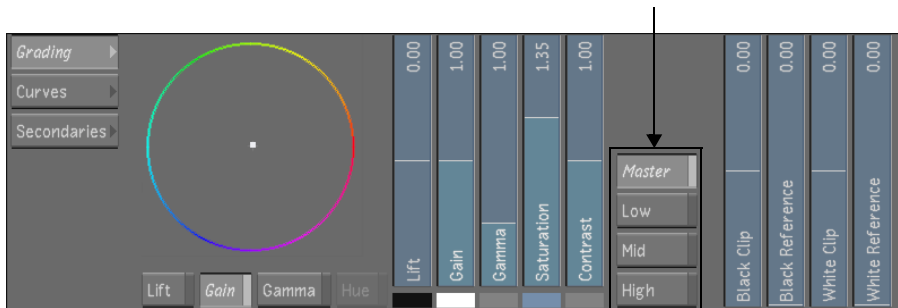
You can modify lift, gain, gamma, and saturation in the low, mid, and high levels of the image, or across the entire image. Select the range you want to modify while performing primary colour grading. You can also define values to clamp minimum and maximum luminance values.

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

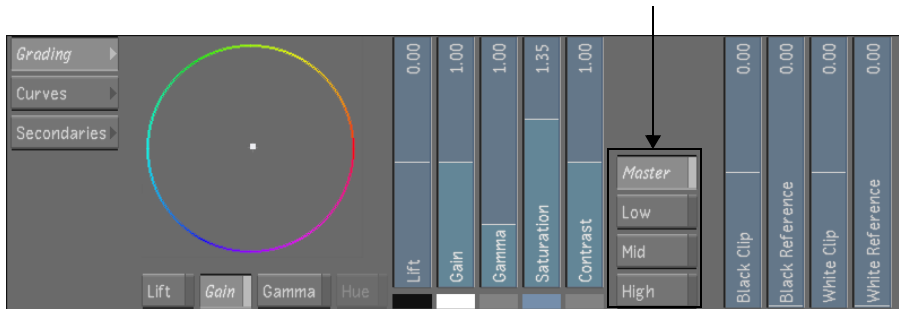
You can lift colour values in a selected range or in the entire image, and then offset ranges of colour. Changes in lift mostly affect black level. Although lift affects the entire image or image level, visually, it mostly affects the shadows.

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:

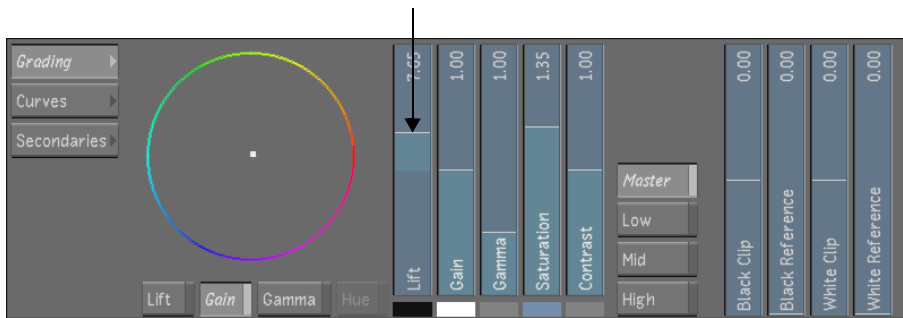
1. Indicate whether this is the initial or final primary grade:
 - To work on the initial primary colour grade, click Input.
 - To make final adjustments to the primary colour grade, click Output.

2. Select the image level you want to modify.

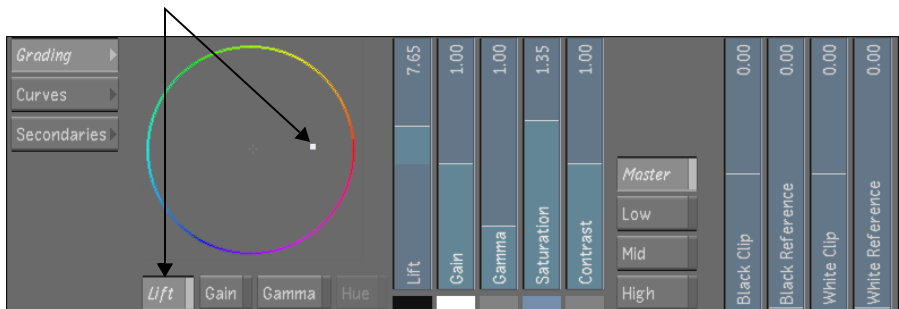


3. Modify the black levels:

- 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.

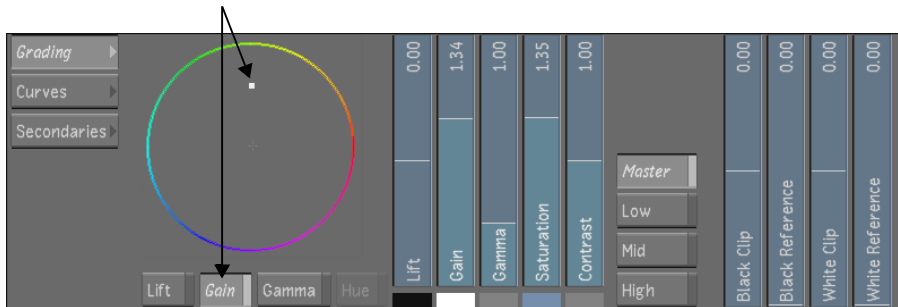
Linear Mode: Modifying Gain

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.

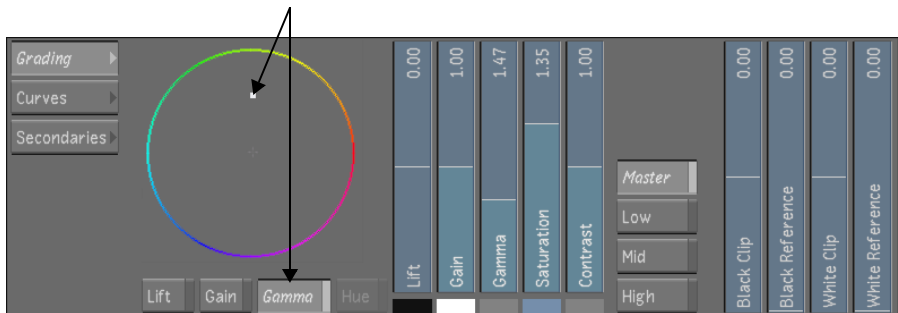
Linear Mode: Modifying Gamma

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.

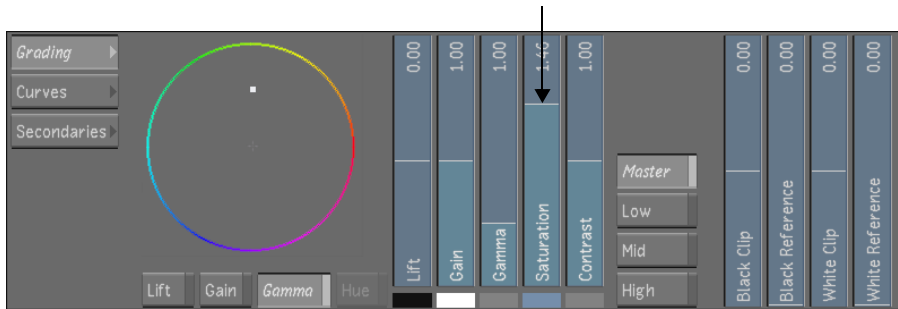
NOTE: Select Low, Mid, or High to further differentiate between the areas of the image that are affected.

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.



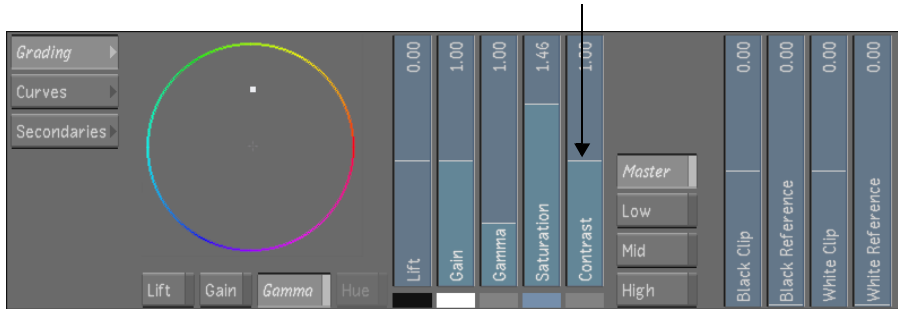
Linear Mode: Modifying Contrast

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:

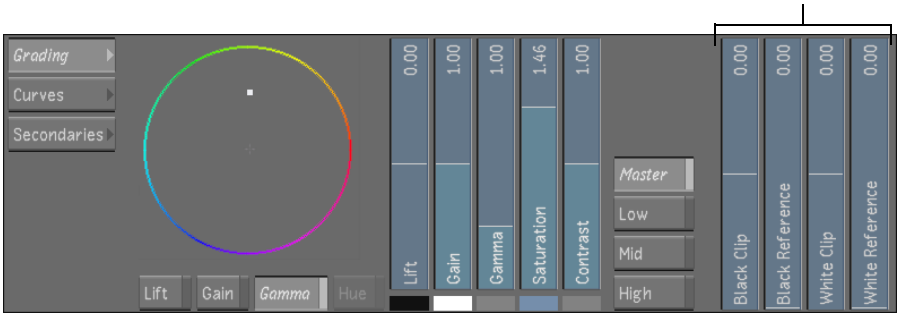
1. Indicate whether this is the initial or final primary grade:
 - To work on the initial primary colour grade, click Input.
 - To make final adjustments to the primary colour grade, click Output.
2. Modify the clip and reference values using the following sliders:

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.

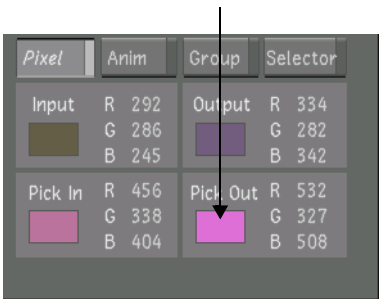
To match colours:

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 147.
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

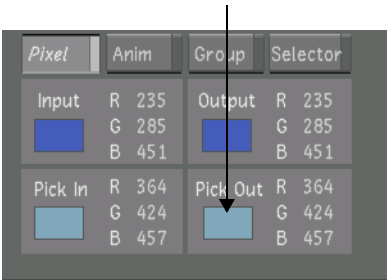
more blue than any other colour, yellow (the colour that is opposite on the colour wheel) is added to the image. To properly autograde a shot, sample an area of the image that should be neutral or have no hue cast. Sample blacks, whites, or greys.

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	315
Accessing the Curves Menu	316
Plotting Colour Values when Adjusting Curves	316
Modifying Red, Green, and Blue Channels	318
Modifying Hue, Saturation, Luminance, and Lightness	319

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 16, “Secondary Colour Grading.”](#) on page 327. 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 Preview Processing”](#) on page 279.

Accessing the Curves Menu

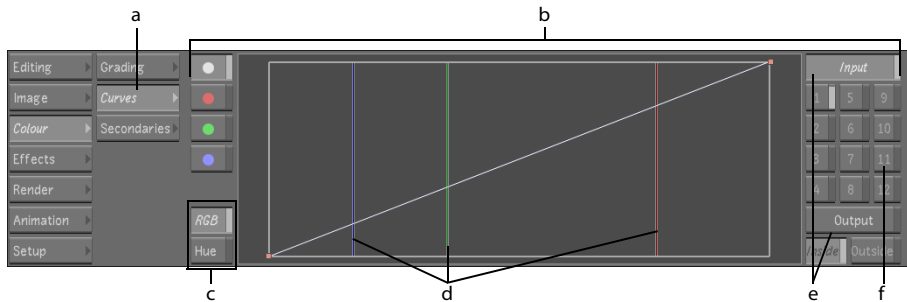
Use the Curves menu to modify RGB and Hue curves.

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

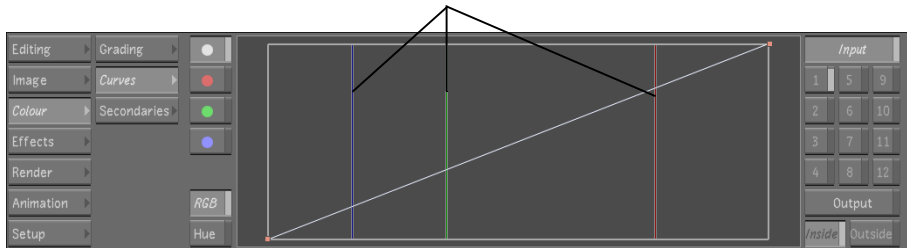
Plotting Colour Values when Adjusting Curves

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.

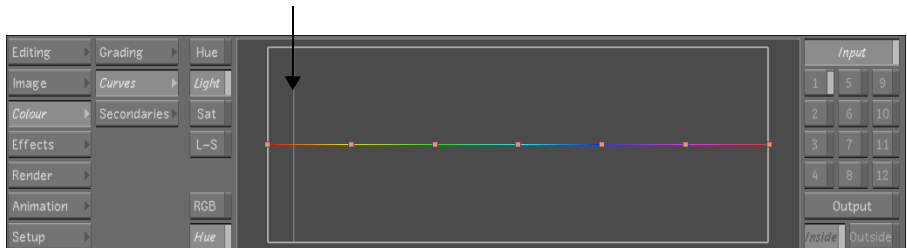
To plot colour values when adjusting curves:

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.



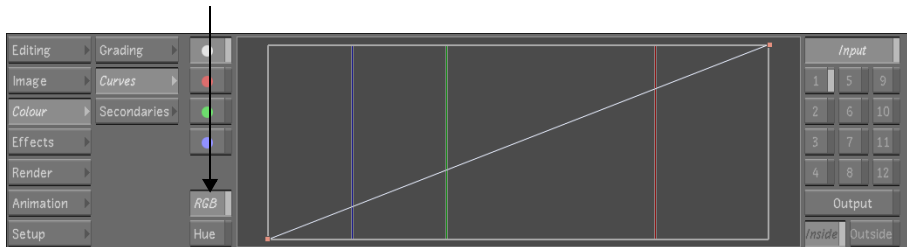
3. To zoom in on a curve using rectangle zoom, drag a selection box around the desired area while pressing **ALT**, and then release the mouse.
The area of the curves contained by the selection box now occupies the entire Curve canvas.
4. To zoom in on a curve using proportional zoom, press **ALT** while 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:

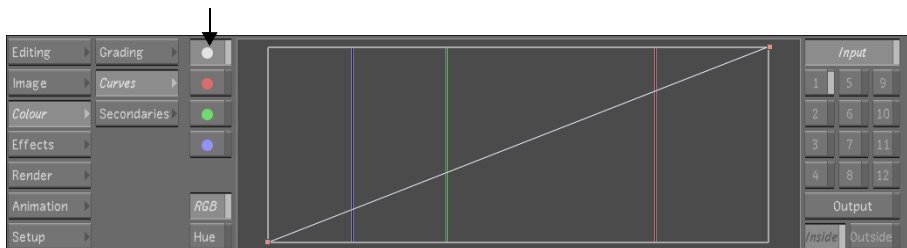
1. Indicate the colour grading stage at which the curve modification is being applied:
 - To work on the initial primary grade, click Input.
 - To make final adjustments to the primary grade, click Output.
 - To make curve modifications for secondary colour grading, enable a secondary layer.
2. Click 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.

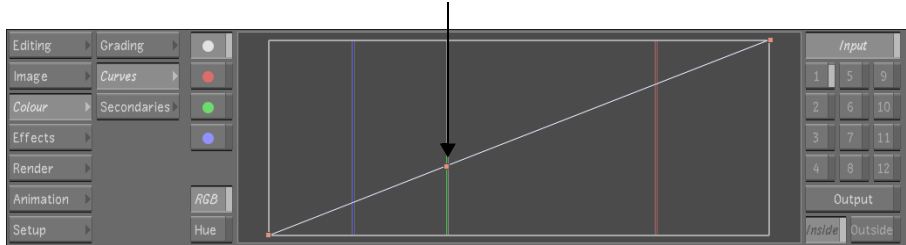
NOTE: To display the RGB curve, click the white button.



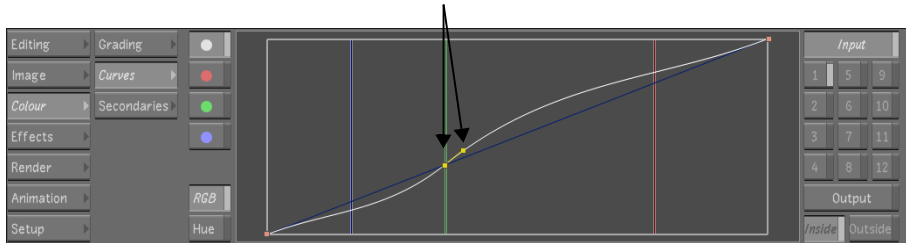
4. Sample the colour you want to use as a reference for your adjustments. See [“Plotting Colour Values when Adjusting Curves”](#) on page 316.

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:

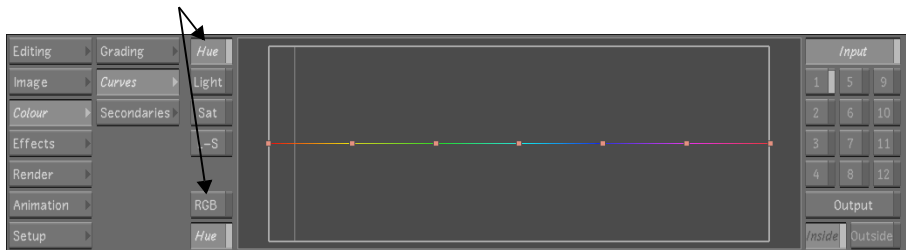
- Perform hue shifts.
- Suppress or saturate colour ranges.
- Suppress or saturate luminance ranges.
- Lighten or darken colour ranges.
- Add vertices to the curves to refine your adjustments.

Shifting Hue

Shift the hue of the entire image or a range of colour in the image.

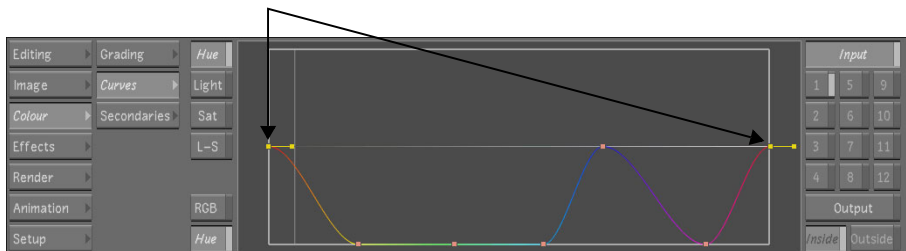
To perform a hue shift:

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 316.
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.

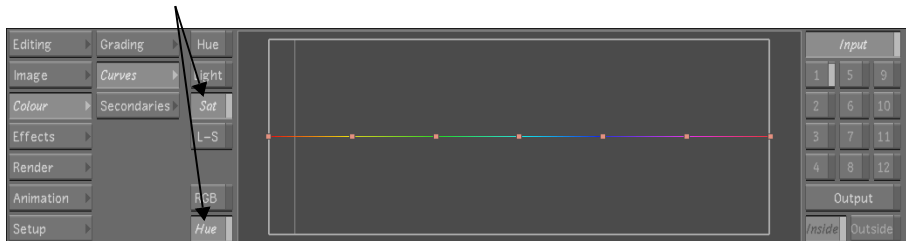
5. Modify the curve until you are satisfied with the results:
 - To select a vertex, click it. To select several vertices, draw a selection box around them.
 - To move several selected vertices, select them and then drag one. To restrict vertex movement to the Y axis, press **SHIFT** while moving the 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.

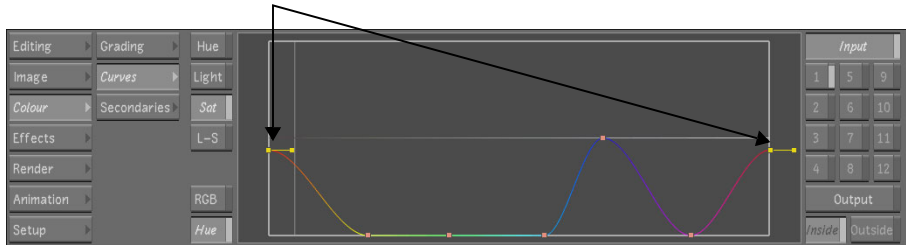


The Saturation curve is plotted to the hue spectrum.

3. Sample the colours you want to use as a reference for your adjustments. See [“Plotting Colour Values when Adjusting Curves”](#) on page 316.
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.

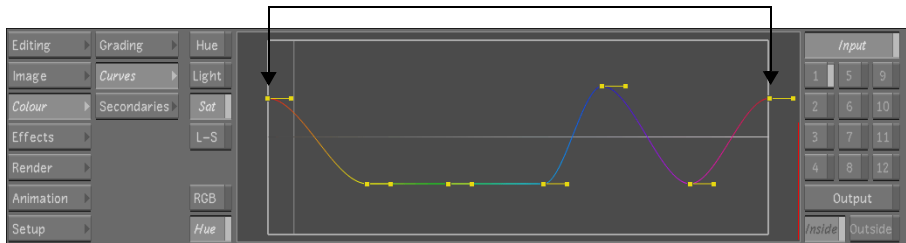
5. Saturate or suppress the colour:

- 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.

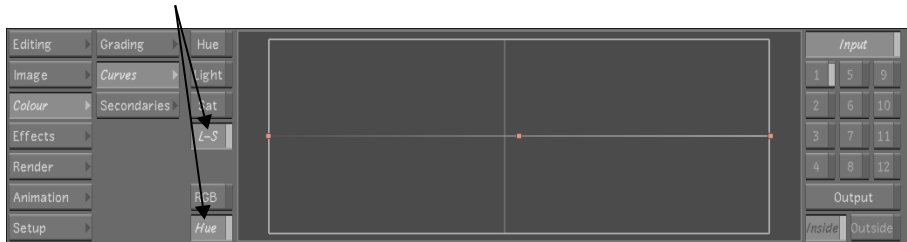
7. To add a vertex to the curve, place the mouse cursor over the area on the curve where you want to add the vertex and then press **A**. To lock the curve in place while adding a vertex, press **SHIFT+A** on the curve.
8. To delete vertices, select them and then press **D**.
9. Edit the curve until 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.

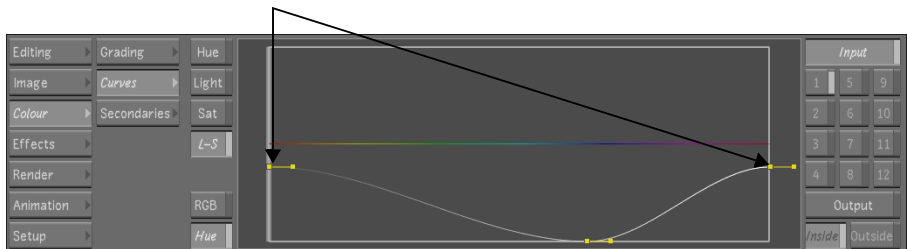
To suppress or saturate luminance areas in the image:

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.



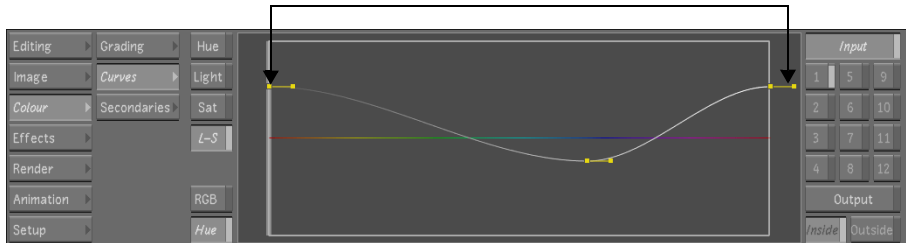
The Luminance curve is plotted to the hue spectrum.

3. Sample the luminance areas you want to use as a reference for your adjustments. See [“Plotting Colour Values when Adjusting Curves”](#) on page 316.
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**.



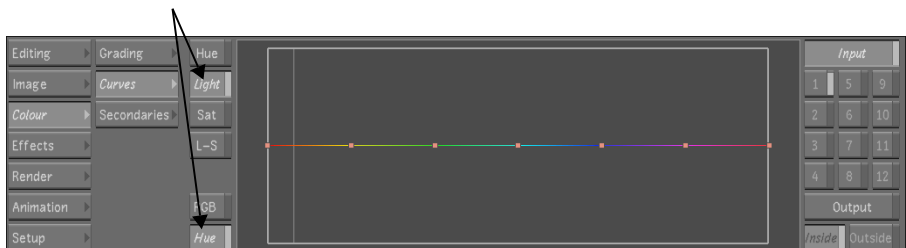
- To add a vertex to the curve, place the cursor over the area on the curve where you want to add the vertex and then press **A**. To lock the curve in place while adding a vertex, press **SHIFT+A** on the curve.
- To delete vertices, select them and then press **D**.
- Edit the curve until you are satisfied with the results.

Modifying Image Lightness

Increase or decrease the lightness in specific colour ranges or across the image.

To modify image lightness:

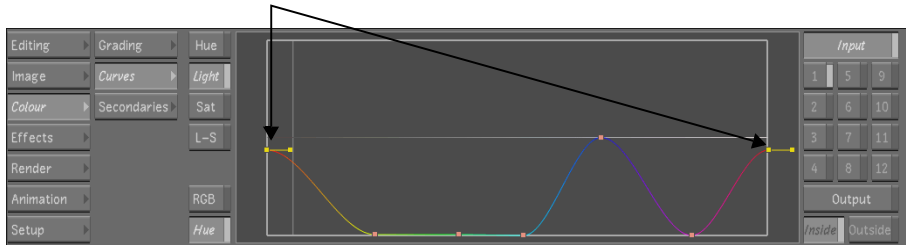
- Indicate the colour grading stage at which the curve modification is being applied:
 - To work on the initial primary grade, click Input.
 - To make curve modifications for secondary colour grading, enable a secondary layer.
- Click Hue and then click Light to display the Lightness curve.



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.

16

Secondary Colour Grading

Summary

<i>About Secondary Colour Grading</i>	327
<i>Accessing Secondary Grading Menus</i>	328
<i>Creating a Secondary by Keying a Range of Colours</i>	329
<i>Saving and Loading Key Presets</i>	340
<i>Importing Mattes</i>	341
<i>About Geometries</i>	342
<i>Accessing the Geometry Controls</i>	343
<i>Drawing Basic Geometries</i>	344
<i>Drawing Free-Form Geometries</i>	345
<i>Modifying Geometries</i>	349
<i>Adding a Split Screen</i>	357
<i>Adding and Deleting Axes</i>	358
<i>Changing the Priority Order of Geometries</i>	361
<i>Showing and Hiding Geometries</i>	362
<i>Inverting Geometries</i>	362
<i>Deleting Geometries and Resetting Secondaries</i>	363
<i>Linking Geometries across Secondary Layers</i>	364
<i>Animating Geometries with the Point Tracker and Shape Tracker</i>	366
<i>Loading Tracker and Stabilizer Data from Autodesk Applications</i>	380
<i>Loading Garbage Mask Setups from Autodesk Applications</i>	382
<i>Colour Grading Secondaries</i>	382

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 17, “Creating Lustre Sparks Effects,”](#) on page 387.

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 processing makes faster playback available for certain secondary grading features. See [“GPU Preview Processing”](#) on page 279.

Accessing Secondary Grading Menus

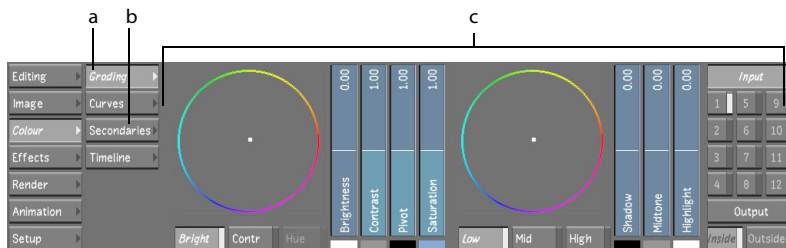
You perform secondary colour grading with the Secondaries and Grading menus.

For control surface mappings, see the *Autodesk Control Surface User’s 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 15, “Colour Grading: RGB and Hue Curves,”](#) on page 315.

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.

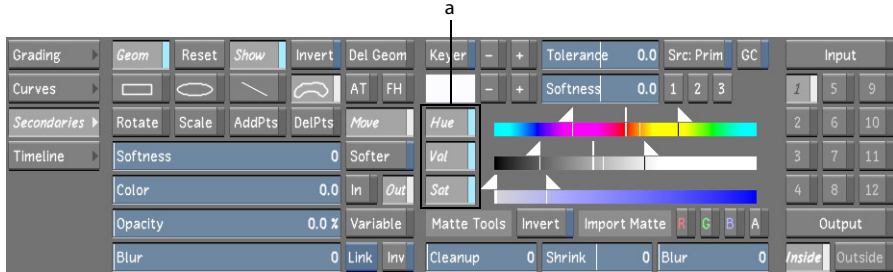
Selecting Channels for a Key: Hue, Luminance, and Saturation

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 288 and [“Saving and Loading Key Presets”](#) on page 340.

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.
3. 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**

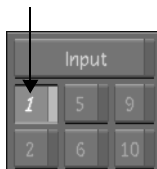
To generate a key, you set the initial softness and tolerance values by sampling a pixel or an average colour in the image.

To set the initial softness and tolerance:

1. Set the view to Output. You can click the O button or press **F12**.



2. 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 \.

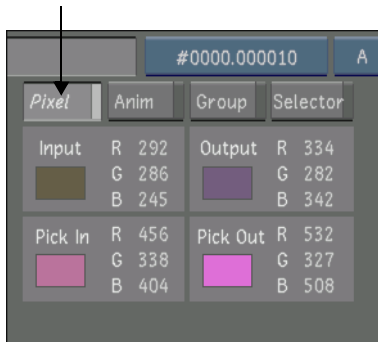
3. Click the colour pot.

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.

- Click Pixel to display the Pixel Analyser.



- 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.

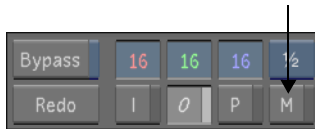


Viewing Secondaries

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**.

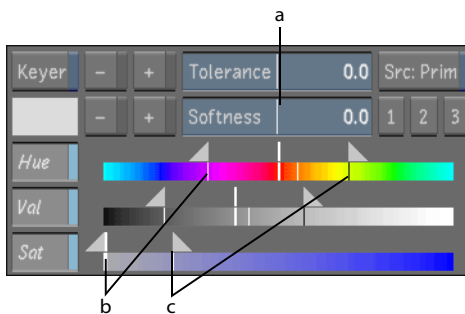
In Matte view, the white parts of the matte are fully keyed and can be selectively colour graded. The black parts are excluded from the secondary. Greys indicate the softness parameters.



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:

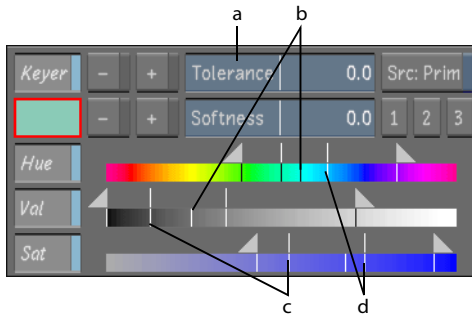
1. View the secondary in the Player. Press **F11** to toggle between Matte view and Secondary view.

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.

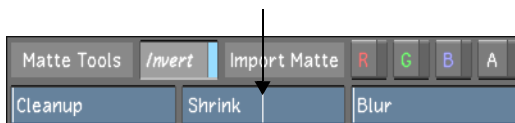
3. For the Hue, Value, or Saturation channel, set the minimum value for the tolerance range using the Minimum Tolerance Range indicator. Drag the indicator to the left to increase tolerance. Drag the indicator to the right to decrease tolerance.
4. For the Hue, Value, or Saturation channel, set the maximum 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.

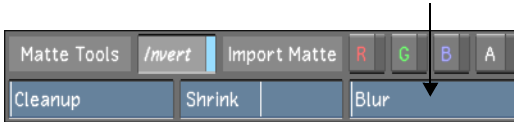
To shrink or erode the edge of the key:

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.

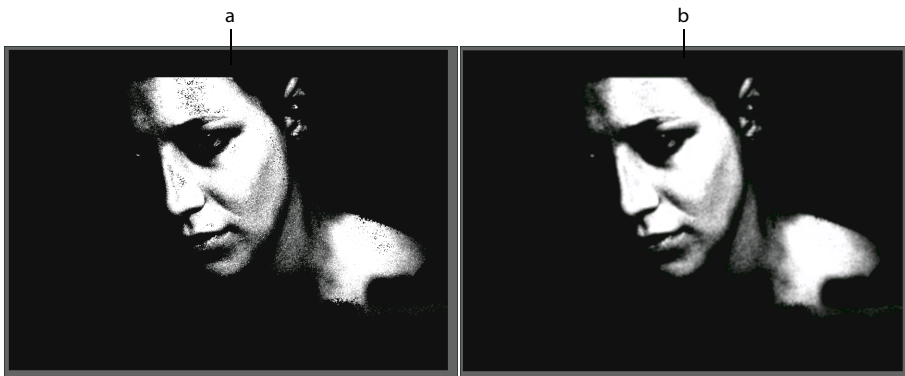


To blur the edge of the key:

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

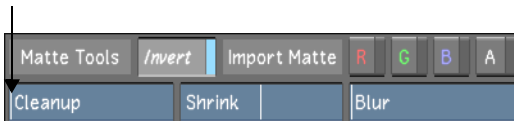
Images courtesy of Hungarian Academy of Film & Theatre, 3rd year

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.

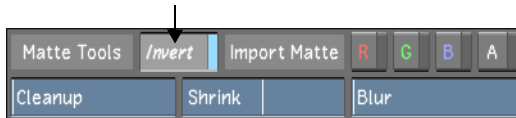


Inverting Keys

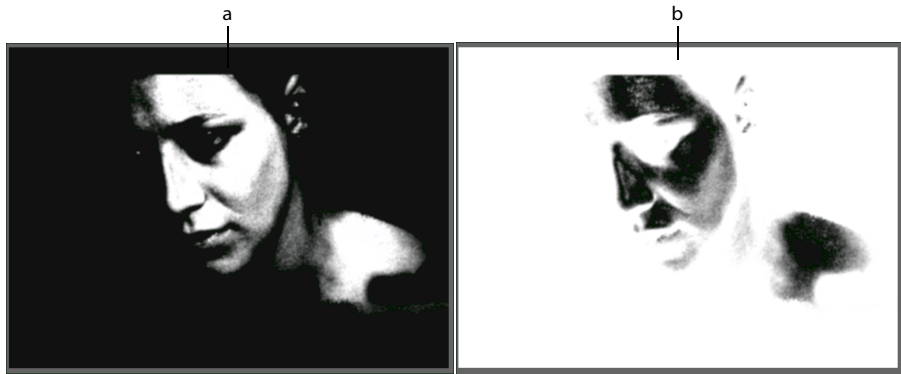
When it is easier to key colours in the image you do not want to include in a secondary than those that you do, you can invert the key.

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

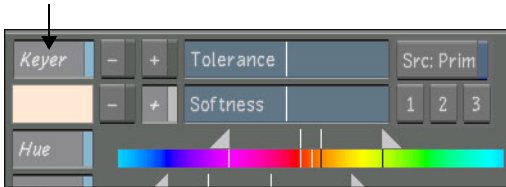
Excluding Keys from Secondaries

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 288.

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*.

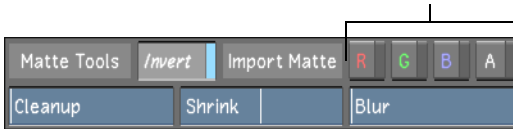
NOTE: The matte file cannot be compressed and must be present at the first image of the file sequence.

To import a matte:

1. In the Storyboard, select a shot with a TIFF or TGA matte.
2. In the Secondaries menu, 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 and Shape Tracker”](#) on page 366.

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.

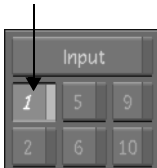
NOTE: Field-based projects viewed on the data monitor will demonstrate a half-pixel downshift. Geometries added to the image will also display this downshift.

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.

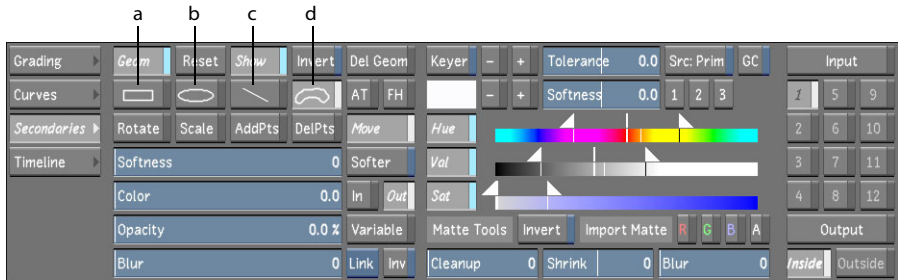


2. Enable Show.



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

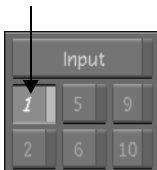
NOTE: The Keyer picker is automatically deactivated when a geometry is being created.

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.

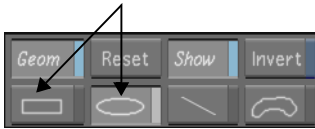


2. Enable Show.



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

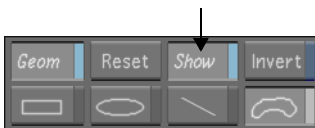
For each secondary, you can draw multiple free-form (Bezier) geometries to isolate parts of the shot or image for secondary colour grading. When you draw a free-form geometry, you can draw point-to-point using the Free-Form (Bezier) button, use Auto-Tangent mode or Freehand mode, or manually add vertices and tangents.

To manually draw a free-form geometry:

1. In the Secondaries menu, click a Secondary button to enable it.



2. Enable Show.



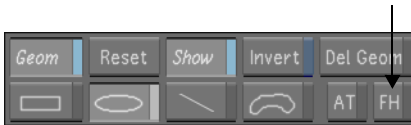
If a geometry exists, it is displayed in the Player.

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.

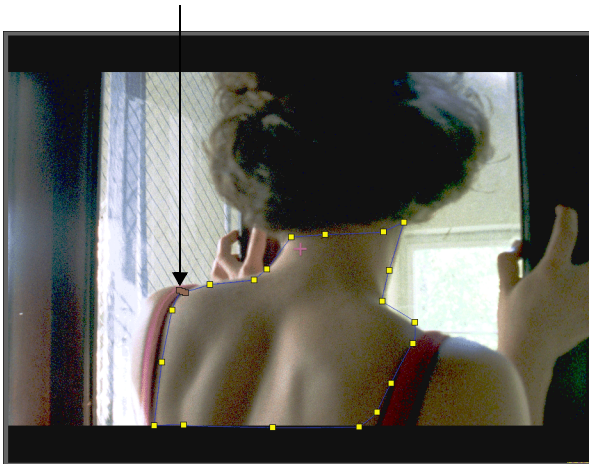


Image courtesy of Hungarian Academy of Film & Theatre, 3rd year

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.

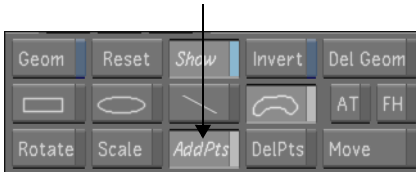
- To modify the shape or curve of a side, move the adjacent tangents. Drag a tangent handle to change its position.

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:

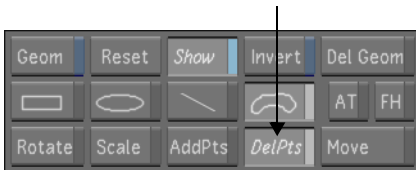
1. If you are working with a basic shape, right-click it in the Player to convert it to a free-form geometry.
2. In the Secondaries menu, enable Add Pts or press **A**.



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:

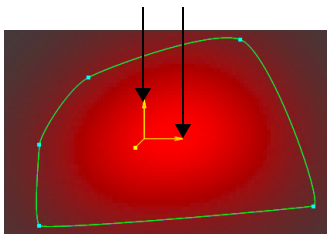
- Move geometries.
- Rotate geometries.
- Scale geometries.

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

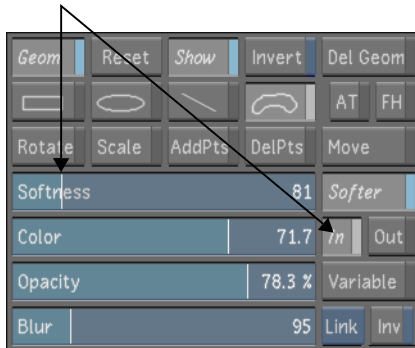
You can apply a constant range of softness to the edge of a geometry on a secondary. You can apply constant softness to the inside or outside of the mask border. You do not need to convert a basic shape to a free-form geometry in order to apply constant softness.

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.



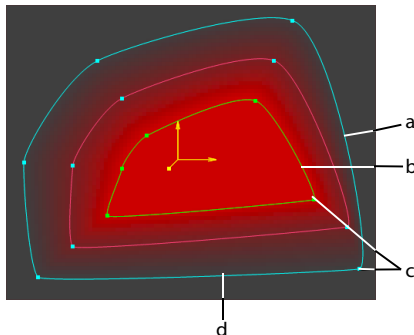
2. In the Secondaries menu, do one of the following:
 - To apply softness to the inside of the mask border, enable In, and then drag the Softness slider. Drag right to increase softness. Drag left to decrease softness.
 - To apply softness to the outside of the mask border, enable Out, and then drag 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.

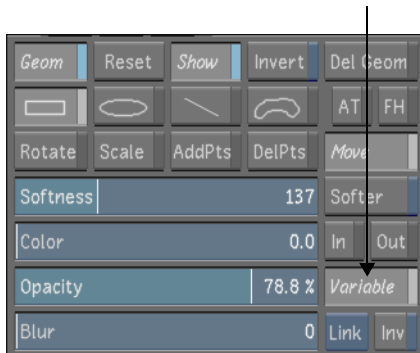
Variable softness can only be applied to free-hand geometries, or to basic geometries that have been converted to a free-form geometry.

To apply variable softness to the edges of the geometry:

1. Do one of the following
 - In the Player, select a geometry.
 - From the Axes list, select a geometry or its axis.

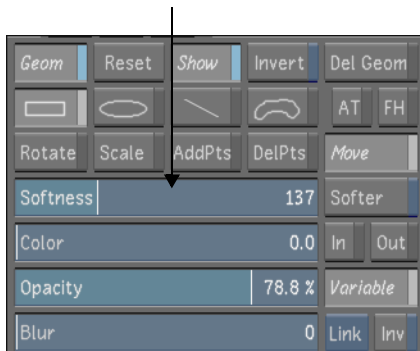


2. In the Secondaries menu, enable Variable.



Softness is applied inside and outside the mask border.

3. Drag the Softness slider. Drag right to increase softness. Drag left to decrease softness.



The softness level set by the Softness slider is applied to the geometry.

4. Drag the vertices and tangents to adjust the shape and vary the softness at the edges of the geometry. See [“Adjusting Tangents and Vertices”](#) on page 350.

Blurring Geometries

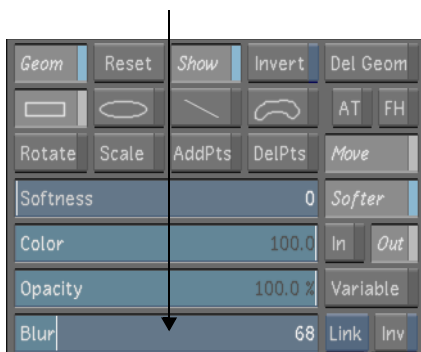
You can apply a Gaussian blur to soften all geometries on the current secondary layer.

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.

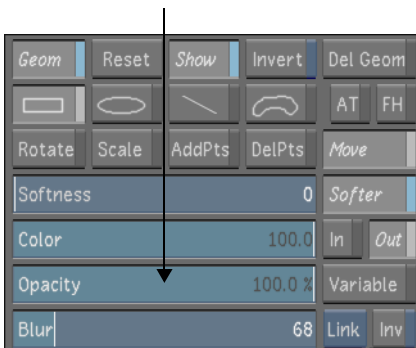
If you are applying only one geometry or several non-overlapping geometries to a secondary, modify either the colour or opacity to adjust the matte—in this case, colour and opacity will have the same effect on the image. If you are applying several overlapping geometries to a secondary, modify both colour 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.

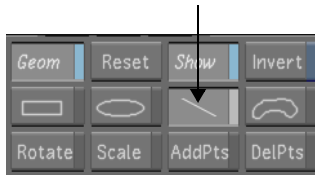
Adding a Split Screen

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.

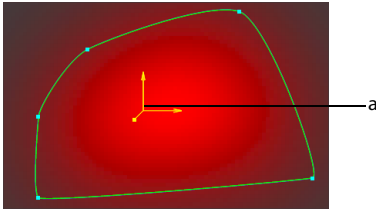


Image courtesy of Technicolor (formerly Toybox)

3. To grade the other half of the split screen, enable Outside (see [“Colour Grading Inside and Outside Secondaries”](#) on page 385).
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.



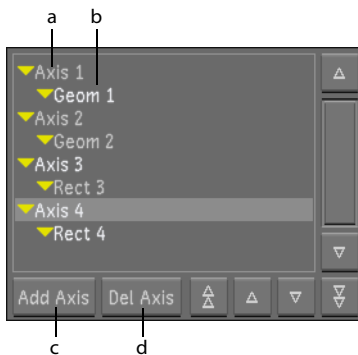
a) Geometry axis

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.

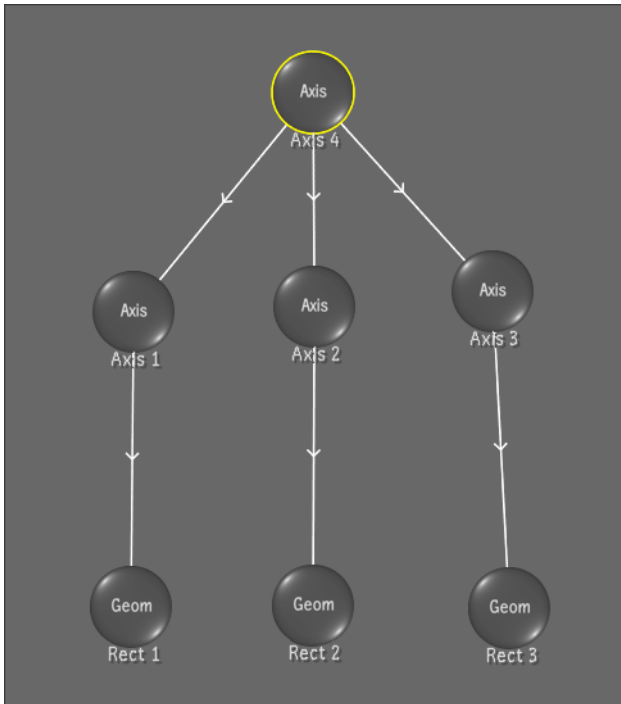
To delete an axis:

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.

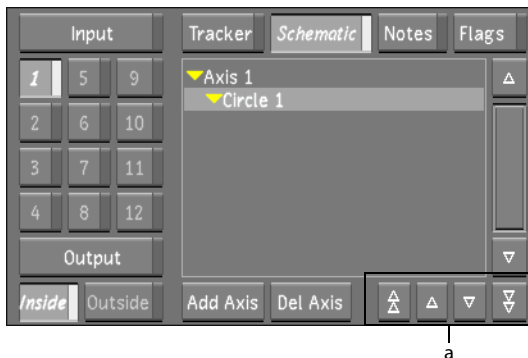
The changed hierarchy is displayed in the schematic.

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.
	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.

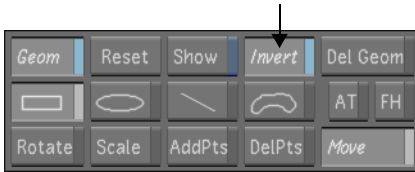


Inverting Geometries

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.

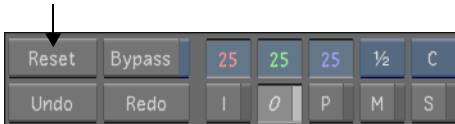
NOTE: You can also delete geometries and their axes from the Axes list. To delete a geometry or axis, select it in the Axis list (or **SHIFT**-click to select several), and then click Del. Axis.

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.

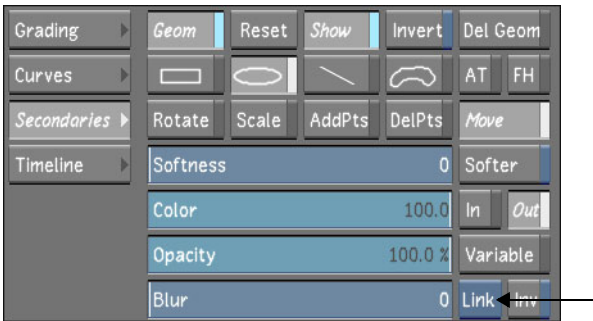
When you use the Link box, a logical operation is applied to determine how the geometries from the previous layer are combined with the key and geometries on the current secondary layer, and what areas of the image are colour graded. Depending on what you want to accomplish with the secondary, you can 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.

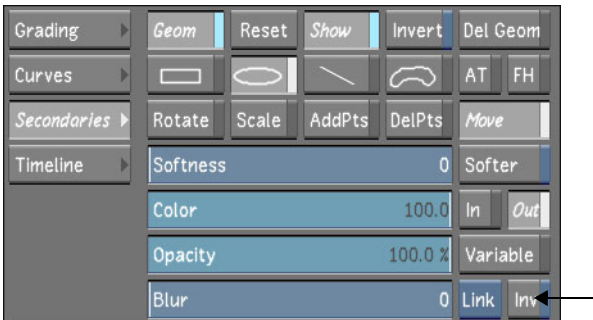
2. Select a logical operation from the Link box. The logical operation you select determines what is colour graded by the current secondary.



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.

Animating Geometries with the Point Tracker and Shape Tracker

Users can animate geometries by using either the Point Tracker or the Shape Tracker tool.

Shape Tracker v.s. Point Tracker

Lustre 2008 introduces the Shape Tracker feature which complements the tracking capabilities of the existing Point Tracker.

Shape Tracker

Shape Tracker allows users to define a Region of Interest (ROI) by drawing a geometry. The ROI inside this geometry becomes the area in which Lustre will search for trackable points: a point cloud. By tracking a cloud instead of a single point, Lustre is able to track shapes through translation, rotation, scaling, and partial occlusions.

The ROI that uses the regular Lustre shape system may be the final graded secondary or else users can hide the shape and parent the resulting axis to a new shape that will appear in the final grade. All Shape Tracker's analysis data is saved in the ROI axis.

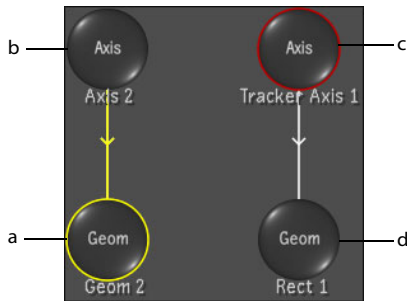
Shape Tracker's Tolerance Slider defines the tracker's tracking tolerance for points in the point cloud. The range is from 0 to 10.

- A low tolerance (e.g.: 0) means that if the shape is not distorting in perspective (in 3 dimensions), and is rigid, the tracker is more able to filter out occlusions successfully. In a lower tolerance setting, a shape distorting in perspective is less tolerated.
- A high tolerance (e.g.: 10) means that the tracker is more likely to allow a shape's distortion in perspective (e.g.: a person's head turning). However, in this setting, the tracker is less able to filter out occlusions.

Shape Tracker allows users to link an ROI resulting axis to another geometry or axis. For example, users can link a small shape they are colour grading to a larger but hidden ROI that moves in parallel to the grading shape. By pressing the tilda key (~) users can access the Schematic view where they can link the axis of an ROI (the Parent Axis) to the axis of a shape to be graded (the Child Axis). For more information about the Schematic view, refer to [“Schematic View”](#) on page 360.

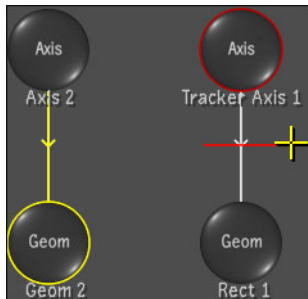
An example of how a user may link Shape Tracker's ROI axis (Parent) to the grading shape's axis (Child) can be seen in the following sequence of diagrams. In the first diagram, the link between

the ROI axis and its own geometry remains unbroken. The same is true for the grading shape's axis and geometry.

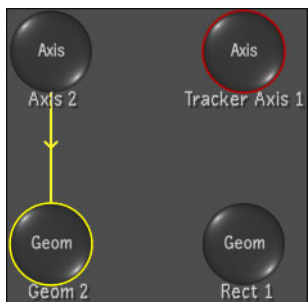


a) Geometry of shape to be graded b) Axis of shape to be graded c) Axis of ROI d) Geometry of ROI

In the second diagram, a user breaks the link between the ROI axis and the ROI geometry by left-click+dragging a line through the link.

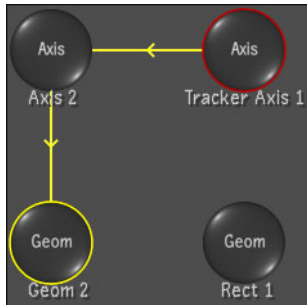


The third diagram shows the ROI axis unattached to its geometry.



In the fourth diagram, a user creates a new link between the ROI axis and the axis of the shape to be graded. This is done by positioning the cursor over the edge of the circle representing the

ROI axis and left-click+dragging a line to the edge of the circle representing the other axis. At this point, the shape to be graded is linked to the ROI.



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.

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

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.

You can also animate the softness of geometries and the position of their individual vertices using the Animation controls. See [Chapter 18, “Animation.”](#) on page 403.

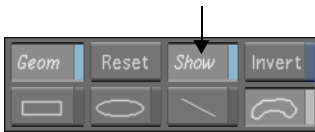
Lustre 2008 further improves the Point Tracker with the introduction of sub-pixel accuracy. Subsequently, any subtle movement (including movements less than a single pixel in dimension) can now be analysed and written to the Lustre Animation Channel.

Animating the Position and Movement of Geometries

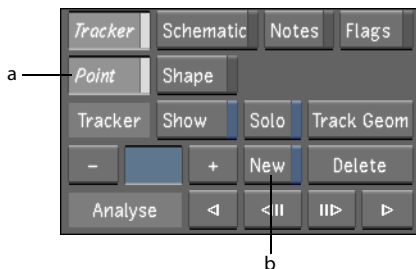
You can animate the position of geometries by defining a Point Tracker and assigning it to the axis of the geometry. Alternatively, you can animate the position, rotation, and scale of a geometry by using the Shape Tracker.

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. Click Point to enable the Point Tracker.

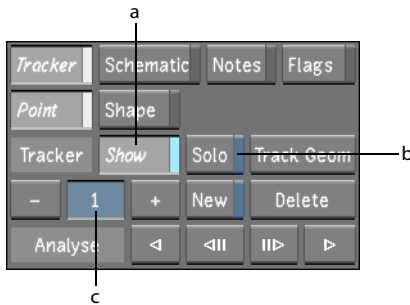


a) Point Tracker button

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.

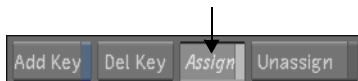
6. Disable the Solo/Gang button to select Solo mode.
In Solo mode, only the current tracker is analysed.
7. Use the Analyse buttons to analyse the shot:
 - To track the tracking point forward in time, click the Analyse Forward buttons.
 - To track the tracking point backward in time, click the Analyse 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.

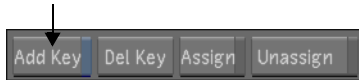
11. In the Player, do any of the following:

- 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.

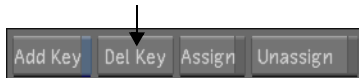
12. Continue using the Analyse buttons to analyse the shot.

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**).

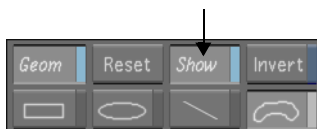


The tracking data is deleted.

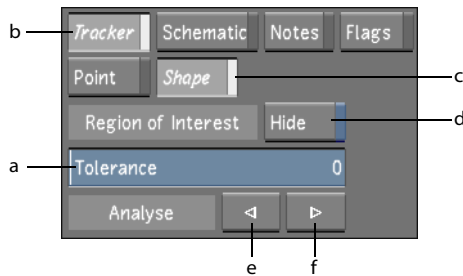
NOTE: Linear interpolation is used to set the tracking position for frames without tracking data. For these frames, the previous defined tracking position and the next defined position are used to interpolate the tracking position.

To animate the movement of a geometry with the Shape Tracker:

1. In the Secondaries menu, enable the Secondary button.
2. Click Show.



3. Select the tracking feature by clicking Tracker.



a) Tolerance slider	d) Hide button
b) Tracker button	e) Analyse backward button
c) Shape button	f) Analyse forward button

4. Click Shape to enable the Shape Tracker.
5. Create a Region of Interest (ROI) by drawing either a basic or free-form geometry. See:
 - [“Drawing Basic Geometries”](#) on page 344
 - [“Drawing Free-Form Geometries”](#) on page 345

NOTE: Make sure your ROI is as large as possible, without including too many non-rigid shapes, in order to give Shape Tracker a sufficiently large area to analyse.

6. Make sure the shot timebar is positioned correctly before you begin analysis. For more information about the shot and scene timebar, see [“Navigating Using the Timebar and Storyboard”](#) on page 139.

NOTE: The shot timebar should be positioned at the beginning of the shot segment you wish to analyse if you plan on analysing forward. The shot timebar should be positioned at the end of the shot segment you wish to analyse if you plan on analysing backward.

7. Select either the shape of the ROI or its axis.
8. Click the Analyse Forward button or Analyse Backward button, as required, to analyse the shot segment.

NOTE: If the analysis stops before the end of the shot or does not begin at all, the ROI may be too small an area for Lustre to see a trackable point cloud. In this case, try making the ROI larger.

NOTE: If the analysis stops before the end of the shot or does not begin at all, the ROI may surround an area whose point cloud is distorting in perspective (3 dimensions) or scaling beyond Lustre’s ability to track. In this case, it may be preferable to employ the Point Tracker on a non-distorting point in the shot.

9. While in the same Secondaries layer as your ROI, create a free-form or automatic tangent shape around an object you wish to grade. See [“Drawing Free-Form Geometries”](#) on page 345.
10. Open the Schematic view by pressing the ~ hotkey.
11. In the Schematic view, left-click+drag a line through the link between the Tracker Axis and the geometry associated with it.
The link between the Tracker axis and its geometry is severed.
12. Left-click+drag a line between the Tracker axis and the axis of the shape you are grading.
A link appears between the two axes.
13. Toggle out of the Schematic view by pressing the ~ hotkey.
14. Left-click+drag the shot timebar to the beginning of the shot segment you are tracking (or the end, if you are tracking in a backwards direction).
15. Click Grading to display grading options.
16. Select the axis of the geometry you wish to grade.
17. Adjust the colour grading of the shape, as required.
18. Play the shot segment to verify the results. See [“Playing Shots”](#) on page 131.

NOTE: If your grading shape is jittery when playing the shot segment, try adjusting the tolerance value of the Shape Tracker, or adjust the shape of the ROI to exclude objects you don't want to track. If both these solutions fail to fix the problem, try enlarging the size of your ROI as much as possible without including objects whose movement you don't want to include.

NOTE: Following interruption or completion of the tracking procedure there is an optimisation step that may take several seconds to complete.

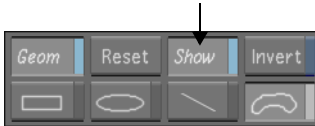
Animating the Shape of Geometries with the Point Tracker

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.

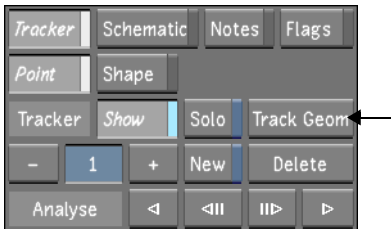
To animate the shape of a geometry:

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.

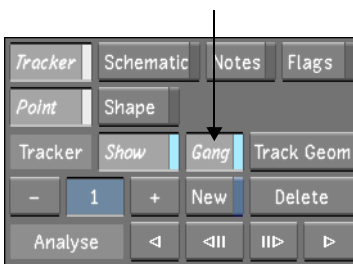


The geometries are displayed in the Player.

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.



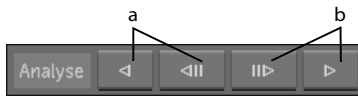
In the Player, a Point Tracker is assigned to, and positioned over, each vertex on the shape. A tracker number is assigned to each tracker. Gang mode is selected and the Point Trackers are turned on.



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 or Shape 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.



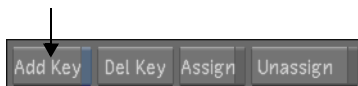
a) Analyse Backward buttons b) Analyse Forward buttons

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.

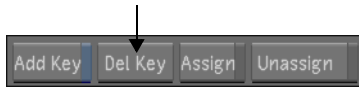
HINT: You can reposition the Point Tracker and reference point when the reference point moves off-screen.

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 373.

To assign a Point Tracker to a geometry:

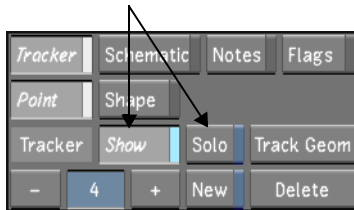
1. In the Secondaries menu, enable the Secondary button that contains the geometry you want to track.
2. Enable Show.



The geometry is displayed in the Player.

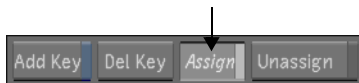
3. Do one of the following:

- 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:

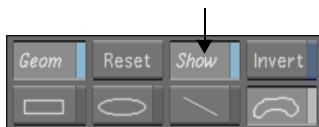
- To animate the entire geometry without changing its shape, click the geometry's axis.
- To animate a single vertex in a geometry, click the vertex.

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.

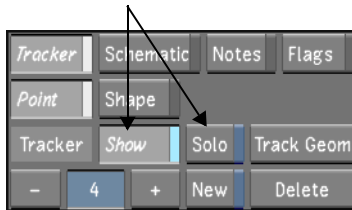
To unassign a Point Tracker from a geometry:

1. Select the shot that contains the tracking data.
2. In the Secondaries menu, enable Show.



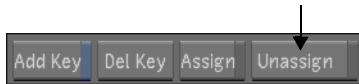
The geometry is displayed in the Player.

- 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.

- Enable Unassign and then click the vertex or axis to which you assigned the tracker.



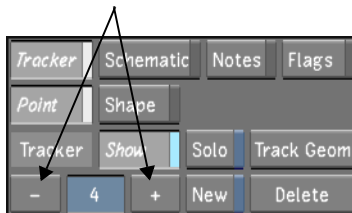
The Point Tracker is unassigned to the vertex but remains in the Tracker list.

Soloing and Ganging Point Trackers

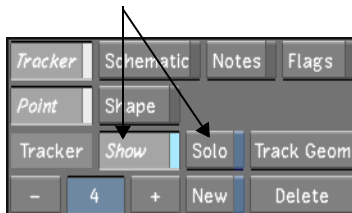
You can solo or gang Point Trackers. Solo a Point Tracker when you want to view and generate tracking data for the current tracker only. Gang Point Trackers when you want to view and generate tracking data for multiple Point Trackers at the same time.

To solo a Point Tracker:

- Click the + or - button to select a Point Tracker from the Tracker list.



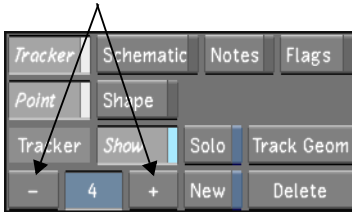
- Make sure the Point Tracker is set to On and is in Solo mode.



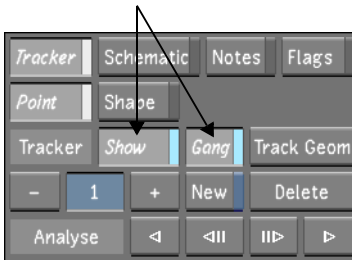
Only the current Point Tracker is displayed in the Player.

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.

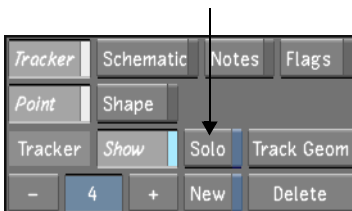
The ganged trackers that are set to On are displayed in the Player and can be analysed as a group.

Deleting Point Trackers

You can delete Point Trackers from a shot.

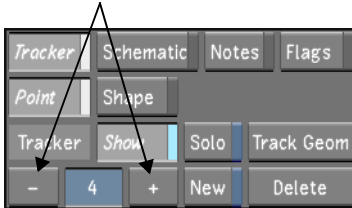
To delete a Point Tracker:

1. Select the shot that contains the tracking data.
2. In the Secondaries menu, select Solo mode.



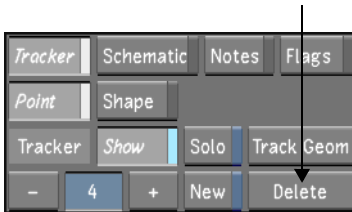
When you use Solo mode, only the current Point Tracker is displayed in the Player. If the tracker does not appear, set the tracker to On.

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.

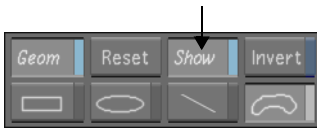
Loading Tracker and Stabilizer Data from Autodesk Applications

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<version number>\settings\geompresets` folder (Windows), or in the `/usr/Autodesk/lustre<version number>/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.

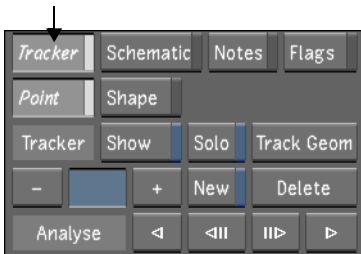
To load tracker or stabilizer data:

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<version number>\settings\geompresets* folder.
 - If using the Linux version of Lustre, store the ASCII file in the */usr/Autodesk/lustre<version number>/settings/geompresets* directory.
3. In the Secondaries menu, enable the Secondary button that contains the geometry you want to track.
4. Enable Show.

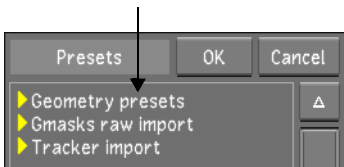


The geometries are displayed in the Player.

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.

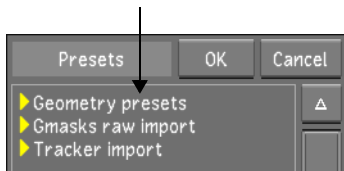
Loading Garbage Mask Setups from Autodesk Applications

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<version number>\settings\geompresets* folder.
 - If using the Linux version of Lustre, store the raw setup file in the */usr/Autodesk/lustre<version number>/settings/geompresets* directory.
3. Right-click or middle-click the Secondaries menu to display the Presets list.



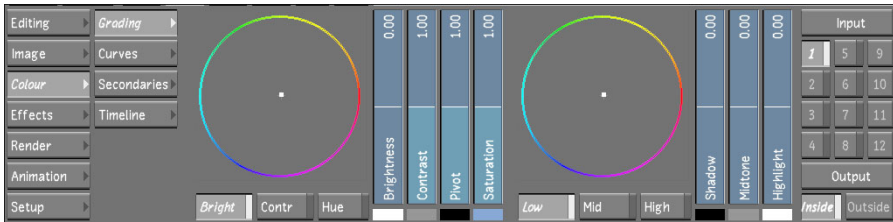
4. Expand GMask's Raw Import, and then select the raw setup file.
5. Click OK to load the garbage mask setup.

The geometry appears in the Player.

NOTE: All garbage mask setups are imported at a 2K resolution.

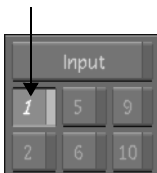
Colour Grading Secondaries

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:

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.
5. Perform colour grading as you would for a primary grade. See [Chapter 14, “Primary Colour Grading,”](#) on page 295.

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's Guide*.

Shifting Hue

You can shift the hue of the area defined by a secondary using the Hue Rotator.

To shift the hue:

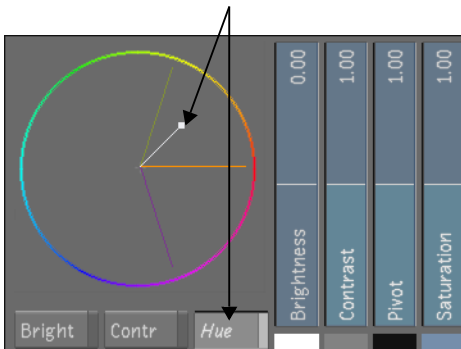
1. In the Secondaries menu, click a Secondary button to enable it.



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.
5. Click Hue and then drag the Hue Rotator to shift the hue. To shift the hue clockwise through the hue spectrum, drag right. To shift the hue counter-clockwise, drag left.

**Adjusting Printer Lights for Secondary Grading**

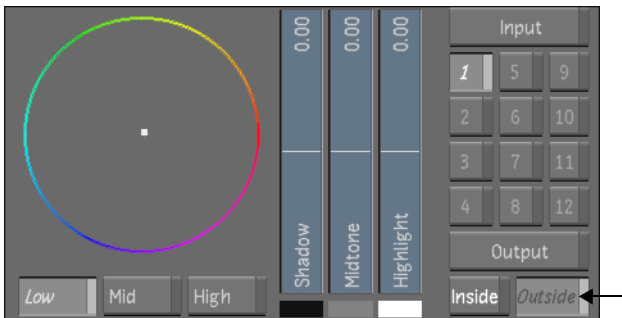
You can make printer light adjustments to secondaries just as you would to primary colour grades. See [“Adjusting Printer Lights for Primary Grading”](#) on page 298.

Colour Grading Inside and Outside Secondaries

Secondaries can have unique colour grades applied both inside and outside a defined shape. The secondary's default state is the inside.

To grade a secondary and its surrounding area:

1. Colour grade the secondary.
2. 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.

Creating Lustre Sparks Effects

Summary

<i>About Lustre Sparks Plugins</i>	387
<i>Installing Lustre Sparks Plugins</i>	388
<i>Licensing Lustre Sparks Plugins</i>	388
<i>Creating an Effect with a Lustre Sparks Plugin</i>	388
<i>Lustre Sparks Plugin Descriptions</i>	391
<i>Disabling and Re-enabling Lustre Sparks Plugins</i>	400
<i>Developing Lustre Sparks Plugins</i>	401

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<version number>\plugins* folder. If running the Linux version of Lustre, save your plugins in the */usr/Autodesk/lustre<version number>/plugins* directory.

Removing a Lustre Sparks plugin consists of deleting it from the plugins directory.

For details on developing Lustre Sparks plugins, see [“Developing Lustre Sparks Plugins”](#) on page 401.

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<version number>\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 (see [“Related Documentation”](#) on page 2).

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.

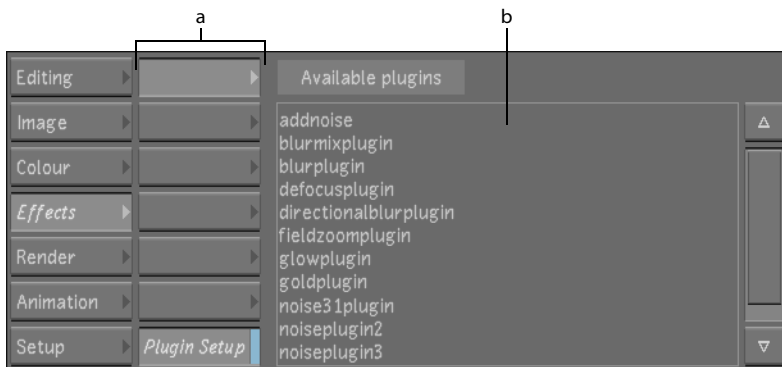
Each plugin has various parameters that you set to create the desired effect. Apply the effect to the whole shot, or an area of a shot defined by a secondary. You can animate the effect to make

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

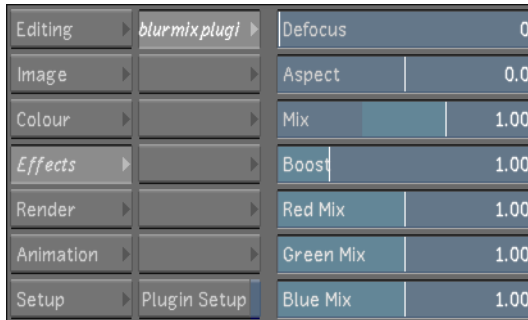
b) Available plugins

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.

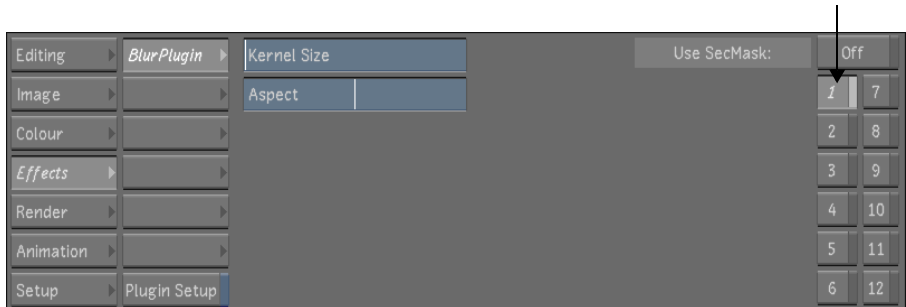


The parameters are applied to the entire shot and the image is dynamically updated to reflect the changes.

- 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.

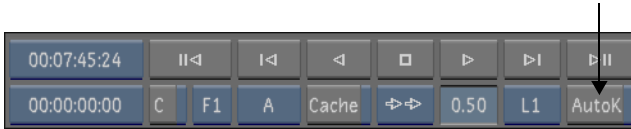


- Create the effect as described in the previous procedure.
The plugin is applied to the area defined in the secondary.

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.

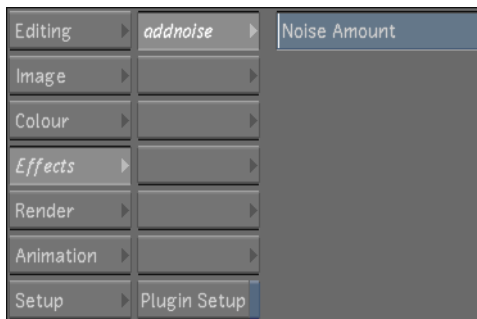


- Set one or more of the parameters.
As soon as you change a parameter, a blue line in the Shot timebar appears indicating a keyframe.
- Move the positioner to the next frame where you want to set another keyframe and adjust one or more of the parameters. Continue to add keyframes as needed.

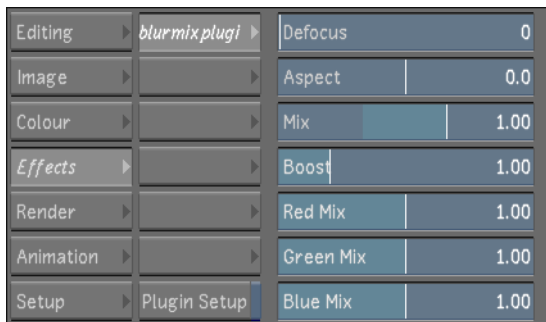
Lustre Sparks Plugin Descriptions

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.

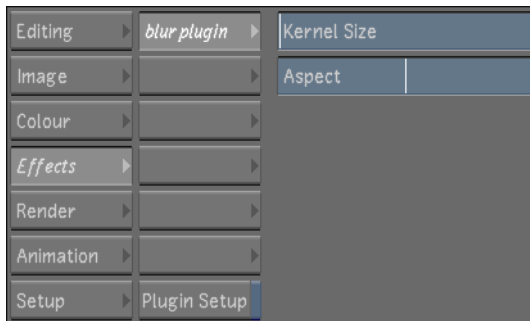


Blur Mix — Blur-type plugin that performs a multiply mix between the original image with the blurred image. You can also use this plugin to sharpen or tint an image.



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.

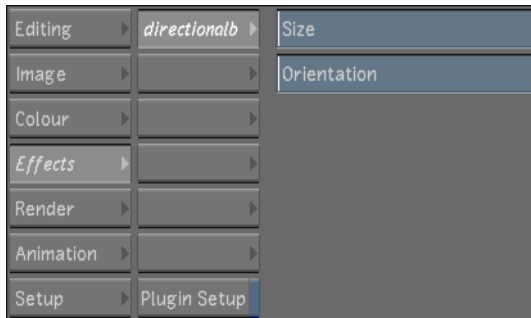
Defocus — Blur-type plugin that simulates a camera out-of-focus effect. Along with the blur parameters, this plugin includes variable iris types and additional highlight controls, as well as a control for softening the edge of a secondary.



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.

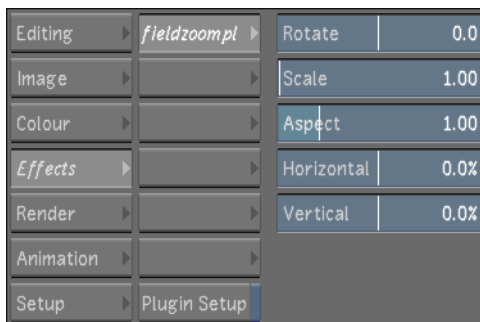
Use:	To:
Circle, Hexagon, Octagon	Specify the shape of the plugin matrix (or camera iris) used to calculate how the defocus is applied.
Rotation	Rotate the plugin matrix (or iris).

Directional Blur — Simple directional blur along a specified direction.



Use:	To:
Size	Adjust the amount of blur.
Orientation	Change the direction of the blur.

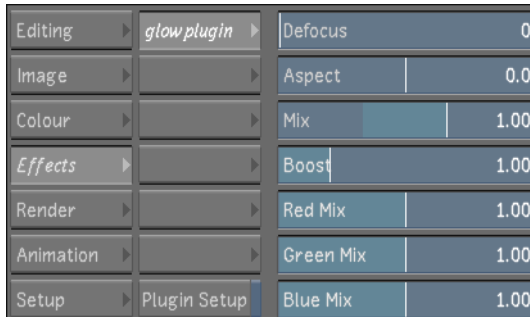
Field Zoom — Pan, scan, scale, and rotate interlaced footage. The parameters in this plugin work in the same way as the tools in the Image | Reposition menu, except that they are applied to individual fields instead of the entire frame. This yields a smooth image from interlaced footage even after repositioning.



Use:	To:
Rotate	Rotate the fields.
Scale	Resize the fields.
Aspect	Adjust the aspect ratio of the fields.

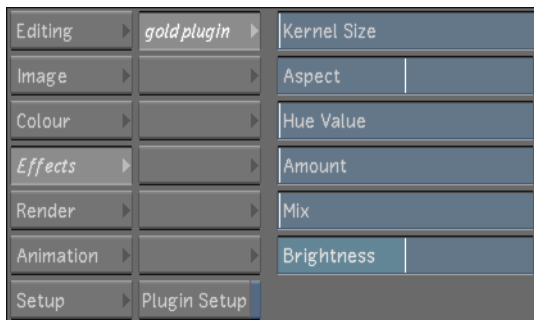
Use:	To:
Horizontal	Adjust the horizontal position of the fields.
Vertical	Adjust the vertical position of the fields.

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.

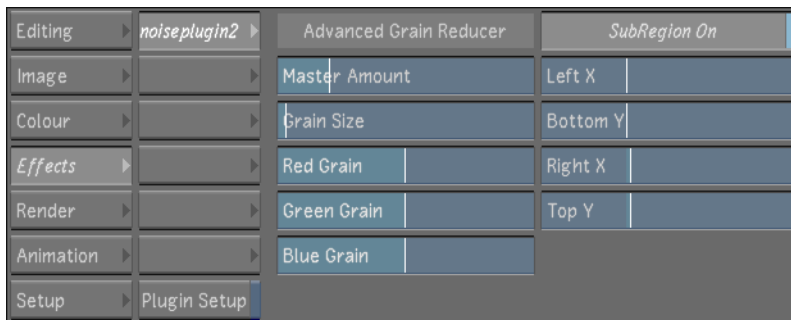
If the plugin processing slows down image playback, you can restrict grain reduction to a subregion of the frame.

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.

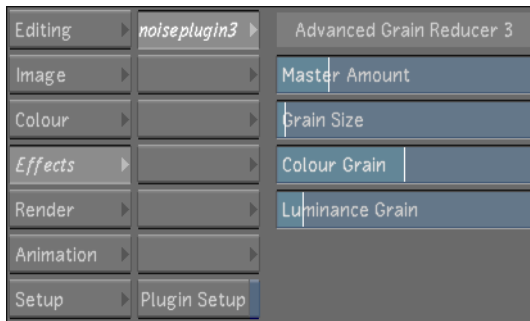
Noise Plugin 2 — This grain-type plugin is a second generation advanced noise reducer that uses a spatial filter with detail preservation control. It is useful for grain, noise, or detail reduction. During playback, this plugin updates dynamically as you adjust the values, so you can see how the grain 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.



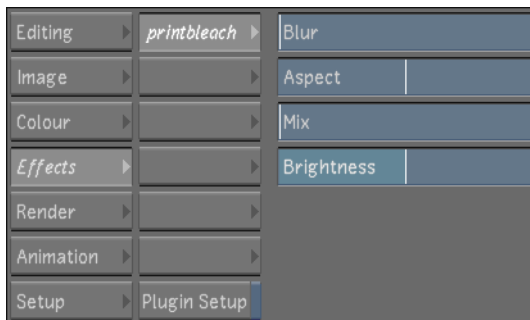
Use:	To:
Master Amount	Set how much the grain is reduced. The default value (or less) generally produces a good result for grain removal.
Grain Size	Set the size of grain that is reduced.
Red Grain	Independently adjust grain removal on the red colour channel.
Green Grain	Independently adjust grain removal 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.

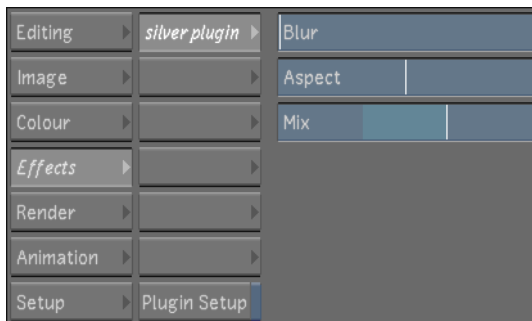
Print Bleach — Blur-type plugin that creates an effect similar to the chemical print bleach process. This plugin increases contrast for the shadows while desaturating the image. When you use this plugin, you first set a mix value so that you mix the image onto itself, and then you blur and brighten the 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.

18

Animation

Summary

About Animation	403
Accessing the Channel Editor	404
Animating with Keyframes	408
Editing Keyframes	418
The Track Editor	428
Animating in Other Menus	432
Specifying Setup Options	434
Undoing and Redoing Operations	436

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

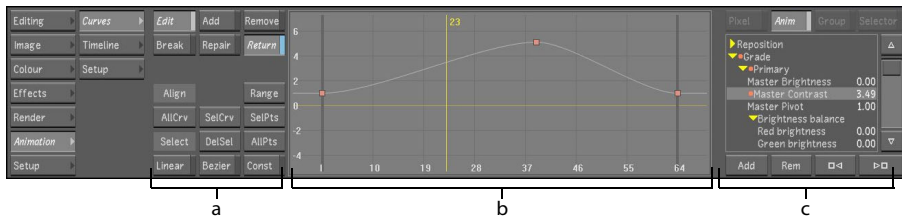
Use the animation curves in the Channel Editor to create animations.

For control surface mappings, see the *Autodesk Control Surface User's 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

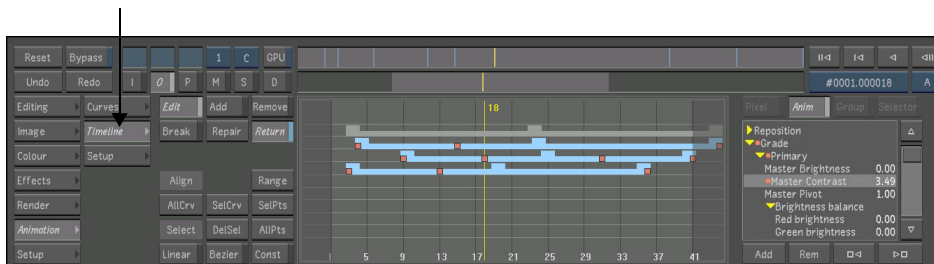
The Channel Editor is made up of the following elements.

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.

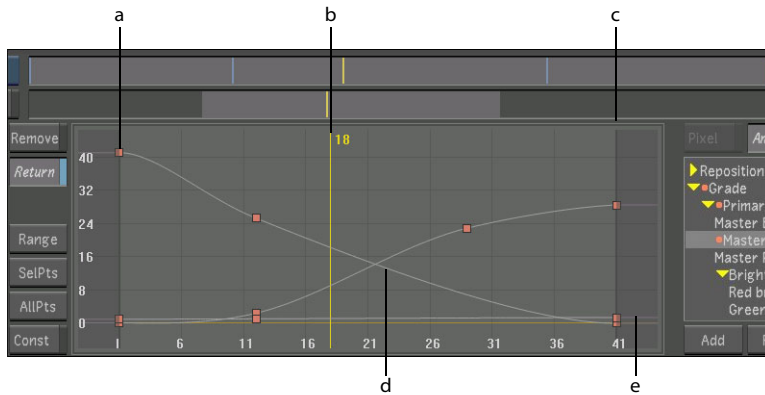
Channel hierarchy — Use to select the directories and channels to animate. You can also refer to the channel hierarchy to quickly see what channels contain keyframes.

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.

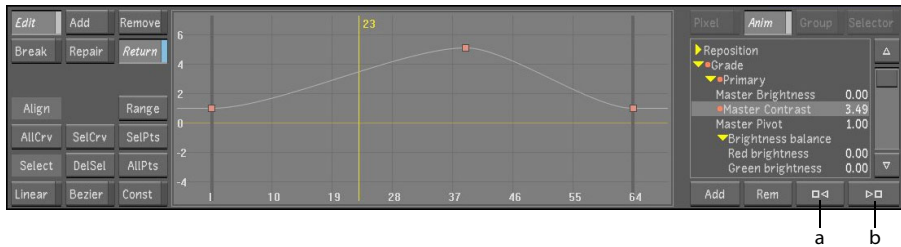
To move to any frame in the Animation Curves window:

- 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.

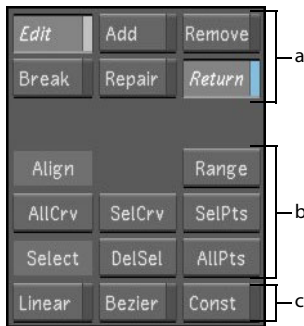
To zoom the Animation Curves window view:

- 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:

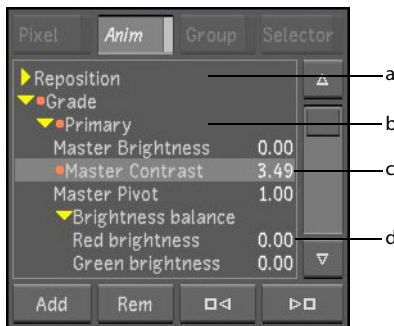
1. In the Animation Curves window, select the keyframes. See [“Selecting Keyframes”](#) on page 419.
2. Click SelPts.

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 directory, all its channels are selected and all its curves appear in the Animation Curves window. You can also expand a directory and select a single channel.



a) Collapsed directory	c) Selected channel
b) Expanded directory	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 channel directory that you want to animate.
2. Move the cursor to the frame at which you want to add a keyframe and then insert a keyframe.
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.
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 428.

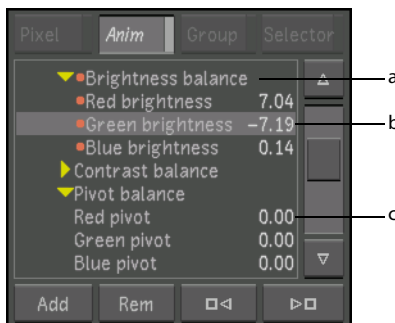
Selecting Channels

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 directory and click a channel to select it. To select all channels in the directory, select the directory.

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 directory 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.

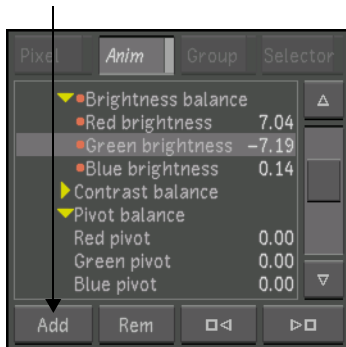
Adding Keyframes

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:

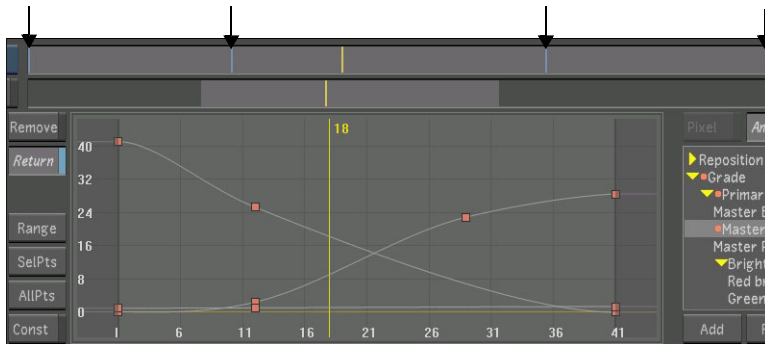
1. In the channel hierarchy, select the channel to which you want to add a keyframe.
2. Position the cursor at the frame at which you want to add a keyframe.
3. Click Add.



When you add a keyframe to a channel, an orange indicator appears to the left of the channel in the channel hierarchy.

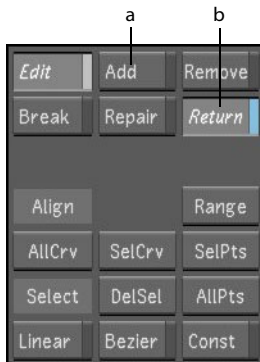
4. Set the channel value as needed by dragging the keyframe.

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.

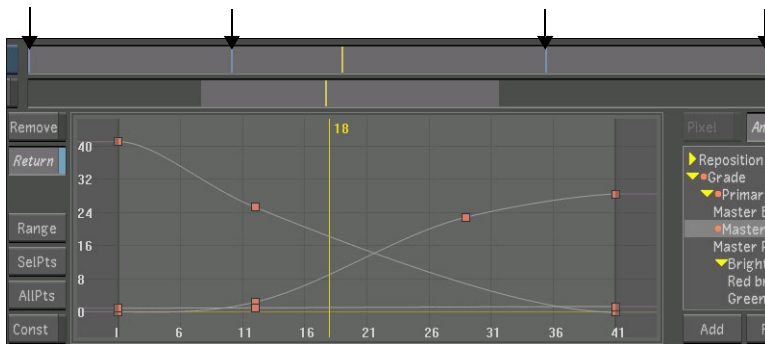
NOTE: If Return mode is enabled, you are returned to Edit mode after adding one keyframe. For information on working in Edit mode, see ["Editing Keyframes"](#) on page 418.

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.

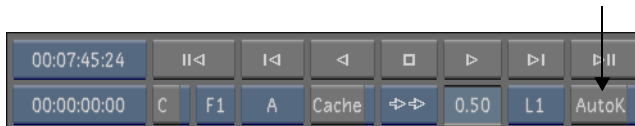
NOTE: The new keyframe is automatically set to Bezier interpolation.

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:

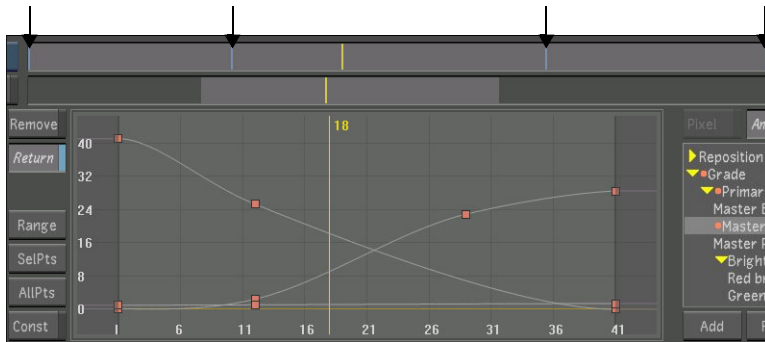
1. Enable AutoK.



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.

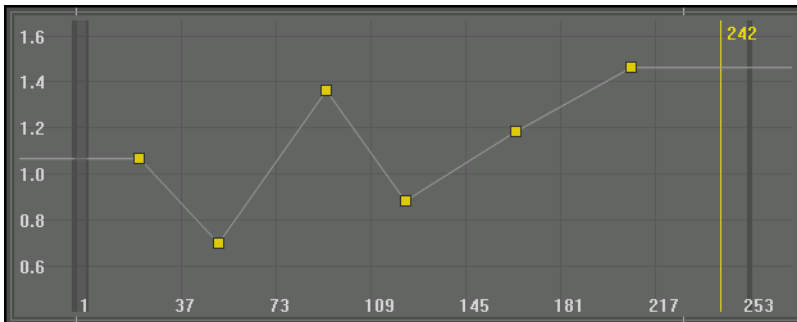
Once you have added keyframes to a shot, markers appears in the Shot timebar indicating the location of the keyframes.



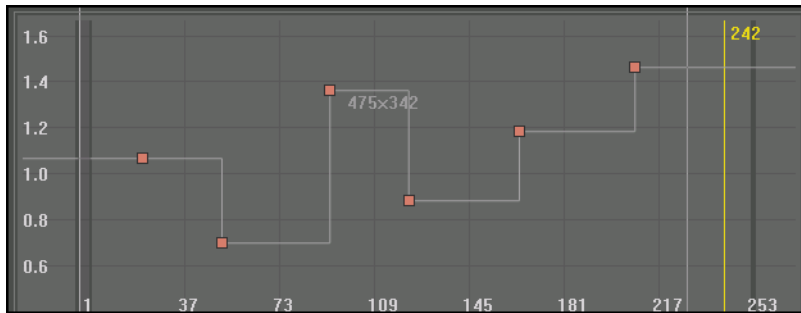
Setting Interpolation

Interpolation defines the shape of an animation curve between keyframes. You can choose from the following interpolation settings depending on how you want the channel values to change over time.

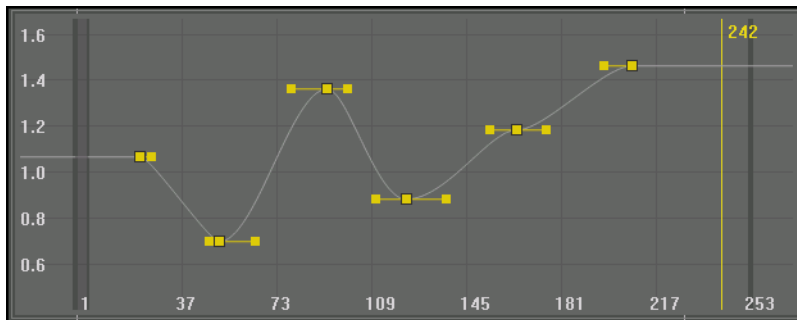
Linear — Joins keyframes with a straight line.



Constant — Produces a square curve. The value of one keyframe is held at a constant value until the next keyframe. This setting can produce abrupt changes between keyframes.



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:

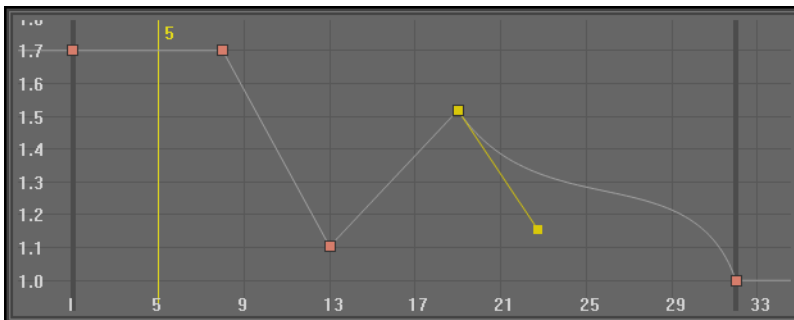
1. Select the keyframe for which you want to set the interpolation.
2. In the Animation controls, click the appropriate interpolation type button.



To create an animation curve with mixed interpolation:

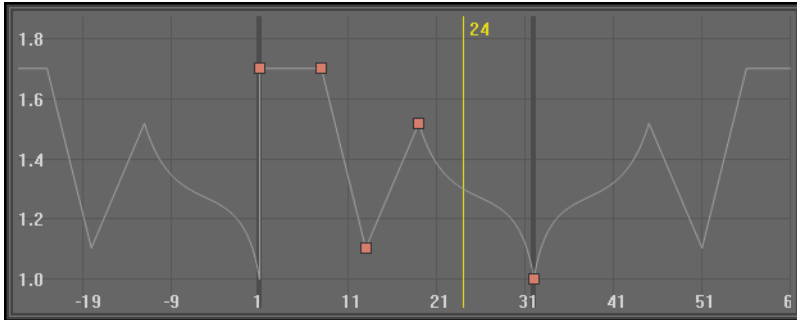
1. Create an animation with at least four keyframes. Do not worry about the interpolation setting at this point.
2. Click Animation, and then click Curves to display the Channel Editor.
3. Make sure you are in Edit mode and then select the first keyframe. See [“Editing Keyframes”](#) on page 418.
4. 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.



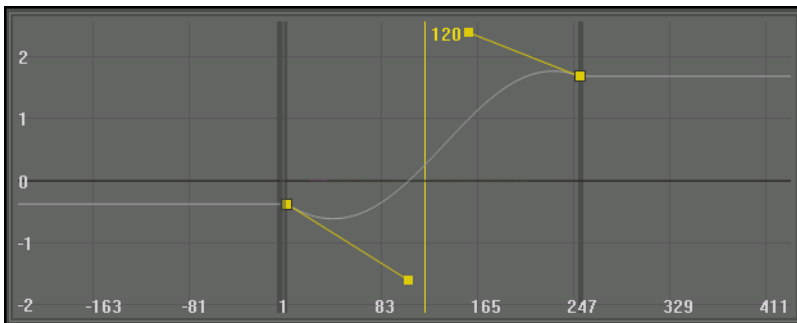
Setting Extrapolation

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.

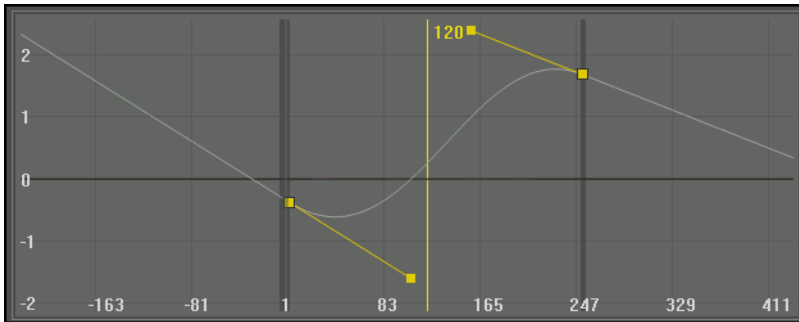


You can specify how keyframes are extrapolated using the following settings.

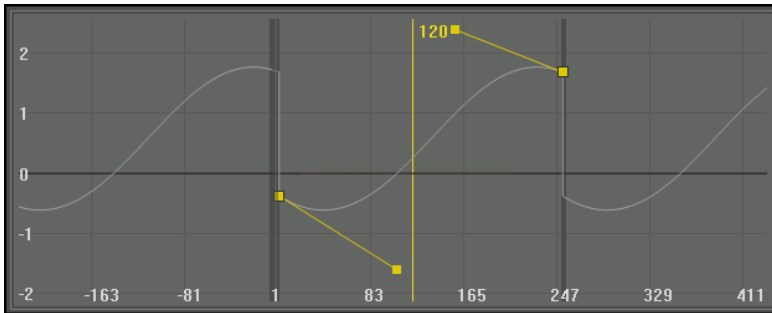
Constant — Creates an extrapolated animation curve that stays at the same value as the first or last keyframe.



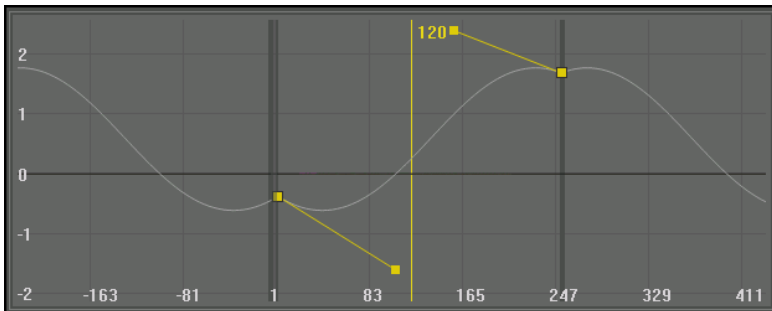
Linear — Creates an animation curve that changes in a linear fashion before the first or after the last keyframe.



Repeat — Creates a mirror image of the animation curve before the first or after the last keyframe.



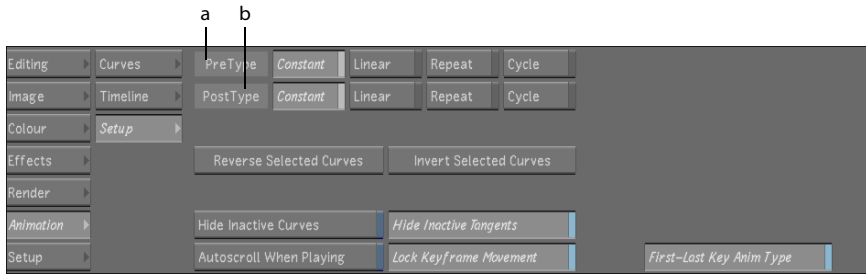
Cycle — Creates a pattern of repeating keyframes before the first of after the last keyframe.



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.

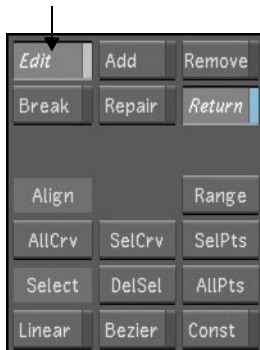
Editing Keyframes

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:

- Click the keyframe.
The keyframe is selected.

To select multiple keyframes:

- Click a keyframe to select it and then hold down **CTRL** and click additional keyframes to add them to 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. 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 controls, click AllPts.

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.

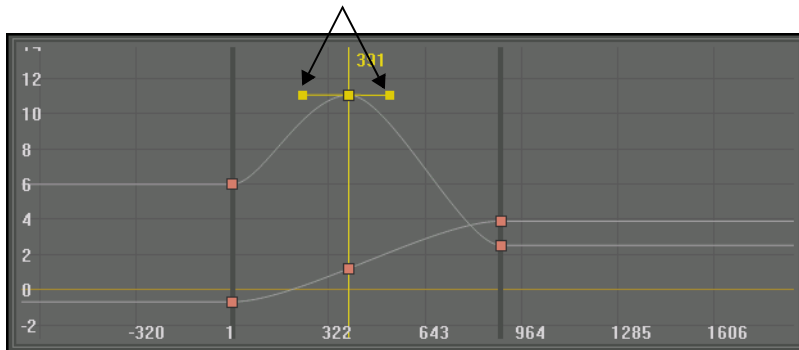
You can hide curves that are not selected. This can make it easier to manage your keyframes and modify animations.

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.



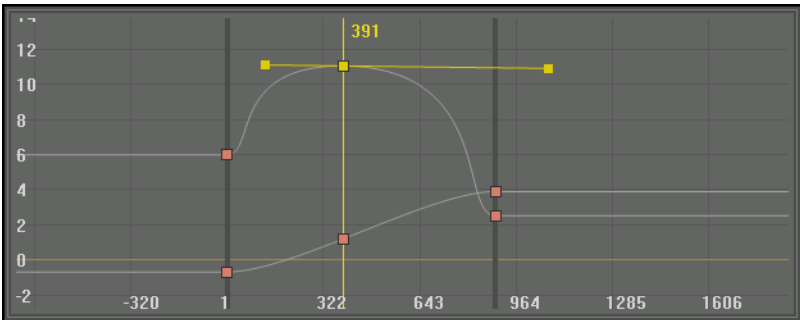
4. Drag one of the tangent handles.

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 422.

- 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.

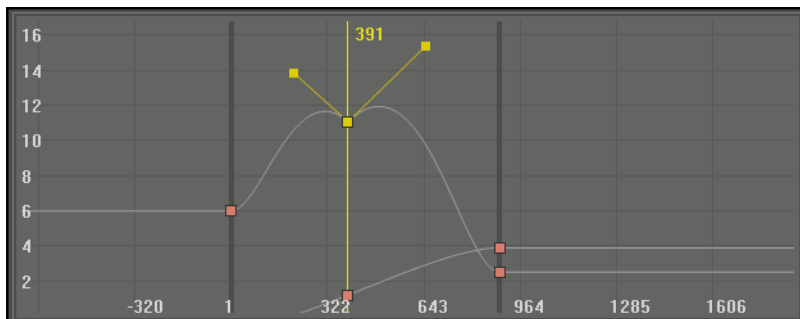
To ensure consistent keyframe functionality between the control surface and the user interface:

1. Click Animation, and then click Setup to display the Setup controls.
2. Enable First-Last Key Anim Type.



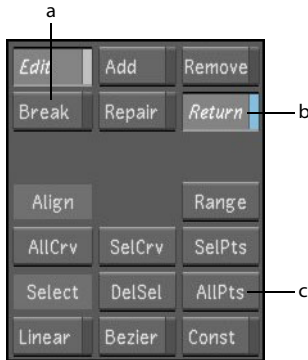
Breaking Tangent Handles

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.



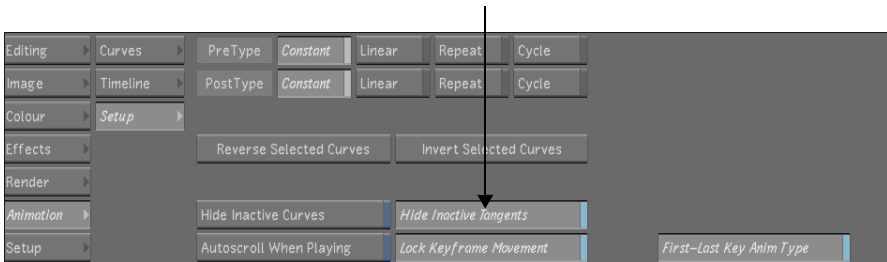
All broken tangent handles for the selected keyframes are restored to an unbroken state.

Hiding Tangent Handles

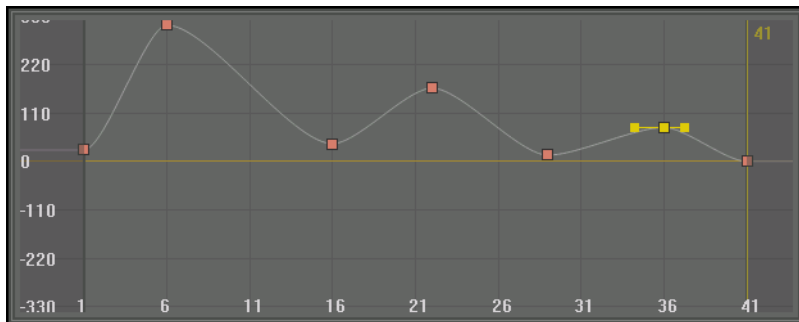
You can hide tangent handles for keyframes that are not selected.

To hide inactive tangent handles:

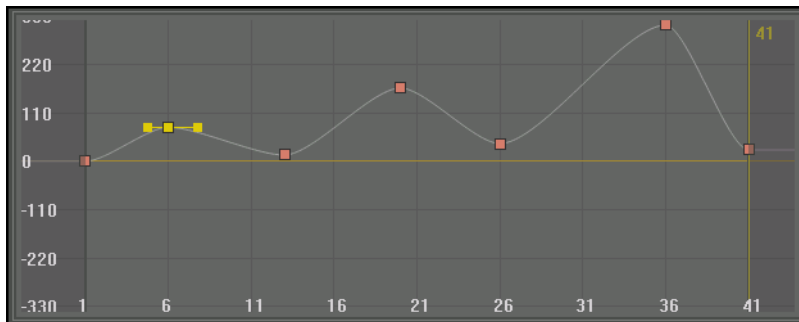
1. Click Animation, and then click Setup to display the Setup controls.
2. Enable Hide Inactive Tangents.

**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.



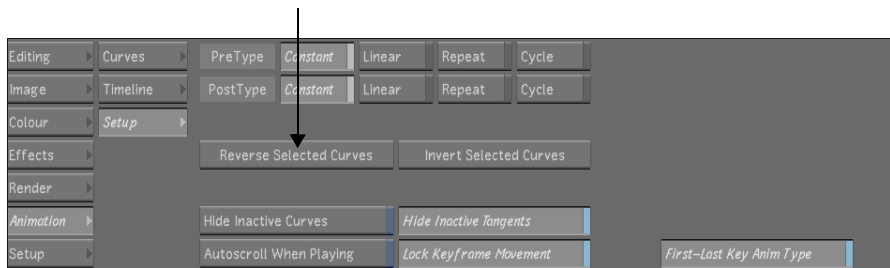
Before reversing the animation curve



After reversing the animation curve

To reverse keyframes for a channel:

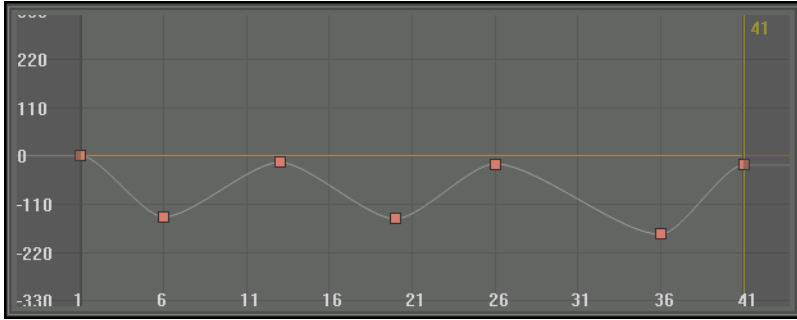
1. Click Animation, and then click Setup.
2. In the channel hierarchy, select the channel you want to reverse.
3. Click Reverse Selected Curves.



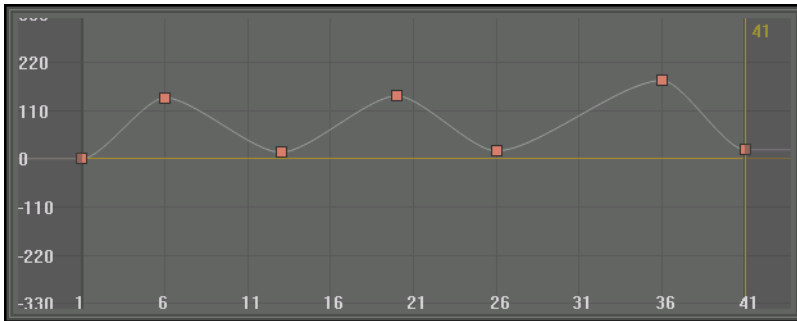
The selected channel's keyframes are reversed.

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.



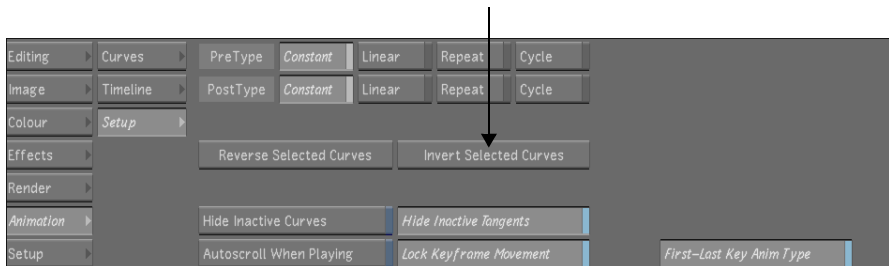
Before inverting the animation curve



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:

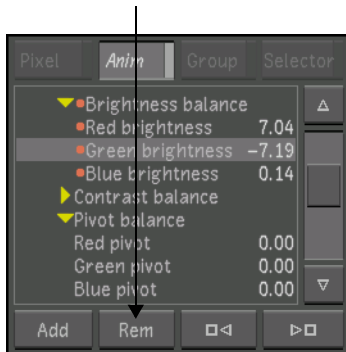
- Delete a single keyframe.
- Delete a selection of keyframes.
- Delete keyframes interactively, enabling you to click directly on keyframes in the Animation Curves window to delete them.

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.

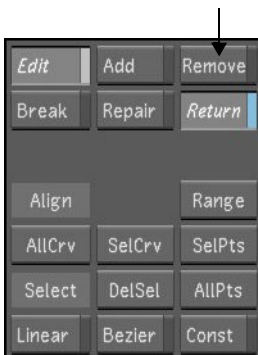
3. Click DelSel and then confirm the deletion. You can also press **DELETE**.



The selected keyframes are deleted.

To delete multiple keyframes interactively:

1. In the Animation controls, enable Remove.



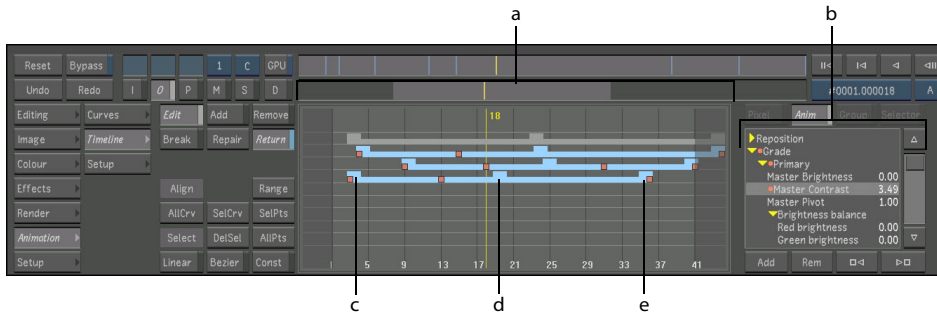
2. Disable Return if you want to delete multiple keyframes.
If Return mode is enabled, you are returned to Edit mode after deleting one keyframe.
3. In the Animation Curves window, click a keyframe to delete it.
4. You can click to delete as many keyframes as needed.

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
- b) Channel hierarchy
- c) First keyframe in channel
- d) A selected channel
- e) Last keyframe in channel

The position and length of a track is determined by the first and last keyframes of the channel it represents. The track extends from the frame number of the first keyframe to the frame number of the last keyframe.

Light blue tracks represent selected channels. Dark blue tracks are for deselected channels. Grey tracks represent 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.

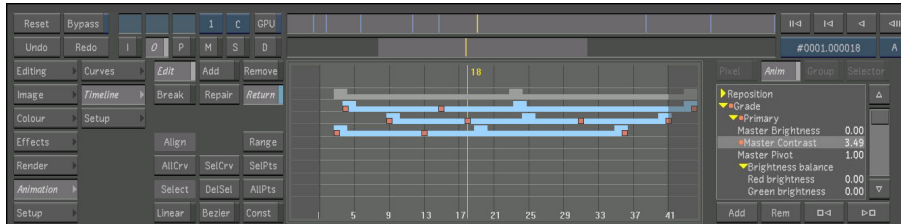
Accessing the Track Editor

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 directory (to select all channels), or expand a directory 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:

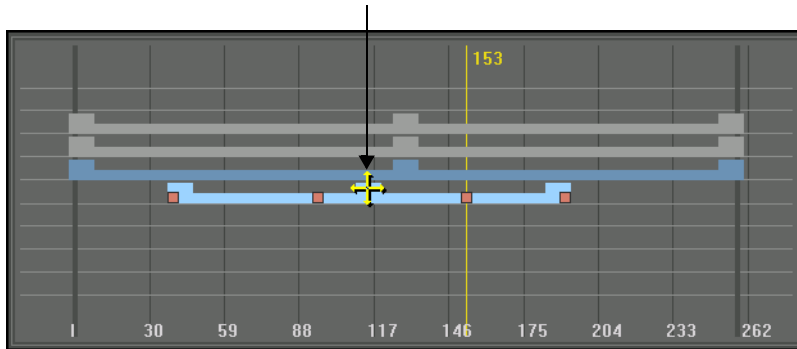
1. Click Animation, and then click Timeline.

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 directory changes the timing for all channels in that directory.

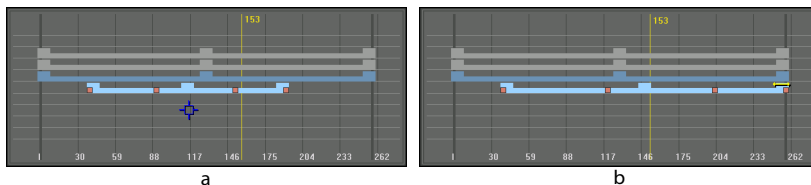
To adjust keyframe timing:

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.

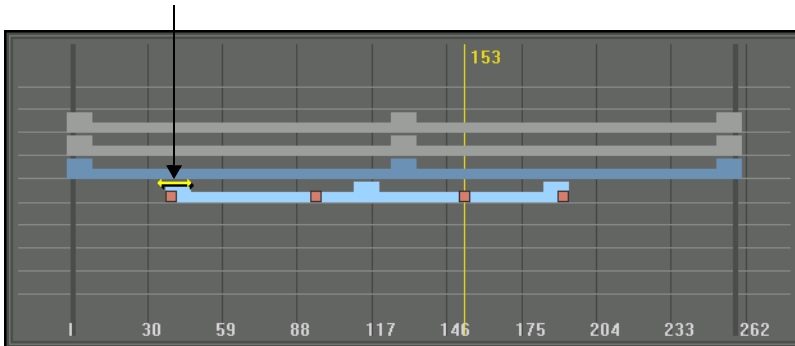
To adjust an animated channel's start or end point:

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 directory changes the timing for all channels in that directory.

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.

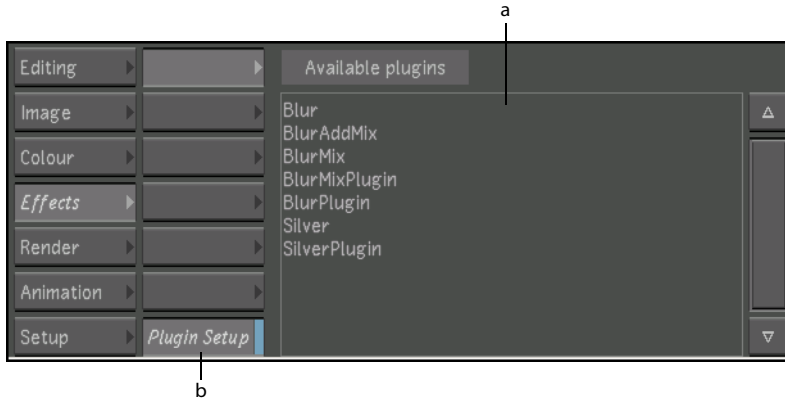
NOTE: Adjusting the timing for a directory changes the timing for all channels in that directory.

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.

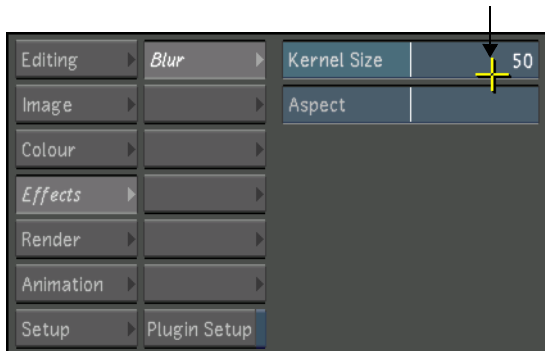


a) Available plugins list

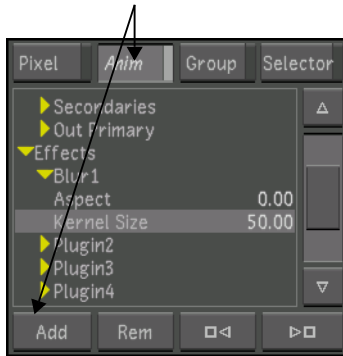
b) Plugin Setup button

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 419.

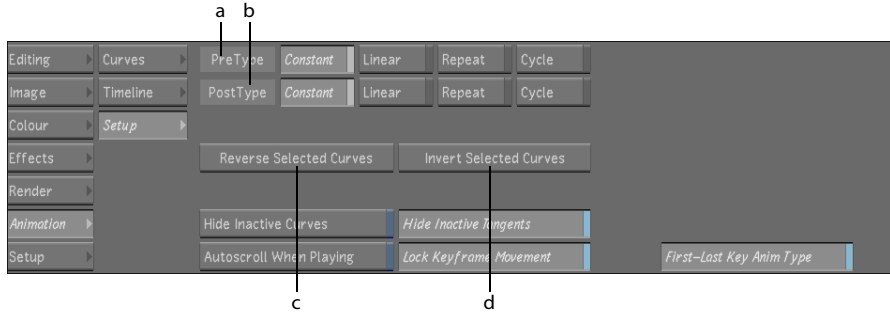
Specifying Setup Options

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
- b) PostType extrapolation options
- c) Reverse Selected Curves button
- d) Invert Selected Curves button

Setting Extrapolation Method

Use the buttons under PreType and PostType to set keyframe extrapolation. See [“Setting Extrapolation”](#) on page 416.

Reversing and Inverting Keyframes

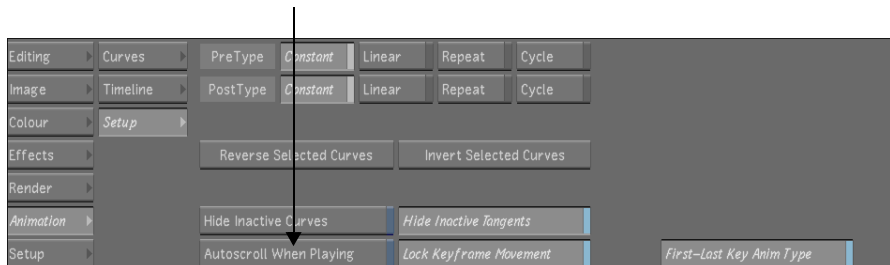
Use the Reverse Selected Curves and Invert Selected Curves options to modify your keyframes for selected curves. See [“Editing Keyframes”](#) on page 418.

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.

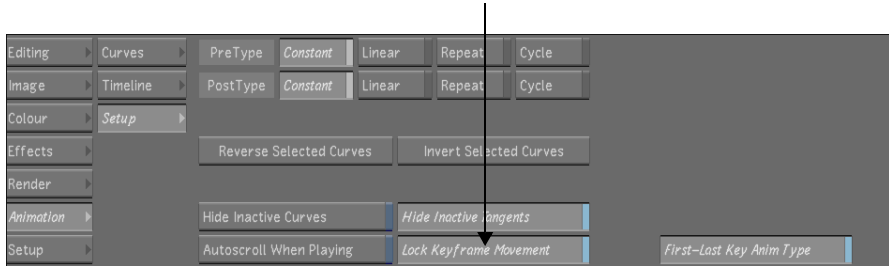


Locking Keyframe Movement

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

19

Rendering

Summary

About Rendering in Lustre	437
Rendering Shots	438
Saving and Loading Colour Mapping Presets	460
Generating and Viewing Proxies	462
Locking Colour Grading	464
Cleaning Up Render Files	465
Rendering Shots as You Work	466
Rendering with Burn for Lustre	468

About Rendering in Lustre

Render individual shots, layer(s) in 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 134. 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 150.

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 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 dot in the upper-left corner of the screen.

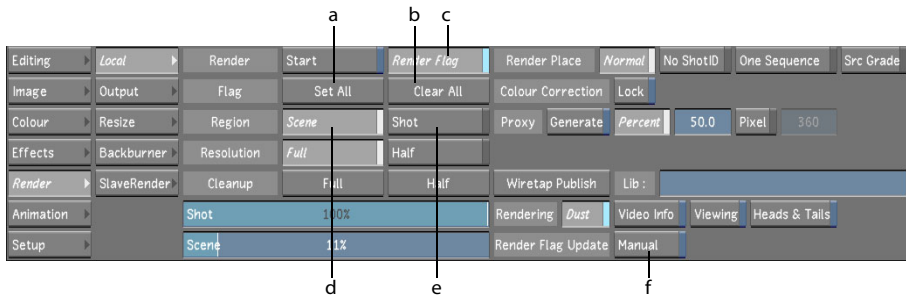


To flag shots, use the Render Local menu.

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's 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.

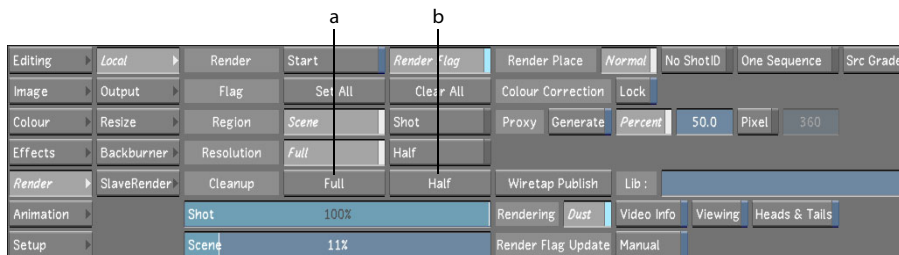
NOTE: In both cases, rendering starts from the current frame and continues forward.

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 441.

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.

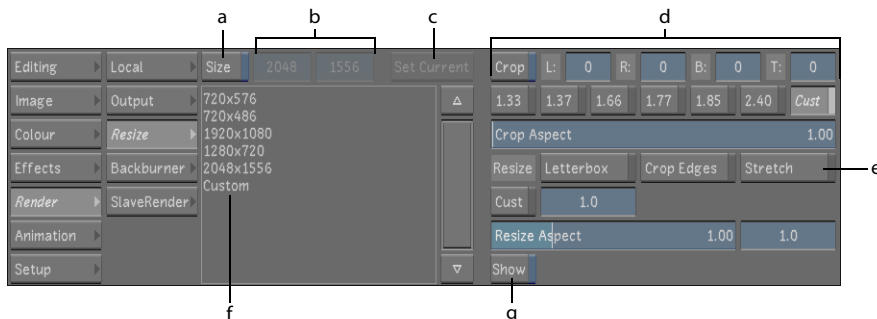
When you resize shots using the Resize menu, all shots in the timeline are resized. To resize individual shots, use the tools in the Reposition menu. See [“Repositioning an Image”](#) on page 255.

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 106.
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.

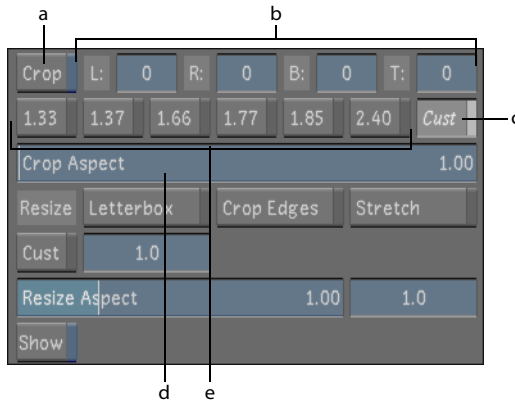
NOTE: You can check the resolution of the original scans in the Shot bin, accessible from the Editing Browse menu. Source resolution is important for quality resizing. For example, you obtain a better quality HD output with 2K full scan sources than with 1K half scan sources.

- 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.

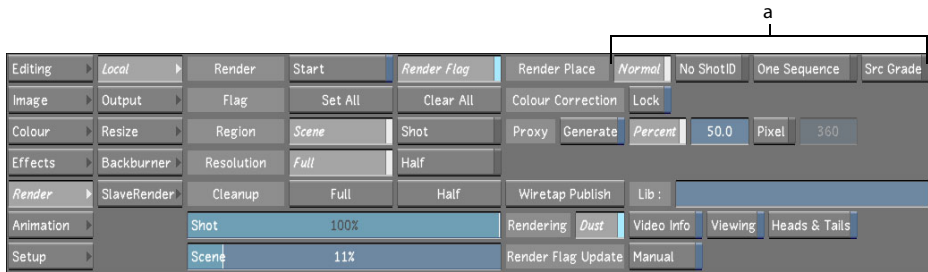
Specifying the Destination for Local Render Files

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 65.

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 248.

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

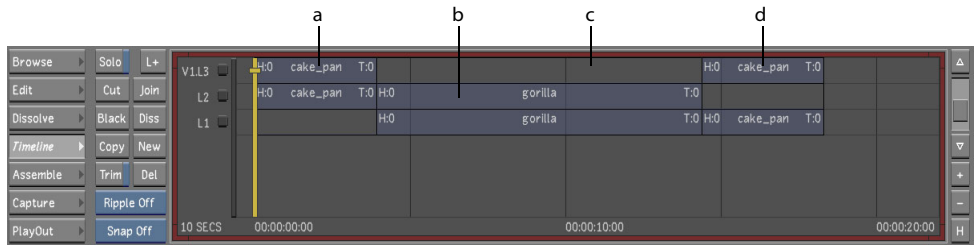
Lustre will render out what is displayed in the Timeline and Player, according to the Solo or Mute status in the Timeline menu.

If Solo is off (this is the default setting), Lustre will render the top-most layer by default. If there are gaps, Lustre will take into account the flattening rule 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 black.

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 will automatically be rendered out as black frames.

Below is an example of what your multi-layer Timeline would look like if you have a grade (grd01) made up of three layers of graded sequences that contain some of the same shots with some gaps.



- a) Shot A c) Gap
b) Shot B d) Shot C

NOTE: The Solo or Mute status is not saved to the grade.

Rendering the Multi-Layer Timeline with Normal

When the Normal button is selected and Solo/Mute is off, Lustre interprets gaps as transparent and flattens layers in a top-down manner. Lustre renders out what is visible in the Timeline (that is, L3-Shot A, L2-Shot B, and L3-Shot C). 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 black to a separate folder in the Renders Home location.

When the Normal button is selected and Solo or Mute is on, users can render one specified layer at a time by soloing a desired layer or muting unwanted layers. 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 (formerly the No Wedge button) is selected and Solo or Mute is off, by default, Lustre sees gaps as transparent and flattens layers in a top-down manner. Lustre renders out what is visible in the Timeline: L3-Shot A, L2-Shot B, and L3-Shot C. 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 as black to a separate folder in the Renders Home location.

When the No ShotID button is selected and Solo or Mute is on, users 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, users 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.

HINT: 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 and Solo or Mute is off, 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 Timeline: L3-Shot A, L2-Shot B, and L3-Shot C. 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 gaps will be rendered as black to the same single sequence in the same scene folder.

When the One Sequence button is selected and Solo or Mute is on, users 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, users can render one layer only for each grade. That is, if a user renders out L1 first, and then within the same grade renders out L2, the L2 render will overwrite the previous L1 render files.

HINT: 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 grd 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 will bypass any dissolve transitions and retime parameters. In other words, dissolves will become hard cuts and retimes will be rendered with a default of 100% value.

Rendering to a Stone Filesystem with Wiretap

When rendering clips to a Stone filesystem, a *scene_grade* reel will be created in the Stone library that was defined when the project was created. See [Chapter 5, “Project Management,”](#) on page 59.

This is the syntax to use when rendering to the Stone filesystem, or publishing soft-import links to your graded sources:

```
<IP address or hostname>@wt:/stonefs/<project_name>/<library>
```

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 Stone filesystem using the Render Place controls in the Render Local menu.

Rendering to the Stone Filesystem with Normal

For shots rendered to a Stone filesystem, a *scene_grade* reel is generated in the Stone library that was defined when the project was created. This is where the rendered material is placed.

For example, if `10.145.43.31@wt:/stonefs/tutor/lustregrades` was entered in the Renders Full Home field, the graded rendered shots are placed in `10.145.43.31@wt:/stonefs/tutor/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.

Rendering to the Stone Filesystem with No ShotID

As outlined in the Normal mode description above, when rendering clips to a Stone filesystem, a *scene_grade* reel is created in the Stone library and all rendered material is placed there. The additional clip with the *_edl* suffix is also generated.

Rendering to the Stone Filesystem with One Sequence

As with Normal and No ShotID mode, when rendering clips to a Stone filesystem, a *scene_grade* reel is created in the Stone 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 Stonefs will be called:

```
/stonefs/HD_project/rendered_grades/interior_003_grd/
```

And, the rendered one sequence clip will contain the grade name: *grd14*

Rendering to the Stone Filesystem with Src Grade

Src Grade mode renders the complete timeline back to the Stone filesystem with Lustre grading applied, but bypasses the dissolves and retimes. In other words, dissolves will become hard cuts and retimes will be rendered with a default of 100% value.

For shots rendered to a Stone filesystem, a *scene_grade* reel is generated in the Stone library that was defined when the project was created. This is where the rendered material is placed.

For example, if *10.145.43.31@wt:/stonefs/tutor/lustregrades* was entered in the Renders Full Home field, the graded rendered shots are placed in *10.145.43.31@wt:/stonefs/tutor/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 a Local or Networked Storage (non-Stone Filesystem)

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 filesystem, or soft-imported into the Stone filesystem, and is based according to your project directory structure. See [“Wiretap-Specific Project Management Configuration”](#) on page 70.

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 organised:

- 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 443.

Rendering Wiretap Clips with Normal

Stone-based 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 Editing or FX 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
```


Rendering Wiretap Clips with No ShotID

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
```

Rendering Wiretap Clips with One Sequence

As with Normal and No ShotID renders, Stone-based clips are rendered to sub-directories of a directory based on the scene name, which Lustre creates automatically when you first render shots. From there, under the sub-directory called *grd*, the shots are rendered as a single sequence to a directory based on the *grd* name. The filenames consist of the scene name, the grade name, and a sequential number. There are no UIDs in the directory naming scheme.

Soft-imported clips are rendered to sub-directories based on the scene name, which Lustre creates automatically when you first render shots. From there, under the sub-directory called *grd*, the shots are rendered as a single sequence to the sub-directory based on the *grd* name. The filenames consist of the scene name, the grade name, and a sequential number. There are no UIDs in the directory naming scheme.

For example:

Stone-based clips are rendered as:

```
<HOME>/<scene>/grd/<grdXX>/<resolution>/<scene_grdXX.xxxx>.dpx
```

Soft-imported clips are rendered as:

```
<HOME>/<scene>/grd/<grdXX>/<resolution>/<scene_grdXX.xxxx>.dpx
```

Rendering Wiretap Clips with Src Grade

As with Normal and No ShotID 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. 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
```

Creating Soft-Imported Clips on the Stone Filesystem

You can create soft-imported clips from rendered material and write the generated timeline back to the Stone filesystem by way of Wiretap. The rendered results reside in the destination specified in the project settings, and a soft-import link is created on the Stone filesystem.

To create a soft-imported clip on the Stone filesystem:

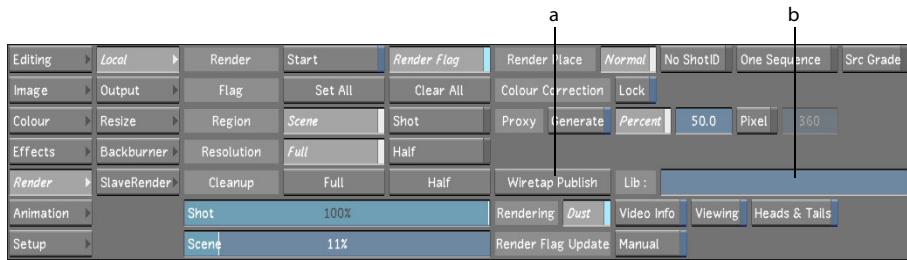
1. Click Setup and then click Project to display the Project menu.
2. In the Renders Full Home field, enter the path for the graded files to be rendered to.
For example, if your Windows drive is mapped to the network storage as S:\, then the renders full home path should be S:/SAN/*lustre_renders*.

NOTE: The shared storage network (i.e., SAN or NAS) must be mounted and accessible from both the Wiretap Linux/Iris host and the Lustre workstation.

If using soft-imported clips, see [“Wiretap-Specific Project Management Configuration”](#) on page 70.

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. Click Start.
This renders the files to the specific path of the shared network storage.
6. Enter the path for the location of the soft-imported clip in the Lib field.
For example, *10.145.43.31@wt:/stonefs/project/library*.



- a) Wiretap Publish button b) Lib (library) field

7. Click Wiretap Publish.
A new scene grade reel containing the soft-imported clips and/or timeline is created in the location specified by the Lib field.

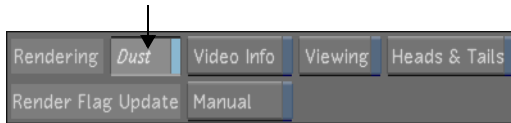
Rendering Dust Metadata

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 248.

To render shots with dust metadata, you must first use the dust removal tools to remove dust artefacts. See [Chapter 10. “Removing Dust.”](#) on page 237.

To render shots with or without dust metadata:

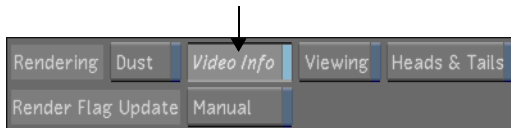
- In the Render Local menu, do one of the following:
 - To render shots without dust metadata, disable Dust.
 - To render shots with dust metadata, enable Dust.

**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.

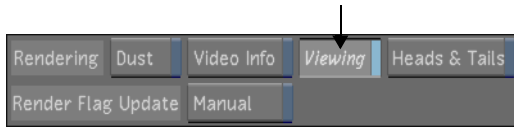
When this option is disabled, the currently applied viewing LUT is not taken into account in the output image, and the files are rendered to the file format specified in the Render Output menu rather than to 8-bit BMP files.

For information on monitor calibration, see [“Calibrating a Monitor”](#) on page 51. For information on output colour space settings, see [“Setting Colour Space Options”](#) on page 456.

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 456.

To set rendering with the viewing LUT:

- In the Render Local menu, enable Viewing.

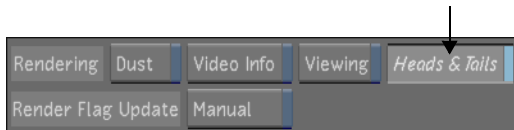
**Rendering Head and Tail Frames**

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**

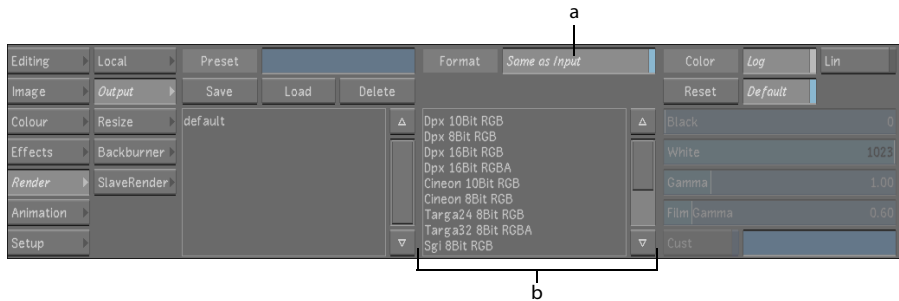
You can render your shots to the same file format as the original footage or select a different one. By default, shots are rendered to the same format as the original footage. Currently supported formats are DPX and Cineon at 10 and 8 bits per channel, TIFF at 8 and 16 bits per channel, and TARGA®, SGI, 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 460.

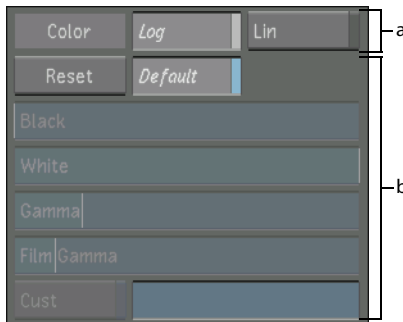
- Select an output LUT from a list of those made available in Lustre.

- 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 454.

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.

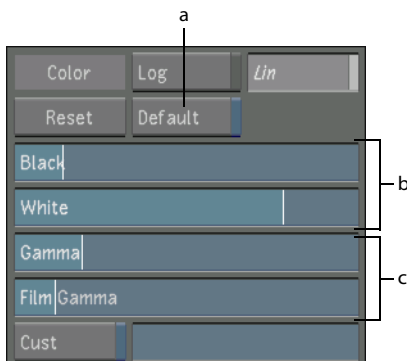
HINT: To ensure that the Linear setting is kept after a video capture, add the `KeepSettingsAfterCapture` keyword to the configuration file. See [“Configuration File Supplemental Options”](#) on page 502.

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.

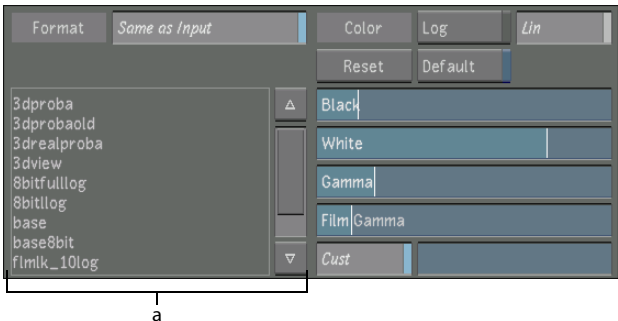


- a) Default button b) Cust button c) LUT box

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.

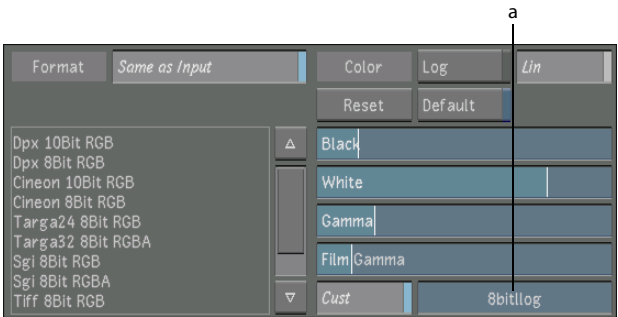


a) Available LUTs

NOTE: The LUTs displayed in the list are all located in the `C:\Program Files\Autodesk\lustre<version number>\luts` folder, if running the Windows version of Lustre, and in the `/usr/autodesk/lustre<version number>/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

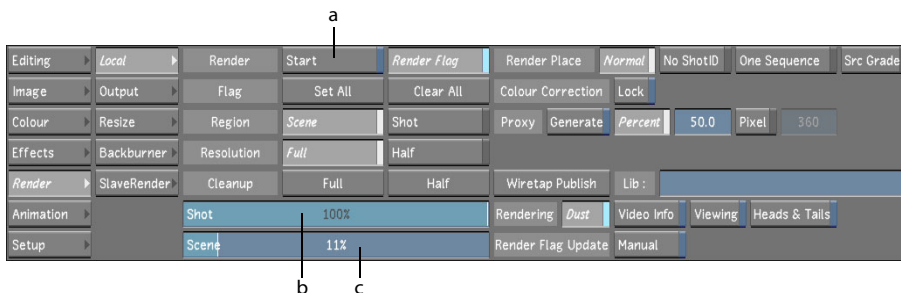
NOTE: If you are using background rendering, place the LUT on all render nodes in the `/usr/autodesk/lustre<version number>/lut/` directory.

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

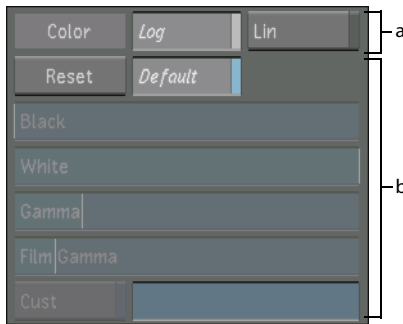
2. Click Start.

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.

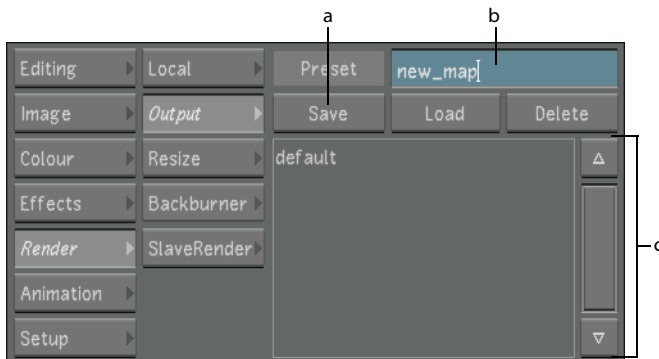


a) Output colour space Log and Lin buttons

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 456.

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.

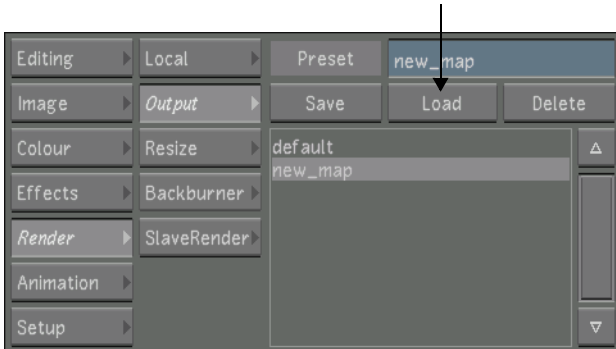
The new preset appears in the Preset list.

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 to be used by adding a keyword to the configuration file. See [“Configuration File Supplemental Options”](#) on page 502.

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 65.

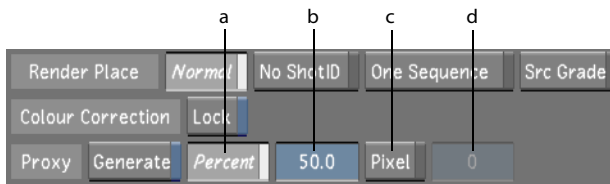
NOTE: Proxy generation differs from rendering a half-resolution render using the Half Resolution button in that grading and other effects data is not applied to proxies as it is when rendering the images.

To generate proxies:

1. If desired, change the proxy generation filter by entering one of the following keywords in the configuration file:
 - ProxyGenFilter Normal (default that uses the Lanczos filter)
 - ProxyGenFilter Mitchell 2 0.2 0.4
 - ProxyGenFilter CatmullRom
 - ProxyGenFilter Lanczos
 - ProxyGenFilter Sinc

See [“Configuration File Supplemental Options”](#) on page 502. 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 474.
2. Load the scene and grade containing the cut for which you want to generate proxies. See [“Selecting Scenes”](#) on page 73 and [“Loading Grades for a Cut”](#) on page 80.
3. Navigate to the first shot in the cut for which you need proxies. You do not have to go to the beginning of the shot, as generation automatically starts at the first frame of the current shot.
4. In the Render Local menu, set the proxy size.

Enable:	To enter:
Percent	The size of the proxy as a percentage in the Percent field.
Pixel	The width of the proxy in pixels in the Pixel field.



- a) Percent button b) Percent field c) Pixel button d) Pixel field

Modifying a value in one field automatically updates the value in the other field.

NOTE: No other options in the Render Local menu affect generated proxies.

5. Click Generate.

The Generate button changes to a Stop button, and proxy generation starts at the first frame of the current shot. It continues through to the end of the cut unless you abort the process by clicking the Stop button.

To view proxies:

- 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.

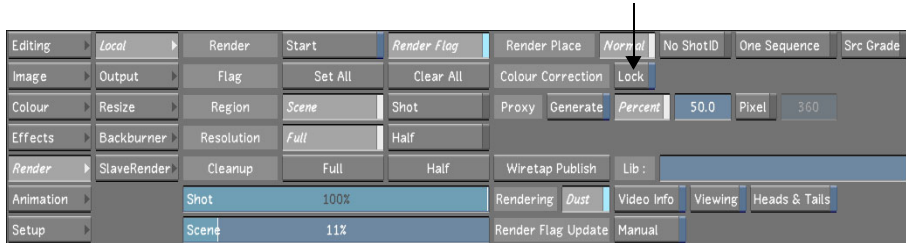
Locking Colour Grading

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 22.

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

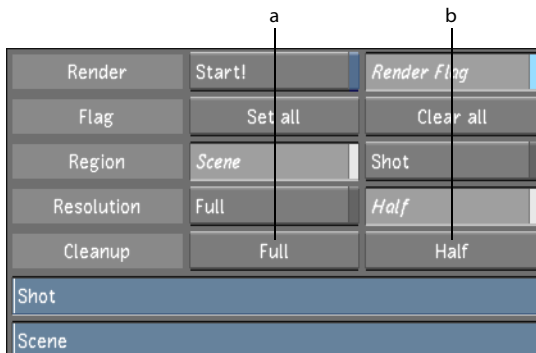
You can remove unneeded render files. Because renders are created separately for each grade (that is, grd01 to grd99), you may want to delete renders for grades that are not used anymore to free up storage.

To remove unneeded render files:

1. Load the grade that has associated unwanted render files. See [“Saving and Loading Grades for a Cut”](#) on page 78.
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.

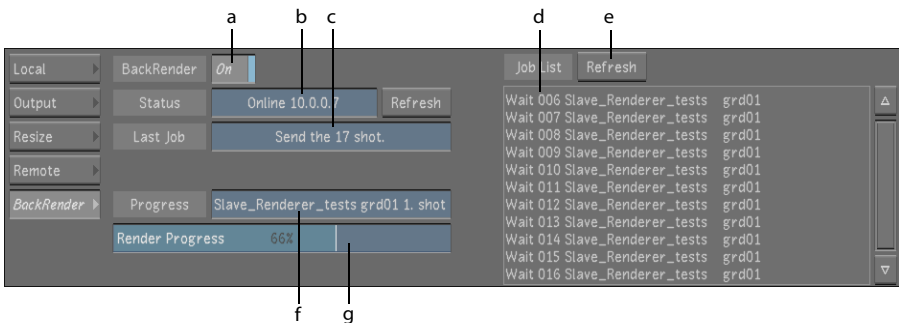
NOTE: When rendering a multi-layer Timeline, Slave renders the shots that are visible from a flattened Timeline, or the shots from the active layer in the Timeline, depending on the mode you are working in. For Solo mode, see [“Accessing the Timeline Menu”](#) on page 176 and for Mute mode, see [“Muting Layers Within the Timeline”](#) on page 185.

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 134.

NOTE: For information on setting up the Slave Renderer, refer to the *Autodesk Lustre Software Installation Guide* for your platform (see [“Related Documentation”](#) on page 2).

To render shots with the Slave Renderer:

1. In the *lustre.config* file or the *init.config* file, uncomment the SlaveRender_HostName line, and then set the IP address for the slave render machine. To render at full resolution when working with proxies, add the RenderFullRes keyword to the configuration file. See [Appendix A, “Software, Project, and Wiretap Configuration Files,”](#) on page 495.
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 438.
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 `BackRender_RefreshTime` keyword in the *lustre.config* file.

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 [“Accessing the Timeline Menu”](#) on page 176) or Mute (see [“Muting Layers Within the Timeline”](#) on page 185) 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 “Specifying the Render Path in Lustre” in the “Configuring Background Rendering” chapter of the *Autodesk Lustre Software Installation Guide* for your platform (see [“Related Documentation”](#) on page 2).

To view and manage jobs sent to the render farm, you use Autodesk Backburner™ Manager and Backburner Monitor.

To render a shot or complete timeline using Burn for Lustre:

1. Start the Backburner server application on all servers (Linux systems). See “Starting Backburner Server” in the “Background Rendering” chapter of the *Autodesk Lustre Software Installation Guide* for your platform (see [“Related Documentation”](#) on page 2).
2. Start the Backburner Manager application on the Windows workstation. See [“Starting Backburner Manager”](#) on page 470.
3. Prepare and submit a job in Lustre. See [“Submitting a Remote Render Job”](#) on page 470.
4. 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. See [“Monitoring the Render Tasks with Backburner Monitor”](#) on page 472.

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 BrowsetD

Enter the Wiretap path in the Renders Full Home field in the Render Backburner menu. See [“Submitting a Remote Render Job”](#) on page 470.

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 474.

Starting Backburner Manager

Before you can start a network render, you must start Backburner Manager on a workstation designated as the Manager, and start Backburner server on each rendering server.

Lustre now supports Autodesk Backburner™ 2007. In Backburner 2007, 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. You can now use Backburner Manager on Linux workstations. See the *Autodesk Backburner 2007 Installation and User's Guide*.

For details on starting and configuring Backburner Manager, see “Starting Backburner Manager” and “Configuring Backburner Manager” in the “Installing Backburner” chapter of the *Autodesk Backburner 2007 Installation and User's Guide*.

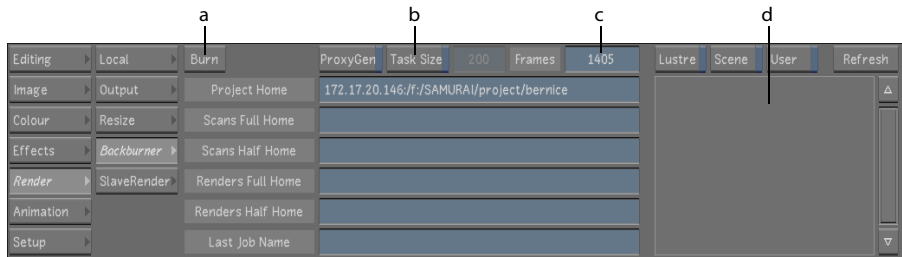
Submitting a Remote Render Job

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 106.
2. Apply the required colour grading changes, and then save the grade.
3. In the Render Local menu, specify the region to render. See [“Specifying the Shots to Render”](#) on page 438.
4. Click Render and then click Remote.

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.

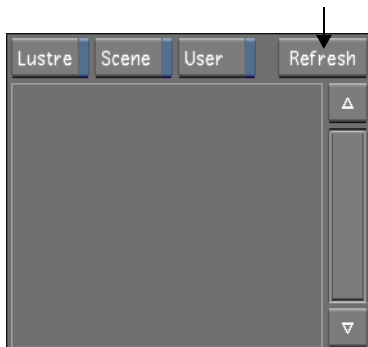
6. Click Burn.

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 472.
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.

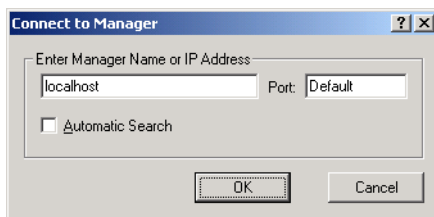
To start Backburner Monitor:

1. On the workstation designated as the render monitor, choose Start | Programs | Autodesk | Backburner | monitor.

The Monitor window appears.

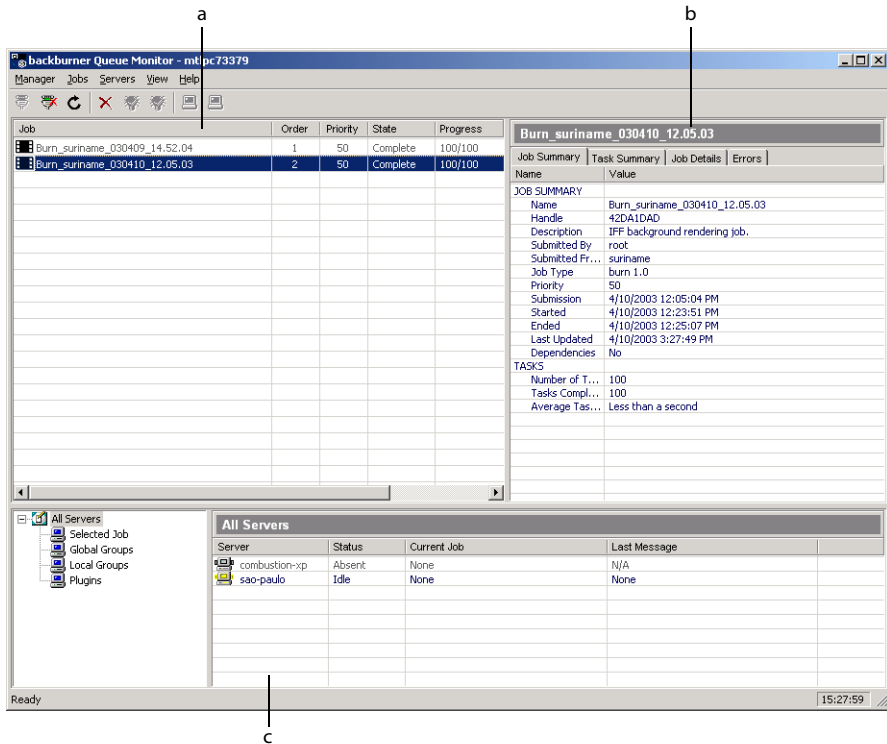
2. If no servers or data appear in the Backburner Queue Monitor, the Backburner Manager was not found with the current settings. Choose Manager | Connect to connect to the Backburner Manager.

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 (see [“Related Documentation”](#) on page 2).

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	

Par.	Modifier	Description	Req'd	Comments
-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 "Creating and Loading Projects" on page 62.
-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.
-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 460.
-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.

Par.	Modifier	Description	Req'd	Comments
-w	None	Renders wedges to the same place.	No	Same as No Wedge option. See "Specifying the Destination for Local Render Files" on page 443.
-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	Overrides the proxy generation filter type specified in the configuration file with the Lanczos (default) filter type.	No	See "Configuration File Supplemental Options" on page 502.
-M	None	Overrides the proxy generation filter type specified in the configuration file with the Mitchell 2 0.2 0.4 filter type.	No	See "Configuration File Supplemental Options" on page 502.
-T	None	Overrides the proxy generation filter type specified in the configuration file with the Catmull-rom filter type.	No	See "Configuration File Supplemental Options" on page 502.
-O	None	Overrides the proxy generation filter type specified in the configuration file with the Lanczos (default) filter type.	No	See "Configuration File Supplemental Options" on page 502.
-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 -l g:/project1
-n 1
```

Where `alabama` is the scene name, `grd21` is the grade name, `dpx` is the output format, and `g:/project1` is the Project Home directory as mapped from the local (rendering) machine.

To render one or more shots with the Command Line Renderer in Windows:

1. Open a Command Prompt window on the machine on which you want to render the shots by choosing Start | Programs | Accessories | Command Prompt.
2. Go to the Lustre application directory by typing:
`cd C:\Program Files\Autodesk\lustre<version number>`
where C is the local drive.
3. Enter the render command using the syntax provided in the previous table.

To render one or more shots with the Command Line Renderer in Linux:

1. Open a Linux shell.
2. Go to the Lustre application directory by typing:
`cd /usr/autodesk/lustre<version number>`
3. Enter the render command using the syntax provided in the previous table.

20

Hot Keys

Summary

Storyboard Navigation	479
View Modes	480
Notes	481
Calculators	482
Dust Removal	482
Editing	483
Colour Grading	485
Numeric Keypad	489
Animation	492
Miscellaneous Functions	492

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.
SHIFT+O	Mark an out point for loop play on the Scene timebar.
SHIFT+L	Clear in and out points from the Scene timebar.

Press:	To:
. (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.
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).
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.

Press:	To:
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.
; (semicolon)	Switch on/off the display of the area of monitor sub-region calculating.
' (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.

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.

Press:	To:
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 enter the `EditingEverywhere` keyword in the configuration file, 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 491.

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.

Press:	To:
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.
SHIFT+ SPACEBAR	Switch to Render View mode, switch off the user interface, and start play.
RIGHT CTRL	Step one frame forward.
RIGHT ALT	Step one frame backward.
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.

Press:	To:
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 the playhead.
]	Zooms in on the Timeline. Zoom is centered on the playhead.
D	Adds a dissolve according to the focus and the positioner.

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 491.

Press:	To:
ESC	Pause the capture. Press again to abort the capture.
E	Restart the capture.
S	Skip the current tape.
DELETE	Delete the last digit of the current timecode value.

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 489.

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.
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.

Press:	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 .
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 selected shots.
ALT-double-click	Load settings specified in the Selector for the selected preset thumbnail to selected 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.
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.

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 Menu — 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.

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.

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.
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.

Press:	To:
} (right brace)	Switch on the render flag of all the shots.
X	Switch on/off background rendering.
T	Set a synchronization point between the currently loaded cut and the audio cut on the DDR. (The DDRSlaveDevice keyword must be enabled.) This hot key can be used from any menu except for the Colour Secondaries menu.
SHIFT+W	Switch on/off Incinerator.

Software, Project, and Wiretap Configuration Files

Summary

Configuration File Overview	495
Configuration File Default Options	496
Configuration File Supplemental Options	502
Configuring Wiretap to Access Remote Stone Filesystems	504
The .lustresetup File	505

Configuration File Overview

The configuration files define all the necessary settings Lustre needs to recognize various hardware and software settings on the Master Station or Lustre HD Station, and on the dedicated render station. Lustre uses three configuration files. Two are identical but are used at different times. The third is used specifically for accessing remote Stone filesystems.

Edit this file:

On Linux:
init.config located in */usr/autodesk/
lustre<version number>/*
On Windows:
*C:\Program Files\Autodesk\lustre<version
number>*

To edit settings for:

All of Lustre. These will be the default settings used if no project is loaded or recognized.

Edit this file:	To edit settings for:
On Linux: <i>lustre.config</i> , located in <i>/usr/autodesk/lustre<version number>/project/<project name></i> On Windows: <i>C:\Program Files\Autodesk\lustre<version number>\project\<project name>\</i>	The specific project. The settings defined in this configuration file will only be used if the project is loaded in Lustre.
On Linux: <i>wt.config</i> , located in <i>/usr/autodesk/lustre<version number>/</i> On Windows: <i>C:\Program Files\Autodesk\lustre<version number>\</i>	Stone filesystem access. The host IP addresses defined here will make it possible to access the contents of remote Stone filesystems from the Lustre file browser.

When creating new projects, the newly created *lustre.config* file will take all the default values from the *init.config* file. Therefore, if you change a value in the *init.config* file, the change must also be made in the *lustre.config* files.

The configuration file can be opened with a text editor and edited quite easily. Once changed, the file must be saved and Lustre restarted.

In addition to the configuration files, a file called *.lustresetup* is documented for here for troubleshooting purposes only. See [“The .lustresetup File”](#) on page 505.

Configuration File Default Options

The following table describes all the options available in the configuration file. Options preceded by a pound symbol (#) are commented out; that is, their value is ignored by Lustre. To enable the parameter, simply delete the # symbol.

Parameter	Default	Function
#Printerlight calibration		
PrinterLight_Step	6.25	Changes the code values corresponding to one lab light.
PrinterLight_HalfStep	3.125	Allows a finer control step for more precision when adjusting the printer light.
PrinterLight_Base	25	This value is used as the base light, corresponding to the unity setting.
#PrinterLight_Inverse	commented out	Enable to switch the printer light density mode.
#ControlPanel config		

Parameter	Default	Function
EthernetCardNumber	1	The port number in which the Ethernet card is installed. If you are experiencing problems with the connection for the Tangent CP100 Control Surface, try entering another port number here.
ControlPanelFreq	24	Defines the communication speed between the Tangent CP100 Control Surface and the Master Station.
#Calibration config		
Calibration_Steps	10	Specifies the number of monitor calibration steps performed.
#Undo mechanism		
#ChangeShot	EmptyUndoBuffer	When enabled, the undo buffer is flushed after changing the shot.
#ChangeShot	Do	When enabled, Lustre forces a Do after changing the shot.
DefaultMaskType	Saturated	Specifies the type of default mask. Choices are grayscale (default) and saturated.
PanScanUpdate	Continuous	When enabled, Lustre immediately updates Pan&Scan changes.
#Backup settings		
#History	commented out	When enabled, a history of all the saved versions of the current scene is saved in the current scene's <i>sacc_data/bak</i> directory.
#Backup_To_System	commented out	When enabled, a copy of all the metadata is saved on the system disk in the Lustre home install directory.
#Autosave	5	When enabled, your work is saved every 5 minutes, unless another minute value is entered for this parameter. This feature is commented out by default.
PixelAspect	1.0	Changes the default aspect ratio of the second display. 1.0 is the default value for film and HD. NTSC uses a 0.9 pixel aspect ratio and PAL uses 1.07. Anamorphic uses a value of 2.

Parameter	Default	Function
DualType	Stretch	Defines how the image is displayed on the secondary monitor. Options are: - Stretch: fits the image to use the entire monitor (aspect ratio change). - Fit: enlarges the image as much as possible and keeps the aspect ratio. - Centre: centres the image without any zooming.
#CalibWait	500	Defines the timeout for the Xrite calibration.
BWMonitorMarkers	95 685	Defines the values of the lines set in the histogram and waveform monitor.
#ReferencelImage	v:/project1/scene1/00163.dpx	Uses a path to set a reference image. Once set, press SHIFT+K to load the image into the single filesystem for reference. Afterward, view the image by pressing L .
#DisableGradeBinMagnifier	commented out	Disables the magnifying pop-up when you hover an item in the Grade bin.
#MustReparse	commented out	When enabled, Lustre is forced to reparse the project structure and source material at start-up.
FirstLastKeyMove	commented out	When enabled, moving a keyed parameter automatically moves the first or last key.
#TimelineCenterDisable	commented out	Disables the timeline in the Animation menu. Selecting a shot will no longer centre the active new shot.
#TimelineBigCenterDisable	commented out	Disables the big timeline in the Animation menu. Selecting a shot will no longer centre the active new shot.
#KeepImagePosition	commented out	When disabled, images can no longer be moved in the interface.
#LinearDefault	commented out	If enabled, inserting a new shot into the timeline will switch off its log-to-lin conversion by default.
#FirstFrameNumber	1	Defines the first frame's number in the timeline display.
#StartShotFrameNumber	0 1	Defines the default frame number at the start of a shot.

Parameter	Default	Function
#SceneConsistency	AutoCheck	This setting has two modes: AutoCheck and AutoRepair. - AutoCheck: when set, this option makes Lustre check the cut files when loading to check for inconsistencies in the metadata. Results are printed out to the console window. - AutoRepair: when set, in addition to the above, inconsistencies will be repaired.
#HideRenderFlag	commented out	When enabled, Lustre does not display the render flag in the upper-left corner of the screen.
#OpenHomeAtStart	commented out	When enabled, the Editing Browse menu will display the content of your project home directory. That is, it will open the project home directory by default instead of displaying files at the drive level.
#SlaveRender	commented out	When enabled, sets the Slave Renderer to be on by default.
#SlaveRender_HostName	192.168.1.2	This address is set to the Slave Renderer's computer name or IP address.
#BurnManager_HostName	192.168.0.12	This address is set to your Burn Manager machine's IP network name or IP address.
#BurnPriority	50	Defines the priority of a job sent to Backburner.
#BurnSkipErrors	commented out	Skips the automatic abort on error when using Backburner.
#Video Sync Modes		
#SyncMode InternalSync	commented out	Enables a free-running, internal sync for SD and HD.
#SyncMode ExternalSync	commented out	Locks to the available SDI input signal.
#SyncMode AnalogSDSync	commented out	Genlocks to the analog sync connected to the sync input. Available for SD only.
#SyncMode DigitalSDSync	commented out	Genlocks to the digital sync connected to the sync input. Available for SD only.
#SyncMode Bilevel03volt	commented out	Genlocks to the bilevel sync. Bilevel signal 0.3V peak. Available for HD only.
#SyncMode Bilevel4volt	commented out	Genlocks to bilevel sync. Bilevel signal 4V peak. Available for HD only.
#SyncMode TrilevelSync	commented out	Genlocks to trilevel sync connected to the sync input. Available for HD only.

Parameter	Default	Function
#SyncMode TTLHFVF	commented out	Genlocks to HV sync signal. Horizontal falling, vertical falling. Available for HD only.
#SyncMode TTLHRVF	commented out	Genlocks to HV sync signal. Horizontal rising, vertical falling. Available for HD only.
#SyncMode TTLHRVR	commented out	Genlocks to HV sync signal. Horizontal rising, vertical rising. Available for HD only.
#HorizontalDelay	0	Sets the number of pixels by which the horizontal sync output is delayed in relation to the incoming sync. Half pixel increments are available for SDTV devices, whereas two-pixel increments are available for HDStationPRO. The maximum delay is one line length.
#VerticalDelay	0	Sets the number of pixels by which the vertical sync output is delayed in relation to the incoming sync. Half pixel increments are available for SDTV devices, whereas two-pixel increments are available for HDStationPRO. The maximum delay is one line length.
#Enabling and configuring the new Autodesk Panel		
#AutodeskPanels	commented out	Instructs Lustre to enable the Autodesk Control Surface. If the keyword is not entered, Lustre will look for the Tangent CP100 Control Surface.
#Panel-BT<Panel_IP>	commented out	Enables the Colour Grading panel. After entering the keyword, enter the panel IP address. This combines the first three numbers of the IP address of the system to which the panel is connected, and the panel ID. For example, if the first three numbers of the Master Station are 192.168.127 and the panel ID is 10, then the panel IP address is 192.168.127.10. This number must be different from the IP address of the Ethernet interface. If they are the same, change the Ethernet interface IP address.

Parameter	Default	Function
#Panel-K <Panel_IP>	commented out	Enables the Function panel. After entering the keyword, enter the IP address. This combines the first three numbers of the IP address of the system to which the panel is connected, and the panel ID. For example, if the first three numbers of the Master Station are 192.168.127 and the panel ID is 9, then the panel IP address is 192.168.127.9. This number must be different from the IP address of the Ethernet interface. If they are the same, change the Ethernet interface IP address.
#Panel-T <Panel_IP>	commented out	Enables the Navigation panel. After entering the keyword, enter the IP address. This combines the first three numbers of the IP address of the system to which the panel is connected, and the panel ID. For example, if the first three numbers of the Master Station are 192.168.127 and the panel ID is 8, then the panel IP address is 192.168.127.8. This number must be different from the IP address of the Ethernet interface. If they are the same, change the Ethernet interface IP address.
#Preventing the user to hit unallowed CC functions while in gpu mode		
GpuNoAutoSwitch	commented out	Prevents you from being switched to CPU mode when you use a feature that is not supported by GPU preview processing while in GPU preview processing mode. When you enable the keyword, all features that are unsupported by GPU preview processing are disabled and made unavailable (greyed-out) in the user interface.

Parameter	Default	Function
ClusterPlayDelay	4	Incinerator-specific keyword that defines the number of frames that will be cached from the timeline before playback begins.
#ClusterSmoothPlayback	commented out	Incinerator-specific keyword that diminishes the “pumping” effect that can occur when grades are slower than realtime. “Pumping” refers to the effect produced when nodes render all frames at an equal rate even when the frame rate is reduced. This keyword estimates the rate at which Lustre receives the rendered frames, and then attempts to adjust the playback speed to the estimated rate.

Configuration File Supplemental Options

The following table describes options that can be added to the configuration file to supplement Lustre functionality.

Parameter	Function
AudioPlaybackRate	Allows users to lock .wav file playback speed (fps) to a specific value, thereby making the audio fps rate independent of the video fps rate. For example: AudioPlaybackRate 25. By default, Lustre plays both video and audio tracks at the same rate.
ControlPanelNoAuto	Disables automatic detection of the Tangent CP100 Control Surface.
DisableEject	Disables automatic tape eject during EDL capture.
DisplayType LCD/CRT	Indicates the type of monitor being used. This keyword is required in order to calibrate an LCD monitor from the Setup Calibrate menu, as the application defaults to CTR setup, unless otherwise configured. For an LCD monitor, enter DisplayType LCD in the <i>init.config</i> file. For a CRT monitor, enter DisplayType CRT.
PlayoutHighSpeed	Disables screen updates during playout.
WtServerNoAuto	Disables automatic detection of Stone filesystems on the network.
NoAutoDustLoad	Ensures that dust metadata is not automatically loaded when you load the grade file.

Parameter	Function
BlackClipColor	Allows the user to define the black clip colour. After entering the BlackClipColor keyword, enter the following four numbers: the colour's bit depth, followed by the red, green and blue bit depth ranges. For example: BlackClipColor 8 152 215 058.
KeepSettingsAfterCapture	Ensures that the colour space architecture you specify in your project-level and output settings are maintained after you perform a video capture.
DPXReelname	Matches the EDL reelname to the reelname in the DPX header during EDL assembly.
MarryType PrimText	Ensures that the Marry files are text based and saves the primary offsets as 10-bit code values.
Backrender_RefreshTime	Sets the number of seconds that elapse before the Slave Renderer progress indicator is updated. For example, to set the update interval to every three seconds, enter: Backrender_RefreshTime 3.
DisableCadenceRedetect	Allows you to disable the automatic pulldown cadence detection on every shot for capture and playout. When the keyword is enabled, Lustre detects the pulldown cadence value once and then applies it to all other capture and playout processes.
DDSlaveDevice	Enables Lustre to control a DDR device.
EditingEverywhere	Makes editing hot keys available from all menus.
RenderFullRes	Allows you to render shots from Burn and the Slave Renderer at full resolution when working with proxies.
MatchCustom	Specifies the XML metadata field to be used by the Custom match option selected from the Browse menu. Enter the MatchCustom keyword followed by the XML field name. For example: MatchCustom DPXKeyCodeEnd.
MatrixMode ccir709digital	Maps colours for ccir709 using the full range of YUV and RGB.
MatrixMode ccir709cgr	Maps the 64-960(Y)/940(UV) range of YUV for ccir709 to the full RGB range.
MatrixMode ccir601digital	Maps colours for ccir601 using the full range of YUV and RGB.
MatrixMode ccir601cgr	Maps the 64-960(Y)/940(UV) range of YUV for ccir601 to the full RGB range. This is the PAL default.
MatrixMode smpte274digital	Maps colours for smpte274 using the full range of YUV and RGB.
MatrixMode smpte274cgr	Maps the 64-960(Y)/940(UV) range of YUV for smpte274 to the full RGB range. This is the HD default.

Parameter	Function
MatrixMode User	Allows you to specify a custom colour space conversion matrix. After the keyword, include the path for the text file that contains the matrix definition. For example: MatrixMode User "D:\Kaktusz2\MatrixDefine.txt". The matrix contains keywords and constants for each conversion. For example: R2Y 0.18256 G2Y 0.61426 B2Y 0.06201 R2V -0.10065 G2V -0.33856 B2V 0.43921 R2U 0.43921 G2U -0.39893 B2U -0.04028 All lines in the matrix definition file that begin with a # are ignored.
UseCGR	Ensures that mapping uses only the 64-940(960) range and not the whole YVU range.
NVidiaFullRange	Allows the NVIDIA card to output full 0-1023 video ranges for either single link or dual link video formats.
RepoType Lanczos	Enables the Lanczos reposition filter.
RepoType Mitchell 3 0.2 0.4	Enables the Mitchell reposition filter.
RepoType CatmullRom	Enables the CatmullRom reposition filter.
RepoType Lanczos3lobe	Enables the Lanczos3lobe reposition filter.
ProxyGenFilter Normal	Enables the Lanczos proxy generation filter.
ProxyGenFilter Mitchell 2 0.2 0.4	Enables the Mitchell 2 0.2 0.4 proxy generation filter.
ProxyGenFilter CatmullRom	Enables the CatmullRom proxy generation filter.
ProxyGenFilter Lanczos	Enables the Lanczos proxy generation filter.
ProxyGenFilter Sinc	Enables the Sinc proxy generation filter.

Configuring Wiretap to Access Remote Stone Filesystems

By default, all Stone filesystems on the network are automatically detected and displayed in the file browser. However, if your facility has numerous Stone filesystems and the automatic detection process is slowing down system performance at startup, you can disable the process by entering the `WtServerNoAuto` keyword in the *lustre.config* file.

When this keyword is used, you must identify each remote Stone filesystem manually. This is done by entering the host IP address of each filesystem in the *wt.config* file.

To define the host IP addresses to be browsed in Lustre:

1. Access the *wt.config* file located in the same directory as the Lustre executable.
2. Enter one IP address per line. For example:

#Wiretap Server Address

192.168.20.56

172.17.20.151

3. Save and close the file.

Once the configuration is complete, a list of available IP addresses will appear in the Lustre browser, located on the right side of the screen.

The .lustresetup File

The *.lustresetup* file contains all user preferences. There are two versions of the file. Each resides in its own location and performs a specific function.

File Location:	Description
On Linux: <i>/root/.lustresetup</i> On Windows: <i>C:\Program Files\Autodesk\lustre<version number>\.lustresetup</i>	Records the user and project last loaded in the application. This file should be modified for troubleshooting purposes only.
On Linux: <i>/usr/autodesk/lustre<version number>/user/<user>/.lustresetup</i> On Windows: <i>C:\Program Files\Autodesk\lustre<version number>\user\<user>\.lustresetup</i>	Contains all user and project information. When you launch Lustre, the most recently loaded user and project are obtained from the setup file described above, and then the detailed settings are provided by this file.

Summary

Lustre XML-Based Files	507
Limitations	507
CUT File Tags	508
CUT File Data Types	517
CUT File Example	518

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 21.

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 517.
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 517.
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 517.
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 517.
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 517.
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 517.
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 517.
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 517.
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 517.
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 517.
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

Numerics

10-bit

- outputting 219
- recording without rendering 223

3D mesh LUTs

- applying 57
- monitor calibration 46

A

ALE files

- assembling 110
- auto-conforming 207
- copying into Lustre 120
- marking shots for later capture 126

Analyse tools

- using to view colour distributions 286

analysing, colours 285

animating

- colour grades 291
- dissolves 422
- geometries with the point tracker 366
- geometries with the shape tracker 366
- reposition values 260
- shape of geometries with the point tracker 373
- Sparks 390

animations

- adding from outside the Animation menu 432
- adding keyframes 410
- adjusting duration 431
- adjusting timing 428
- breaking tangent handles 422
- editing keyframes 418

hiding inactive tangents 424

hot keys 492

modifying curves 419

overview 403

panning and zooming 406

resetting tangents 423

reversing keyframe order 424

selecting channels 409

setup options 434

undoing and redoing 436

using the channel hierarchy 408

workflow 408

aspect ratio, setting for the output frame 253

assembling

EDL files 121

EDLs to the current Timeline 193

EDLs, ALE and Cutlist files 110

audio, importing and offsetting 40

Autodesk control surface, using 17

autograding, shots based on sampled pixels 313

automated dust removal

overview 238

performing 242

axes

adding and deleting from geometries 358

moving within geometries 349

viewing in schematic view 360

axes list

creating a parent axis 359

deleting an axis 359

view of geometries and their axes 359

B

Backburner Manager, starting 470

- Backburner Monitor
 - Linux workstation, accessing from 473
 - monitoring render tasks 472
- background rendering
 - overview 466
 - with the Slave Renderer 466
- batch, digitizing 205
- black clip, adding to the Storyboard 174
- black level
 - adjusting 49
 - modifying for input LUTs 270
- black media, adding to the Timeline 184
- blurring
 - geometries 356
 - keys 337
 - with Sparks 391, 395, 399
- boxes, adjusting for dust removal 245
- brightness
 - adjusting printer lights 298
 - modifying for low, mid, or high levels 300
 - modifying for the entire image 297
 - resetting 301
- building
 - change lists 105
 - cuts 105
- Burn for Lustre
 - overview 468
 - rendering locally 469
 - rendering remotely 470
- bypassing menu, in the UI 18

C

- caching, for real-time playback 150
- calculator
 - applying values to UI elements 30
 - hot keys 482
 - using to calculate timecode 141
- calibrating, see monitor calibration
- cancelling, actions 20
- capturing
 - at a selected video format 230
 - from a batch list 205
 - from a VTR 202
 - from an EDL, ALE or Cutlist file 207
 - from timecode 202
 - hot keys 491
 - performing live from non-controlled source

- 209
- preparatory steps 228
- removing pulldown 209
- setting options 228
- shots after the initial conform 126
- supported video formats 231
- video, overview 199
- CDL
 - adding data to EDL file 118
 - example workflow 118
 - loading data in EDL panel 115
 - overview 111
 - panel 113
 - parameter in Selector 114
- CDL files
 - 110
- change lists
 - building 105
 - creating 109
 - deleting 110
 - loading 110
 - using to track changes 108
- channel hierarchy, using to animate channels 408
- channels
 - displaying in colour view mode 136
 - selecting 409
 - selecting for a key 329
 - viewing animated 407
 - viewing selected animated 407
- cleaning up, render files 465
- collapsing, Storyboard 145
- colour grading 275
 - accessing Colour menus 278
 - accessing secondary grading menus 328
 - animating grades 291
 - concepts 276
 - creating continuity grades 278
 - curves grading 277, 315
 - hot keys 485, 489
 - keying ranges of colours to create secondaries 329
 - loading presets 289
 - primary grading 277, 295
 - saving presets 288
 - secondary grading 278, 327, 382
 - selecting Logarithmic or Linear mode 277
 - using control surfaces 283

- using printer lights 278
 - workflows 276
- colour space
 - rendering 456
 - setting for projects 51, 75
 - UI differences between Logarithmic and Linear mode 76
- colourimeter, calibrating a monitor 52
- colourist
 - Timeline 291
- colours
 - adjusting for geometries 356
 - analysing 285
 - comparing when creating conversion LUTs 268
 - sampling values 285
 - viewing distributions with the Analyse tools 286
- Command Line Renderer
 - example 476
 - rendering with 474
 - syntax 474
- comparing
 - shots using a reference 284
 - shots using multi-view 284
- confidence check, performing 194
- configuration file
 - additional options 502
 - default options 496
 - overview 495
 - Wiretap 504
- confirming, actions 20
- continuity grades, overview 278
- continuity, matching colours between shots 312
- contrast
 - modifying in Linear mode 310
 - modifying in Logarithmic mode 302
 - setting the pivot point 303
- control surfaces
 - using 17
 - using for colour grading 283
- controlling
 - DDR 201
 - VTR 199
- converting
 - data with input LUTs 264
 - from logarithmic to linear 265
- copying
 - EDL, ALE, and Cutlist files into Lustre 120
 - parameters with the Selector 28
 - shapes and grades 385
- creating
 - change lists 109
 - cuts 106
 - projects 62
 - scenes 71
 - users 60
- credits, adding to the timeline 42
- cropped frames, repositioning 252
- curves
 - adjusting saturation 321
 - colour grading 315
 - measuring for monitor calibration 53
 - modifying lightness 324
 - modifying luminance 322
 - modifying RGB channels 318
 - plotting colour values 316
 - shifting hue 320
- curves colour grading
 - hot keys 490
 - overview 277
 - view mode 134
- CUT files
 - data types 517
 - example 518
 - modifying 508
- cut point, sliding between shots 166
- Cutlist files
 - assembling 110
 - auto-conforming 207
 - copying into Lustre 120
 - marking shots for later capture 126
- cuts
 - adding overlays 87
 - building 105
 - changing without affecting grade settings 107
 - creating new 106
 - deleting 108
 - inserting between shots 167
 - loading 106
 - managing 106
 - saving 107
 - tracking changes 108

D

- DDR, controlling 201
- deleting
 - change lists 110
 - cuts 108
 - geometries 363
 - grades 85
 - keyframes 427
 - shots 158
 - shots from Shot bin 96
- destination, specifying render files for 447
- developing, Sparks 401
- directories, setting up for projects 65
- disabling
 - dissolves 174
 - reframing settings 256
 - Sparks 400
- dissolves
 - adding between shots 171
 - adding in the Timeline 191
 - animating 422
 - changing duration 172
 - changing orientation in the Timeline 192
 - changing speed 173
 - creating 171
 - creating additional handles for 173
 - disabling and re-enabling 174
 - editing 173
 - editing in the Timeline 192
 - in the Timeline 191
 - overview 171
 - removing in the Timeline 191
 - retrimming 173
- documentation
 - library, on the Autodesk Web site 5
 - set of guides 2
- drawing
 - basic geometries 344
 - free-form geometries 345
 - split screens 357
- dust removal
 - choosing a repair type 238
 - choosing manual or automated 238
 - dealing with problem dust artefacts 244
 - drawing boxes 245
 - hot keys 482
 - optimizing parameters 239

- overview 237
- performing automated 242
- performing manual 241
- rendering 248, 453
- saving metadata 247
- using the infrared channel 246

E

- edges, modifying for keys 337
- editing
 - accessing the edit menu 154
 - duplicating shots in the Storyboard 158
 - frames by trimming 159
 - hot keys 156, 483, 491
 - in and out points interactively 165
 - in Timeline 181
 - overview 153
 - shot order in the Storyboard 157
 - shots from the Storyboard 158
 - shots from the Timeline 158
 - start timecode 194
 - with the Storyboard 156
 - XML-based files 507
- EDL files
 - and Wiretap, overview 193
 - assembling 110, 121
 - assembling to the current Timeline 193
 - assembly workflow 118
 - auto-conforming 207
 - copying into Lustre 120
 - EDL display and functionality 116
 - exporting 129
 - marking shots for later capture 126
- effects, creating with Sparks 388
- engineering options, setting 234
- exporting, EDL files 129
- extrapolation, setting 416

F

- field-based processing, setting the active field 136
- file browser, accessing 91
- file formats, supported for playback 93
- flagging shots 22
 - automatically 440
 - for rendering 438
 - manually 439
- flipping, shots 256

- flopping, shots 256
- format, setting the output for render files 455
- full-resolution images, displaying 135
- full-screen Player
 - activating 133
 - panning and zooming 133
 - using hot keys 151

G

- gain, modifying 308
- gamma
 - adjusting for the monitor manually 55
 - measuring for monitor calibration 52
 - modifying 308
 - modifying for input LUTs 271
- garbage masks, loading setups 382
- generating, proxies 462
- geometries
 - adding and deleting axes 358
 - adjusting shapes 350
 - animating with the point tracker 366, 373
 - animating with the shape tracker 366
 - assigning tracking data 376
 - blurring 356
 - changing priority order 361
 - copying shapes and grades 385
 - creating 342
 - creating a parent axis 359
 - deleting 363
 - deleting an axis 359
 - deleting point tracker data from 379
 - drawing basic 344
 - drawing free-form 345
 - drawing split screens 357
 - grading inside and outside 385
 - inverting 362
 - linking across secondaries 364
 - modifying 349
 - moving axes 349
 - setting colour and opacity 356
 - showing and hiding 362
 - softening edges 353
 - soloing and ganging point tracker 378
 - transforming 352
- GPU preview processing
 - GPU compatibility 283
 - GpuNoAutoSwitch parameter 283

- playing out from source with 218
 - real-time 279
- Grade bin 32
 - applying intermediary grades 34
 - deleting intermediary grades 37
 - saving intermediary grades 32
- grade bin storage 291
- Grade menu, selecting a scene 73
- grades
 - applying from the Grade bin 34
 - applying to a new assembled timeline 128
 - copying unrendered changes to real-time deliverables 226
 - deleting 85
 - deleting from the Grade bin 37
 - emulating telecine-style tape 220
 - loading backup copies 81
 - locking 464
 - maintaining when swapping cuts 107
 - saving and loading for a cut 78
 - saving and loading for single shots 82
 - saving to the Grade bin 32
- grouping
 - applying changes to shots as a group 26
 - creating a shot group 25
 - soloing shots in a group 27

H

- head frames, rendering 455
- help, accessing online 3
- hiding, Storyboard 143
- History list, loading for a grade 81
- hot keys
 - accessing editing hot keys globally 156
 - assigning notes to shots 481
 - capturing and playing out with the numeric keypad 491
 - colour grading 485, 489
 - creating animations 492
 - curves colour grading with the numeric keypad 490
 - editing 483
 - editing with the numeric keypad 491
 - for Player and full-screen Player 151
 - miscellaneous functions 492
 - navigating the Storyboard 479
 - navigating with the playback controls 139

- numeric keypad 489
 - removing dust 482
 - switching view modes 480
 - using calculators 482
- hues
 - shifting for secondaries 383
 - shifting with curves 320
- I
- infrared channel, using for dust removal 246
- input LUTs
 - applying existing 264
 - comparing colour values 268
 - creating conversion LUTs 265
 - modifying gamma of film footage 271
 - modifying reference black and white levels 270
 - modifying reference soft clip values 272
 - overview 263
 - viewing reference images 266
- installing, Sparks 388
- Interface menu, customizing Lustre's appearance 86
- interlaced video, setting the active field 136
- interpolation, setting 413
- inverting
 - geometries 362
 - keyframe values 426
 - keys 339
- J
- joining, shots together 168
- K
- keycode
 - assembly 124
 - assembly workflow 118
- keyframes
 - adding 410
 - adding from outside the Animation menu 432
 - deleting 427
 - editing 418
 - inverting values 426
 - locking movement 436
 - modifying curves 419
 - navigating between 405
 - reversing order 424
 - scaling to adjust animation duration 431
 - selecting 419
 - setting extrapolation 416
 - setting interpolation 413
 - viewing channel ranges between 407
 - viewing selected 407
- keys
 - creating to isolate ranges of colours 329
 - excluding from secondaries 339
 - importing mattes 341
 - inverting 339
 - removing stray pixels from 338
 - saving and loading presets 340
 - selecting channels 329
 - setting initial softness and tolerance 330
 - setting softness ranges 334
 - setting tolerance ranges 336
 - shrinking edges 337
 - viewing results 332
- L
- letterbox, scaling 251
- levels, selecting in the image for colour grading 305
- licensing
 - options for Lustre 2
 - Sparks 388
- lift, modifying colour values 306
- lightness, modifying using the Lightness curve 324
- limitations, XML-based files 507
- Linear mode
 - defining luminance 311
 - improving contrast 310
 - modifying gain 308
 - modifying gamma 308
 - modifying lift 306
 - modifying saturation 309
 - monitor calibration 48
 - using in Lustre 277
- Linear mode, selecting levels 305
- linking, geometries across layers 364
- loading
 - change lists 110
 - cuts 106
 - grades 78

- projects 62
 - shots to the Shot bin 93
 - shots to the storyboard 97
 - users 60
- locking
 - colour grade 464
 - keyframe movement 436
- Logarithmic mode
 - adjusting printer lights 298
 - improving contrast 302
 - modifying brightness for low, mid, or high levels 300
 - modifying brightness for the entire image 297
 - modifying saturation 304
 - monitor calibration 48
 - resetting brightness 301
 - setting the pivot point 303
 - using in Lustre 277
- lookup tables, see LUTs
- LTC, outputting while working on the storyboard 219
- luminance
 - modifying in Linear mode 311
 - modifying using the Luminance curve 322
- Lustre
 - configurations 2
 - exiting 13
 - setup file 505
 - starting 13
- LUTs
 - 3D mesh 46
 - monitor calibration 46
 - See also input LUTs
 - viewing 137
- M**
 - manual dust removal
 - overview 238
 - performing 241
 - Marry controls, saving the grade for a single shot 82
 - matching
 - colours between shots 312
 - grade settings with a new assembled timeline 128
 - shots between the storyboard and Shot bin 105
 - mattes, importing 341
 - metadata, saving for dust removal 247
 - monitor calibration 45
 - 3D mesh LUTs 46, 57
 - adjusting black level and picture settings 49
 - adjusting manually 55
 - calibrating 51
 - curves 53
 - for dual-head mode 55
 - gamma 52
 - Logarithmic and Linear mode 48
 - LUTs 46
 - Print LUTs 56
 - reverting to previous settings 54
 - selecting project colour space 51
 - using a colourimeter 52
 - workflow 48
 - monitoring location of current frame 140
 - moving, geometries 352
 - Multilayer Timeline, See Timeline 176
 - multi-view, viewing in the Player or SDI monitor 149
 - muting, layers within Timeline 185
- N**
 - navigating
 - between keyframes 405
 - the Storyboard 138, 142
 - using the timebar and Storyboard 139
 - with playback controls and hot keys 139
 - with Timeline navigation controls 179
 - noise, adding with Sparks 391, 396
 - notes
 - assigning to shots 21
 - hot keys 481
- O**
 - online help, accessing 3
 - opacity, adjusting for geometries 356
 - output LUTs
 - creating 457
 - selecting 458
 - output, view mode 134
 - outputting, LTC while working on the storyboard 219
 - overlays, adding to cuts 87

P

- panning
 - animation curves 406
 - in Player and full-screen Player 133
- parameters
 - bypassing 18
 - configuration file add-ins 502
 - configuration file defaults 496
 - optimizing for dust removal 239
 - resetting 17
 - setting for projects 74
- PDF documentation, accessing 4
- picture settings, adjusting 49
- Pixel Analyser, sampling images 285
- pixels, removing from keys 338
- playback, changing speed 169
- Player
 - activating full-screen 133
 - displaying the user interface 137
 - setting to split or multi-view 149
 - using hot keys 151
 - using Split View 148
 - viewing multiple shots 147
- playheads
 - assigning reference images 147
 - assigning user interface to active playhead 137
- playing
 - shots in the Player 131
 - shots rendered with the Slave Renderer 468
- playing out
 - accessing Playout Menu 212
 - adding pulldown 217
 - defining in and out points when 210
 - from source with GPU preview processing 218
 - hot keys 491
 - in Assemble mode 215
 - in Insert mode 214
 - in Record mode 213
 - real-time deliverables 226
 - setting options 228
 - supported video formats 231
 - to a VTR 211
 - using the Direct playout option 219
 - video, overview 199
 - workflow 212
- Playout Menu, accessing 212
- plotting, sampled colour values 316
- Plugins, see Sparks
- point tracker
 - animating a geometry 368
 - animating geometries with 366
 - animating the shape of geometries 373
 - assigning a tracker to each vertex 368
 - deleting 379
 - loading data 380
 - soloing and ganging 378
 - tracking points 368
 - tracking without rigid, trackable shapes 368
- post-production pipeline
 - integrating Lustre into 101
- preferences, configuring in Lustre 505
- presets
 - colour mapping, saving and loading 460
 - for real-time deliverables 224
 - loading colour grading parameters 289
 - saving and loading for keys 340
 - saving colour grading parameters 288
- primary colour grading
 - applying secondaries to 386
 - overview 277, 295
 - view mode 134
- Print LUTs, applying preset gamma correction 56
- print, view mode 134
- printer lights
 - accessing the calculator from 30
 - adjusting brightness 298
 - adjusting for secondaries 384
 - using in Log mode 278
- processing, image pipeline order 8
- projects
 - creating and loading 62
 - creating the directory structure 65
 - reparing the structure 88
 - setting colour space architecture 75
 - setting parameters 74
 - setting the resolution 76
 - setting up for footage accessed via Wiretap 64
 - setting up for soft-imported clips 70
 - setup workflow 60
- proxies
 - displaying 135
 - generating 462

- generating locally from Wiretap footage 63
- viewing 284
- pulldown
 - adding for playout 217
 - removing for capture 209
- R**
 - racking, to reframe footage 254
 - rdesktop, monitoring results 473
 - real-time deliverables
 - copying changes to or from the grade file 226
 - creating presets 224
 - overview 223
 - playing out 226
 - resizing 226
 - real-time playback
 - using GPU preview processing 279
 - with frame caching 150
 - recording
 - Assemble mode 215
 - Insert mode 214
 - Record mode 213
 - redoing
 - actions 19
 - in animation 436
 - reference images
 - assigning to playheads 147
 - viewing 145
 - viewing results of logarithmic to linear conversion 266
 - reference, setting the view mode 134
 - reframing
 - aspect ratio output frame 253
 - disabling settings 256
 - flipping and flopping shots 256
 - racking the image 254
 - using reposition controls 255
 - regions, rendering selected 460
 - removing, dust and other artefacts 237
 - render files, removing 465
 - rendering
 - BackBurner Manager, starting 470
 - burning in timecode 454
 - colour space, setting 456
 - dust removal data 248, 453
 - file destination, specifying 447
 - flagging shots automatically 440
 - flagging single shots manually 439
 - head and tail frames 455
 - in the background 466
 - locally with Burn for Lustre 469
 - monitoring with Backburner Monitor 472
 - output format, specifying 455
 - output LUTs, creating 457
 - output LUTs, selecting 458
 - overview 437
 - presets for colour mapping, saving and loading 460
 - region, specifying 438
 - remotely with Burn for Lustre 470
 - resolution, setting 440
 - selected region 460
 - setting quality for repositioned shots 261
 - shots 438
 - Timeline 193
 - with Burn for Lustre 468
 - with the Command Line Renderer 474
 - with viewing LUTs 454
 - repair type, selecting for artefact removal 238
 - reparsing, project structure 88
 - replacing, shots in the Storyboard 170
 - repositioning
 - after adjusting the project aspect ratio 255
 - cropped frames 252
 - interlaced footage using Sparks 394
 - setting rendering quality after 261
 - using animation controls 260
 - with the tracker 257
 - resetting
 - parameters 17
 - secondaries 363
 - resizing, real-time deliverables 226
 - resolution
 - capture 230
 - display 135
 - project 76
 - rendering 440
 - retiming, playback speed for shots 169
 - reverting to previous, calibration settings 54
 - RGB channels, adjusting individually with curves 318
 - rotating, geometries 352

S

- sampling, pixels for autograding 313
- saturation
 - adjusting with curves 321
 - modifying in Linear mode 309
 - modifying in Logarithmic mode 304
- saving
 - cuts 107
 - grades 78
- scaling
 - geometries 352
 - letterbox format 251
- Scene menu, selecting a scene 73
- scenes
 - creating 71
 - detecting 195
 - selecting 73
 - working with 74
- schematic view, toggling to 360
- SDI monitor
 - viewing multiple shots 149
- SDI monitor, setting to split or multi-view 149
- secondary colour grading
 - accessing grading menus 328
 - adjusting printer lights 384
 - applying Sparks 390
 - applying to the entire image 386
 - changing layer order 361
 - creating geometries 342
 - excluding keys 339
 - grading outside the secondary 385
 - keying ranges of colours 329
 - loading garbage mask setups 382
 - loading tracker and stabilizer data 380
 - overview 278, 327
 - performing colour grading 382
 - resetting 363
 - shifting hue 383
 - showing and hiding geometries 362
 - turning on or off 386
 - view mode 134
 - viewing key results 332
- selecting, keyframes 419
- selective colour grading, see secondary colour grading
- Selector
 - CDL parameter 114
- Selector, copying parameters 28
- setup options, Animation menu 434
- shape tracker
 - animating geometries with 366
 - point cloud 366
 - region of interest 366
 - tracking through translation, rotation, scaling, partial occlusions 366
- sharing, changes between grades and real-time deliverables 226
- Shot bin
 - deleting shots from 96
 - loading shots from the file browser 91, 93
 - selecting shots for the Storyboard 156
- shots
 - adding dissolves 171
 - adding in Timeline 181
 - adding to the Storyboard 156
 - applying changes to a group 26
 - assigning notes 21
 - comparing multiple in the Player 284
 - comparing using a reference 284
 - deleting 158
 - deleting from Shot bin 96
 - dividing into scenes 195
 - duplicating in the Storyboard 158
 - flagging 22
 - grouping 25
 - identifying selection colours on the Storyboard 144
 - joining back together 168
 - matching between the storyboard and Shot bin 105
 - moving to the storyboard 97
 - playing in the Player 131
 - rendering 438
 - reordering in the Storyboard 157
 - replacing in the Storyboard 170
 - selecting in the Storyboard 143
 - setting to half resolution 284
 - slipping in the timeline 164
 - soloing 27
 - trimming head or tail frames 160
 - viewing information 97, 159
- size, setting the Storyboard view 143
- Slave Rendering
 - background rendering 466

- playing shots rendered with 468
- sliders, modifying with the calculator 30
- sliding, cut point in a shot 166
- slipping, shots in the timeline 164
- softclip values, modifying for input LUTs 272
- softness
 - applying to geometries 353
 - setting initial for keys 330
 - setting ranges for keys 334
- Sparks
 - animating 390
 - applying to secondaries 390
 - creating effects with 388
 - descriptions 391
 - developing 401
 - disabling and re-enabling 400
 - installing 388
 - licensing 388
 - overview 387
 - using to add noise 391, 396
 - using to blur images 391, 395, 399
 - using to reposition interlaced footage 394
- speed
 - dissolves 173
 - playback 169
- splicing
 - by adding a shot to the Timeline 190
 - shots 167
- split screens, drawing 357
- split view, activating in the Player or SDI monitor 149
- Stone filesystem
 - linking to media on StoneFS via Wiretap 103
 - playing clips located on 104
- Stone filesystem, accessing footage via Wiretap 100
- Storyboard
 - adding a virtual clip 174
 - adding shots 156
 - collapsing 145
 - deleting shots from 158
 - duplicating shots in 158
 - editing shots 156
 - hot keys 479
 - inserting cuts between scenes 167
 - joining shots together 168
 - navigating 138, 142
 - navigating using Timebar and Storyboard 139
 - reordering shots 157
 - replacing shots 170
 - selecting shots 143
 - selection colours 144
 - setting view size 143
 - showing and hiding 143
 - sliding the cut point between shots 166
 - trimming frames interactively 165
 - updating thumbnails 144
 - viewing shot information 159
- supported file formats, in Lustre 93
- syntax
 - example 476
 - for the Command Line Renderer 474
- T**
- tail frames, rendering 455
- Tangent CP100 control surface, using 17
- tangents
 - adjusting to reshape geometries 350
 - breaking for animations 422
 - hiding handles 424
 - resetting broken handles 423
- telecine tape
 - emulating grading with 220
 - writing to, enhancements for workflows using NTSC 221
- thumbnails, updating in the Storyboard 144
- timecode
 - burning into render files 454
 - calculating 141
 - modifying the start value 194
 - viewing information 159
- Timeline
 - accessing the Timeline menu 176
 - adding a dissolve in 191
 - adding black media to 184
 - adding shots to 181
 - assembling an EDL to 193
 - changing orientation of a dissolve in 192
 - checking against an offline cut 194
 - colourist 291
 - copying elements in 182
 - creating 181
 - deleting elements in 184

- deleting shots from 158
- dissolves 191
- editing 181
- editing a dissolve in 192
- gaps, overview 185
- modifying the starting timecode 194
- moving elements in 181
- muting layers within 185
- navigation controls 179
- overview 175
- removing a dissolve in 191
- rendering 193
- selecting elements in 181
- sliding an element 188
- slip-and-slide Ripple Mode 189
- slipping an element 187
- slipping-and-sliding an element 189
- splices 190
- transitions 190
- trimming 186
- trimming cursors 186
- trimming with Ripple Mode 186
- view in Lustre 176
- viewing element information 180
- Timeline menu, accessing 176
- timing, adjusting for animations 428
- tolerance
 - setting initial for keys 330
 - setting ranges for keys 336
- tracking
 - geometries 376
 - repositioned shots 257
- trimming
 - cursors in the Timeline 186
 - frames in the timeline 159
 - head or tail frames 160
 - in and out points 166
 - interactively 165
 - shots for dissolves 173
 - Timeline 186
 - with Ripple Mode in the Timeline 186
- troubleshooting, problem dust artefacts 244

U

- undoing
 - actions 19
 - in animation 436

- user interface
 - differences in Logarithmic and Linear mode 76
 - setting appearance of 86
- user's guide, conventions 5
- users, creating 60

V

- versioning, for film and video 235
- video
 - capture and playout, overview 199
 - capturing from timecode 202
 - formats, supported 231
 - outputting multiple versions 235
 - preparatory steps for capturing from 228
 - setting engineering options 234
- view modes
 - hot keys 480
 - setting 134, 136
- viewing
 - animated curves 407
 - channel ranges between keyframes 407
 - multiple shots in the player 147
 - multiple shots on the SDI monitor 149
 - selected animated channels 407
 - selected keyframes 407
 - shot information 97
- viewing LUTs
 - rendering with 454
 - selecting 137
- VTR
 - capturing from 202
 - controlling for video I/O 199
 - emulation 227
 - playing out to 211

W

- WAV files, importing and offsetting 40
- white level, modifying for input LUTs 270
- Wiretap
 - accessing footage on the Stone filesystem 100
 - and EDL files, overview 193
 - detecting servers 103
 - generating proxies locally 63
 - grading shots through 193
 - linking to media on StoneFS via 103
 - playing clips located on StoneFS 104

- rendering 104
- rendering options 447
- setting up projects for footage accessed via 64
- setting up projects for soft-imported clips 70
- supported image files from StoneFS 104
- wt.config file 504
- workflows
 - creating animations 408
 - EDL and keycode assembly 118
 - enhancements to writing to Telecine-style tape for workflows using NTSC 221
 - integrating Lustre into post-production 101
 - Lustre general 7
 - monitor calibration 48
 - playout 212
 - processing pipeline 8
 - project setup 60
 - working with scenes 74
- X
- XML
 - CUT file data types 517
 - CUT file example 518
 - limitations 507
 - modifying CUT files 508
 - viewing and editing files 507
- Z
- zooming
 - animation curves 406
 - in Player and full-screen Player 133
 - with the calculator 30

